## Contents

### Chapter 1. Takeoff Module

<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>TakeOff Menu</td>
<td>2</td>
</tr>
<tr>
<td>Define Layer Target/Material/Subgrade</td>
<td>2</td>
</tr>
<tr>
<td>Edit Selected Layer</td>
<td>7</td>
</tr>
<tr>
<td>Set Layer For Existing</td>
<td>7</td>
</tr>
<tr>
<td>Set Layer For Design</td>
<td>7</td>
</tr>
<tr>
<td>Set Layer For Other</td>
<td>8</td>
</tr>
<tr>
<td>Boundary Polyline</td>
<td>8</td>
</tr>
<tr>
<td>Areas Of Interest</td>
<td>10</td>
</tr>
<tr>
<td>Hatch Subgrade Areas</td>
<td>12</td>
</tr>
<tr>
<td>Erase Subgrade Hatches</td>
<td>14</td>
</tr>
<tr>
<td>Draw Subgrade Hatch Legend</td>
<td>14</td>
</tr>
<tr>
<td>Label Subgrade Areas</td>
<td>15</td>
</tr>
<tr>
<td>Erase Subgrade Labels</td>
<td>15</td>
</tr>
<tr>
<td>Topsoil Removal and Replacement</td>
<td>16</td>
</tr>
<tr>
<td>Make Existing Ground Surface</td>
<td>20</td>
</tr>
<tr>
<td>Make Design Surface</td>
<td>23</td>
</tr>
<tr>
<td>Make Overexcavate Surface From Strata</td>
<td>24</td>
</tr>
<tr>
<td>Make Overexcavate Surface From Screen Entities</td>
<td>24</td>
</tr>
<tr>
<td>Make Overexcavate Surface From Existing/Design Surfaces</td>
<td>25</td>
</tr>
<tr>
<td>Adjust Overexcavate Surface</td>
<td>25</td>
</tr>
<tr>
<td>Draw Overexcavate Surface 3D Faces</td>
<td>26</td>
</tr>
<tr>
<td>Erase Overexcavate Surface 3D Faces</td>
<td>26</td>
</tr>
<tr>
<td>Draw Overexcavate Cut Color Map</td>
<td>27</td>
</tr>
</tbody>
</table>
Change Elevations ........................................ 72
Set Polyline to Elevation ................................. 73
Edit-Assign Polyline Elevations ........................... 74
2D to 3D By Surface Model ............................... 75
2D to 3D Polyline by Points .............................. 77
2D to 3D Polyline-By Text ............................... 78
2D to 3D By Text With Leader ............................ 80
2D to 3D Polyline by Start-End Elevations ............... 83
Draw Building Envelope Polyline ......................... 83
Pad Polyline By Interior Text ............................ 85
Convert Spot Elev To Points ............................ 88
Assign Contour Elevation - Multiple in Series ........... 91
Assign Contour Elevation - From Contour Labels ......... 93
Assign Contour Elevation - Single Elevation Group ....... 94
Drape 3D Polyline On Surface ........................... 95
Edit Polyline Vertex .................................... 96
Edit Contours ............................................ 97

Digitize Menu ............................................. 99
Tablet On .................................................. 99
Tablet Off ............................................... 99
Tablet Calibrate .......................................... 99
Digitizer Setup .......................................... 101
Save Tablet Calibration .................................. 103
Load Tablet Calibration .................................. 104
Digitizer Settings ........................................ 104
Digitize Existing ......................................... 106
Digitize Design .......................................... 107
Digitize Other .......................................... 107

Contents

iii
Digitize Point ................................ 107
Digitize Spot Elevation .............................. 108
Digitize 2D Polyline .................................. 110
Digitize 3D Polyline .................................. 112
Digitize Perimeter ..................................... 113
Digitize Areas .......................................... 114
Digitize Contour Polyline ............................ 115
Digitize Sections ....................................... 118
Digitize End Areas ..................................... 121

Drillhole Menu ........................................... 124
Drillhole Strata Settings ............................... 124
Drillhole Import ......................................... 127
Place Drillhole .......................................... 130
Edit Drillhole ........................................... 132
Drillhole Reports ........................................ 135
Make Strata Surface .................................... 137
Clear Strata Surface .................................... 137
Draw Strata Cut Depth Contours ....................... 137
Erase Strata Cut Depth Contours ....................... 138
Draw Strata Cut Color Map ............................ 139
Erase Strata Cut Color Map ............................ 139
Draw Strata Surface .................................... 140
Erase Strata Surface .................................... 140

Trench Menu .............................................. 141
Input Trench From Polyline ........................... 141
Create Trench Network Structure ...................... 143
Edit Trench Network Structure ....................... 146
Remove Trench Network Structure .................... 147
Field Menu .................................................. 185
  Configure Field ........................................... 186
  Equipment Setup .......................................... 199
  Align GPS To Local Coordinates ......................... 226
  Point Store .................................................. 231
  StakeOut .................................................... 240
  Auto Points at Interval .................................... 245
  Track Position ............................................. 247
  Satellite SkyPlot .......................................... 248
  Monitor GPS Position ..................................... 249
  Benchmark ................................................... 250
  Resection ................................................... 251
  Building Face Surface .................................... 252
  Pattern Point Survey ..................................... 253
  Point Check By Robotics .................................. 253
  Carlson Field Icon Menu .................................. 253
  Typical Alignment Scenarios ............................... 255
Surface Menu .................................................. 256
  Elevation Difference ....................................... 256
  Prepare Story Stake ....................................... 259
  Story Stake By Points ..................................... 261
  Story Stake Along Polyline ............................... 262
Roads Menu .................................................... 264
  Centerline Position ....................................... 264
  Offset Stakeout ........................................... 267
  Slope Staking .............................................. 272
  Slope Inspector ............................................ 278
  GIS Menu ..................................................... 279
<table>
<thead>
<tr>
<th>Equipment Menu</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Apache Lightbar</td>
<td>280</td>
</tr>
<tr>
<td>CSI GBX Pro</td>
<td>280</td>
</tr>
<tr>
<td>Depth Sounder</td>
<td>281</td>
</tr>
<tr>
<td>Geodimeter</td>
<td>283</td>
</tr>
<tr>
<td>InnerSpace Tech depth sounder</td>
<td>285</td>
</tr>
<tr>
<td>Laser Atlanta</td>
<td>285</td>
</tr>
<tr>
<td>Leica Disto</td>
<td>286</td>
</tr>
<tr>
<td>Leica GPS System 500</td>
<td>286</td>
</tr>
<tr>
<td>Leica TC Series</td>
<td>288</td>
</tr>
<tr>
<td>Manual Total Station</td>
<td>289</td>
</tr>
<tr>
<td>Mikrofyn Lightbar</td>
<td>290</td>
</tr>
<tr>
<td>Navcom Configuration Guide</td>
<td>291</td>
</tr>
<tr>
<td>Navcom GPS Setup</td>
<td>293</td>
</tr>
<tr>
<td>Nikon Total Stations</td>
<td>298</td>
</tr>
<tr>
<td>OmniStar Otto</td>
<td>299</td>
</tr>
<tr>
<td>Simulation GPS</td>
<td>299</td>
</tr>
<tr>
<td>Sokkia</td>
<td>300</td>
</tr>
<tr>
<td>Topcon Total Stations</td>
<td>303</td>
</tr>
<tr>
<td>Trimble</td>
<td>306</td>
</tr>
</tbody>
</table>
Takeoff Module
Define Layer Target/Material/Subgrade

Function

The Define Layer Targets dialog box (shown here) offers many functions that will ultimately make up the surface models used in volume and material calculations. Every entity (line, polyline, point, etc) in a drawing is assigned a layer name. Carlson Takeoff uses the entity layer names to define which entities represent the existing ground surface, the design surface or no surface. These surfaces are referred to as the "Target" surfaces. The drawing entities are assigned their target surface by their layer name. For example, if polylines representing design contours are on the layer "Final", then "Final" will be set as a layer for the design surface. For layers of entities that are for neither existing nor design surfaces (such as text labels for street names), the layer target is set to Other. The Define Layer Targets dialog has three lists for layer targets: Existing, Design and Other. To switch between lists, pick the tabs at the top of the dialog. To move a layer to a target destination, highlight the desired layer, choose the target from the Move To list and pick the "Move To" button. All layers populate the "Other" target before being assigned to "Existing" or "Design".
Besides the basic three layer targets (Existing, Design and Other), you can add more target surfaces with the Add Target button. When another target is defined, there will be another tab along the top of the Define Layer Targets dialog. Then layers can be assigned to this additional target surface. The only pre-defined additional surface is Overexcavate. The layers that are assigned to the Overexcavate target can be modeled into the Overexcavate surface using the Make Overexcavate Surface command. Besides Overexcavate, the other additional targets are user-defined. The layer targets can be modeled using the Make User-Defined Surface command. Then these surfaces can be used in Takeoff commands by assigning them to a Takeoff existing or design surface using the Set Active Surfaces command.

Edit Materials

The "Edit" button activates the Edit Material dialog box (shown here) and allows you to define the Material name and Subgrade depths and names. Besides assigning target surfaces by layer, layers are also used to define material names and subgrades depths. By assigning a material name, Subgrade names and depths to layers, the volume, area, length and count for entities on these layers can be reported. Also the depth is used to vertically adjust the design surface. The polylines on the layer used for a Materialmust be closed polylines. Carlson Takeoff supports nested Subgrade polylines for exclusion areas such as islands by counting how many Subgrade polylines surround an area. If the number is odd, then the area is included in the Subgrade. The even count regions in the area are not part of the Subgrade. To activate the Edit Material, select a layer from the list and then choose "Edit".
Include in Material Quantities Report

With this option checked on, the material that is named will appear in the Material Quantities Report. The report will include either the area of the material, the linear length of the material, or the number of items counted on the layer defining the material. This is accomplished by choosing "Area", "Linear", or "Count" for the Material Type.

Set Color For 3D Drive

This options checked allows you to assign a color for this particular material for display purposes during the 3D view/drive simulator.

Material Type

This will report the subgrade by area, linear length, count, or as curb area. If you choose Back of Curb/Pavement then you can pick on the Curb Dimensions button and bring you to the below
This option will calculate your curb volume as well as act as the limit of the pavement. The pavement limit will be from the Back of Curb polyline offset by the length of the Curb base. In the above case the base is 30 inches wide. Therefore, the pavement area will stop 30 inches before the Back of Curb polyline.

**Material Cost Per Cost Unit**

Use this field to add the value of the multiplier for the unit cost of your material. If the material type is an area that has multiple subgrades, use the available fields below to add each individual subgrade name, depth and cost value per unit type. If a linear or count type material type option is selected, use the "length in feet", or the "count" unit options.

**Adjust Design Surface by Depth**

This determines whether the subgrade depths are incorporated in the design surface or not.
Area Subgrades

Depth Units

Select the "feet" or "inches" as the unit value desired for depth of subgrades.

Subgrade Name Depth Shrink Cost Per Cost Unit Density

Use these options for areas that are represented with a single/multiple closed polygon/polygons in the drawing, but have multiple material types defining the surface. Simply name each "lift" in the area, issue a depth value and add a cost unit if desired, or click on select and choose a material from the Materials Library (see Define Materials Library for more). Carlson Takeoff will report each subgrade material value in the material quantities report. The Shrink factor is multiplied by the subgrade volume in the material quantities report and represents the fill shrinkage. A Density factor can be entered in when using Cost Per Tons.

If user entered values are needed in the report use the "Edit User-Fields" button to activate the "User Defined Features" dialog box shown here. Choose the "Add" button to define needed fields such as TONS of material or BAGS OF GRASS SEED for reporting options.

Once all of the material subgrades, depths and cost units or user defined units have been defined, select save to preserve the settings in a .trg file, the "save as" function allows the user to name the
file to load later.

**Prerequisite:** none  
**Keyboard Command:** define\_tk\_layers

---

**Edit Selected Layer**

**Function**

Use this command to click on any layer and advance to the Edit Materials dialog from the Define Layer Target/Material/Subgrade command.

**Prerequisite:** none  
**Keyboard Command:** edit\_tk\_layer

---

**Set Layer For Existing**

**Function**

Set Layer For Existing allows the user to pick the layers from objects on the screen and assign them to the Existing Layer.

**Prerequisite:** none  
**Keyboard Command:** set\_existing\_layer

---

**Set Layer For Design**

**Function**

Set Layer For Design allows the user to pick the layers from objects on the screen and assign them to the Design Layer.

**Prerequisite:** none  
**Keyboard Command:** set\_design\_layer
Set Layer For Other

**Function**

Set Layer For Other allows the user to pick the layers from objects on the screen and assign them to the Other Layer.

**Prerequisite:** none  
**Keyboard Command:** set_other_layer

Boundary Polyline

**Function**

The Boundary Polyline options allow the user to Set the Boundary Polyline, Set the Exclusion Polylines, Clear Exclusion Polylines, Hatch the Boundary Area, Erase the Boundary Hatched area.

**Set Boundary Polyline**

Use this command to select the "CLOSED" polyline that defines the outer most limit of the disturbed area. This boundary should dissect the site at the point where the design contours meet the existing contours, or where the limit of work will occur. If your site contains seperated areas (such as different phases or isolated sections of work), then multiple Boundry Polylines can be used. Volume calculation will take place inside this boundary.

**Prerequisite:** a closed polyline  
**Keyboard Command:** tag_inclu

**Set Exclusion Polylines**

Use this command to select the "CLOSED" polylines the define the areas inside the Boundary Polyline that will not be disturbed. These boundarys should also be at the intersection of the proposed and existing surface. A pond or wetland that will not be removed during construction is a good example of an Exclusion Area.
Prerequisite: a closed polyline
Keyboard Command: tag_exclu

Clear Exclusion Polyline

Use this command to select polylines that were previously defined as exclusion polylines but are no longer needed as exclusion areas.

Prerequisite:
Keyboard Command: untag_exclu

Highlight Boundary Polyline

This command highlights the polyline you set as the Boundary Polyline.

Prerequisite: a boundary polyline
Keyboard Command: highlight_boundary

Hatch Boundary Area

Use this command to confirm the boundary polylines that have been selected are correct. This hatched area can also be utilized in exhibits of the drawing.

Prerequisite: a boundary polyline
Keyboard Command: hatch_boundary

Erase Boundary Hatch

This command erases the hatch drawn in the plan view.

Prerequisite: a boundary hatch
Keyboard Command: erase_boundary
Areas Of Interest

Function

Areas of Interest can be used to calculate volumes and material quantities within a specified area. The Area Of Interest perimeters are defined by user-selected closed polylines and each area is assigned a name. The Area Of Interest polylines can be assigned either as inclusion or exclusion perimeters for the area. You can have any number of exclusion perimeters within an inclusion but inclusion perimeters cannot be inside exclusions.

The Areas Of Interest (AOI) commands allow you to Tag/Untag Areas of Interest, Identify/Report Areas of Interest and Hatch/Label Areas of Interest.

Tag Area Of Interest

This command allows the user to select polylines and exclusion perimeters that define phases of a project. Carlson Takeoff will separate each area of interest in the volume and material reports.

Prerequisite: a desired polyline
Keyboard Command: tag_aoi

Area Of Interest by Interior Text

This command allows the user to select text from the screen to name AOIs and linework to determine the area.

Prerequisite: area linework and text
Keyboard Command: txt2aoi

Untag Area Of Interest

This command allows the user to remove previously tagged areas.

Prerequisite: an area of interest
Keyboard Command: untag_aoi
Identify Area Of Interest

This command allows users to identify AOI by either picking on a polyline(s) or by searching the entire drawing. The command will then report the AOI name, layer, type, starting point, and highlight the polyline in the plan view.

**Prerequisite:** an area of interest  
**Keyboard Command:** id_aoi

Report Area Of Interest Areas

Use this command to report the Inclusion or Exclusion area (SF), the name, the layer, and the starting point.

**Prerequisite:** an area of interest  
**Keyboard Command:** report_aoi

Hatch Area Of Interest Areas

This command allows the user to visually see AOIs in the plain view.

**Prerequisite:** an area of interest  
**Keyboard Command:** hatch_aoi

Erase Area Of Interest Hatch

This command erases AOI hatching.

**Prerequisite:** hatched area of interest  
**Keyboard Command:** erase_aoi_hatch

Label Area Of Interest Areas
This command labels the AOI name and area in the plain view.

**Prerequisite:** an area of interest  
**Keyboard Command:** label_aoi

### Erase Area Of Interest Labels

This command erases AOI labeling.

**Keyboard Command:** erase_aoi_labels  
**Prerequisite:** hatched area of interest

### Hatch Subgrade Areas

#### Function

This command draws a hatch with a specified color and pattern for the area that the selected subgrade area applies to. The purpose is to allow you to visually review a subgrade area to make sure that the area coverage is correct.

The command displays a dialog to select which subgrade to hatch. The list of available subgrades comes from the layers with subgrade depths as set in the Define Layer Target/Material/Subgrade command. Then there is a dialog for the hatch pattern, color and scale. The scale determines how spread out the pattern is within the hatch. The Automatic Hatch Scale option checks the size of the subgrade areas and sets the scale to make the pattern fit.

The resulting hatch areas show where the subgrade is applied. In the example below, notice how the islands are not hatched because they are curb polylines that are already inside another curb polyline. Also note that the smaller pad area is not hatched because this polyline layer is different than the bigger pad polyline.
Prompts
**Keyboard Command:** hatch_subgrade  
**Prerequisite:** subgrades

---

**Erase Subgrade Hatches**

**Function**

This command removes from the screen the subgrade hatches created by the command Hatch Subgrade Area.

**Keyboard Command:** erase_subgrade  
**Prerequisite:** hatch subgrade areas

---

**Draw Subgrade Hatch Legend**

**Function**

This command draws a legend for the subgrade areas currently in the drawing. The legend includes the subgrade names and squares of the hatch patterns. The size of the labels, size of the hatch squares, layer for the legend entities and the legend title are set in the dialog shown below. The subgrade hatches to include in the legend are automatically selected from all the subgrade hatches currently in the drawing that were created by the Hatch Subgrade Areas command.
Prompts

Keyboard Command: draw_subgrade_legend
Prerequisite: hatched subgrade areas

Label Subgrade Areas

Function

This command lets you label the subgrade depth and area (in sq. ft. or meters). The label is placed at the center of the subgrade area, but can be moved with the Move command under Edit.

Pull-Down Menu Location: Inquiry-> Subgrade Areas
Keyboard Command: label_subgrade
Prerequisite: Subgrade Areas

Erase Subgrade Labels

Function

This command erases subgrade labels.

Pull-Down Menu Location: Inquiry-> Subgrade Areas
Keyboard Command: erase_subgrade_labels
Prerequisite: subgrade labels
Topsoil Removal and Replacement

Function

The Topsoil Removal and Replacement options (shown here) allow the user to Define Topsoil removal and replacement depths, Set topsoil removal and replacement areas by selecting closed polylines, Clear the selected boundary polylines if needed, Hatch the topsoil removal and replacement areas and Erase the hatched areas.

Define Topsoil Depths

This command requires user input to define the depth, or strata, of topsoil removal and replacement. Fill in the options available in the Define Topsoil Depths dialog (shown here). Carlson Takeoff will preform four functions with these values. First, the value set for the Removal Depth, or the Top Strata if selected, will be the "defined" removal amount from the Existing Ground Surface. Second, the calculated volume of topsoil removed will be included in the reporting.
options. Third, the value set for the Replacement Depth will be added "BELOW" the Finished Ground Surface model. Fourth, the amount of topsoil replaced will be included in the reporting options.

When topsoil depths are defined, the volume report routines will include the topsoil quantities. These topsoil quantities are in addition to the cut/fill for the existing to design surfaces for the site.

The Removal Swell Factor and Replacement Shrink Factor are multiplied by the topsoil removal and replacement quantities respectively in the volume report routines. The Density is used to report topsoil tons when the volume report option for tons is active.

The Topsoil Offset Method choose between offseting the topsoil depth vertically or perpendicular to the surface. The perpendicular method will result in more topsoil quantities since it represents applying the topsoil depth to the slope area of the surface whereas the vertical method represents applying the topsoil depth to the horizontal area.

**Prerequisite:** topsoil depths

**Keyboard Command:** define_topsoil

**Identify Topsoil Polylines**

This command allows users to identify topsoil polylines by either picking on a polyline(s) or by searching the entire drawing. The command will then report the layer name and starting point for
both removal and replacement polylines. These polylines are also highlighted in the plain view.

**Prerequisite:** topsoil polylines  
**Keyboard Command:** id_topsoil

### Report Topsoil Areas

Use this command to report the Inclusion or Exclusion area (SF), the type, the depth, the layer, and the starting point.

**Prerequisite:** topsoil areas  
**Keyboard Command:** report_topsoil

### Label Area Of Interest Areas

This command labels the topsoil type and area in the plain view.

**Prerequisite:** topsoil area  
**Keyboard Command:** label_topsoil

### Erase Area Of Interest Labels

This command erases topsoil labeling.

**Prerequisite:** hatched area of interest  
**Keyboard Command:** erase_topsoil_labels

### Set Topsoil Removal Polylines

Use this command to select the "CLOSED" polylines defining the extents of topsoil removal and any "CLOSED" interiors that will not have topsoil removed. The outer boundary usually is the same polyline that defined the "Boundary Polyline" selected earlier. The internal polylines usually are those that have been selected as the "Exclusion Polylines". The layer names for these boundaries is irrelevant. You will be prompted to use the Removal Depth defined in the Define Topsoil Depths command or to customize your depth.
Prerequisite: polylines for removal
Keyboard Command: tag_topsoil_remove

Clear Topsoil Removal Polylines
This command allows the user to remove and previously selected Topsoil Removal Polyline boundaries.

Prerequisite: topsoil polylines
Keyboard Command: untag_topsoil_remove

Hatch Topsoil Removal Area
Use this command to display a hatch pattern over the entire area designated for topsoil removal.

Prerequisite: topsoil areas
Keyboard Command: hatch_topsoil_remove

Erase Topsoil Removal Hatch
Use this command to remove the hatch pattern that defined the topsoil removal area.

Prerequisite: hatched topsoil
Keyboard Command: erase_topsoil_remove

Set Topsoil Replacement Polylines
Use this command to select the "CLOSED" polyline boundary defining the extents of topsoil replacement, and any "CLOSED" interior polylines that will not have topsoil replaced. The layer names for these boundaries is irrelevant. You will be prompted to use the Topsoil Replacement amount defined in the Define Topsoil Depths command or to customize your amount.

Prerequisite: polylines for replacement
Keyboard Command: tag_topsoil_replace
Clear Topsoil Replacement Polylines

This command allows the user to remove and previously selected Topsoil Replacement Polyline boundaries.

**Prerequisite:** topsoil polylines

**Keyboard Command:** untag_topsoil_replace

Hatch Topsoil Replacement Area

Use this command to display a hatch pattern over the entire area designated for topsoil replacement.

**Prerequisite:** topsoil areas

**Keyboard Command:** hatch_topsoil_replace

Erase Topsoil Replacement Hatch

Use this command to remove the hatch pattern that defined the topsoil replacement area.

**Prerequisite:** hatched topsoil

**Keyboard Command:** erase_topsoil_replace

Make Existing Ground Surface

**Function**

This command makes the triangulation models for the existing ground surface. There are three surfaces that are created: initial original ground, original ground after applying subgrade zones, and original ground after subgrade zones and topsoil removal. These surface files are automatically named as "filename-og.tin", "filename-ze.tin" and "filename-ex.tin" respectively. The "filename" is set to the name of the current drawing (dwg) file. Also, the file extension will be .tin for the binary format triangulation and .flt for the ASCII format triangulation. This file format is set in Configure->Takeoff. The subgrade zones are defined in the Define Layer Target/Material/Subgrade command. If there aren't any subgrade zones for the Existing surface, then the original ground after subgrades surface will be the same as the initial original ground surface. The topsoil removal depths and areas are set with the commands in the Topsoil Removal/Replacement sub-menu. The topsoil removal areas will lower the ground surface by the topsoil depth. If there aren't any topsoil removal areas, then the original ground after subgrade
and topsoil surface will be the same as the original ground after subgrade surface.

Before running this command, the layer names for the entities on the Existing layer target must be set in the Define Layer Target/Material/Subgrade command. Also these entities must be at their proper elevations. The entity elevations can be reviewed using commands from the Inquiry menu and the elevations can be assigned if needed using command from the Elevate menu. Another prerequisite is that the Boundary Polyline must be set for the site. If the boundary has not been set, the following error message will appear.

![Carlson TakeOff Message](image)

If this error message appears, run the "Set Boundary Polyline" command and pick the CLOSED polyline representing the boundary of the site.

When the program finds errors in the existing entities, a Data Error Log dialog reports these errors. Three types of conflicts are reported: Crossing Breaklines, Vertical Edges, and Breakline T-Intersections. Crossing Breaklines indicates that the intersection of two entities does not have a common elevation. Vertical Edges indicates that two entities or vertices of differing elevations have the same x-y location, thus forming a vertical plane. Breakline T-Intersections indicates that a 3d entity is abutting another entity, but the second entity doesn't have a vertex at the point of intersection. Each type of conflict is listed in its own category.

The Data Error Log shows the amount of elevation difference at each error. You can use the Data Error Log to review, report and draw markers at these error locations. Then you can exit the Data Error Log and fix the data errors with the commands in the Elevate menu or other drafting tools. After these errors are fixed, you can run Make Existing Ground Surface again.
Clicking to the "plus" sign beside a category will display the individual conflicts within that category. When a line item error is selected, a highlighted arrow is temporarily placed in the drawing to indicate the exact location of the specific conflict. Zoom functionality allows the user to more closely inspect the specific problem area, and if needed a marker can be drawn or a report generated for an individual conflict or conflicts.

**Zoom To** pans the drawing to move the selected conflict to the center of the screen. The zoom functions are only active when a single line item is selected.

**Zoom In** zooms in on the highlighted area for closer inspection. Multiple picks on the zoom button will increase the magnification.

**Zoom Out** zooms out away from the highlighted area.

**Report All/One** toggles between One and All depending whether a single line item conflict or a category is selected from the error log. An error report is generated listing the x-y position and the elevation difference of the entities in conflict.

**Draw All/One** toggles between One and All depending whether a single conflict or a category is selected from the list. This option draws an "X" symbol at each selected conflict. The layer and size of the symbol is controlled in the fields below.

**Continue** closes the Error Log and proceeds with the contouring operation.
Settings has controls for the tolerances for error reporting and for the Layer Name and Symbol Size to use with the Draw function.

**Keyboard Command:** mk_exist_tin  
**Prerequisite:** a boundary polyline and elevated entities on the Existing layer target

---

### Make Design Surface

#### Function

This command makes the triangulation models for the design surface. There are three surfaces that are created: initial design, design after applying subgrade zones, and design after subgrade zones and topsoil replacement. These surface files are automatically named as "filename-bs.tin", "filename-zn.tin" and "filename-fn.tin" respectively. The "filename" is set to the name of the current drawing (dwg) file. Also, the file extension will be .tin for the binary format triangulation and .flt for the ASCII format triangulation. This file format is set in Configure->Takeoff. The subgrade zones are defined in the Define Layer Target/Material/Subgrade command. If there aren't any subgrade zones for the Design surface, then the design after subgrades surface will be the same as the initial design surface. The topsoil replacement depths and areas are set with the commands in the Topsoil Removal/Replacement sub-menu. The topsoil replacement areas will lower the design surface by the topsoil depth to leave room for the topsoil replacement. If there aren't any topsoil replacement areas, then the design after subgrade and topsoil surface will be the same as the design after subgrade surface.

Before running this command, the layer names for the entities on the Design layer target must be set in the Define Layer Target/Material/Subgrade command. Also these entities must be at their proper elevations. The entity elevations can be reviewed using commands from the Inquiry menu and the elevations can be assigned if needed using command from the Elevate menu. Another prerequisite is that the Boundary Polyline must be set for the site.

When the program finds errors in the existing entities, a Data Error Log dialog reports these errors. Refer to the Make Existing Surface command for more information on the Data Error Log dialog.

**Keyboard Command:** mk_final_tin  
**Prerequisite:** a boundary polyline and elevated entities on the Existing layer target
Make Overexcavate Surface From Strata

Function

This command sets the Overexcavate surface to a selected strata surface. Before running this command, the strata surface must be created with the Make Strata Surfaces command in the Drillhole menu. The resulting overexcavate surface is stored in a triangulation file that is named with "-ox" appended to the current drawing name.

Prerequisite: Strata surfaces
Keyboard Command: overx_by_strata

Make Overexcavate Surface From Screen Entities

Function

This command makes the overexcavate surface from entities on the layers defined as Overexcavate in the Define Layer Target/Material/Subgrade command. The resulting surface of Make Overexcavate Surface is stored in a triangulation file that is named with "-ox" appended to the current drawing name.

Prerequisite: overexcavate entities
Keyboard Command: mk_overx_tin

Make Overexcavate Surface From Existing/Design Surfaces

Function

The Initialize Overexcavation Surfaced dialog box shown here allows the user to select which surface model to overexcavate and to enter in the depth value for the desired adjustment. Use the Min Existing/Design option to set the overexcavate as the minimum of the existing and design surfaces. If a single surface is selected the value entered will be applied to that surface only. The resulting surface of Make Overexcavate Surface is stored in a triangulation file that is named with "-ox" appended to the current drawing name.

Prerequisite: Existing and/or Design surfaces
Keyboard Command: set_overx

Adjust Overexcavate Surface

Function
This command adjusts the overexcavate surface vertically within the selected perimeter polylines. This command allows the site to be overexcavated at a variety of depths in specified areas represented with CLOSED polyline boundaries. Select the desired areas to be adjusted when prompted at the command line.

**Keyboard Command:** adjust_overx  
**Prerequisite:** an overexcavate surface

---

## Draw Overexcavate Surface 3D Faces

### Function

Use this command to draw the 3D faces of the overexcavated surface model on the screen. The 3D faces will be drawn in the TK_OVERX_SURFACE layer and will depend on the latest surface created using the make and adjust routines.

**Prerequisite:** An overexcavate surface  
**Keyboard Command:** draw_overx

---

## Erase Overexcavate Surface 3D Faces

### Function

Use this command to remove the previously drawn 3D Faces from the screen.

**Prerequisite:** 3D Faces  
**Keyboard Command:** erase_overx
Draw Overexcavate Cut Color Map

Function

Use this command to display a cut color map on the screen that shows the areas of overexcavate cut. The colors will graduate from white to red based on zero cut depth to maximum cut depth. This command also offers the user to place a legend of the cut depths on the screen. Pick the desired location and type the desired scale of the legend when prompted at the command line.

**Prerequisite:** An overexcavate surface

**Keyboard Command:** overx_cfmap

Erase Overexcavate Cut Color Map

Function

Use this command to remove the previously drawn Cut Color Map and Legend from the screen.

**Prerequisite:** An overexcavate cut color map

**Keyboard Command:** overx_cfmap2

Clear Overexcavate Surface

Function

Use this command to remove the overexcavate surface. When the overexcavate surface is removed, the rest of the Takeoff commands will not calculate overexcavate volumes. You will be prompted to confirm before the remove is done.

**Prerequisite:** An overexcavate surface
**Keyboard Command:** clear_overx

## Surface Manager

### Function

This command allows the user to name and manage multiple surface models. The Surface Manager dialog shown here has options to name and save the current "existing and design" surface models. The "current" surface is dictated by the layers that populate a target and the Make Surface command. If layers are removed from a target, and others assigned, multiple surfaces can be created and stored. Highlight a named surface and select the Set Current From List option to make that model active. Use the Remove From List option to remove a named surface model from the list.

![Surface Manager Dialog](image)

Selecting the Save Current To List options brings up the Surface Name dialog box shown here. Type the desired name that describes a particular surface model and select OK.
Prerequisite: none
Keyboard Command: surf_mgr

Make User Defined Surface

Function

This command makes a surface from the entities on the layers defined as user-defined targets in the Define Layer Target/Material/Subgrade command. The purpose of user-defined surfaces is for modeling surfaces besides existing ground and design. The drawing needs to contain entities that represent the elevations of the user-defined surface. For example, the user-defined surface could be for alluvial soil and the drawing has contour polylines for this surface.

There is a dialog to select which surface to make. The surface is stored in a triangulation file that is named after the current drawing name with the user-defined surface name appended.

This user-defined surface can be applied to Takeoff routines by running the Set Active Surfaces command.

Prerequisite: Define Layer Target/Material/Subgrade command
Keyboard Command: mk_user_tin
Set Active Surfaces

Function

This command assigns which surfaces to use for initial and final. These surfaces are used by all the Takeoff routine that compare surfaces including:
- Calculate Total Volumes
- Calculate Volumes Inside Perimeter
- 3D Drive Simulation
- Cut/Fill Contours/Labels/Color Map
- Surface Inspector
- Quick Profile
- etc.

The surface created by the Make Existing Ground Surface command is called "Existing" and is the default for the Initial Surface. The surface created by the Make Design Surface command is called "Design" and is the default for the Final Surface.

![Set Active Surfaces dialog box](image)

The purpose of this routine to for selecting user-defined surfaces to use in place of the existing ground or the design surface. For example, there could be a user-defined surface for alluvial soil that is set as the initial surface while design is set to the final surface. Then the calculate volume routines will report the quantities between alluvial soil and design. Also the Display->Cut/Fill Color Map routine will make the map for the difference between the alluvial soil and design surfaces.

These user-defined surfaces can be created using the Add Target function in the Define Layer Target/Material/Subgrade command combined with the Make User-Defined Surface command.
Prerequisite: a surface model

Keyboard Command: set_active_tins

Design Surface Vertical Offset

This command can be used to lower or raise the design surface within a defined perimeter or by the entire surface.

Prerequisite: a design surface

Keyboard Command: adjust_final

Existing Surface Vertical Offset

This command can be used to lower or raise the existing surface within a defined perimeter or by the entire surface.

Prerequisite: an existing surface

Keyboard Command: adjust_exist

Merge Existing With Design

This command allows you to merge the existing surface with design surface within perimeter polylines. The resulting merged surface can be saved to update either the Existing or Design surfaces. The program prompts for inclusion and exclusion perimeter polylines. These polylines must be closed. The merge will be applied inside the inclusion perimeters and not inside the exclusion perimeters. The exclusion perimeters are optional.

For example, if a portion of the site is completed, you can update the existing surface to match the design for the completed area. First, draw a closed polyline around the completed area. Then run Merge Existing With Design and choose the merge results target as Existing. Then select the perimeter polyline.
Prerequisite: existing and design surfaces and an inclusion perimeter polyline

Keyboard Command: merge_final

Calculate Total Volumes

Function

Use this command to report total volume calculations within the site boundary polyline. The report includes the cut and fill quantities as well as the strata and topsoil quantities if the site has strata and topsoil defined. Besides reporting the total quantities for the site boundary, Area Of Interest polylines can be used to report quantities within named perimeters.

Before running this command, the existing and design surfaces must be created and the boundary polyline must be assigned. Also, the strata surfaces, topsoil and Area Of Interest polylines need to be set before this command if those features are to be reported.

The Volume Options dialog box shown here offers options for the final report. Here you can select four different types of reports: Standard Report Viewer, Custom Report Formatter, Expanded Auto Format, and Compressed Auto Format. The Cut Swell Factor is multiplied by the cut volume and the Fill Shrink Factor is multiplied by the fill volume. The Report Units setting chooses between English and Metric quantities for the report. In Drawing Setup in Takeoff, you set the drawing units as either English or Metric. The Report Units will default to match the drawing units but you can change the Report Units to the other mode and the program will apply the conversion between English and Metric for the report. So you can have a drawing in English units and create a report with Metric quantities.

Note: As the quantities are calculated within each area, the area is hatched with a solid fill as a visual verification that the right area is being processed.
Shown here is an example of a Standard Report Viewer.

Use Customs Report Formatter to customize or "user define" the reporting options. The Report

Chapter 1. Takeoff Module
Formatter Options dialog box shown here offers a variety of output options including Excel. You can choose the fields to report from the Available list and set their report order under the Used list.

The Expanded Auto Format is shown in this report preview. You can also send the report to MS Word in this command.
The Compressed Auto Format puts all the area information on one row.
If drillholes have been located on the drawing and strata types and depths have been defined, a calculate Strata Depth Zones Volume option becomes available as shown here.

![Volume Options](image)

Shown here is an example of the report if strata depth intervals have been defined.
The Balance Cut/Fill option shown here allows an import or export volume in cubic yards option. Use these options if waste material is available or needed elsewhere. If this option is used the resulting report indicates the vertical movement of the site needed to satisfy the balance option.
Shown here is a report with a 500 CY importation of material and suggests that the site be vertically raised 0.859 feet.

If the adjusted surface is satisfactory, Carlson Takeoff offers the option to save the adjusted surface as shown here in the Balance Cut/Fill dialog box.

**Prerequisite:** Existing and design surfaces and a boundary polyline  
**Keyboard Command:** tin_volume

**Calculate Volumes Inside Perimeter**

**Function**
Use this command to create volume reports inside the selected closed perimeter polyline. The same reporting options are available for this command as are for the Calculate Total Volumes command.

**Keyboard Command:** `tin_volume2`  
**Prerequisite:** Existing and Design surfaces and a closed perimeter polyline

---

### Material Quantities

#### Function

The Material Quantities flyout shown here offers many options for quantity reporting including the option for user defined attributes. Entities with attributes can be drawn, edited, and identified. Standard and custom report options are also available.

Material Quantities are counted from the entities in the drawing. Several entity properties can be reported including entity count, length, area and volume. Also user-defined attributes can be assigned to the entities and reported. The type of material for each entity is determined by the layer for the entity. In the Define Layer Target/Material/Subgrade, you can assign the material types by layer.
Standard Report

Use this command to display all or a selected set of material quantities and user-defined information with the standard Carlson Takeoff report format shown here.
Prerequisite: Defined materials
Keyboard Command: materials_report2

Custom Report

Use this command to customize or "user define" the reporting options. This command first prompts whether to report quantities for all the entities in the drawing or selected entities. Then if the drawing contains Areas Of Interest polylines, there is an option report quantities by these areas which adds the area name to the available report fields to allow sorting and grouping by area name. The Report Formatter Options dialog box shown here offers a variety of output options. You can choose the fields to report from the Available list and put them in report order under the Used list.
Selecting the Display option shows the report in the standard Carlson Takeoff report viewer. Reports can be exported to an Excel spreadsheet as well.

**Prerequisite:** Defined materials  
**Keyboard Command:** materials_report

### Define Materials Library

Define Materials Library allows you to Add, Remove, Load, Save, and Report a list of material costs. Costs can be set per area, count, volume, ton, or length by using the Edit function at the bottom of the dialog. The left side of the dialog can be used to set categories for different material costs.
**Prerequisite:** pricing for materials

**Keyboard Command:** define tk materials

---

**Edit-Assign Block Materials**

This command scans the current drawing to find and report block/symbol names and their count. For example, when the drawing contains different symbols for different types of utilities, this command identifies each type of symbol and the number. From this command, you can set the Description and Cost of the block by using the Edit button. You can also set the Description and Cost by predefined Materials by using the Set By Library button. When a block name is highlighted from the list, the drawing is zoomed to the location of one of those blocks so that you can see what it looks like. To Report these materials as part of the Standard Report, check on Include Materials Quantities Report in the Edit Materials dialog of the block layer found in the Define Layer Target/Material/Subgrade command. You can also just click on the Report button for a simple report.

![Edit-Assign Block Materials dialog](image)

---

**Prerequisite:** Blocks

**Keyboard Command:** edit all blocks

---

**Define Material Attributes**

Use this command to define all the material attributes that will be assigned to objects in the drawing for reporting purposes. The Define Attribute dialog box shown here allows the user to...
"Add", "Edit", or "Remove" attributes and save the definitions for later use. Simply "Load" a saved attribute definition file with the "tkd" extension for future use.

Selecting the Add or Edit options produce the edit attribute dialog box shown here. Use this command to define the Data name and the layer the objects currently reside on and the layer that future objects will be drawn on. Two entity types can be used, polyline data or point data. If the symbol option is selected the user has the option of which symbol will represent the object. Attribute fields must be defined for material reporting.
Selecting the Add or Edit button on the Edit Attribute dialog box brings up the Edit Field dialog box shown here. Use this dialog to define the field name and type. If the Value option is selected, only numeric values will be allowed when prompted. If the String option is selected, the user will have the ability to type in a text message when prompted.

**Prerequisite:** attributes

**Keyboard Command:** define_tk_data

**Draw Materials Entities**
Use this command to apply attribute data to objects as you draw or digitize them. Select the predefined attribute type to draw from the list available in the Select Attribute to Draw dialog box shown here.

The command line will prompt the user to pick the points of the desired location of the object and allow the attribute data fields to be filled out upon completion or each "enter".

**Prerequisite:** defined attributes  
**Keyboard Command:** draw_tk_data

**Input-Edit Material Attributes**

Use this command to assign predefined attribute information to an object already existing in the drawing. The command line prompt will require the user to select the object that attribute information is to be applied, and offer the Input-Edit Attribute dialog box shown here. This dialog box will display all predefined fields for that particular attribute type.
Prerequisite: predefined attribute information
Keyboard Command: edit_tk_data

Identify Materials Entities

Use this command to display all the objects that have attribute data assignments. The user will have the options of selecting the objects by picking them individually or by searching the entire drawing database. The objects that have attribute information assignments will "highlight" on the screen and the command line will display the attribute information.

Prerequisite: attributes
Keyboard Command: id_tk_data

Tools Menu

3D Drive Simulation

Function

This command allows you to view and move around the design surface in 3D mode.
Use the arrows on your keypad to move around the drawing.

At the very bottom of the window you will find the basic commands: Run will start to drive your vehicle around the surface, once your vehicle is moving the Run button turns into the Stop button. The arrows moves your vehicle left and right. The magnify glass zooms in and out. Click and drag up to zoom in and click and drag down to zoom out. When your vehicle is stopped the icon can be used to rotate the vantage point of the viewer by the x, y, or z axis. When you move the cursor to the screen it will change into a x, y symbol or a z symbol. Move the cursor around to move it from one to the other. If you have the x, y cursor move right or left to change the x axis view, or to change the y move the cursor up or down. If you have the z cursor than move it in a circular fashion to rotate the view point according to the z axis. The hand icon allows you to pan around the viewer. Click and drag the direction you want to move. The icon toggles the shading of the surfaces. The icon exits 3D Driver Simulation.

Above the basic command buttons you can change the Elevation and Distance away from your vehicle. Also, you can set the speed at which your vehicle travels. For a smaller drawing you may want to move around slower, for a larger drawing faster. Note: Unrealistic speeds such as 500 mph in a dozer may cause 3D Drive Simulation to freeze.
View Direction: Sets the direction of the view from the Front, Back, Left, or Right.
Vehicle Icon: You can select which Vehicle you want to use whether: Doozer, Hummer, School Bus or none at all.
View Position: Sets the elevation and distance to either that of the driver, a pedestrian, or bird.
Shading: Here you can set the shading of the surface to either Flat, Smooth, Elevation, Cut/Fill, or none. Flat just shades the contours as the are. Smooth smooths contours to look for realistic. Elevation colors different elevations in different colors so differences can visual be seen. Cut/Fill colors areas of cut differently than areas of fill so they can be visually seen. None merely shows the triangulation and does not shade in a surface.
You can select the Surface, High, and Low color by enter in an AutoCAD defined color number or you can choose Select to pick a color. The circle on the right determines the shade of the color.

In the top right of the 3D viewer is an aerial map of your surface. Below that the Elevation, Slope percentage, Azimuth, and Roll are updated as your vehicle moves around the surface. Slope and Roll are shown visually here as well.

On the bottom right you can set the Vertical Scale and check to Ignore Zero Elev, Display Trail, and Display Cut/Fill. If you increase the Vertical Scale than elevation differences can be seen easier. Ignore Zero Elev does not display elevations of zero in the 3D viewer. Display Trail draws a line where your vehicle has driven. Display Cut/Fill displays the cut and the fill.

**Prompts**

Loading edges...
Loaded 5057 points and 14923 edges
Created 9866 triangles
Loading reference file...

Loading edges...
Loaded 574 points and 1393 edges
Created 820 triangles

Loading entities...
Loading vehicle icon file...

Loading edges...
Loaded 926 points and 2150 edges
Prerequisite: a design surface
Keyboard Command: tk_flyby

Existing Surface 3D Viewer

Function

This command allows you to view the existing surface in 3D mode.

In the top right of the control bar you can check to Ignore Zero Elev and Color By Elevation and change the Vertical Scale. If you increase the Vertical Scale than elevation differences can be seen easier. Ignore Zero Elev does not display elevations of zero in the 3D viewer. Color By Elevation shows elevation change with the change of colors. Note: Color By Elevation is used in the above example. To adjust the color use the color circle on the right.

The magnify glass icons can be used to zoom in and out. Click on the plus magnify glass to zoom in and the minus magnify glass to zoom out. With the icon click and drag up
to zoom in and drag down to zoom out. The hand icon below the color circle allows you to pan around the viewer. Click and drag the direction you want to move. The icon can be used to rotate the vantage point of the viewer by the x, y, or z axis. When you move the cursor to the screen it will change into a x, y symbol or a z symbol. Move the cursor around to move it from one to the other. If you have the x, y cursor move right or left to change the x axis view, or to change the y move the cursor up or down. If you have the z cursor than move it in a circular fashion to rotate the view point according to the z axis. The icon toggles on and off the shading of the surface. The arrow icon reports the elevations at the bottom of the screen as you move around the surface. The icon restores the surface viewpoint to flat. The icon exits 3D Driver Simulation.

Rotation Axis: These three control bars rotate the surface around the x, y, and z axis. Clip plane trims the size of the surface shown in the viewer.

Prompts

Loading entities...
Loading edges...
Loaded 574 points and 1393 edges

Loading edges...
Loaded 574 points and 1393 edges

Loading edges...
Loaded 574 points and 1393 edges

Loading edges...
Loaded 574 points and 1393 edges

Prerequisite: an existing surface
Keyboard Command: cube_exist

Design Surface 3D Viewer

Function
This command allows you to view the design surface in 3D mode.

In the top right of the control bar you can check to Ignore Zero Elev and Color By Elevation and change the Vertical Scale. If you increase the Vertical Scale than elevation differences can be seen easier. Ignore Zero Elev does not display elevations of zero in the 3D viewer. Color By Elevation shows elevation change with the change of colors. Note: Color By Elevation is used in the above example. To adjust the color use the color circle on the right.

The magnify glass icons can be used to zoom in and out. Click on the plus magnify glass to zoom in and the minus magnify glass to zoom out. With the icon click and drag up to zoom in and drag down to zoom out. The hand icon below the color circle allows you to pan around the viewer. Click and drag the direction you want to move. The icon can be used to rotate the vantage point of the viewer by the x, y, or z axis. When you move the cursor to the screen it will change into a x, y symbol or a z symbol. Move the cursor around to move it from one to the other. If you have the x, y cursor move right or left to change the x axis view, or to change the y move the cursor up or down. If you have the z cursor than move it in a circular
fashion to rotate the viewpoint according to the z axis. The icon toggles on and off the shading of the surface (the shading is shown in the above drawing). The arrow icon reports the elevations at the bottom of the screen as you move around the surface. The icon restores the surface viewpoint to flat. The icon exits 3D Driver Simulation.

Rotation Axis: These three control bars rotate the surface around the x, y, and z axis. Clip plane trims the size of the surface shown in the viewer.

Prompts

Loading entities...
Loading edges...
Loaded 5057 points and 14923 edges

Prerequisite: a design surface
Keyboard Command: cube_design

FlyOver Along 3D Polyline

Function

This command allows you to view a self guided animation of following a path through a 3D surface model. There are two variations to this command. When the command is started, you must specify whether you want to use a surface model from file or screen entities.

Surface model from file: Using this method, you can select either a triangulation (.TIN) file or a grid (.GRD) file, then you have the option of following a polyline or following a "free" path. If you choose the polyline method, then the animation is limited to following the polyline. If you choose the "free" path method, you first specify two points to obtain a starting direction, the while inside the viewer you can point the animation in any direction.

Screen entities: Using this method, you must select a 3D polyline to follow. The animation is limited to following the polyline.

After making the above selections, the 3D graphics window is opened. The main window is for the animation, the smaller upper right window shows you the overall plan view, and the smaller
window located at middle right shows you the current elevation, slope and azimuth. While following a "free" path, you will have a 3rd small window located at lower right which shows you the amount of roll at your current position.

This button raises the elevation of your viewing position.

This button lowers the elevation of your viewing position.

This button turns your viewing position to the left.
This button turns your viewing position to the right.

This button allows you to zoom in and out.

This button allows you to rotate the main animation window in any X, Y or Z direction.

This button allows you to pan.

This button toggles shading on and off.

This button starts the animation in the main window.

This button stops the animation.

This button exits the 3D Surface FlyOver command

Control for position of the light source, viewed from above.

**Prerequisite:** Surface Model and optionally a 3D Polyline

**Keyboard Command:** flyby
Surface Inspector

Function

This command allows you to report and optionally label elevations from your drawing. You can analyze all of your different surface files at one time. After running the command, Surface Inspector will begin showing you real-time elevations for each surface as you move the cursor on the screen. If you pick a point or enter coordinates, the elevation will be labeled along with the surface name.

![Surface Inspector](image)

Surface inspector shows you real-time elevations as you move the cursor over your surface.

Prerequisite: Surface Model (s)
Keyboard Command: surfvals

Surface Report

Function
This command reports a variety of information on each of your different surfaces. This is useful for checking for bad data and the file names of your surfaces. An example is below.

**Surface Report 3/10/2005 15:34**

Max Cut: 18.327 at 409269.984,207196.674
Max Fill: 1.943 at 409389.586,207248.866

**Original Ground After Topsoil Removal**
File: C:|Documents and Settings|Todd Carlson|Desktop|Takeoff|Drawings|demo3-ex.flt
Date Modified: Thu Feb 10 10:02:05 2005
File Size: 64,028
Points: 259, Edges: 744, Triangles: 486
Min Z: 184.000 at 409299.790,206879.287
Max Z: 210.000 at 409571.562,207177.240

**Design With Subgrade and Topsoil Replacement**
File: C:|Documents and Settings|Todd Carlson|Desktop|Takeoff|Drawings|demo3-fn.flt
Date Modified: Thu Feb 10 10:02:08 2005
File Size: 153,038
Points: 609, Edges: 1,779, Triangles: 1,171
Min Z: 176.000 at 409357.096,206821.604
Max Z: 206.000 at 409551.532,207185.124

**Original Ground Before Topsoil Removal**
File: C:|Documents and Settings|Todd Carlson|Desktop|Takeoff|Drawings|demo3-og.flt
Date Modified: Thu Feb 10 10:02:05 2005
File Size: 64,028
Points: 259, Edges: 744, Triangles: 486
Min Z: 184.000 at 409299.790,206879.287
Max Z: 210.000 at 409571.562,207177.240

**Design Without Subgrade or Topsoil Replacement**
File: C:|Documents and Settings|Todd Carlson|Desktop|Takeoff|Drawings|demo3-bs.flt
Date Modified: Thu Feb 10 10:02:08 2005
File Size: 153,038
Points: 609, Edges: 1,779, Triangles: 1,171
Min Z: 176.000 at 409357.096,206821.604
Max Z: 206.000 at 409551.532,207185.124

Chapter 1. Takeoff Module
Prerequisites: A Surface
Keyboard Command: SURF_STATS

Quick Profile

Function

This command allows you to create, view, edit, and report profiles from the TakeOff surfaces.

Pick starting point (CL-Centerline, P-Polyline): To make a profile you need to define the alignment by: 1) picking points on the screen; 2) typing in CL in the command prompt, and selecting a centerline file; or 3) typing in P and choosing a polyline from the screen. After doing so, the above profile viewer is created.

The far right dialog box allows you to toggle on and off different Surfaces to view in the profile viewer including: Original Ground, Topsoil Removal, Design Surface, Final Subgrade, Overex Surface, Strata Surfaces. If a surface is not defined in the current TakeOff project, like Topsoil Removal in this example, than you will not have the option to display it. In this example,

Chapter 1. Takeoff Module  58
the three Surfaces that can be displayed, Original Ground, Design Surface, and Final Subgrade, 
are displayed in the profile viewer.

When you move the cursor around the profile viewer a crosshair follows along the surface 
and reports the Station, Slope %, and Elevation at each point. It is displayed towards the 
bottom-right side of the screen next to Adjust Alignment. In this example the station is 2+16.650, 
the Slope is -5.6%, and the Elevation is 818.133. A crosshair can been seen in the profile drawing 
and along the alignment in the main drawing as well.

Vertical Exaggeration: x1 is the actual appearance of the surface(s). Depending on the 
flatness of the surface(s), you can select x2, x5, x10 vertical exaggerations to better see the 
elevation differentiation and different surfaces. The option Fit automatically exaggerates the 
vertical to best fit the profile viewer.

Drag Action: This dialog allows you to zoom in and out, and pan around the profile. To 
zoom in click and drag up, to zoom out click and drag down. To Pan, click amd drag the direction 
you want to move.

The Adjust Alignment icon allows to pick the polyline or centerline that you used and 
move it to your liking. If you selected an endpoint vertex, you can pivot that vertex around 360 
degrees and the profiles will update in real time. This is helpful when checking for spikes. If you 
select the middle vertex then you can shift the entire centerline around.

If you created a profile alignment by picking points and you want to save that polyline 
you created then toggle on Draw Plan View Polyline. If you do not choose Draw Plan View 
Polyline than the polyline will be lost when you exit out of the Quick Profile command. Grid 
Ticks Only marks elevations and distances but does not draw them into grids.

The Save icon allows you to save the profile as a (.pro) file by whatever name you give it. 
The Draw icon allows you to draw the profile right on your drawing. Set the layer name, vertical 
and horizontal scale as desired, pick a starting point to draw, and the profile is created. Note: the 
below example has a veritcal scale of 5 feet per grid and a horizontal scale of 50 feet per grid.
Prompts

Pick starting point (CL-Centerline,P-Polyline): p
Polyline should have been drawn in direction of increasing stations.
CL File/<Select polyline that represents centerline>: 
Loading edges...
Loaded 5057 points and 14923 edges
Created 9866 triangles

Prerequisite: a surface
Keyboard Command: TK_QUICKPRO, QUICKPRO

Cut/Fill Centroids

Function

This command visually shows cut/fill areas in your drawing and finds the center of mass or centroids for every cut and every fill region. Also, a report can be created to show volumes in each region.
Points are created to show the centroids locations. Also, labels can be created to display the volumes in the different regions.
Here is the Cut and Fill Centriod Report for the above example. It shows the volumes, the coordinates of the centroids, and the Earth Movement Report. The Earth Movement Report shows the minimal distances for moving Cut to Fill areas.

**Prompts**

Loading edges...
Loaded 1199 points and 3391 edges
Created 2193 triangles

Loading edges...
Loaded 574 points and 1393 edges
Created 820 triangles

Loading edges...
Loaded 8732 points and 25651 edges
Created 16920 triangles

Generating report:
Region #2
3053.7 Fill
Reading the selection set ...  
Joining ...

---

Chapter 1. Takeoff Module
Prerequisite: Existing and Design surfaces
Keyboard Command: tk_cutfillc

Cut/Fill Map Legend

Function

This command will draw a Cut/Fill Map Legend on your drawing. It will display the cut/fill amount, color, and range, as seen below.

Prompts:

Select point for color legend:
Legend size <10.0>:

Prerequisite: Cut/Fill amounts
Keyboard Command: CF_MAP_LEGEND
Perimeter Polylines Properties

Function

This command allows you to control the properties of any perimeter polyline (Note: Perimeter polylines also have to be closed polylines). Select a polyline and the following dialog appears. Here you can define the functionality of the polyline in regards to, the Site Boundry, Areas of Interest, and Topsoil Removal/Replacement. These properties can also be set separately using the Boundry Polyline, Areas of Interest, and Topsoil Removal/Replacement commands found under the Tools menu of Carlson Takeoff.

Prerequisite: a polyline
Keyboard Command: perim_prop

Update Colors For Set Elevations

Function

This command refreshes the color of entities depending on their elevation and layer target. For
entities assigned to the Existing or Design layer targets, if the entities are at zero elevation then their color is set to grey. Otherwise the entities have their true, original color. If the Automatic Update Colors command under Settings->Configure->Takeoff Module is toggled off, then this command is the way to update the entity colors after editing elevations.

**Prerequisites:** none  
**Keyboard Command:** update_tk_colors

---

### Convert LDD-AEC Contours

**Function**

This command allows you to convert LandDesktop contours (known as AECC
CONTOUR objects) into polylines. You must have the AEC Object Enabler installed before using this command. If you do not have the object enabler installed, download the latest version from www.autodesk.com.

You can use the *List* command to determine if contours are polylines or AECC_Contour objects. Here is an example listing:

```plaintext
AECC_CONTOUR Layer: "CONT-MJR"
Space: Model space
Handle = 429
Major Contour Interval
Elevation: 1005.00
Smoothing: None
Number of Vertices: 48
Open
Length: 560.25
Constant width: 0.00
Style Name: Standard
```
Export Polyline File

Function

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

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

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

Prompts

Polyline file format [Carlson>/DTM/Idan/MicroStation]? press Enter for Carlson format
Specify File to Write dialog create a new file or append to existing
Polyline file for Grid File Utilities macro [Yes/<No>]? press Enter The option will write a polyline file that can be used with Grid File Utilities for inclusion/exclusion perimeters.
Include Z coordinate in polyline file [Yes/<No>]? press Enter This option controls whether the polyline vertices are written in 2D or 3D.
Specify Exclusion/Warning Polylines [Yes/<No>]? press Enter This option applies to machine control for warning areas.

Chapter 1. Takeoff Module 66
Specify WorkZone Polylines [Yes/<No>]? press Enter This option applies to machine control for working areas.

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

Enter reduce offset cutoff <0.1>: press Enter

Decimal places for coordinates <2>: press Enter

Select polylines, lines and arcs to write.

Select objects: pick the entities to process

Done.

Sample Polyline File:

POLYLINE,15
47639.82,74540.11,0.00
47670.49,74565.79,0.00
47701.08,74591.49,0.00
49375.61,76358.47,0.00
50066.86,76846.75,0.00

POLYLINE,15
47633.24,74547.97,0.00
47663.90,74573.65,0.00
etc...

Keyboard Command: polywrite

Prerequisite: Polylines in the drawing

Import/Export Carlson Triangulation Files

Function

Import Carlson Triangulation Files allows you to import an external surface file into TakeOff to use as a named surface. Export Carlson Triangulation Files allows you to take a Surface Triangulation file and save it independent of the drawing.
**Prerequisite:** .TIN or .FLT files

**Keyboard Command:** import_tin, export_tin

---

**Export Topcon TIN File**

**Function**

This command writes a Topcon TIN file (.TN3) from a Carlson Takeoff surface. The first prompt at the Command line chooses whether to export the surface from a file or the screen. The file option will prompt for a triangulation file (.tin or .flt) and then the Topcon TIN file to create. The screen option can be used to export a surface from 3d Faces or TIN lines from the drawing. The screen option has the following prompts:
Choose Type of Export

- **Triangle Faces**: Triangulation networks that consist of edge matched 3DFaces defined by three points.
- **Triangle Edges**: Triangulation Networks that consist of edges drawn with Lines defined by two points.
- **Triangulation File**: A Carlson triangulation file (with a DOS extension of .flt).

**Source Units**: Select the appropriate unit type of the entities to be exported, then press NEXT.

If the Triangulation File option is chosen, a standard windows file selection dialog will open to allow for the selection of the .flt file. If Triangulation Edges or Faces are used for the export, you must select the entities to be exported from the screen. The following dialog opens:
Manually Select allows on-screen selection with any of the various methods (Window, Crossing, Crossing Polygon, etc.)

All on a Selected Layer allows selection of a single entity. The command then selects all Lines or 3DFaces on the layer of the selected entity for export, filtering entities not on selected layer.

Generally, you can select any entities drawn on the screen without special care. The command filters out all types except Lines or 3DFaces depending on the type selected.

When Processing Lines: Crossings, crossing polygon, and fence selection sets can often result in "spikes" around the edges. The command generally discards these spikes, or reconnects them. In any case, better results are developed with clean selections of drawing entities.

Pressing NEXT will open the following dialog:
**Other Features to Export:** Often times, landmark lines, or other features will assist in orientation to the user when using the TN3 file in the field. For example: Property lines, project centerlines, or other well established reference lines. These features may be added and displayed on the TN3 by choosing them at this prompting. The Other Features dialog opens with all three Source Formats. Selections are made as described above. Once the selections are made and the dialog reappears, press NEXT.

**Naming the TN3 File:** After all the triangulation and feature items are selected from the graphics screen, the command requests a name for the output file. Once a name is given, the following dialog reviews all the selections made. A description for the job can be added at this time. Press Finish when ready to proceed.

The program will process the selected entities and/or files and report as shown here:
To view the final output, press the Run Simulation button, or press Exit to return to the Carlson program.

Run Simulation Dialog

**Elevate Menu**

**Change Elevations**

**Function**

This command will change the elevation of selected Entities. It can move the entity to a specified elevation from it's current elevation (absolute) or do a differential change by adding or subtracting a value from it's current elevation. If Carlson TakeOff points are selected, their attribute text and z axis coordinate are changed.

**Prompts**
Ignore zero elevations (<Yes>/No)? Press Enter. If you answer No, then entities with elevation 0 will be changed.

[A]bsolute or [D]ifferential Change <A>: A
Select/<Enter Elevation <0.0000>>: 125
Change Layer for changed entities [Yes/<No>]: No

Elevation to change to:

By using the Absolute option all entities selected are changed to the elevation 125.

Select Entities for elevation change.
Select objects: C
First corner: (pick point)
Other corner: (pick point)
Select objects: [Enter]

Keyboard Command: chgelev
Prerequisite: Something to change

Set Polyline to Elevation

Function

This command allows you to assign elevations to one or more polylines. The elevation can be assigned by entering in the value or by picking a text entity that has the elevation.

Prompts

Select/<Enter Elevation <0.0000>>: Select a text entity or type in an elevation. Press enter for the default elevation in brackets.

Select Polyline for elevation change. Pick on the screen a polyline you wish to change such as: LWPOLYLINE
Done.
Set another polyline [<Yes>/No]? Press Y to pick another polyline to assign an elevation to. Type in N to finish the command.

Keyboard Command: set_pline_z
Prerequisite: A polyline and an elevation to assign it.
**Edit-Assign Polyline Elevations**

**Function**

This command allows very precise control of 3D polylines, specifically in the ability to edit vertex elevations, as well as add, delete, or move vertices. You can also control the location of polyline vertices as defined by the station and offset of the vertices relative to a Centerline.

Polyline vertices are designated as either Control or Free vertices. The elevation of Control vertices are set and held, the elevations of Free vertices are interpolated. In the drawing, control vertices are shown with red boxes, along with their vertex number and elevation. Free vertices are displayed with blue boxes and are not annotated.

When you run the command, you are first prompted to select a polyline to edit. When you pick a polyline to work with, the following control panel appears on the left side of your screen.

![Control Panel for Edit-Assign Polyline Elevations](image)

The top row of buttons across the top of the control panel are used to manipulate the view in the drawing with various Zooming and Panning options. The second row of buttons will change...
as you select different tabs, but are essentially used to add vertices, delete vertices, or pick elevations or locations for vertices.

The four tabs in the panel provide access to control of polyline vertex Elevation, Position, Offset and Settings.

**Elevation:** This tab displays the vertices of the polyline, each with a check box to set whether it is a control vertex or free, its assigned number, its elevation, and the slope from the previous vertex to that vertex. Selecting a vertex highlights its grip in the drawing. Once selected, you can enter an elevation or slope for that vertex in the spaces below the list, thereby automatically setting the vertex to a control vertex. The Base Elevation is used to adjust the elevations of all the vertices simultaneously.

**Position:** The Position tab displays the coordinates of each vertex. To move a vertex, you can type in new coordinates, use the Pick Position icon to specify a new location for the vertex on the screen, or you can grip the vertex and drag it to a new location.

**Offset:** The Offset tab requires the selection of a Centerline to reference. Once a Centerline is designated, the Station, Slope, and Offset of each vertex relative to the Centerline is displayed and can be edited.

**Settings:** The Settings tab provides control over various overall options pertaining to the use of the command. For example, hiding free vertices and setting how to report your slopes between vertices.

**Right-click menu:** There is a right-click menu available at all times which also gives access to a variety of functions and settings.

**Keyboard Command:** edit_pline_z

**Prerequisite:** Polylines with vertexes

### 2D to 3D By Surface Model

#### Function
This command converts a 2D polyline into a 3D polyline by calculating 3D polyline vertices at all the intersects of the 2D polyline with surface entities (contour polylines, triangulation lines) and by interpolating elevations from these intersections at the original vertices locations. An application for this command is to create breaklines. For example, a ridge breakline could be generated from contour lines by drawing a 2D polyline along the ridge and across the contours. Then this command could grab the contour line elevations along the polyline to make a ridge breakline.

In addition to using entities in the drawing, the 2D polyline can be converted to 3D using a surface model stored in triangulation (.flt or .tin) file. If you use a file, then you can also use the polyline's current elevation as a vertical offset from surface.

**Prompts**

*By Screen Entities:*

**Source of surface model [File/<Screen>]?** Type S for Screen

Select polylines to convert.

Select objects: select the polyline(s) to convert

Select surface 3DFaces, lines and polylines.

Select objects: select the surface entities (contour polylines, breaklines, triangulation lines, etc)

Reading points ... 692

Keep existing polylines [Yes/<No>]? Press Enter

This command creates a new 3D polyline, and this prompt allows you to keep the old polyline.

Set layer name for converted polylines [Yes/<No>]? Press Enter

This allows you to assign the new polyline to a layer.

Converting polylines ...

Converted 1 polylines.

*By a .flt or .tin File:*

**Source of surface model [<File>/<Screen]?** Type F for File

Select polylines to convert.

Select objects: select the surface entities (contour polylines, breaklines, triangulation lines, etc)

Use current polyline elevations as vertical offset from surface [Yes/<No>]? Press Enter

This will offset the new polyline by its current elevation. That is, if a polyline has an elevation of -4 and the surface you are converting it to has an elevation of 800, then saying Yes will drape the
polyline at an elevation of 796.

**Keep existing polylines [Yes/<No>]? Press Enter**
This command creates a new 3D polyline, and this prompt allows you to keep the old polyline.

**Set layer name for converted polylines [Yes/<No>]? Press Enter**
This allows you to assign the new polyline to a layer.

**Keyboard Command:** 2dto3dp

**Prerequisite:** A polyline and surface lines or grid file or triangulation file.

---

### 2D to 3D Polyline by Points

#### Function

This command converts a 2D polyline into a 3D polyline by using the elevations of points. At each vertex of the polylines, the program looks for a point with elevation at the same x,y location. The points can be Carlson point blocks or AutoCAD POINT entities. This routine can be useful if the linework is created in 2D at zero elevation, and points with elevation are located along the linework. Then the linework can be converted into 3D polylines with this command. For example, a centerline polyline with arcs may need to be created in 2D for stationing because AutoCAD does not allow arcs on 3D polylines. To use this polyline as a breakline in surface modeling, this command can convert the polyline into a 3D polyline.

#### Prompts

**Select points and polylines.**

**Select objects:** select polylines to convert and the points with elevation
**Keyboard Command:** 2dto3dpt

**Prerequisite:** A polyline and points

### 2D to 3D Polyline-By Text

#### Function

This command allows you to change 2D polylines to 3D polylines by elevation labels. This command will prompt you for samples of the elevation labels and the polylines to convert. The program uses these samples to know the layer names for the labels and linework to process. Then select all the polylines with their labels you want to convert.

You will then be prompted to enter in an elevation to add to label values. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 539.97, 540.02, 540.11 sometimes, like in the example on the side, they are listed as 39.97, 40.02, 40.11. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing. This command will assign elevations from the labels to nearby vertices. If vertices do not have a close elevation label than they will be interpolated from vertices that are nearby elevation labels.
Prompts

Select sample of elevation text: Pick a text label
Select sample of a polyline to convert: Pick a polyline
Select polylines to convert and elevation labels.

Select objects: Select all the entities to process
19 found, 19 total

Enter elevation to add to label values <0.00>: 500
Pre-processing entity #19 of 19
Filtering text entities
Processing elevation text #18
Remaking polyline #1

Keyboard Command: elevfb
Prerequisite: 2D polyline and elevation labels

2D to 3D By Text With Leader

Function

This command will assign elevations from the labels to the polylines by following the label leaders to their corresponding vertices on the polyline.
This command will prompt you for samples of the elevation labels, the leaders, and the polylines to convert. The program uses these samples to know the layer names for the labels and linework to process. Then select all the labels and leaders for the polylines you want to convert. You will then be prompted to enter in an elevation to add to label values. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 800s instead of labeling every elevation 817.85, 817.40, 817.30 sometimes, like in the above example, they are listed 17.85, 17.40, 17.30. This command allows you to add a given amount, such as 800, to every label elevation to produce the correct elevation in the drawing.

Carlson TakeOff searches for all leaders and gathers their associated text. If the program finds
different labels in the elevation text, then this dialog box allows you to select the text you want to create 3D polylines. In this example you might want to use elevations followed by TC. This dialog box allows you to select that text and exclude the other text which is not to be used in the elevations of the polyline, such as FS.

![Image of dialog box](image)

If you are creating 3D polylines from multiple elevation labels than this dialog box will allow to offset certain labels by a given amount. In the above example you can offset an elevation labeled FS by .50 so that it matches vertices set by TC labeled elevations.

**Prompts**

**Select sample of elevation text:** Pick a text label  
**Select sample of an annotation leader:** Pick an annotation leader  
**Select sample of a polyline to convert:** Pick a polyline  
**Select polylines to convert, leaders and elevation labels to process.**  
**Select objects:** Select the desired entities
22 found
3 were filtered out.

Select objects:

Enter elevation to add to label values <0.00>: 800
Pre-processing entity #19 of 19
Filtering text entities
Processing leader #6
Remaking polyline #1

Keyboard Command: elevfl
Prerequisite: 2D polyline, elevation labels, and leaders

2D to 3D Polyline by Start-End Elevations

Function

This command allows you to convert a 2D polyline to a 3D polyline by specifying the starting and ending elevations of the polyline. All intermediate polyline vertex elevations are linearly interpolated from these end point elevations.

Prompts

Select polyline to assign elevations:
Enter starting elevation: 109.85
Percent/Ratio/<Enter ending elevation>: 112.16
Select polyline to assign elevations (Enter to End): press enter to end

Keyboard Command: 2dto3dpl
Prerequisite: A polyline

Draw Building Envelope Polyline

Function

This command creates a rectangular polyline around selected linework. This can be used to give a building all one elevation.
Select the entities that make up the building. Next you will be prompted to name the layer and to set an offset distance (The above example is offset by 5 feet). Also, you can set the elevation of the envelope and trim crossing linework to ensure you have a flat pad.

Prompts

Draw Building Envelope dialog
Select building lines.
Select objects: pick the linework that makes up the perimeter of the building
Draw another building envelope [Yes]/No? N
Keyboard Command: bldg_perim
Prerequisite: a pad

Pad Polyline By Interior Text

Function

This command allows you to set one or more pad elevations using interior text labels.

After running the command you will be prompted to select the layers you want to use for the pad elevation and for the boundary of the pad. Sometimes pads are drawn with linework from two different layers and Carlson TakeOff allows you to pick all the correct linework.

This dialog box allows you to create a new layer with the correct x,y coordinates and elevations. If the pad shares the same coordinates with other linework with different elevations than this dialog box allows you to offset the new polyline to avoid the problem of shared occupied points with different elevations. You can choose to have an interior offset or an exterior offset and also decide how much to offset the new polyline. Selecting Both will give both the interior pad elevation and the exterior contour elevations. This helps the transition from you pad elevation to the design contouring. The Snap Tolerance field joins linework which falls within the range you set to create a pad. Trim Outside Elevated Polylines will trim out contour elevations that go through your pad that you are not using elevations from within the pad.
Elevation to add to text values adds to the values from the elevation labels. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 523.5, 543.3, 537.2 sometimes they are listed as simply 23.5, 43.3, 37.2. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing.

After running the command you will be prompted to select the layers you want to use for the pad elevation and for the boundary of the pad. Sometimes pads are drawn with linework from two different layers and Carlson TakeOff allows you to pick all the correct linework. In addition, if your text has multiple Prefixes and Suffixes you will be prompted to select the ones you want to use the elevation from.

After clicking <OK> select all the pads and their elevation labels that you wish to change, press <Enter>, and the new layer with elevations will be created and placed in the Design target.
Prompts

Select layer sample of elevation text: Pick a label text
Selected text layer —-TX07
Select layer sample of boundary linework:
Selected linework layer PAD
Select another layer sample of boundary linework (Enter to continue):
Select text and linework to process.
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects:
Analyzing entire selection...
Set elevation for 1 polylines.

Keyboard Command: pad_by_text
Prerequisite: Pad polylines and elevations
Convert Spot Elev To Points

Function

This command takes spot elevation entities with zero elevations and assigns them elevations according to corresponding elevation labels. This dialog box allows you to choose the format of the spot elevations entities that you want to convert.

![Assign Spot Elevations dialog box]

Output:

Carlson points: creates Carlson points at elevation of spot and stores them in coordinate file

AutoCAD points: creates AutoCAD point objects at elevation of spot

Is spot indicator a part of the elevation label?

If set to "Yes", four choices for Spot indicator are available to select from:
Text **insertion point**: uses the insertion point of the text for the location of the new point

**Text decimal point**: uses the decimal point in the text for the location of the new point

**Text plus sign**: uses the plus sign in the text for the location of the new point

**Text letter x**: uses the letter x in the text for the location of the new point

If set to "No", five choices for **Spot indicator** are available to select from:

**Linework leader**: creates a data point at the end of a leader

**Linework cross**: creates a data point at the intersection of a linework cross

**Text plus sign**: creates a data point at the insertion point of a text plus sign

**Text letter x**: creates a data point at the middle of a text letter x

**AutoCAD point**: creates a data point at the node of an AutoCAD point

**Block References**:

**Process Block References**: If check box is cleared, Carlson Civil searches only text entities for
elevations, but if checked, Carlson Civil will search block references for elevations that are stored as attributes of a block. Use this option if the elevation is an attribute and the symbol designating the location of the spot elevation are both part of the block definition.

**Expand Block References:** Use this option to search block references when the elevation is stored as an attribute of a block, but the symbol designating the location of the spot elevation is a different block or even other geometry that is not defined within a block.

**Base elevation:** The value entered here is added to the existing spot elevations for all newly created points. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 523.5, 543.3, 537.2 sometimes they are listed as simply 23.5, 43.3, 37.2. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing. Note: The base elevation will not be added to any elevations that are closer to the base elevation value than they are to 0; e.g. if a base elevation of 500 is specified, 500 will be added to elevations like 23.4, 45.5, etc, but will not be added to elevations like 456.4 or 468.9.

**Prefix Filter:** Carlson Civil examines all selected spot elevations for prefixes or suffixes. If they are all the same, the command proceeds, but if there are different prefixes and/or suffixes found, the Prefix Filter dialog box is invoked. This dialog box allows you to select which prefixes and/or suffixes to use to create spot elevations, and also allows you to use different offset values for each.
**Prompts**

Starting point number \(<1>\): press Enter

Select TEXT, MTEXT spot elevations to process and any associated leader lines:

Select objects: pick entities to process

Pre-processing entity #40 of 40...

Filtering text entities

Processing elevation #40...

Converted 40 spot elevations.

**Keyboard Command:** spotelv2

**Prerequisite:** Spot elevations

---

**Assign Contour Elevation - Multiple in Series**

**Function**

This command can be used to quickly and accurately assign the elevation of series of AutoCAD polylines that have been converted from raster or digitized without correct elevations. Ther-
outline will automatically assign elevations to the polylines crossing the fence line selected by two points. At the same time the elevations are changed, the program can assign it a new layer, color, linetype, and polyline width. This process usually works best if contours are in a temporary (white) layer to start. When they are processed, they will take on the color of the new layers making it easy to distinguish which polylines have been processed.

**Prompts**

**Settings/First Point:** *(Press S to change settings or pick first point.)*

**Second Point:** *(Pick second point)*

**Beginning Elevation** <0.00>: 1020

**Increment Direction U/D** <U>: (enter)
Keyboard Command: grpcelev
Prerequisite: digitized polylines

Assign Contour Elevation - From Contour Labels

Function

This command allows you to set elevations to contours from elevation labels.

Select a sample of the elevation text to be used on the contouring. Next, select a sample of the contouring that you want to add the elevations to. Now select all the contours and their corresponding elevation labels and press <Enter>. Carlson TakeOff will then add elevations to all the contours. You may be prompted to distinguish what contour goes with what elevation label. You can either press <Enter> to accept the contour that Carlson TakeOff has selected or you can Press <N> to choose another contour.

Prompts

Select sample of elevation text:
Select sample of a contour line:
Select contour lines and elevation text to process.
Select objects: all
5049 found
4041 were filtered out.
Select objects:

Joining adjacent polylines...
Reading the selection set ...
Joining ...
Pre-processing entity #1008 of 1008
Filtering text entities
Processing elevation text #518

Conflict detected: pick contour corresponding to current elevation text
Press N for next selection or Enter to accept current:
Remaking polyline #311

Keyboard Command:  TXTCELEV
Prerequisite: contours and contours labels

Assign Contour Elevation - Single Elevation Group

Function

This command changes the elevations of polylines and can be used to set the elevations of contour polylines. The routine begins at a specified elevation and prompts for a selection set of polylines to set to the elevation. Then the routine repeats using the last elevation plus the elevation increment. Enter an empty selection set to exit the routine.

Prompts

Starting elevation <0.0>: 500.0
Contour interval (negative for down) <1.0>: 5.0
Select polylines to set to elevation 500.0.
Select objects: pick the polylines
Select polylines to set to elevation 505.0.
Select objects: pick the polylines
Select polylines to set to elevation 510.0.
Select objects: Press Enter

Keyboard Command: setcelev
Prerequisite: polylines
Drape 3D Polyline On Surface

Function

This command converts a 2D polyline into a 3D polyline by calculating 3D polyline vertices at all the intersects of the 2D polyline with surface entities (contour polylines, triangulation lines) and by interpolating elevations from these intersections at the original vertices locations. An application for this command is to create breaklines. For example, a ridge breakline could be generated from contour lines by drawing a 2D polyline along the ridge and across the contours. Then this command could grab the contour line elevations along the polyline to make a ridge breakline.

In addition to using entities in the drawing, the 2D polyline can be converted to 3D using a surface model stored in triangulation (.flt or .tin) file. If you use a file, then you can also use the polyline's current elevation as a vertical offset from surface.

Prompts

By Screen Entities:
Source of surface model [File/<Screen>]? Type S for Screen
Select polylines to convert.
Select objects: select the polyline(s) to convert
Select surface 3DFaces, lines and polylines.
Select objects: select the surface entities (contour polylines, breaklines, triangulation lines, etc)

Reading points ... 692

Keep existing polylines [Yes/<No>]? Press Enter
This command creates a new 3D polyline, and this prompt allows you to keep the old polyline.

Set layer name for converted polylines [Yes/<No>]? Press Enter
This allows you to assign the new polyline to a layer.
Converting polylines ...
Converted 1 polylines.

By a .flt or .tin File:
Source of surface model [<File>/Screen]? Type F for File
Select polylines to convert.
Select objects: select the surface entities (contour polylines, breaklines, triangulation lines, etc)

Use current polyline elevations as vertical offset from surface [Yes/<No>]? Press Enter
This will offset the new polyline by its current elevation. That is, if a polyline has an elevation of -4 and the surface you are converting it to has an elevation of 800, then saying Yes will drape the polyline at an elevation of 796.

Keep existing polylines [Yes/<No>]? Press Enter
This command creates a new 3D polyline, and this prompt allows you to keep the old polyline.

Set layer name for converted polylines [Yes/<No>]? Press Enter
This allows you to assign the new polyline to a layer.

Keyboard Command: 2dto3dp
Prerequisite: A polyline and surface lines or grid file or triangulation file.

Edit Polyline Vertex

Function

This tool allows you to make changes in the coordinates of vertices on all polyline types. Upon execution you will be asked to select a polyline to edit. Upon selection a temporary marker will be placed at all of the vertices of the polyline, making them easy to distinguish. Then pick near the vertex you wish to edit, and the following dialog appears.

At the top of the dialog it identifies the type of polyline, being 2D or 3D. In the case of 2D polylines it allows you convert the polyline. You have the ability to type in new northing, easting or elevation values. You can also determine the 3D coordinate position by using distances and slope to/from adjacent points. As you change the values in the dialog, new values for derivatives are being calculated. For example if you change the horizontal distances, the coordinates will change.
Prompts

Select polyline vertex to edit: pick a polyline at the point to be modified
Pick or enter position \(<5264.23,5048.21>\): pick a point
Enter elevation \(<0.00>\): Press Enter
Select polyline vertex to edit: Press Enter to end

Keyboard Command: editpl
Prerequisite: A polyline.

Edit Contours

Function

This command revises a segment of a contour polyline. Begin by picking a point on the contour where you want to start editing. Then pick new points for the polyline. When finished picking new points, press Enter and then pick a point on the contour to connect with the new points. The polyline segment between the start and end points is then replaced with the new points.

Prompts

Select contour to edit: pick the contour polyline at the place to start editing
Pick intermediate point (Enter to End): pick a point
Pick intermediate point ('U' to Undo, Enter to End): pick a point
Pick intermediate point ('U' to Undo, Enter to End): Press Enter
Pick reconnection point on contour: pick the contour polyline at the place to join

Edit this contour by picking new points
Contour with segment replaced with new points

**Keyboard Command:** editctr
**Prerequisite:** polylines with elevation (contour polylines)
Digitize Menu

Tablet On

Function

Executes AutoCad's TABLET command to set the tablet on. Refer to the AutoCad Reference manual for further information.

Note: Function key [F4] can toggle on/off tablet.

**Keyboard Command:** tablet

**Prerequisite:** Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

Tablet Off

Function

Executes AutoCad's TABLET command to set the tablet on. Refer to the AutoCad Reference manual for further information.

Note: Function key [F4] can toggle on/off tablet.

**Keyboard Command:** tablet

**Prerequisite:** Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

Tablet Calibrate

Function

You can calibrate the tablet/digitizer in one of two ways: **Known Reference Points** or **Drawing Scale with New Reference Points**. Reference points are the foundations of whatever data you
digitize into the computer. Takeoff bases everything from drawing location to drawing scale on
the reference points you digitize.

**Drawing Scale with New Reference Points** method is very convenient when you don't
know the precise coordinates of the entities on your drawing. As long as you can obtain the
drawing scale from your plan, this method can establish a coordinate system relative to the
position of the plan on the digitizer board. In addition to the drawing scale, you are required to
enter a random coordinate for the first reference point, the default coordinate is (1000,1000).
Takeoff would computer the coordinate of the second reference point that you pick based on the
first point. The coordinates of these two reference points would be saved and will be display on
the **Tablet Calibration Dialog** next time when you calibrate the tablet, so you can digitize back
to the previous coordinates using **Know Reference Points** method if you are working on the
same drawing, though you might have moved or rotated your drawing on the digitize board..

If you know the precise coordinates of two points, you can select **Known Reference Points**
method, which establishes a coordinate system that is exactly match the coordinates in the field
or on your drawing. Furthermore, Takeoff saves the coordinates of the two reference points
from previous calibration and displays them on the **Tablet Calibration Dialog** next time when
you calibrate the tablet. If you want to continue to work on the same drawing, you can use the
**Know Reference Points** method with the saved coordinates to digitize back to your previous
coordinates although you might have moved or rotated your drawing on the digitizer board.

For accurate takeoff calculations, choose two points that can be easily found in the field
and are farther apart rather than closer together.

![Tablet Calibration Dialog](image)

---

**Chapter 1. Takeoff Module**
Prompts

Tablet Calibration Dialog
Specify the Calibration Methods. If you select Drawing Scale method, enter the drawing scale and the coordinate of the first reference point. Otherwise enter the exact coordinates of the first and second reference points.

Pick first reference point: pick a point on the drawing
Pick second reference point: pick another point on the drawing

Keyboard Command: digsetup
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

Digitizer Setup
Digitizing is the act of inputting data into the computer by tracing the data from a plan sheet. You need to have a digitizer board, puck, Carlson Takeoff, your computer and your plan to do digitizing. Wintab is a digitizer driver that lets you to use the digitizer cursor as both a digitizer cursor and a mouse. You need to install Wintab when you install Carlson Takeoff. Wintab can be downloaded from GTCO web site: http://www.gtcocalcomp.com/supportgtcosoftware.htm. Select the driver version that suits the type of your digitizer board well.

After you installed Wintab driver on your computer, you set up you digitizer to the correct
point mode. In Windows 2000/XP, go to **Start**->**Settings**->**Control Panel**->**TabletWorks**, high light the **16-Btn Cursor**, and select **Mouse** as the **Pointing Mode**, which lets the digitizer cursor moves relatively to the screen coordinates. This step is indicated in the following **TabletWorks Control Panel** dialog.

The next is to set up the pointing device in Carlson Takeoff. Open up Takeoff and go to pull-down **Settings**->**Preferences**, click tab **System**, select **Wintab Compatible Digitizer** as **Current Pointing Device**, and set the **Accept input from** to **Digitize and mouse**. Please refer to the following **Options** dialog.
Now, you are ready to use your digitizer. On the bottom of the screen, there is a tray icon TABLET on the right side of MODEL. You can use accelerator key **F4** to toggle on/off the tablet.

**Save Tablet Calibration**

**Function**

This command saves current tablet calibration to a file. You are prompted to enter a file name.

**Keyboard Command:** tablet1

**Prerequisite:**
Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.
Load Tablet Calibration

Function

This command restores the tablet calibration parameters from a file and load it into the current drawing. You are prompted to specify a file name.

**Keyboard Command:** tablet2

**Prerequisite:** Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. The calibration file should be associated to the current drawing, and the current drawing shouldn't have been moved on the digitizer board since last calibration.

Digitizer Settings

This command allows you to select the puck layout and set Auto On/Off features.

![Digitizer Settings](image)

**Auto Tablet On For Digitize Commands** means after you select a digitize command your puck will automatically be put in Digitize Mode. If this is toggle off, then you will need to turn Tablet on seperately from running a digitize command.

**Auto Tablet Off After Digitize Commands** means you will return to Mouse Mode after running a digitize command. Read below for more on Mouse and Digitize Mode.
Puck Layout

The 16-button puck can be used as either a mouse or a digitizer. It's very important to understand how the 16 buttons are mapped in both modes.

**Mouse Mode:**

When the tablet is off, the puck is in Mouse Mode. The top-left button is the left mouse click, and the top-right button is the right mouse click. The labels on the other buttons do not mean anything. All buttons are mapped as same as the buttons of the default pointing device in AutoCad. Please refer to AutoCad Reference manual for further information.

**Digitize Mode:**

When tablet has been calibrated and is on, the puck is in digitize mode. It is mapped as a small keyboard, which enables you to enter numerous values such as elevation, thickness and offset etc., and also provide you some functionality to digitize various entities. Currently there are two puck layouts in Takeoff, shown in the figure below. After you install Carlson Takeoff and finish setting up the digitizer, you go to the pull-down menu **Digitize->Puck Layout** to select a 16-button puck layout. A button mapping would be created and Takeoff would recognize the buttons as represented.
Layout 1 is Carlson Puck Layout, which is the most common layout used in Carlson Takeoff. Layout 2 is for users who don't have a Carlson Puck. If your puck is different than these two layouts, please contact Technical Support for help setting the mapping for your 16 button puck.

**Prompts**

**Digitizer Settings Dialog**
Specify the Digitizer Puck Layout to layout 1 or 2

**Prerequisite:** Have a digitizer board and a puck connected to your computer, and have Wintab driver installed.

**Keyboard Command:** dig_config

**Digitize Existing**

**Function**

This sets the layer target to existing. Set this prior to running any digitizing command and anything you digitize will be assigned for your existing surface. Checkout the Define Layer Target/Material/Subgrade command under Tools for more on targets.

**Keyboard Command:** set_digit_exist

**Prerequisite:** none
Digitize Design

Function

This sets the layer target to design. Set this prior to running any digitizing command and anything you digitize will be assigned for your design surface. Checkout the Define Layer Target/Material/Subgrade command under Tools for more on targets.

Prerequisite: none

Keyboard Command: set_digit_final

Digitize Other

Function

This sets the layer target to other. Set this prior to running any digitizing command and anything you digitize will be assigned to the Other target. Checkout the Define Layer Target/Material/Subgrade command under Tools for more on targets.

Keyboard Command: set_digit_other

Prerequisite: none

Digitize Point

Function

This command allows you to digitize individual points one at a time. The first time it prompts you the Digitize Points Dialog for entering point symbol styles, point prompt settings and number settings, starting point number and layer name. If you want to enter the elevation and description for each point, select Prompt for Descriptions and Prompt for Elevations. After having digitized a point, you can continue to digitize next point by picking the point on the drawing. The command defaults to the last layer name, point symbol, elevation, description and the last point number plus 1. If you have finished digitizing points, press Enter to finish.
Prompts

Digitize Points Dialog
Specify a layer name and select the point symbol, point prompt settings and number settings.
Pick point to create (Enter to end): pick a point on the drawing
Select/<Enter Point Elevation < > : enter the elevation or type <Select> to select the elevation text on the screen
Enter Point Description < > : enter the point description
Result like "N: 1231.16 E: 1099.17 Z: 30.00" would be display on the command line, and a point would be drawn on the screen with the text of its number, elevation and description.
Pick point to create (Enter to end): pick next point or press Enter to finish digitizing points

Keyboard Command: dig pt
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Digitize Spot Elevation

Function
This command allows you to label points with their elevation. The point can either be digitized from a drawing, picked on a screen or specified by a point number. The command first prompts you the **Label Spot Elevation Dialog** for entering layer name, label prefix and suffix and symbol types etc. Click OK to start. After specifying the point, the command prompts you to enter the elevation if its elevation is unknown and then pick an angle from the location of the point to label the elevation. You can repeat labeling points until you press **Enter** to finish.

![Label Spot Elevation Dialog](image)

**Prompts**

**Label Spot Elevation Dialog**  
Specify a layer name, label prefix and suffix and select the spot symbol.

**Point to Label ?**

**Pick point or point number:** 2 *(enter a point number)*

PointNo. Northing(Y) Easting(X) Elev(Z) Description  
2 1231.16 1099.17 30.00 bb

*Note:* if the point number you entered is not in the drawing, you will be prompted again to pick point or enter a point number.

**Elevation <30.000>:** press enter

**Pick angle for label:** pick an angle from the spot
Point to Label (ENTER to End)?
Pick point or point number: pick a point on the drawing
Elevation <0.000>: enter elevation
Pick angle for label: pick an angle from the spot
Point to Label (ENTER to End)?
Pick point or point number: press enter to finish

Keyboard Command: labspot
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab
driver installed. The digitizer has been correctly set up. Have done tablet calibration for current
drawing.

Digitize 2D Polyline

Function

A 2D polyline is a line of connected points that have the same elevation. This command lets you
digitize a 2D polyline by picking points along the lines on the drawing. It prompts you first the
Polyline 2D Options Dialog for entering the layer name. Prompt For Polyline Elevation option
allows you to enter the elevation for each polyline, otherwise all 2D polylines have 0.0 elevation.
Auto-Zoom mode would automatically zoom the display to center around the last point when
you get near the edge of the screen while picking points. There are three ways to enter a layer
name, Use current drawing layer, Select from a list of layer name, or Pick an entity on the
screen to get its layer name. While digitizing a polyline, the command keeps prompting you to
pick next point until your press Enter to finish digitizing, or press A on the puck or enter Close
on the keyboard to close the polyline on itself. If you make a mistake, press B on the puck or
enter Undo on the keyboard to remove the mistake and then continue to digitize. After finishing
a polyline, the command prompts your to digitize another polyline until you press B or enter No.
Prompts

Polyline 2D Options Dialog
Enter the layer name and select the options of Prompt For Polyline Elevation and Auto-Zoom mode etc.

Enter default elevation <0.00>: 100
First point: pick a point on the drawing using puck
Segment length: 0.00, Total length: 0.00
Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Segment length: 119.03, Total length: 119.03
Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Segment length: 121.76, Total length: 240.80
Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Segment length: 115.23, Total length: 356.03
Close[A]/Undo[B]/Pick next point (Enter to end): press enter to finish digitizing or press A to close the polyline

Digitize Another FINAL Polyline [Yes(A)/<No(B)>]? press A on the puck or enter Yes on the keyboard to digitize next 2D polyline, press B on the puck or enter No on the keyboard to finish digitizing 2D polyline.

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.
**Keyboard Command:** dig_2dp

**Digitize 3D Polyline**

**Function**

A 3D polyline is a line of connected points that have various elevations, and the slope between points is constant. It can be used in defining pads, excavations, drainage ditches and slopes from proposed design features to meet existing site conditions. This command lets you digitize a 3D polyline by picking points along the lines on the drawing. It prompts you first the **Polyline 3D Options Dialog** for entering the layer name. **Elevation Adder** allows you to truncate the elevations you have to enter in by add a given amount to them. There are five ways to enter elevations: known elevation of the point, interpolate, slope from previous point, ratio from previous point and degree from previous point. You can choose one of the methods between picking points. **Auto-Zoom** mode would automatically zoom the display to center around the last point when you get near the edge of the screen while picking points. While digitizing a polyline, press A to interpolate the elevation or B to enter it in. The command keeps prompting you to pick the next point until your press **Enter** to finish digitizing, or press A on the puck or enter **Close** on the keyboard to close the polyline on itself. You can also use AutoCAD's OSNAP command to pick points by pressing the decimal [.] button on the digitizer puck. If you make a mistake, press B on the puck or enter **Undo** on the keyboard to remove the mistake and then continue to digitize. After finishing a polyline, the command prompts you to digitize another polyline until you press B or enter **No**.
Prompts

First point:
Interpolate[A]/screen Pick/<Elevation[B]> <0.00>: 256

Z: 256.00
Close[A]/Undo[B]/Osnap[.] Pick next point (Enter to end): Pick point
Slope/Ratio/Interpolate[A]/Degree/screen Pick/<Elevation[B]> <256.00>: A

Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.] Next point or elevation<Interpolate>: Pick point
This point elevation will be interpolated upon completion.
Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.] Next point or elevation<Interpolate>: 279

Z: 279.00, Hz dist: 30.01, Slope dist: 37.81, Slope: 76.6% Ratio: 1.3:1
Close[A]/Undo[B]/Osnap[.] Pick next point (Enter to end): Pick point
Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.] Next point or elevation<Interpolate>: Press Enter

Z: 279.00, Hz dist: 24.18, Slope dist: 24.18, Slope: 0.0% Ratio: 0.0:1
Close[A]/Undo[B]/Osnap[.] Pick next point (Enter to end): A

Digitize Another EXIST_PLINE Polyline [Yes(A)/<No(B)>]? B
<Tablet Off>

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.
Keyboard Command: dig.3dp

Digitize Perimeter

Function

Perimeter is a 2D polyline that all points on it have the same elevation. It can be used as boundary polyline of your targets on your drawing. This command allows you to digitize a perimeter by
picking points on the drawing. While digitizing a polyline, the command keeps prompting you to pick next point until your press Enter to finish digitizing, or press A on the puck or enter Close on the keyboard to close the polyline on itself. If you make a mistake, press B on the puck or enter Undo on the keyboard to remove the mistake and then continue to digitize. After finishing a perimeter, the command prompts your to digitize another polyline until you press B or enter No.

Prompts

First point: pick a point on the drawing using puck
Segment length: 0.00, Total length: 0.00
Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Segment length: 104.27, Total length: 104.27
Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Segment length: 153.14, Total length: 257.41
Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Segment length: 104.89, Total length: 362.30
Close[A]/Undo[B]/Pick next point (Enter to end): press Enter to finish the perimeter, or press A to close the perimeter

Digitize Another PERIMETER Polyline [Yes(A)/<No(B)>]? press A or enter Yes to continue digitizing another perimeter, press B or enter No to finish digitizing perimeters.

Keyboard Command: dig_perim

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Digitize Areas

Function

This command allows you to find an area in digitize mode. With the puck, pick around the area you wish to calculate. If Draw Perimeter Polyline is toggled on then the linework of your perimeter will be displayed. You can then set the Layer Name and choose to label the Perimeter and Area and enter in an Area Description. You can also set the area you created as a Boundary,
Prerequisite: a digitizer
Keyboard Command: dig_area

Digitize Contour Polyline

A contour is a line of points with a constant elevation, representing the natural contour of the site. In Takeoff, there are two layer targets: Existing Ground Surface and Design Surface. Contour Polyline has two sub-command to digitize contour lines into Existing Contour and Final Contour layers directly for assigning them easily into Existing Ground Surface and Design Surface in the future analysis.

There are two ways to digitize contour lines: sketch mode or point mode. You can start digitizing a contour with one mode and switch to the other during digitizing the contour. Sketch mode uses more points than pick mode. In general, we recommend using pick mode to digitize the straight parts of lines because it reduces the number of points and speeds up Takeoff’s calculations, but using sketch mode to digitize the curved parts because it is fast and accurate.

Function

This command lets you digitize contours as polylines one at a time. The first time it prompts
you the Digitize Contours Dialog. Enter the layer name or select it from a list of existing layer. Look at your plans and determine an elevation interval that is between most of the contours and enter it in the Elevation Interval field. You are able to modify both the value and the direction of the elevation interval between digitizing contour lines, using the buttons on the puck. To have Takeoff automatically close contours whose beginning and ending points are within a specified range, check the Auto Detect Close Contour. Draw Labels would draw the elevation at the starting point of the contour. In Pick mode, if you want the Takeoff to automatically zoom the display to center around the last point when you get near the edge of the screen while picking points, check the Auto Zoom Center. Click OK to start digitizing.

If this is your first time digitizing a contour, you are defaulted to the Pick Mode digitizing, otherwise you would be defaulted to the last digitize mode. If you want to use the other digitize mode, press 0 on the puck or enter 0 from the keyboard. Place your cursor at one end of the contour line and begin digitizing the line. While digitizing a line, you can force a contour to close on itself by pressing A on the puck to end the contour and connect the last point to the first point, remove a mistake by pressing B on the puck, or switch to the other digitize mode by pressing 0. During Sketch Mode digitizing, you can stop digitizing by pressing Pick or Enter button on the puck, take some rest or changes, and start sketching again. At the end of the contour line, press Enter on your puck or keyboard. The contour is completed, and the elevation for the next contour is automatically incremented. You would be asked to digitize next contour. If you press A on the puck or enter Yes on the keyboard, you can digitize another contour, or press B on the puck or enter No on the keyboard to finish digitizing contours.

Prompts

Chapter 1. Takeoff Module 116
Digitize Contours Dialog
Enter Layer Name, Elevation Interval, and toggle on/off Auto Detect Close Contour etc.
Increment(1.00)[A]/Direction(+)[B]/Elevation <573.00>: 450 (enter elevation or press Enter to accept current value)

Start Digitizing...

Sketch[0]/Pick the first point: pick a point to start Pick Mode digitizing (press 0 to switch to Sketch Mode)
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): 0(press 0 on the puck or enter 0 on the keyboard to use Sketch Mode)

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): pick and drag
Drag to digitize (Pick or press Enter to stop sketching)... pick or press Enter to stop sketching

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): B (undo the last point)
Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): B (undo the last point)
Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): pick and drag again
Drag to digitize (Pick or press Enter to stop sketching)... pick or press Enter to stop sketching

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): 0 (press 0 on the puck or enter 0 on the keyboard to use Pick Mode)

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): press Enter to finish digitizing

Digitize Another Contour [Yes(A)>/No(B)]? B(press B to finish digitizing)

**Prerequisite**: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

**Keyboard Command**: digcont_exist, digcont_final

Chapter 1. Takeoff Module
Digitize Sections

Function

This command allows you to digitize section lines and store the section data in the section file you have specified. The command first prompts you the Digitize Section Dialog. Enter the section file name and determine if you want to digitize second and third sections at the same station. Look at your plans and determines the station interval, which is used to automatically default to the next station value when digitizing a series of stations. If the grids at all the stations have the same base elevation, toggle on Use Fixed Base Grid Elevation. You can also toggle on Interpolate Zero Offset Elevation, Prompt for Subgrades, Prompt for Save for Each Section and Use Beeps with Prompts. Click OK to start digitizing.

Takeoff prompts you to calibrate the section sheet before you digitize the section lines. You pick three points and specify their offsets to the centerline and elevations in order to determine the horizontal and vertical intervals. Corners on the section grid are preferred reference points. Place your cursor at one end of the section line and begin digitizing the line. While digitizing a line, you can remove a mistake by pressing A on the puck or entering Undo on the keyboard. At the end of the section line, press Enter on your puck or keyboard. The station is completed, and the station value is automatically incremented. The command would prompts to digitize next section. You can press A on the puck or enter Exit on the keyboard to finish digitizing. If you want to continue to digitize next section, press Enter or enter the new station number. For every station after the first one, you can calibrate the grid sheet by picking one reference point and specify its offset and elevation. After you digitize the section lines on your drawing, all the section data would be saved in a section file (.sct).
Prompts

Digitize Section Dialog
Enter Section File Name, Station Interval, and toggle on/off Use Fixed Base Grid Elevation etc.
Section station to digitize <0.000>: press Enter to start with station 0.0 or enter a station number
Calibrate section sheet
Pick First section sheet reference point: pick a grid point of this station on your drawing
Enter offset <0.0>: press Enter to accept the offset or enter the offset of the point to the centerline
Enter elevation: 1030 (enter the Elevation of the reference point)
Pick Second section reference point: pick the second grid point
Enter offset: 0 (enter the offset of the point to the centerline)
Enter elevation: 1040 (enter the Elevation of the reference point)
Pick Third section reference point: pick the third grid point
Enter offset: 50 (enter the offset of the point to the centerline)
Enter elevation: 1040 (enter the Elevation of the reference point)
3 calibration points

Transformation type: Orthogonal Affine Projective

Outcome of fit: Success Exact Impossible
RMS Error: 11.49
Standard deviation: 2.38
Largest residual: 14.08
At point: 2
Second-largest residual: 14.08
At point: 1

Digitize break point for DRAWING1 section 0.000 (Enter to end): pick a point on the section line
Offset: -39.81 Elev: 1028.80
Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): pick a point on the section line
Offset: -9.94 Elev: 1030.03
Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): pick a point on the section line
Offset: 49.44 Elev: 1034.93
Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): press Enter to finish

Save changes to DRAWING1 section 0.000 [<Yes(A)>/No(B)]? A (press A or B)

Exit[A]/Section station to digitize <50.000>: 200 (enter next station number)

Calibrate next section
Pick section reference point: pick a grid point of the station on your drawing
Enter offset <0.00>: press Enter to accept the offset or enter the offset of the point to the centerline
Enter elevation <1030.00>: 1020 (enter the Elevation of the reference point)
Digitize break point for DRAWING1 section 200.000 (Enter to end): pick a point on the section line
Offset: -40.40 Elev: 1008.07
Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): pick a point on the section line
Offset: -5.38 Elev: 1019.98
Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): pick a point on the section line
Offset: -5.38 Elev: 1019.98

Chapter 1. Takeoff Module
Digitize break point for DRA WING1 section 200.000 (Undo[A],Enter to end): pick a point on the section line
Offset: 27.86 Elev: 1030.02
Digitize break point for DRA WING1 section 200.000 (Undo[A],Enter to end): press Enter to finish
Offset: 50.33 Elev: 1035.80
Digitize break point for DRA WING1 section 200.000 (Undo[A],Enter to end): press Enter to finish
Save changes to DRA WING1 section 200.000 [<Yes(A)>/No(B)]? A (press A or B)

Exit[A]/Section station to digitize <250.000>: A (press A to finish or enter the station number to continue)

Keyboard Command: digxsec
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Digitize End Areas

Function

There are two types of end areas: cut area and fill area. This command allows you to digitize both cut area and fill area on the drawing and writes data to a .ew file. The command first prompts you to calibrate the section sheet by picking three points and specify their offsets to the centerline and elevations in order to determine the horizontal and vertical intervals. Corners on the section grid are preferred reference points. Then it prompts you to digitize the cut area and fill area respectively. Place your cursor at one end of the end area and begin digitizing the outline of the area. At the end of the section line, press Enter on your puck or keyboard. The end area is completed, and its area is printed on the command line, and you are prompted to digitize next end area. After you finish all the end area at one station, accumulated cut area and fill area are computed and printed out on the screen. All data of cut area and fill area at every station would be saved in the area file (.ew) that you have specified.

Prompts
Calibrate section sheet

Pick First section sheet reference point: pick a point on the drawing
Enter offset <0.0>: press Enter to accept the offset (or enter the offset of the point to the centerline)

Enter elevation: 1020 (enter the Elevation of the reference point)
Pick Second section reference point: pick a point
Enter offset: 0 (enter the offset of the point to the centerline)
Enter elevation: 1030 (enter the Elevation of the reference point)
Pick Third section reference point: pick a point
Enter offset: 50 (enter the offset of the point to the centerline)
Enter elevation: 1030 (enter the Elevation of the reference point)

3 calibration points

Transformation type: Orthogonal Affine Projective

Outcome of fit: Success Exact Impossible
RMS Error: 11.69
Standard deviation: 2.40
Largest residual: 14.29
At point: 2
Second-largest residual: 14.29
At point: 3

Digitize cut area (Enter to end): pick a point that is on the outline of the cut area, 0*(0.211129 1030.76)
Digitize cut area (Enter to end): pick a point that is on the outline of the cut area, 1*(11.5804 1030.49)
Digitize cut area (Enter to end): pick a point that is on the outline of the cut area, 2*(17.8643 1030.73)
Digitize cut area (Enter to end): pick a point that is on the outline of the cut area, 3*(19.0216 1032.35)
Digitize cut area (Enter to end): pick a point that is on the outline of the cut area, 4*(-0.777246 1030.75)
Digitize cut area (Enter to end): press Enter to finish

End area: 17.2312
Accumulated Cut Area: 17.2312

More Cut Areas [Yes(A)/<No>(B)]? press A to digitize more Cut Areas, or press B to finish digitizing Cut Areas.

Chapter 1. Takeoff Module
Accumulated Cut Area: 17.2312

**Digitize fill area (Enter to end):** pick a point that is on the outline of the fill area, 0*(-18.9614 1029.65)
**Digitize fill area (Enter to end):** pick a point that is on the outline of the fill area, 1*(-18.1315 1030.75)
**Digitize fill area (Enter to end):** pick a point that is on the outline of the fill area, 2*(-11.9592 1030.49)
**Digitize fill area (Enter to end):** pick a point that is on the outline of the fill area, 3*(-2.0676 1030.72)
**Digitize fill area (Enter to end):** pick a point that is on the outline of the fill area, 4*(-10.0082 1030.01)
**Digitize fill area (Enter to end):** pick a point that is on the outline of the fill area, 5*(-18.531 1029.67)

**Digitize fill area (Enter to end):** press enter to finish

End area: 8.64646
Accumulated Cut Area: 8.64646

**More Fill Areas [Yes(A)/<No>(B)]?** press A to digitize more Fill Areas, or press B to finish digitizing Fill Areas.

Accumulated Cut Area: 8.64646
Total Cut Area: 17.2312
Total Fill Area: 8.64646

**Store data to file [<Yes>(A)/No(B)]?** press A or B

Opened file: C:\Program Files\Carlson TakeOff 2004\DATA\Drawing1.ew

**Station Number: 1 (enter Station Number)**

Data Stored in file: C:\Program Files\Carlson TakeOff 2004\DATA\Drawing1.ew

**Digitize another station [Yes(A)/No(B)]?** B (press A or B)

**Prerequisite:**
Have a digitizer board and a puck connected to your computer, and have Wintab driver installed.
The digitizer has been correctly set up. Have done tablet calibration for current drawing.

**Keyboard Command:** digendar
Drillhole Menu

This chapter provides information on using the commands from the Drillhole menu to produce, import and edit drillhole strata settings, place drillholes, make strata surfaces and draw strata surfaces.

Drillhole Strata Settings

Function

This command selects drillhole symbols, defines strata, and determines how you place drillholes.

Note: The order in which the Strata are defined in the Strata Definitions list will be the default order for the strata when you create new drillholes through Place Drillhole.

The dialog box below shows the layout of the Drillhole and Strata Settings.

![Drillhole and Strata Settings Dialog Box](image)
• **Select Symbol:** Select a symbol to represent the drillhole location on the screen.

• **Symbol Name:** This name corresponds to the symbol selected.

• **Symbol Size:** This field can be edited to adjust the symbol's size displayed on the screen.

• **Strata Definitions:** This is not directly editable. Select the Strata you are interested in by highlighting it, then select the Edit button.

• **Add:** This option adds additional strata to the available Strata name list. See Edit Strata dialog box below.

• **Edit:** Similar to Add, this option is available to make changes to the Strata, including adding a swell factor.

• **Strata Name:** The name of the strata.

Density (lbs/ft³): The Strata Density field is the default density used to calculate strata tons. Density is strata-specific.

• **By Depth:** This option will generate a strata surface by modeling the strata depth values in the drillholes. This strata surface will follow the existing ground surface at the model depth.
• **By Strata Elevation:** This option will generate a strata model that connects strata irrespective of the upper surface elevation changes.

• **Remove:** This will remove a strata name from the available strata.

• **Move Up:** This option will move the selected strata name up one place in the strata name list.
• **Move Down:** This option will move the selected strata name down one place in the strata name list.

• **By Strata Elev:** This method will generate a strata surface by modeling the strata elevation values from the drillholes. This strata surface is independent of the existing ground surface.

• **Place Drillhole Prompts:** If Depth is selected, then when you run Place Drillholes you will be prompted for the depth of each strata in your drillhole. If Thickness is selected, you will be prompted for the thickness of each strata. If Dialog is selected, you will go straight into the Place Drillhole dialog when you create a drillhole.

• **Default Last Thickness:** Will set the thickness of your bottom strata to the same amount for all your drillholes.

**Keyboard Command:** tk_chdef

**Prerequisite:** strata information

## Drillhole Import

### Function

This command allows you import existing drillhole files. When you select Drillhole Import from the Drillhole menu, a command prompt shows:

"Use seperate drillhole and strata files [Yes/<No>]?" If you have two separate files, one with strata info, and the other file has drillhole locations, select Yes. If you enter Yes, the dialog box below appears.

This command creates drillholes from the data contained in text files. Currently there are many company-specific formats. A Drillhole Data File Formatter that is flexible to handle almost any drillhole text file format is below. The format to use is chosen in the dialog shown here.
The import text can have comma delimited, space delimited or fixed width columns of data. All the data for a record should be on one row. For the fixed width format, choose the Fixed Width toggle and then enter the column numbers separated by spaces in the edit box. For example, "8 15 24 32".

The Custom format can import all the drillhole and strata data from one text file or the drillhole data from one file and the strata data from another file. The method to use is set at the Use separate drillhole and strata files prompt.

Use the following commands to prepare a file format that will match the *.imp imported file.

- **Add**: Moves the selected entry from Available to Used.
- **Add Attribute**: Allows user input attributes into the Used section.
- **Add Skip**: Adds a "Skip" place holder in the Used List
- **Remove**: Moves a selected item from Used to Available list.
- **Move Up**: Moves the selected item up one place in the list.
- **Move Down**: Moves the selected item down one place in the list.
• **Comma Delimited:** Select this if your *.imp file has commas separating each field.

• **Single space delimited:** Select this if your *.imp has a space separating each field.

• **Tab delimited:** Select this if your *.imp file has tabs separating each field.

• **Fixed widths:** Select this if your *.imp has a defined width of space separating each field.

• **Auto Fixed widths:** Select this to automatically determine the fixed widths that separate each field in the *.imp file.

• **Header Lines to Skip:** If your *.imp file has header lines, enter the number of header lines here.

• **Load:** Takes you to select/brows for your *.imp file.

• **Save:** This command will save your imported file as a *.imp file.

The dialog box below details the drillhole import options.

In addition to the previously listed import commands above, this dialog box also has the
following prompts:

- **Avoid Duplicate Strata Names:** Select this to prevent having more than one strata with the same name.

- **Strata on one row:** Select this option if all of your strata info is on one row.

**Keyboard Command:** tk_chimport  
**Prerequisite:** drillhole files

---

**Place Drillhole**

**Function**

This command allows you to screen pick or enter coordinates the placement of a drillhole.

Go to **Drillhole/Strata Settings, Place Drillhole Prompts**, to determine how you would like to be prompted. When you select **Place Drillhole** from the **Drillhole menu**, the command line prompt shows:

"Pick Drillhole Location:" Type in x-y coordinates or move your pointer around the screen to pick the placement of the drillhole. If you are in Dialog Mode defined in **Drillhole/Strata Settings**, once a location was picked on the screen, the following dialog box appears:
Place DrillHoles generates drillholes in the drawing that are required to run strata surface application routines. Each drillhole consists of a surface elevation, strata, and optional description(s). Every strata has a name, bottom elevation, thickness. Within a drillhole, the strata names must be unique, but each real-world strata should have the same strata name across all the drillholes. This is because strata surface applications connects together the strata with the same name.

The drillhole data can be entered in the dialog shown here, or if Depth or Thickness Mode is selected under **Drillhole/Strata Settings**, then the data can be entered in on the command line when you place each drillhole. Make sure to specify the surface elevation and drillhole description. While in Dialog Mode or to change data, use the Edit and Insert/append buttons to enter strata data. The symbols are defined in DrillHole/Strata Settings and drillhole may be changed in Edit DrillHole. Pick Save when done and a drillhole symbol is drawn.

- **Edit**: Make changes to the highlighted strata name. Thickness, Bottom Elevation, Depth.

![Edit Strata](image)

When placing drillholes, every strata must be assigned a bottom elevation and a thickness. The bottom elevation is the elevation of the bottom of the strata. There are different methods for entering this information.

- **Insert Above**: To add a Strata above the highlighted strata name.
- **Append to Bottom**: To add a strata to the bottom of the available strata name list.
- **Remove**: Removes a strata from the available Strata Name list.

- **Surface Elevation**: This field can be set by you to establish the surface elevation of the drillhole.

- **Drillhole Name**: The name of the drillhole

- **Description**: Drillhole descriptions are intended for storing of drillhole specific information in the drillhole. One general drillhole description is predefined and user may define any number of specific drillhole descriptions. Typical additional descriptions are DRILLER, DATE, TOWNSHIP, and etc. You will be prompted for values of these descriptions in Place DrillHole.

- **Adjust Bottom Elevations**: Will make adjustments to the bottom elevation based on thickness changes.

- **Adjust Next Thickness**: Will adjust the next thickness to hold the bottom elevation unchanged.

- **Save**: This command saves this drillhole as listed.

- **Zoom In**: This increases the magnification of the black view window, cross-section view of the drillhole.

- **Zoom Out**: This decreases the magnification of the black view window, cross-section view of the drillhole.

- **Cancel**: Ends Drillhole placement routine without making changes.

**Keyboard Command**: tk.chplace  
**Prerequisite**: drillhole information

### Edit Drillhole

#### Function

This command allows you to screen pick an existing drillhole and edit its properties. When you select **Edit Drillhole** from the **Drillhole menu**, a command prompt shows:
"Select Drillhole to edit:" Move your pointer around the screen to pick the drillhole you want to edit. Once a drillhole is picked on the screen, the following dialog box appears:

- **Edit:** Make changes to the highlighted strata name. Thickness, Bottom Elevation, Depth.
• **Insert Above:** To add a Strata above the highlighted strata name.

• **Append to Bottom:** To add a strata to the bottom of the available strata name list.

• **Remove:** Removes a strata from the available Strata Name list.

• **Surface Elevation:** This field can be set by you to establish the surface elevation of the drillhole.

• **Drillhole Name:** The name of the drillhole

• **Description:** The screen display description of the drillhole

• **Adjust Bottom Elevations:** Will make adjustments to the bottom elevation based on thickness changes.

• **Adjust Next Thickness:** Will adjust the next thickness to hold the bottom elevation unchanged.

• **Save:** This command saves this drillhole as listed.

• **Zoom In:** This increases the magnification of the black view window, cross-section view of the drillhole.

• **Zoom Out:** This decreases the magnification of the black view window, cross-section view of the drillhole.

• **Cancel:** Ends Drillhole placement routine without making changes.

**Keyboard Command:** tk_chedit  
**Prerequisite:** drillhole information
Drillhole Reports

Function

This command allows you to generate a report of selected drillholes. When you select Reports from the Drillhole menu, a sub-menu choice of Standard Drillhole Report or Custom Drillhole Report, is displayed.

Standard Drillhole Report

If this is selected, several prompts are asked at the command line. They are as follows:

Select objects:

Add Page break between drillholes [Yes/<No>]?
Report Strata depth to [Top/<Bottom>]?
Report Strata elevation of [Top/<Bottom>]?
The report is then displayed accordingly.

![Drillhole Report]

Chapter 1. Takeoff Module
Custom Drillhole Report

This function allows you to customize your report format.

Prompts:

Command: tk_chreport2
Select the Drillholes for report.
Select objects: Specify opposite corner: 271 found
262 were filtered out.

Keyboard Command: tk_chreport, tk_chreport2
Prerequisite: drillholes
Make Strata Surface

Function

This command generates multiple strata surfaces based on strata definitions and placements of drillholes. Strata surfaces are generated at the bottom of each strata. These strata surfaces can then be used in other TakeOff commands like Calculate Total Volumes. They can be viewed on screen, through the command Draw Strata Surface.

Note: By observing the command line, one can see the status of each strata surface generation.

Keyboard Command: tk_chgrid
Prerequisite: Define Drillhole/Strata Settings, Place Drillhole

Clear Strata Surface

Function

This command clears the strata surfaces previously generated with Make Strata Surface. This removes the strata surfaces from processing in other takeoff commands.

Note: This command will not remove the surface from the screen view. You must use the command Erase Strata Surface to remove them from view.

Keyboard Command: tk_chclear
Prerequisite: Make Strata Surface

Draw Strata Cut Depth Contours

Function
This command will draw the **Strata Cut Depth Contours**. This command creates contours for the cut depth between the design surface and strata.

You must have created Strata Surfaces through the **Make Strata Surface** command.

Then select **Draw Strata Cut Depth Contours** from the **Drillhole menu**. You will be prompted to select the Strata from the dialog box below.

You can assign a contour interval and contour layer for the contours to be drawn.

**Keyboard Command:** tk_chdepth  
**Prerequisite:** Make Strate Surface

---

**Erase Strata Cut Depth Contours**

**Function**

This command will erase the Strata Cut Depth Contours from the screen display.
Keyboard Command: tk_chdepth2
Prerequisite: Strata Cut Depth Contours

Draw Strata Cut Color Map

Function

This command will generate a map of areas where the design surface cuts into the selected strata.

Prompts

Select point for color legend: - Use your pointing device to select the top left corner of where you want the cut color legend to be displayed.

Legend size <10.0>: Screen display size.
Label all zones or summary [All/<Summary>]? This pertains to the the number of elevation labels on the legend.

Keyboard Command: tk_chmap
Prerequisite: Make Strata Surface

Erase Strata Cut Color Map

Function
This command will erase all Strata Cut Color information from the screen display.

**Keyboard Command:** tk_chmap2  
**Prerequisite:** Draw Strata Cut Color Map

---

**Draw Strata Surface**

**Function**

This command will display the selected strata surfaces as 3D faces. The bottom elevation of the strata is drawn.

![Select Strata To Draw](image)

A color can be selected to distinguish each strata.

**Keyboard Command:** tk_chplot  
**Prerequisite:** Make Strata Surface

---

**Erase Strata Surface**

**Function**

This command will erase all strata surface 3D faces from the screen display.
Keyboard Command: tk_chplot2
Prerequisite: Draw Strata Surface

Trench Menu

Input Trench From Polyline

Function

This command allows you to input a trench sewer network structure from polylines. It first prompts you the Input Trench from Polyline Dialog where you specify the Trench Type, Trench System, and the System Name. The Individual Profile option lets you input one trench reach at a time and save its information to a profile (.pro). The Connected Network option lets you input all the trench polylines on the drawing, merge them into a trench network structure and save the whole structure to a .sew file. For trenching or utilities without Invert-Ins, uncheck Prompt For Invert-In Elevation. If you want to set the Rim Elevation to any surface elevations, check on Default Rim Elev to Surface Elev and then use the Surface Button to select the desired .tin or .flt surface file. Click OK to start inputting trench structure.

Pick a polyline that represents a trench reach on your drawing. Takeoff extracts the coordinates of all the vertices of the polyline and prompts you for the starting station number. Takeoff would computer the station values based on the starting station number. Next, you are prompted to enter the Manhole ID (Sewer Trench) or Station ID (Pipe Trench), Invert Elevation, Manhole Elevation (Sewer Trench), and Pipe Size for every station. You can either enter the values manually or select the texts that represent these values on the drawing. When you finish inputting a polyline, the command would ask you for a profile name to store the profile data if you are doing Individual Profile; otherwise the command would ask you to pick next polyline that is in the same trench network.
Prompts:

Pick a polyline that represents a trench reach: pick a polyline on your drawing
Starting Station of trench reach <0.0>: press Enter to accept 0.0 as the starting station or enter a value

For station 0.00 ...
Enter/<Select text of Manhole ID>: select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually
Enter/<Select text of invert elevation>: select the invert elevation text on the drawing or enter Enter on the keyboard to enter the invert elevation value manually
Enter/<Select text of manhole elevation>: select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manhole elevation value manually

For station 270.22 ...
Enter/<Select text of Manhole ID>: select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually
Enter/<Select text of invert elevation>: select the invert elevation text on the drawing or enter Enter on the keyboard to enter the invert elevation value manually
Enter/<Select text of manhole elevation>: select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manhole elevation value manually
Enter/<Select text of pipe size>: select the pipe size text on the drawing or enter Enter on the keyboard to enter the pipe size value manually

For station 425.02 ...
Enter/<Select text of Manhole ID>: select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually
Enter/<Select text of invert elevation>: elect the invert elevation text on the drawing or enter Enter on the keyboard to enter the invert elevation value manually
Enter/<Select text of manhole elevation>: select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manhole elevation value manually
Enter/<Select text of pipe size>: select the pipe size text on the drawing or enter Enter on the keyboard to enter the pipe size value manually

For station 649.73 ...

Another Polyline [Yes]/No? enter Yes to input another trench reach from a polyline or enter No to finish

At the end of the command, a file opening dialog would be prompted to you to specify a .sew file name to store the trench network structure.

Prerequisite: A drawing with one or more polylines that represent the trench structure.
Keyboard Command: pline_trench

Create Trench Network Structure

Function

This command allows you to create or modify a trench network structure on a drawing. Before you are able to locate the trench structure, the drawing has to be open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. You can locate the trench structure by one of three methods: picking points on the drawing, entering the point number, or specifying the station and offset of a centerline. If you use centerline method, you need to specify a centerline file. After you locate a point on the drawing, you are prompted the Sewer Structure Data Dialog
for entering the sewer structure information, such as Structure Name, System Name, Symbol Name, and Elevations. Take a look at the list of the trench points that have been defined. If there is any point that is connected upstreamly to the current point, you add it to the Upstream Connections list. The Invert Elevation and the Pipe Size fields will be filled with the information of the upstream point. Click OK to finish entering the trench structure data. The command will repeatedly ask you to pick a structure point until you hit Enter to finish. The trench network structure data is saved in a .sew file.

![Trench Structure Data](image)

**Prompts**

**By Pick:**

Locate by pick point, point number or station-offset [<Pick>/Number/CL]? press Enter to do Pick point

Loading edges...
Loaded 4 points and 5 edges
Created 2 triangles
Pick structure location: pick a point
Sewer Structure Data Dialog: enter trench structure information
Pick structure location (Enter to end): pick a point
Sewer Structure Data Dialog: enter trench structure information
Pick structure location (Enter to end): pick a point
Sewer Structure Data Dialog: enter trench structure information
Pick structure location (Enter to end): pick a point
Sewer Structure Data Dialog: enter trench structure information
Pick structure location (Enter to end): press Enter to finish

By station-offset of CL:

Locate by pick point, point number or station-offset [<Pick>/Number/CL]? CL (enter CL to do locating trench structure by station-offset of a centerline)
Specify a centerline file.

Loading edges...
Loaded 4 points and 5 edges
Created 2 triangles

Structure Station: 0 (enter the station number on the centerline)
Structure Offset: 200 (enter the offset from the centerline)
Sewer Structure Data Dialog: enter trench structure information
Structure Station (Enter to end): 100 (enter the station number on the centerline)
Structure Offset: 200 (enter the offset from the centerline)
Sewer Structure Data Dialog: enter trench structure information
Structure Station (Enter to end): press Enter to finish

Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface.
Keyboard Command: locate_trench
Edit Trench Network Structure

Function

This command edits the existing trench structure data on the drawing. There has to be a trench network structure that has been created beforehand and its data is stored in a .sew file whose name is as same as the drawing name. The command first prompts you to pick a sewer structure on the drawing. If there is no such structure in the .sew file, you would get an error message like this: "Error: unable to locate structure in file C:\temp\takeoff\SANI1x.sew, otherwise this command will restore the trench structure data from the corresponding .sew file and display it on the Sewer Structure Data Dialog for editing. Click OK to confirm your modification. You are prompted to edit another structure point until you press Enter to finish. All modifications are saved in the .sew file.

Prompts

Loading edges...
Loaded 4 points and 5 edges
Created 2 triangles

Select sewer structure to edit: pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): press Enter to finish

Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.

Keyboard Command: edit_trench
Remove Trench Network Structure

Function

This command removes the existing trench structure data. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name. The command first prompts you to pick a sewer structure on the drawing or to select from a List of your Sewer Structures. If there is no such structure in the file, you will get a error message like this: "Error: unable to locate structure in file C:\temp\takeoff\SANI1x.sew, otherwise this command removes the structure from both the drawing and and the .sew file immediately. You are prompted to remove another structure point until you press Enter to finish. The removed trench structure points would no longer be found in the .sew file.

Prompts

Select structures to erase by screen pick or name list [<Pick>/List]? Pick to choose from the screen, or List to choose from the below dialog.

Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.

Keyboard Command: remove_trench
Find Trench Network Structure

Function

This command will center the screen and draw an arrow to the structure you specify.

Prerequisite: a Trench Network
Keyboard Command: findswr

Export Trench Network Data

Function

Export to Points

This command will add points at your trench structures and add them into your coordinate file by either the Rim Elevation or the Invert-Out.

Export to Profiles
This command will create a profile file (.pro) of your trench either going Upstream or Downstream. The (.pro) file can then be drawn under Roads->Draw Profile.

**Prerequisite:** a Trench Network  
**Keyboard Command:** swr2pts, swr2pro

---

**Trench Network File Backup**

**Function**

*Save Trench Network File* saves your trench network as a (.sew) file. *Load Trench Network File* loads a previously saved (.sew) file.

**Prerequisite:** none  
**Keyboard Command:** save_trench, load_trench

---

**Plain View Label Settings**

**Function**

This command allows you to set the labeling for your structures and piping. The below dialog box gives you the option to display the Structure Name, the Rim Elevation, the Invert-In, and Invert-Out. In addition, you can set the Prefixes, Suffixes and labeling location as you so desire. The Use Structure Data Table will create linework around each Structure's labeling.
This below dialog box gives you the option to display the Length, Size, Material, and Slope for your Piping. In addition, you can set the Prefixes, Suffixes and labeling location as you so desire. To specify to which structure the label is intended for, select Arrow On Pipe, Parallel Leader, or None. You can also set the type of linework to draw.
In this dialog you can set the properties for your Symbol and Linework as well as the decimal places to report.

Prerequisite: a trench network

Keyboard Command: swrsetup

Draw Trench Network - Plan

Function

This command draws a trench network structure on the screen, based on the Plain Vew Label Setting command and the trench network structure data in the .sew file whose name is as same as the drawing name. If Takeoff couldn't find such file in the same directory where the drawing locates, nothing would be drawn on the screen.

Prerequisite: A open drawing
**Keyboard Command:** plan_trenc

**Draw Trench Network - Profile**

**Function**

This command allows you to draw a branch of the trench network structure as a sewer/pipe profile. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name, otherwise you would get a error message like "Error: no data in sewer network file". The command first prompts you the **Draw Sewer Network Dialog**. Select the Upstream and Downstream Struct that you want to draw. If you want to draw the existing and final design surface, as well as Strata Surfaces, toggle on Draw Existing Ground Surface, Draw Final Design Surface, and Draw Strata Surfaces options. If your profile is from upstream to downstream, then select the Profile Direction as Downstream, otherwise Upstream. You can also choose to save the profile data to a profile file. Click OK to draw.

![Draw Sewer Network Dialog](image)

**Prompts**

Loading edges...
Loaded 41041 points and 122901 edges
Created 81859 triangles

Loading edges...
Loaded 5487 points and 16332 edges
Created 10846 triangles

Initializing Draw Profile command ...
Draw Sewer Profile Dialog

Enter drawing parameters such as Grid scale, text scaler, starting and ending stations etc. for drawing the sewer profile.

Enter general sewer profile settings such as elevations (Rim, Invert-In, Invert-Out) to draw and label.
Use the Manhole tab to define what manhole information is labeled in your trench profile.
Use the Pipe tab to define what piping information is labeled in your trench profile.
The command will find the elevation range of your profile and display it at the top of this dialog. Here you can set the elevation top and bottom of the profile's grid.

Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.

Keyboard Command: profile_trench

Input-Edit Trench Template

Function

This command lets you create a new trench template or modify an existing trench template. It prompts you the Input-Edit Trench Template Dialog. If you are modifying a trench template, click the Load button on the dialog to open a trench template file and display the template data on the dialog. Enter the dimensions of the trench: bottom offset, trench width and vertical side height. The Edit Trench Benches button will bring up the below dialog, and allows you to enter in up to four benches into your trench.
There are three methods for entering the cut slope, Percent, Ratio and Degree. Choose one of the methods and enter the slope value. Display Sewer Structure allows you to see your pipe or manhole as part of the trench. This is for display purposes only, calculations will be drawn from the pipe size you set in the Trench Network Structure commands. Add Pipe Diameter To Trench Width will increase the size of your trench by the diameter of your different pipe sizes. There are three trench bottom backfill layers that can be defined. Enter the layer label in the material name field, the depth of the layer in the thickness field. Click Save or SaveAs to save the template information in a .tch file, and Click Exit to quit this command.
Prompts:

Input-Edit Trench Template Dialog
Enter the dimensions of the trench template, save the information to a template file (.tch).

Prerequisite: None
Keyboard Command: make_trench_tpl

Draw Typical Trench Template

Function

This command draws a trench template on the screen. After you select a trench template file (.tch) to draw, a Typical Trench Template Dialog is prompted for entering the layer name, drawing
scale, text size scaler and selecting how many decimal points you want. You can also hatch the backfill on the drawing. Click OK to draw the template at the position that you pick on the screen.

![Typical Trench Template](image)

**Prompts**

**Pick position to draw template:** pick a position on the screen

**Prerequisite:** None.

**Keyboard Command:** draw_trench_tpl

**Trench Subgrade Areas**

**Function**

The purposes of Trench Subgrades is to assign a different type of trench template when a trench passes under a road, building pad, etc.

**Set Trench Subgrade Polyines**

Choose a closed polyline that defines the area that you want a different trench template for, ie a building pad polyline.

**Prerequisite:** None.
**Keyboard Command:** tag_trench_subgrade

**Clear Trench Subgrade Polyines**

This command untags selected polylines for trench subgrade use.

**Prerequisite:** Trench Subgrade Polylines  
**Keyboard Command:** untag_trench_subgrade

**Hatch Trench Subgrade Area**

This command hatches trench subgrades for easy viewing.

**Prerequisite:** Trench Subgrade Polylines  
**Keyboard Command:** hatch_trench_subgrade

**Erase Trench Subgrade Hatch**

This removes previous made trench hatching.

**Prerequisite:** Trench Subgrade Hatching  
**Keyboard Command:** erase_trench_subgrade

---

**Trench Network Quantities**

**Function**

This command calculates the trench volumes. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name, otherwise you would get a error message like "Error: no data in sewer network file". The command loads the trench network data and split them into individual trench lines and display them on the Calculate Trench Quantities Dialog. You can choose to calculate the trench volume of one trench line or several trench lines at a time. If you select **Use Trench Template for Volumes**, you need to **Set Trench Main Template** and **Subgrade Template** if you have one. If you select Use Design Surface Profile to Minimize Cut, then the Depth Zones are depths from the final design surface, otherwise the Depth Zones are depths from the existing ground surface. If you have Strata Surfaces defined then the program can calculate cut volumes for a strata you...
select. Report Stations Depth Summary will report you your Trench depth zones by stations along the trench network. Trench Depths can be reported by either the bottom of the trench or bottom of the pipe by using the Depth Target pull-down. You can also color the trench in the drawing by defined zones. Click OK to compute the template volumes. Backfill quantities take into account pipe size. A report would be shown after the calculation.

Prompts

Loading edges...
Loaded 41041 points and 122901 edges
Created 81859 triangles
Loading edges...
Loaded 5487 points and 16332 edges
Created 10846 triangles

Trench Quantities Report Window

Draw zone map color legend on the screen [Yes/<No>]? y for Yes
Pick a point for color legend: pick a point away from site
**Prerequisite:**
Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.

**Keyboard Command:** `calc_trench`

**Report Trench Network**

**Function**
This command will a report the Name, Station Distance, Invert-In Slope, Invert-Out Width, the Rim Elevation, Trench Type, Manhole Depth, and the Area Direction for the selected Trench. You may also choose to report the Trench Network from Downstream or Upstream, or just the Structures.
Prerequisite: a sewer line
Keyboard Command: reportswr
Display Menu

Existing Drawing

Function

This command allows you to display all the entities on the layers that are grouped as part of the Existing Drawing.

Carlson TakeOff allows you to assign layers into three different "Target" surface groups: Existing, Design, and Other. For more about assigning layers to different "Target" surface groups see Define Layer Target/Material/Subgrade under the tools menu. Once layers have been assigned, the display menu allows for easy viewing of each "Target" surface. When Existing Drawing is checked than the existing drawing will be displayed. If it is not checked it will not be displayed. You can check on and off the other "Target" surfaces to view the existing drawing in isolation or in accordance to the other drawings.

Keyboard Command: set_display_exist_dwg
Prerequisite: Define Layer Target/Material/Subgrade

Existing Contours

Function
This command displays all the contours that represent the existing surface (For contouring options see Display Options). Clicking on Cut/Fill Labels from the menu runs the command and puts a check mark on the menu. Picking again turns it off.

When Existing Contours is checked than all the contours for the existing surface will be displayed. If it is not checked they will not be displayed.

**Prerequisite:** existing surface  
**Keyboard Command:** set_display_exist_ctr

### Existing Surface

### Function

This command allows you to display the surface triangulation for the existing drawing.
When Existing Surface is checked than all the triangulation for the existing will be displayed. If it is not checked, they will not be displayed.

**Keyboard Command:** set_display_exist_grd  
**Prerequisite:** an existing surface

---

**Design Drawing**

**Function**

This command allows you to display all the entitles on the layers that are grouped as part of the Design Drawing.
Carlson TakeOff allows you to assign layers into three different "Target" surface groups: Existing, Design, and Other. For more about assigning layers to different "Target" surface groups see Define Layer Target/Material/Subgrade under the tools menu. Once layers have been assigned, the display menu allows for easy viewing of each "Target" surface. When Design Drawing is checked than the design drawing will be displayed. If it is not checked it will not be displayed. You can check on and off the other "Target" surfaces to view the Design drawing in isolation or in accordance to the other drawings.

**Keyboard Command:** set_display_final_dwg  
**Prerequisite:** Define Layer Target/Material/Subgrade

---

**Design Contours**

**Function**

This command displays all the contours that represents the design surface (For contouring options see Display Options). Clicking on Cut/Fill Labels from the menu runs the command and puts a check mark on the menu. Picking again turns it off.
When Design Contours is checked than all the contours for the design will be displayed. If it is not checked they will not be displayed.

**Keyboard Command:** set_display_final_ctr

**Prerequisite:** design surface

---

### Design Surface

#### Function

This command allows you to display the surface triangulation for the design drawing.
When Design Surface is checked than all the triangulation for the design will be displayed. If it is not checked, they will not be displayed.

**Keyboard Command:** set_display_final_grd  
**Prerequisite:** a design surface

---

### Cut/Fill Contours

**Function**

This command compares the existing and design surfaces and shows the cut/fill contours in blue for fill and red for cut. There is a Draw Only Cut/Fill Daylight option as apart of the Display Options command (See Display Options for more information). Clicking on Cut/Fill Contours from the menu runs the command and puts a check mark on the menu. Picking again turns it off.
**Keyboard Command:** set_display_cf_ctrl  
**Prerequisite:** elevation differences between existing and design

---

**Cut/Fill Labels**

**Function**

This command displays the design elevation, the existing elevation, and the amount to either cut or fill right on the screen (See Display Options for information about labeling options). Picking on Cut/Fill Labels from the menu runs the command and puts a check mark on the menu. Picking again turns it off.
Keyboard Command: set_display_cf_txt
Prerequisite: existing and design surfaces

Cut/Fill Color Map

Function

This command compares the existing and design surfaces and shows the cut/fill regions in blue for fill and red for cut (See Display Options for information on pixel resolution). Clicking on
Cut/Fill Color Map from the menu runs the command and puts a check mark on the menu. Picking again turns it off.

**Keyboard Command:** set_display_cf_map

**Prerequisite:** existing and design surfaces

---

**Other Drawing**

**Function**

This command allows you to display all the entities on the layers that are grouped as part of the Other drawing.
Carlson TakeOff allows you to assign layers into three different "Target" surface groups: Existing, Design, and Other. For more about assigning layers to different "Target" surface groups see Define Layer Target/Material/Subgrade under the tools menu. Once layers have been assigned, the display menu allows for easy viewing of each "Target" surface. Typically, most layers are listed under Other before they are assigned to Existing or Design. Some layers, like perimeter, are neither apart of the Existing or the Design drawing so they remain under Other. When Other Drawing is checked than the entities grouped under Other will be displayed. If it is not checked it will not be displayed. You can check on and off the other "Target" surfaces to view the Other surface in isolation or in accordance to the other surfaces. In this example, Existing, Design, and Other are all shown.

**Keyboard Command:** set_display_other_dwg  
**Prerequisite:** Define Layer Target/Material/Subgrade

---

**Display Options**

**Function**

This command allows you to change the features of the different display commands. Note: You can toggle on/off the Existing, Design, and Other surfaces by right clicking with your mouse. To activate this feature type in "shortcutmenu" in the command line and then <1>. To turn off the feature type in <0>.
**Display Setup:** Here is the master list for the major things you can display, including: the Entities, Contours, and Surface for both the Existing and Design, Cut/Fill Displays, and Other Drawing Entities.
Contour Options: Here you can set the interval, the elevation difference between each contour, for both the Existing, Design by clicking on Design and Existing Contour Settings. The Smooth Contours option will apply smoothing to the contour polylines. You can also choose to draw only the daylight line between Existing and Design instead of the Cut/Fill contours at an interval.

Draw Contours

When this box is checked, the program will draw contour lines after triangulating. Otherwise, only the designated triangulation operations are performed. Specify the layer for contours in the edit box to the right.

Contour by Interval or Contour an Elevation

Select whether to contour by interval (ie: every 10 feet) or to contour a certain elevation. The elevation option allows you to contour specific values. For example, if you want just the 100ft contour, then select elevation and enter 100. The default mode is by interval.
Contour Interval

Specify the interval to contour. Note: If the above option is set to Contour an Elevation, then this field is used to specify the elevation to contour.

Minimum Contour Length

Contour lines whose total length is less than this value will not be drawn.

Reduce Vertices

This option attempts to remove extra vertices from the contour polylines which has the advantages of a faster drawing and smaller drawing size. Default is ON

Offset Distance

When the Reduce Vertices option is enabled, This value is the maximum tolerance for shifting the original contour line in order to reduce vertices. The reduced contour polyline will shift no more than this value, at any point, away from the original contour line. A lower value will decrease the number of vertices removed and keep the contour line closer to the original. A higher value will remove more vertices and allows the contour line to shift more from the original.

Hatch Zones

When activated, this option will allow you to hatch the area between the contours sequentially. A secondary dialog will load allowing the user to specify the hatch type and color.

Draw Index Contours

This option creates highlighted contours at a specified interval. When enabled, the fields for Index Layer, Index Interval and Index Line Width are activated.

Contour Smoothing Method

Select the type of contour smoothing to be performed. Bezier smoothing holds all the contour points calculated from the triangulation and only smooths between the calculated points. Polynomial smoothing applies a fifth degree polynomial for smooth transition between the triangulation faces. The smoothing factor described below affects the smoothing bulge.

Bezier Smoothing Factor

The contour preview window shows you an example of how much smoothing can be expected at each setting. Sliding the bar to the left results in a lower setting which have less looping or less freedom to curve between contour line points. Likewise, moving the slider to the right results in a setting that increases the looping effect.

Subdivisional Surfaces / Subdivisions Generation
This option causes each triangle in the triangulation surface model to be subdivided into an average of three smaller triangles per subdivision generation, with the new temporary vertices raised or lowered to provide smoother contours. More generations increases the smoothness of the algorithm at a cost of increased processing time. If Straight Lines are chosen as the contouring drawing method, then the contours are guaranteed never to cross. The original points of the surface model are always preserved. These modifications to the surface model are only for contouring purposes and are not written to the triangulation (.FLT) file or inserted into the drawing. If some contour movement is too small for appearance's sake, consider enabling Reduce Vertices.

Label Tab

![Takeoff Contour dialog box](image)

**Label Contours**

When activated, contours will be labeled based on the settings below.

**Label Layer**

Specifies layer name for intermediate contour labels.

**Index Label Layer**
Specifies layer name for index contour labels.

**Label Style**
Specifies the text style that will be used for the contour label text.

**Label Text Size Scaler**
Specifies the size of the contour labels based on a multiplier of the horizontal scale.

**Min Length to Label**
Contours whose length is less than this value will not be labeled.

**Break Contours at Label**
When checked, contour lines will be broken and trimmed at the label location for label visibility. When enabled, the Offset box to the right activates. The Offset determines the gap between the end of the trimmed contour line and the beginning or ending of the text.

**Draw Broken Segments**
When checked, segments of contours that are broken out for label visibility will be redrawn as independent segments. Specify the layer for these broken segments in the box to the right of this toggle.

**Label Contour Ends**
When checked, contour ends will be labeled.

**Draw Box Around Text**
When checked, a rectangle will be drawn around contour elevation labels.

**Label Index Contours Only**
When checked, only the index contours will be labeled. This option is active only when "Draw Index Contours" has been selected in the Contour tab of the main dialog.

**Hide Drawing Under Labels**
This option activates a text wipeout feature that will create the appearance of trimmed segments at the contour label, even though the contour is fully intact. This feature provides the user with the best of both worlds; you have clean looking contour labels, and the contour lines themselves remain contiguous. This feature will also hide other entities that area in the immediate vicinity
of the contour label.

**Align Text with Contour**

When checked, contour elevation labels will be rotated to align with their respective contour lines. This option also activates the Align Facing Uphill feature explained below.

**Align Facing Uphill**

When checked, contour elevation labels will still be rotated to align with their respective contour lines, but the labels will be flipped in such a manner that the bottom of the text label will always be toward the downhill side of the contours. So as the labels are read right side up, you are always facing uphill.

**Internal Label Intervals**

Choose between label intervals or distance interval. Label intervals will label each contour with a set number of labels. Distance interval lets you specify a distance between labels.

![Display Options](image)

**Cut/Fill Label Options**: Here you can customize the Cut/Fill labels. Text can be added
either before or after the Cut/Fill amount, the Existing elevation, and the Design elevation with the Prefix and Suffix fields. You can also choose whether or not to display the Existing Surface elevations, the Design Surface elevations and Strata Cut Thickness. Carlson TakeOff gives you the option to draw a marker symbol for where each label represents. You can also hide the drawing under the labels so that you can read the labels clearly. Text Size chooses the text size for each line of the label. Decimal Places sets to how many decimal places the labels will report. The Spacing of the labels can be determined by intervals or by a selected number of spaces. The size of each space is determined by the Text Size.

![Display Options](image)

**Cut/Fill Color Map Options:** Number Of Subdivision Rows is the number of blocks both horizontally and vertically in the Color Map. If the box reads 100 that means 100 blocks left to right and 100 blocks up and down or 10,000 total pixels. A higher the Number Of Subdivision Rows will make the Color Map sharper, however too high number can cause Carlson TakeOff to run slower. Auto Set Range will automatically set the red to blue scale for your cut/fill levels. However, if you desire greater contrast, then use Max Cut/Fill Range to manual set the range. Use lower numbers for greater contrast. The Daylight Color can be set to be either White or Green.

**Keyboard Command:** tk_display_options

**Prerequisite:** a drawing
Field Module
COGO Menu

The most of the commands in the COGO pull-down menu are described in the Survey manual. Only the few commands that are specific to Field are described here.

Tape Baseline

Function

This command creates points or linework along a baseline that is defined by two points. After specifying the baseline start and end points by either entering point numbers from the coordinate file or screen picking points, the program has a dialog with different methods for creating the points. The Tape method creates points at the specified chainage (distance) and offset from the baseline. reports the cut or fill between your current position and a design surface. The design surface can be one flat elevation, a grid file, a triangulation file, a road design file, or a section file. The Rectangle method draws a rectangle as a closed polyline using two points specified by chainage and offset from the baseline. The Square method draws a square as a closed polyline with a starting point at a baseline endpoint and the other corner specified by a distance along the baseline. The Divide method creates points at an interval between the baseline endpoints.
Pulldown Menu Location: COGO
Prerequisite: None
Keyboard Command: tape_bline
File Name: \lsp\gpsutil.arx

Cutsheet Spreadsheet Editor

Function

This command edits and reports cut sheet data that is stored in an Excel (.xls) file. To create this data with Field stakeout routines, the option to Store Cutsheet Data In Spreadsheet must be set active in Configure Field->Stakeout Settings.
Pulldown Menu Location: COGO
Prerequisite: None
Keyboard Command: editcutsheet
File Name: \lsp\gisprt.arx

Field Menu

The Field pull-down menu has the main functions for Field including equipment setup, storing points and stakeout.
Configure Field

Function

This command sets the equipment type, communication parameters and other Carlson Field options. Make sure the *Equipment Type* box shows the correct GPS or Total Station equipment that you'll be using. The down triangle button to the right of this box brings up a list of the equipment types to choose from. The eight buttons in *Configure* bring up the dialog boxes which are used to change Carlson Field's default settings. Explanations for each are shown below.
General Settings

If you are using a total station, *Rod Height* is the distance from the prism to the ground. For GPS, *Rod Height* is the distance from the center of the GPS antenna to the ground.

The *Show Carlson Field Startup Icon* controls whether the Carlson Field Startup Icon is displayed in the lower right of the screen. This startup icon brings up the Carlson Field function menu for launching Carlson Field commands without having to pick them from the pull-down menu.

The *Use Bold Font* toggles between using standard or bold font for the Carlson Field dialogs.

The *Twist Screen In Direction Of Movement* will rotate the drawing view so that your current direction of movement is facing straight up in the view. This rotate is for the view only and does not change the coordinates. This option only applies to GPS and robotic total stations in commands that show the arrow icon such as Track Position.

The *Station Type* chooses the format of centerline station labels. Typically 1+00 is used for feet units, 1+000 is used for metric and 100 has no plus symbol in the number.

*Serial COM Port* - The GPS receiver or total station attaches to your Carlson Field computer using a serial cable. This cable is plugged into a serial COM port on your computer called 1, 2, 3 or 4. Check the circle denoting the COM Port to be used.

The *Baud Rate*, *Parity*, *Char Length* and *Stop Bits* are the serial port communication parameters for the Carlson Field computer. These parameters need to match the parameters on the instrument that you are using. The *Defaults* button will set these communication parameters to the standard parameters for the current equipment type.
GPS Settings

The *RMS Tolerance* checks the RMS values when reading GPS positions. The RMS is the accuracy value reported by the GPS receiver. There are separate settings for the horizontal and vertical RMS values. The RMS (root mean squared) value means that the reported coordinate is within +/- the RMS value of the true coordinate to a certain confidence level. The confidence level depends on the GPS receiver. Typically it is a 98% confidence. If either RMS value exceeds the user-defined tolerance while storing points, Carlson Field will default to "No" when it asks if you want to store the point. You are required to choose yes to override the tolerance check and store the point.

Suggestion: When GPS RTK systems lose lock and go "Float", both the horizontal and vertical RMS values typically jump up to sub-meter (1' or higher) values. In Carlson Field, one foot is the default for the *GPS RMS Tolerance*. Some operators set the *GPS RMS Tolerance* low to 0.2 to check for high RMS values while still "Fixed".

*Store Fixed Only* - The position of the GPS rover is considered either "Autonomous", "Float" or "Fixed" based on the solution status from the GPS base corrections. When you are storing points and the *Store Fixed Only* box is checked, Carlson Field will only store points if your position is "Fixed". We suggest you leave this box checked. It ensures that you do not record inaccurate points.
Suggestion: When walking in light to heavy canopy, the rover might remain "Float" and display RMS accuracies of over a foot, sub-meter or more. Setting your GPS RMS Tolerance high and turning off Store Fixed Only will allow storing wetland and LOD (limits of disturbance) points under canopy that require only sub-meter tolerances. (USCG beacon DGPS sub-meter RTK GPS will always use these settings.)

**Projection Type** - defines the datum coordinate system to be used for converting the latitude/longitude from the GPS receiver into cartesian coordinates. For the United States two separate horizontal control systems have been developed by the Federal Government: State Plane 1927 and State Plane 1983. For international use the UTM (Universal Transverse and Mecator System) should be selected. The Lat/Lon option will convert the latitude/longitude from degrees minutes seconds format into decimal degrees. This option is useful when working in a decimal degrees lat/lon coordinate system.

**Zone** - For State Plane projections, you must select the correct state zone that you are working in. For UTM, the Automatic Zone option will have the program automatically use the correct UTM zone for your location. Otherwise for UTM, you can manually set a specific UTM zone. This manual option applies to working on the border between zones and you want to force the program to always use one of those zones.

Important: Coordinates of surveyed points will be inaccurate if the Projection Type and Zone settings are wrong. If you have done survey work and then realize that they are set wrong, then your point coordinates are wrong, but your work is not wasted. Carlson Field records the latitude, longitude and height of every point in a *.RW5 file. You can input the correct projection zone...
settings later and reprocess your data using the Edit-Process Raw File command.

**Model** - For UTM, this option sets the ellipsoid constants for converting the lat/lon to UTM coordinates. The following is a list of the models:

**Model Earth Radius(m) Flattening factor**
- Airy 1830 6377563.396 0.00334085064038
- Modified Airy 6377340.189 0.00334085064038
- Bessel 1841 6377397.155 0.00334277318217
- Clarke 1866 ellipsoid 6378206.4 0.00339007530409
- Clarke 1880 6378249.145 0.00340756137870
- Everest(EA-India 1830) 6377276.345 0.00332444929666
- Everest(EB - Brunei & E.Malaysia) 6377298.556 0.00332444929666
- Everest(ED - W.Malaysia & Singapore) 6377304.063 0.00332444929666
- International 1924 6378388.0 0.00336700336700
- Helmert 1906 6378200.0 0.00335232986926
- Hough 1960 6378270.0 0.00336700336700
- Geodetic Reference System 1980 6378137.0 0.00335281068118
- South American 1969 and Australian National 6378160.0 0.00335289186924
- World Geodetic System 1972 6378135.0 0.00335277945417
- World Geodetic System 1984 6378137.0 0.00335281066475

**Transformation** - The transformation in the Align Local Coordinates command can either be by plane similarity or rigid body methods. Both methods use a best-fit least squares transformation. The difference is that the rigid body method does a transformation with a translation and rotation and without a scale. The plane similarity does a rotation, translation and scale. This option only applies when two or more points are used in Align Local Coordinates.

**One Pt Align Azimuth** - This option applies to the rotation when using one point in Align Local Coordinates. For this alignment method, the state plane coordinate is translated to the local coordinate. Then the rotation can use either the state plane grid or the geodetic as north. No scale is applied in this transformation. The state plane and geodetic true north diverge slightly in the east and west edges of the state plane zone. This option allows you to choose which north to use.

**Two Point Align Method** - This option applies only two point alignments. Possible values are *Fit & Rotate and Rotate Only*. Fit & Rotate (the default) will use the second alignment point for rotation, translation, and scale (depending on the value set for Transformation). The Rotate Only option will use the second point of a two point alignment for rotation only.

**Geoid To Apply** - This option will account for the geoid undulation in determining the orthometric elevation of the measurement. The definition of the geoid model as currently adopted by the National Geodetic Survey is the equipotential surface of the Earth's gravity field which best fits,
in a least squares sense, global mean sea level. Orthometric elevation measurements are used in survey calculations. In order to convert ellipsoid heights (He) as measured by GPS into orthometric elevations (Eo), you must provide for a correction between the GPS-measured ellipsoid (reference ellipsoid) and a constant level gravitational surface, the geoid. This correction is the geoid undulation (Ug). The formula is \( He = Eo + Ug \).

The Geoid models are essentially large elevation difference models in grid format. Carlson Field has two geoid models available. Geoid99 covers the United States at 1 minute grid intervals. EGM96 covers the entire globe at 15 minute intervals. These Geoid models are huge and take a lot of disk space and memory. The Geoid model files are not installed automatically and instead need to be installed by going to the Geoid folder on the Carlson Field installation CD. Once installed onto Carlson Field, you then need to specify your location by lat/lon so that the program only needs to load a local portion of the Geoid model. To set your local Geoid area, pick the Set Geoid Area button. Setting the Geoid area will carve out a Geoid model around the specified lat/lon covering a square area of 2 degrees by 2 degrees which is about 100 miles by 100 miles.

Carlson Field applies the Geoid model by subtracting the Geoid undulation from the GPS elevation.

The resulting elevation is then used and displayed. In the Monitor function, the Geoid undulation is displayed.

In practice, the Geoid model is most applicable to two types of alignment scenarios. One of these types is when setting up the base over a known point and having no alignment control points. The other is when there is one alignment control point. When using multiple alignment control points, the Geoid model is not as important because Carlson Field can model the elevation difference which can generally pick up the local Geoid undulation.

**Project Scale Factor** - After converting the LAT/LONG from the GPS to the state plane coordinates and applying the Align Local Coordinates, the Project Scale Factor is applied as the final adjustment to the coordinates. This adjustment is used on the X,Y and not the Z. The Project Scale Factor is applied by dividing the distance between the coordinate and a base point by the Project Scale Factor. The coordinate is then set by starting from the base point and moving in the direction to the coordinate for the adjusted distance. The base point is the first point in Align Local Coordinates. If there are no points specified in Align Local Coordinates, then 0.0 is used as base point. The Project Scale Factor can be entered directly or calculated using the grid factor and elevation for the current position. When using the current position, the program will read the LAT/LONG from the GPS receiver. The scale factor is then calculated as: \( \text{State Plane Grid Factor} - (\text{Elevation/Earth Radius}) \).

**Default Alignment** - This option sets the alignment file to use by default for new drawings. This
feature applies when you will keep working at the same site with the same base receiver setup.

*Helmert 7-Parameter Transformation* - These settings apply when the Transformation is set to Helmert. The Helmert 7-parameters can either be calculated by the program using the control points in the localization or user-entered.

*Laser Offset Settings* - There is an option to use a laser for reading the distance and angle for offset points. When this option is enabled, you can choose the laser equipment type and communication parameters. The serial port for the laser must be different than the GPS which requires at least two serial ports on the computer. When using a laser for offsets, the program will read the current position from the GPS and then read the laser for the distance and angle to the point. This combination allows you to calculate points that cannot be directly reached by the GPS. There are two methods in the Point Store command to use the laser when this option is enabled. The Point Store dialog will have a new Laser button which will bring up another dialog that allows you to take multiple shots from the laser. The other method is to click on the Offset toggle in the Point Store dialog. Then when you do the Read function, the program will read the GPS position and then pop-up a dialog for taking one offset shot.

**Point Settings**

*Beep for Store Point* - This option will make a triple beep to indicate when a point is stored in the coordinate file.

*User-Entered Point Notes* - Point Notes are additional descriptions that can be stored with a point. A regular point consists of a point number, northing, easting, elevation and 32 character description. These points are stored in a .CRD file. Point Notes are a way to add an unlimited number of lines of text to a point number. With Point Notes ON in the Store Point command, the program will prompt for notes after collecting a point. The notes are stored in a file that uses the name of the coordinate file with a .NOT extension. For example, a coordinate file called JOB5.CRD would have a note file called JOB5.NOT.
Coordinates in Point Notes - When storing a point, this option will store the point number, northing, easting, elevation and description in the point notes as well. This may be used as a backup or reference to coordinate data as it was originally stored.

GPS RMS in Point Notes - When storing a point, this option will store the horizontal and vertical RMS values in the note field for the point. This offers a good check on the quality of the shot.

GPS DOPs in Point Notes - When storing a point, this option will store the DOP (dilution of precision) values as reported from the GPS receiver.

Rod Height in Point Notes - When storing a point, this option will store the rod height value in the note field for the point.

Project Scaler in Point Notes - When storing a point, this option will store the project scale factor in the note field for the point.

Time/Date in Point Notes - This option will store the time and date that the point was stored in the note file. Carlson Field will read the time from the computer.

Drawing Options control how points are drawn by default. It controls the layer, symbol number and whether points will be drawn with descriptions and elevations. Carlson Field's Field to Finish code table can override these defaults.

The symbol used for default points is displayed. You can choose another symbol by changing the
Symbol name or by selecting one from the table that the Select Symbol button brings up. Default point settings are used for points whose descriptions don't correspond to any category on the Field to Finish code table.

Label Descriptions and Label Elevations control whether these two items of information appear on your drawing next to each point.

Locate on Real Z Axis will record points with their true elevations. If this setting is off, all points recorded will have an elevation of zero.

Layer for Points indicates the layer where all default points will be drawn. For points using a code on the code table, the code table will determine their layer.

Number of Readings specifies how many times Carlson Field will read from the instrument in the Read function of the Point Store command. This applies to both GPS and total stations. The readings will be averaged to find a more accurate position.

Direct-Reverse Tolerances are used with total stations to check the pairs of direct and reverse horizontal angles, vertical angles and distances. When these values are off by more than the tolerance, the program will display a warning.

Field to Finish is explained fully in the Field to Finish command definition. Basically it uses a code table which holds information on types of points (ie. Man Hole or Edge of Pavement). When the settings Use Code Table...For Symbols, For Layers and For Descriptions are selected, Carlson Field will look to the code table for how to draw points of a particular code description.

The file containing the active code table appears after Code File: You can change this with the button Select File.

The Split Multiple Codes option will draw multiple points from the same point when that point description has multiple codes. For example, a point with description "EP DR" will draw the point twice: once with the properties of code EP and a second time using code DR. When this option is off, the program will use the first code and draw the point once.

The Check Descriptions With Code Table option will display a warning before storing a point if that point description is not found in the code table. With this option off, the program will go ahead and store the point and the point will be drawn using the default point properties.

Stakeout Settings

Display GPS RMS in Stakeout causes Carlson Field to report the constantly updating horizontal RMS accuracy values while staking a point. The only disadvantage to having this option active is that it slows down a little the stakeout position update.

Draw Trail displays a line in the stakeout screen showing where you've been as you move towards the stakeout point. This option only applies to GPS.
Auto Zoom will zoom the drawing display in or out so that both your current position and stakeout target are visible on the screen.

Zero Horizontal Angle To Target will set the horizontal angle of the total station to zero in the direction towards the stakeout point. When stakeout is completed, the horizontal angle is set back to the original value. This option only applies to total stations.

Store Cutsheet/Stakeout Data in Note File will store stakeout data in the note file (.NOT) for the current coordinate file. At the end of staking out a point, there is an option to store the staked coordinates in the current coordinate file. This stakeout note file option allows you to store more stakeout data in addition to the staked coordinates. This additional data includes the target coordinates and horizontal and vertical difference between the staked and target points. This stakeout note data can be used in reports with the List Points or CutSheet Report commands.

Store Cutsheet/Stakeout Data in Excel Spreadsheet will display a cutsheet report in an Excel spreadsheet. The spreadsheet will pop-up at the end of each point stakeout. The report can be saved in Excel format and processed by Excel.

Store Stakeout Points To Separate Coordinate File will store the staked points to a different coordinate file besides the current coordinate file. This allows you to use the same point number for the target and staked points. The staked point coordinate file can be specified by picking the Select Coordinate File button.

Check Total Station Turn Angle will compare the angle from the instrument and the angle to the target point. If this difference is greater than the specified tolerance, then Carlson Field will display a warning message.

Stakeout Tolerance controls the maximum difference between the target location and actual staked point. When the staked point is beyond the tolerance, Carlson Field displays a warning dialog.
**GPS Number of Reads for Final Avg** specifies how many times Carlson Field will read the GPS receiver position for the final staked point. These readings are averaged. Averaging several readings while occupying one point yields a more accurate result, but inevitably takes longer.

**Total Station Scale Settings**

These settings apply only to total stations. The *Project Scale Factor* is multiplied by the measured distance from the total station when calculating the foresight point coordinates. A typical project scale factor for working in state plane coordinates is slightly less than one. Factors greater than 2.0 or less than 0.5 are not allowed. The Project Scale Factor can be entered directly or choose the Calculate button. The Calculate function takes a state plane coordinate and calculates the project scale factor as the state plane grid factor minus the elevation factor (Grid Factor - elevation/earth radius). The state plane coordinate is specified by a point number from the current coordinate file.

The *Calculate State Plane Scale Factor At Each Setup* option will calculate the scale factor for each shot as the combined grid and elevation factors (see above equation). The scale factor is calculated at both the occupied and foresight points and then averaged. To use this option, you must be working in state plane coordinates and set the state plane zone in this dialog.

The *Correct For Earth Curvature* option adjusts the horizontal distance and vertical difference to the foresight point to account for the earth curvature.
**Depth Sounder Settings**

Carlson Field can use depth sounders in combination with GPS to collect points of underwater surfaces. Carlson Field supports depth sounders that output standard NMEA data, the Odom Digitrace model and the Hydrotrac model. For the Odom Digitrace, you also need to specify the depth unit mode that the instrument is set to.

The depth sounder must be connected to a separate serial port than the GPS. The *Baud Rate* between the computer serial port and the depth sounder is also specified here. The *Store Depth In Notes* option will record the water depth in the current note file (.NOT) when a point is stored to the coordinate file. The *Debug* number can be used when contacting technical support if the depth sounder is not communicating to Carlson Field.
Elevation Difference Settings

These settings apply to the Elevation Difference command. Grading Tolerance is the target difference between the actual elevation and the design surface. Carlson Field can use an external Light Bar to indicate whether your current position is in cut, fill or on-grade. Currently Carlson Field supports light bars made by Apache and Mikrofyn. The Light Bar must be connected to a separate serial port than the GPS.

![Image of Elevation Difference Settings dialog box]

GIS Settings

A standard point is stored in the coordinate file with a maximum 32 character description. The GIS Settings allow you to store more data with each point.

The Store Data In Note File option will record additional fields for each point in the note file. The note file has the same name as the current coordinate file except with a .NOT instead of .CRD file extension. The fields that are recorded are defined by the GIS File (.GIS). This file defines a sequence of field names and prompts. For example, a GIS file for manholes could contain Location, Depth and Condition fields. Choose the Select File button to choose the GIS file to use. Or use the Select GIS File Automatically by Point Description to use different GIS files depending on the point description. With this option, the program will look for a GIS file with the same name as the point description. For example, if the point description is MH, then the GIS file will be MH.GIS. See the Define Note File Prompts command for more information.

The Store Data Direct To Database option will store additional fields for each point in a Microsoft Access database. The database to store the data is set in the Output File line. The Template File is
a database that defines the fields to record. See the Define Template Database command for more information.

![GIS Options dialog box]

**Equipment Setup**

**Function with Total Stations**

Selecting the Equipment Setup command will send the user directly to a settings window that corresponds with the instrument selected in Configure Field. Equipment Setup for total stations will be discussed first, followed by GPS equipment.

This function for Total Stations lets you tell Carlson Field how you have positioned your total station. The setup information in this command is required before taking shots. Besides running this command from the Field pull-down menu, you can also reach this command with the Setup(F3) button from many of the other Carlson Field functions.

*Occupied Point* refers to the point your total station is setup on. This point is defined by a point number that references the current coordinate file. The coordinates and description of this point are displayed below the point number. The *List* button will bring up a list of the points in the coordinate file which you can review or select from. If the coordinates for the occupied point are not yet in the coordinate file, then you can pick the *Create Point* button to enter these coordinates.

The backsight can reference either a point or an azimuth. *Backsight Point* is only used if *Point Number* is selected as your *Backsight Method*. If you want to use an azimuth instead of a backsight point, select the *Azimuth* toggle and specify the azimuth in the *Bksight Azi* box.
Set the *Instrument Height* and *Rod Height*. These values will use whatever units your drawing uses: feet or meters.

Carlson Field expects the instrument to have the horizontal angle zeroed on the backsight. Part of the station setup procedure needs to include zeroing the instrument on the backsight. To do this, first specify the occupy point and backsight in this dialog. Then orient the instrument to the backsight and pick *Zero Hz* to zero the gun.

![Total Station Setup dialog]

The *BS Check* button runs a backsight check. The program will take a shot and compare the calculated point to the expected backsight point and report the results to you. This will help you establish if the point you are using as the backsight point is really the point that you think it is. For some robotic total stations, the Backsight Check routine has an option to automatically turn the instrument to the backsight. Then after the check is done, the instrument can be automatically turned back to the previous direction. The purpose of this auto turn is to speed up the steps to check the backsight in the middle of surveying points in a different direction.

For some types of total stations, the Total Station Setup dialog will also contain different options that are specific to that type of total station.

**Geodimeter Total Station Setup**

The three methods of connecting to the Geodimeter include: *Station, RPU* and *GeoRadio*. The Station option is for connecting directly to the instrument. The RPU is a remote control panel. The GeoRadio is a radio for remote control of the instrument. For the GeoRadio, the *Station Address* and *Remote Address* set the radio addresses and the *Radio Channel* sets the radio channel.
The intensity of the instrument Tracklight can be set to Off, Low or High.

The Geodimeter On and Off buttons are for putting the instrument in sleep mode to save power.

There are four different read methods. STD mode has a 3.5 second measurement time for each point. It is usually used when a normal degree of angle and distance accuracy is required. TRK mode uses automatic, measured values that are updated 0.4 seconds after making a contact with the prism. Rep STD mode measures distance automatically every 4 seconds. Fast STD mode measures distance in 1.3 seconds. It is used when the demands on precision are low.

**Leica Total Station Setup**

The Connection Mode chooses between connecting Carlson Field directly to the instrument or to a radio for remote control.

The EDM Mode sets the instrument distance measurement mode for standard shots. All the possible modes are listed in this dialog including tracking and reflectorless. Be sure to choose a mode that is supported by your instrument. When using the reflectorless mode, the Rod Height should typically be set to zero. When tracking is selected in Carlson Field functions, the program will automatically put the instrument in IR Rapid Tracking mode during tracking and then return to the specified EDM Mode when tracking is done.

The intensity of the instrument Tracklight can be set to Off, Low, Medium or High.
Topcon 800 Remote Total Station Setup

The *Radio Type* can either be Satel 3AS, Satel 2AS or Other. With Other, Carlson Field does not send any radio setup commands. So these radios must be configured before running Carlson Field. For the Satel 3AS radios, you can set the radio frequency by Channel ID or by manually typing a frequency between 468.5 and 470.5.

The *EDM Mode* sets the instrument distance measurement mode for standard shots. When tracking is selected in Carlson Field functions, the program will automatically put the instrument in Coarse mode during tracking and then return to the specified EDM Mode when tracking is done.

*Wait Time* is switched on when the instrument cannot track a prism due to an obstruction. If after the wait time have elapsed the instrument does not switch back to tracking mode, then searching mode is set.

*Vertical range* and *Horizontal range* set the search area. Vertical range can be anywhere from 0-90 degrees, and horizontal range can be anywhere from 0-180 degrees.

*Track Indicator On* if checked turns on the light which is mounted below the telescope.

*Joystick Speed* sets the instrument turning speed from the arrow keys in Robotic control.

Chapter 2. Field Module
Equipment Setup with GPS

Carlson Field works with the following RTK GPS manufacturers: Ashtech, Javad, Leica, Novatel, Sokkia and Trimble. Each RTK GPS brand has its own GPS Setup control window. To get the window which matches the GPS equipment you are using, go to Configure Field and under the Equipment Type pulldown menu select the correct equipment. A brief explanation is given below for each brand's controls.

For RTK (real-time kinematic) GPS work, the base sends GPS corrections to the rover. To setup a base receiver, you should attach the computer running Carlson Field to the base receiver and run the Equipment Setup. After this is done and the base is outputting corrections, you should detach the base receiver and attach the rover receiver and do Equipment Setup again.

If your base radio has a TX light, it should be flashing while it's sending out corrections. This is a convenient way to tell if the base is configured.

**Ashtech GPS Setup**

The *Ashtech Type* specifies the model of Ashtech equipment to be used. Carlson Field works with the following Ashtech high precision, centimeter accurate RTK GPS equipment: Z12/Sensor, Z-Surveyor, GG24 and Z-Extreme. Carlson Field also works with the Ashtech Reliance USCG/DGPS RTCM sub-meter RTK GPS receivers.
The previous Ashtech Control settings are default. Changing these settings will change the internal settings of the Ashtech receiver.

*Ashtech Data Port* is the port on the GPS receiver where the Carlson Field computer is connected, usually Port A.

*Ashtech Radio Port* is the port on the receiver where the radio modem is connected, usually Port B.

*Message Type* for high precision centimeter RTK GPS set message type to Ashtech (CPD). If you are using the USCG/RTCM DGPS message type for sub-meter accuracy then set the message type to RTCM (USCG).

*Multipath Type* is used to filter out interference in the satellite signals caused by nearby objects. The choices are No Multipath, Low: Open Field, Medium: Default, High: Building and Severe: Forest, Urban.

*Dynamics* settings are Static, Quasistatic, Walking and Automobile. Static is selected only when the Rover receiver is stationary. The default is Walking.

*Elevation Mask* is the cutoff vertical angle above the horizon. Any satellites below this angle will be left out of calculations.

*Site Name* and *Record Interval* are all setting for post processing use only, not for use with RTK GPS. Site Name is the Point ID name for post processing. Record Interval is the epoch interval to
record post processing information. RTK GPS updates every second but post processing epochs are usually 5, 10, 15, 20 or 30 second intervals.

Ambiguity Fixing Parameter (90 - 99.9): controls the confidence level of fixed positions. The default is 99.0. At a lower confidence interval the system solves much faster. If the system incorrectly solves the position, then the position error will be much greater than the reported RMS value.

Position Update Rate is the frequency that GPS positions are calculated and reported.

Fast CPD is a toggle On or Off. Fast CPD toggled On will allow approximating the rover’s position if your position is lost briefly. Off is the default. Fast CPD is generally toggled on when Dynamics is set to Automobile.

When Carlson Field functions start, the program uses the settings specified in Equipment Setup to configure the GPS receiver. The Save Settings to Receiver uploads the settings in the Carlson Field dialog to the receiver so that the next time the receiver is turned on these settings are still set even without connecting to Carlson Field. Otherwise, Carlson Field must be connected to the receiver to setup these options.

Send Command to Receiver allows experienced users to type in commands using Ashtech GPS receiver commands to set or report internal settings. (See the Ashtech operations manuals for a complete list of Ashtech GPS receiver commands.)

Reset Sensor Memory will reset the receiver memory, reinitialize the communications ports and reset the modem. Saved settings on the receiver will be returned to their default values.

Radio Baud Rate allows you to change Pacific Crest radio baud settings through the receiver. The default baud rate is 9600. (Note: If there are communication problems with either port A or B on the Ashtech ZSurveyor receiver, turn off receiver and turn it back on with both keys depressed to reset receiver to factory defaults.)

For the Z-Extreme, the Configure Internal Radio button allows you to change the radio channel and settings. This function will attempt to establish a connection with the internal radio, reporting an error if it is unable to do so. Otherwise, it will open a dialog which will display the current radio channel as well as the valid range of radio channels. Enter the desired radio channel in the edit box and then click on "Program Radio" to set the changes to the radio. Carlson Field will communicate with the radio for a few seconds, and will then request that you power the receiver down, then turn it on again before continuing. It is very important that this is done, or else Carlson Field will be unable to communicate with the ZExtreme. Also note that if the programming of the radio is canceled for any reason, the receiver will still need to be powered down, then powered up again in order for Carlson Field to be able to communicate with it.

Create Base REF File takes a reading from the GPS receiver and stores this lat/lon to a reference
file (.ref) that can be used later in Configure Base Station. The purpose is to allow moving the base station based on the current base setup. In this case, Create Base REF File would be run from the rover receiver while in "fixed" position. Then the base could be moved to this point without having to redo the local coordinate alignment.

Configure Base Station initiates the receiver connected to Carlson Field to be a base and begin broadcasting its stationary position and satellite corrections to the rover. (See Configure Base Station for All GPS Brands at end of this section.)

Topcon GNSS/Javad GPS Setup

Radio Port on the Javad base and rover receiver is usually C. Data Port is always A. When using Pacific Crest radios, Javad recommends the new PDL Pacific Crest radios. These must be set to 38,400 baud rate. Javad also uses Spread Spectrum radios which work at 119,200 baud rate.

Receiver Model selects between Legacy, Odyssey and Regency. Currently the Receiver Model does not effect the Carlson Field interface except to determine the default Antenna Type.

Position Update Rate sets the frequency that the receiver calculates and reports position. The faster rates are an option that must be purchased for the receiver.

Antenna Type chooses between an internal and external GPS antennas. This option applies to receivers with built-in antennas.

The RTK Message Type determines the format of the GPS correction message that is used from the base to the rover.

RTK Calculation Mode chooses between Delay and Extrapolate. The Extrapolate mode is needed for fast Position Update Rates.

Satellite Elevation Cutoff is the cutoff vertical angle above the horizon. Any satellites below this angle will be left out of calculations.

Ambiguity Fixing Parameter (95 - 99.9): controls the confidence level of fixed positions. The default is 99.0. At a lower confidence interval the system solves much faster. If the system incorrectly solves the position, then the position error will be much greater than the reported RMS value.

Power Cycle Receiver is the same as turning the Javad receiver off and then on.

Restore Factory Defaults resets the Javad receiver to factory settings the receiver stops acting as base or rover. The baud rate of Port A will be set to 115,200. Reset this to 9600 by turning the receiver off and then on while holding down the FN button. Watch the REC light go from orange to green to red and then let up the FN button. This method can be used if Carlson Field cannot establish communications at any time.
Clear Non-Volatile Memory does everything Restore Factory Defaults does and also wipes out the almanac data that tells it where to look for the satellites. The receiver then downloads a new almanac from the satellites.

Send Command to Receiver allows experienced users to type in commands using Javad GPS receiver commands to set or report internal settings. (See the Javad operations manuals for a complete list of Javad GPS receiver commands.)

Create Base REF File takes a reading from the GPS receiver and stores this lat/lon to a reference file (.ref) that can be used later in Configure Base Station. The purpose is to allow moving the base station based on the current base setup. In this case, Create Base REF File would be run from the rover receiver while in "fixed" position. Then the base could be moved to this point without having to redo the local coordinate alignment.

Configure Base Station initiates the receiver attached to be a base and begin broadcasting its stationary position and satellite corrections to the rover. (See Configure Base Station for All GPS Brands at end of this section.)

Leica GPS Setup

Carlson Field works with the following Leica GPS receivers: System 500, GS50, MC1000 and MK31. The type of Leica receiver is set in the Configure Field command. The options available in the GPS Setup dialog depend on the current type of receiver.
*Leica Radio Port* is the port on the receiver where the radio is attached, usually 2 or 3. Port 1 is usually the one attached to the computer. For the System 500 receivers, you can also set the radio baud rate, stop bits and parity parameters.
For the system 500 receivers, you need to specify the antenna types used at both the base and rover receivers. This antenna type sets the phase center offsets for the antennas which can effect the reported elevations by as much as 0.25 foot if not set properly.

Cell phones can also be used with Leica GPS equipment instead of radios for RTK work.

For GS50 receivers, you can choose between US Coast Guard or Racal for the corrections.

*Power Cycle Receiver* shuts the receiver off and turns it back on. This forces the receiver to reinitialize tracking satellites and the position solution. This routine is useful if the receiver is stuck in float solution.

*Send Command to Receiver* allows experienced users to type in Leica commands or send a file to set or report internal settings. (See the Leica operations manuals for a complete list of Leica GPS receiver commands.)

*Create Base REF File* takes a reading from the GPS receiver and stores this lat/lon to a reference file (.ref) that can be used later in Configure Base Station. The purpose is to allow moving the base station based on the current base setup. In this case, Create Base REF File would be run from the rover receiver while in "fixed" position. Then the base could be moved to this point without having to redo the local coordinate alignment.

*Configure Rover* sets the receiver to rover mode.

*Configure Base Station* initiates the receiver attached to be a base and begin broadcasting its stationary position and satellite corrections to the rover. (See Configure Base Station for All GPS Brands at end of this section.)

**Novatel GPS Setup**

Carlson Field works with the original Novatel Outriders and the just released Outrider DL's including the centimeter accurate RT-2 RTK receivers and the sub-meter accurate Gismo USCG/satellite RTCM/DGPS beacon receivers.
Radio Port for external radio connection is typically COM 2 on the receiver. The Data Port connected to Carlson Field is typically COM 1.

Differential Mode toggles the Novatel GPS receiver to use RTCA, RTCM or CMR message types. RTCA is proprietary to Novatel and is used only for centimeter accuracy RTK GPS surveying. RTCM can be used with USCG/DGPS beacon signals for sub-meter accuracy. Novatel receivers work with Trimble CMR proprietary message signal type and can be either a base or rover working with Trimble RTK GPS receivers.

Dynamics toggles the rover between Kinematic or Static. The base is always in Kinematic mode. Kinematic is used for surveying while walking with the receiver. Static is for stationary use only at the rover and gives better accuracies. Since Static mode is for more precise measurements, it can be used for GPS alignment points and for any control points. The receiver should not be moved while in Static mode.

Elevation Cutoff is the vertical cut-off angle above the horizon. Any satellites below this limit will be ignored in calculations. 15 is a common setting.

Elevation Type chooses between Mean Sea Level or Ellipsoid for the elevation model used by the receiver.

Solution Reset (Soft Reboot) resets the Novatel receiver in a few seconds. This is used when the rover receiver is locked up or not properly reporting its position in the Monitor function.

Receiver Reset (Full Initialize) essentially does a factory reset and a power off and on cycle.
A Receiver Reset (Full Initialize) takes three to five minutes to get back on line and become fixed after a full initialize.

*Set Radio Channel* allows you to change Pacific Crest radio channels through the receiver. The base and rover must operate using the same radio channel.

*Send Command to Receiver* allows experienced users to type in commands using Novatel GPS receiver commands to set or report internal settings. (See the Novatel operations manuals for a complete list of Novatel GPS receiver commands.)

*Check Communication Status* checks the radio port operation and reports the status as working or not communicating.

*Configure Base Station* initiates the receiver connected to be a base and begin broadcasting its stationary position and satellite corrections to the rover. (See Configure Base Station for All GPS Brands at end of this section.)

**Sokkia Radian GPS Setup**

*Radio Port* for external radio connection is typically COM 2 on the receiver. The Data Port connected to Carlson Field is typically COM 1.

*Differential Mode* toggles the Sokkia GPS receiver to use RTCA, RTCM or CMR message types. RTCA is proprietary to Sokkia and is used only for centimeter accuracy RTK GPS surveying. RTCM can be used with USCG/DGPS beacon signals for sub-meter accuracy. Sokkia receivers
work with Trimble CMR proprietary message signal type and can be either a base or rover working with Trimble RTK GPS receivers.

![Diagram of Radian Controls](image)

**Dynamics** toggles the rover between Kinematic or Static. The base is always in Kinematic mode. Kinematic is used for surveying while walking with the receiver. Static is for stationary use only at the rover and gives better accuracies. Since Static mode is for more precise measurements, it can be used for GPS alignment points and for any control points. The receiver should not be moved while in Static mode.

**Elevation Cutoff** is the vertical cut-off angle above the horizon. Any satellites below this limit will be ignored in calculations. 15 is a common setting.

**Solution Reset (Soft Reboot)** resets the Novatel receiver in a few seconds. This is used when the rover receiver is locked up or not properly reporting its position in the Monitor function.

**Receiver Reset (Full Initialize)** essentially does a factory reset and a power off and on cycle. A Receiver Reset (Full Initialize) takes three to five minutes to get back on line and become fixed after a full initialize.

**Set Radio Channel** allows you to change Pacific Crest radio channels through the receiver. The base and rover must operate using the same radio channel.

**Send Command to Receiver** allows experienced users to type in commands using Sokkia GPS receiver commands to set or report internal settings. (See the Sokkia operations manuals for a complete list of Sokkia GPS receiver commands.)
Check Communication Status checks the radio port operation and reports the status as working or not communicating.

Configure Base Station initiates the receiver connected to be a base and begin broadcasting its stationary position and satellite corrections to the rover. (See Configure Base Station for All GPS Brands at end of this section.)

Trimble Controls

Carlson Field works with the following Trimble receivers: 4000 series, 4700, 4800, 7400, NT300D, GeoExplorer and Pathfinder. The type of receiver is set in the Configure Field command. The options available in the GPS Setup dialog depend on the current type of receiver.

For the Pathfinder and GeoExplorer, the Altitude Measurement Type chooses between using Ellipsoid or Mean Sea Level as the elevation model in the receiver.

With the Pathfinder, Carlson Field will activate the receiver when the first Carlson Field command is run and the receiver will stay active until Carlson Field is exited. The reason is that the Pathfinder will turn off as soon as the COM port is turned off. If you need to make Carlson Field turn off the receiver, then use the Close Communication With Pathfinder button.

With the Pathfinder, DGPS Correction Source selects whether the Pathfinder will get its Corrections from a local Coast Guard Radio Beacon or from the Racal Satellite Correction service. Note that the Racal option must be enabled on the receiver in order to use Racal satellite corrections. (See your dealer for details as to how to do so). If Racal Service is selected as the correction source, the Racal Region selection will be enabled. The region corresponding to the relative location of the receiver should be selected to ensure proper reception of corrections.

The Pathfinder and 4700/4800 also feature the ability to select a Satellite Elevation Cutoff. All satellites with elevations below this setting will not be used in the final position calculations, even if they are otherwise visible to the receiver.

For 4700/4800 series receivers, the Receiver Type option must be set to the correct model in order for Carlson Field to communicate with the receiver. RTK Correction Type selects what format of RTK corrections between the Base and Rover receivers. CMR and RTCM formats are available. Radio Baud Rate should be set to the same setting as the communication port of the radio connected to the receiver. 4800 bps, 9600 bps, 19,200 bps and 38,400 bps rates are supported. Configure Base Station will configure the receiver as a base and begin transmitting corrections via the radio.

Configure Base for All RTK GPS Brands

Within Equipment Setup, the Configure Base Station button is the command that starts the base receiver broadcasting GPS corrections to the rover. You must click the Configure Base Station button in Equipment Setup while your are connected to the base receiver. The base needs a set
of coordinates to use as its stationary position. There are five methods to set the stationary base position: Read from GPS, Enter Lat/Lon, Enter State Plane Coord, and Read From Reference File and Read From Alignment File.

Read from GPS - This method takes one GPS reading from the base receivers autonomous position and uses it as its "true" position. The autonomous position can be off of the actual position by 200 feet. The base will calculate corrections based on this autonomous position. If you set up the base with this method, the rovers must be aligned since the corrections they are using are based on a "true" position that is not really true.
Enter Lat/Lon - requires you to enter the latitude and longitude for the position of the base antenna. This is useful if you are setting up over a USGS monument whose lat/lon you know. It can also be used over a control point whose position is known from GPS post-processing.

Enter State Plane Coord - requires you to enter the State Plane northing and easting for the point that the base is occupying. This is useful if you are setting up over a USGS monument whose coordinates you know.

Read From Reference File - reads a previously saved base position file. All of the other methods of setting up the base let you save the base position at the end of setup. If you return to a site, set up the base in exactly the same position, use Read From Reference File to use the same base position and you don't have to re-align the rover: the old alignment is still valid.

Read From Alignment File - reads a position file from one of the control points in an alignment file. This allows you to setup the base on one of the control points from the alignment. Then you don't have to re-align the rover: the old alignment is still valid.
Method 1 - Read from GPS

Step 1
Pick Read from GPS

Step 2 - Station ID (Optional)
If you plan on doing post-processing, you can input a Station ID for the base GPS Antenna location. Otherwise just hit OK.

Reminder Pop-Box
You are reminded to connect the radio to the correct port.

Base GPS receiver's autonomous position
Carlson Field takes a reading and displays the latitude, longitude and ellipsoid height. This is the position that the base will use as its "true" position. The base is now configured. If you are using Pacific Crest radios, the TX light on the radio should begin blinking.

Error Message if incorrect
If the GPS receiver is not properly connected, is turned off, or hasn't determined a position yet, you will see an error message. Check all connections and try again.

Step 3 - Save Settings to File?
You have the option to save this base position as a file. You'll be able to use this file if you set up in the same spot in the future.
Method 2 - Enter Lat/Lon

Step 1
Pick Enter Lat/Lon.

Step 2 - Enter Lat/Long/Ellipsoid Height
Input the Latitude, Longitude and Ellipsoid Height for the base position. Pick North or South for the Latitude and East or West for Longitude. Important Note: The Latitude and Longitude entered must be within 100 meters of its true location on the globe. Ideally the entered base position should be a Latitude, Longitude and Ellipsoid Height from an accurate post processed static GPS point or a published NGS monument.

Step 3 - Station ID (Optional)
If you plan on doing post-processing, you can input a Station ID for the base GPS Antenna location. Otherwise just hit OK.

Reminder Pop-Box
You are reminded to connect the radio to the proper port.

**Base's Lat/Lon/Hgt position**
The Lat/Long and Ellipsoid Height for the base position are displayed. These will be used for corrections and broadcast to the rover. If your radio has a TX light, it should begin flashing.

**Step 4 - Save Settings to File?**
You have the option to save this base position as a file. You'll be able to use this file if you set up in the same spot in the future.
Method 3 - Enter State Plane Coord Step 1
Pick Enter State Plane Coord.

Reminder Pop-Box - Current Zone & Datum
You are reminded what State Plane Zone and Datum is loaded. If this is incorrect, exit Equipment Setup and input correct State Plane Zone and Datum in Configure Field > GPS Settings.

Step 2 - Enter Northing/Easting/Elevation
Input the State Plane coordinates (northing, easting and elevation) for the base position. Important Note: The State Plane coordinates entered must be within 100 meters of its true location on the globe. Ideally, the entered State Plane coordinates (N,E,Z) should be from an accurate post processed static GPS survey point or from a published NGS monument data sheet.

Step 3 - Station ID (Optional)
If you plan on doing post-processing, you can input a Station ID for the base GPS Antenna location. Otherwise just hit OK.

Reminder Pop-Box
You are reminded to connect the radio to the correct port.

Base's Lat/Lon/Hgt position
The Lat/Long and Ellipsoid Height for the base position are displayed. This position will be used for the corrections that are sent to the rover. If your radio has a TX light, it should begin flashing.

Step 4 - Save Settings to File?
You have the option to save this base position as a file. You'll be able to use this file if you set up in the same spot in the future.
Method 4 - Read From Reference File

Step 1
Pick Read From Reference File to select an existing base position REF file.

Step 2 - Select Base Reference File to Load
Pick the base position REF file to be loaded. Use the up arrow folder to browse elsewhere for the REF file.

Position as Read from File
The latitude, longitude and elevation are read from the selected file and displayed.

Step 3 - Base Antenna Height
Enter the vertical height of the base antenna.

Step 4 - Station ID (Optional)
If you plan on doing post-processing, you can input a Station ID for the base GPS Antenna location. Otherwise just hit OK.

Reminder Pop-Box
You are reminded to connect the radio to the selected port.

Base's Lat/Lon/Hgt Position
The Lat/Long and Ellipsoid Height for the base position are displayed. This position will be used to calculate the correction that are sent to the rover. If your radio has a TX light, it should begin flashing.
Method 5 - Read From Alignment File

Step 1
Pick Read From Alignment File.

Step 2 - Select Alignment File to Load
Pick the alignment DAT file to be loaded. Use the up arrow folder to browse elsewhere for the DAT file.

Step 3 - Select Alignment Point
The program will display a list of points in the alignment file. Pick the point from this list.

Step 4 - Base Antenna Height
Enter the vertical height of the base antenna.

Step 5 - Station ID (Optional)
If you plan on doing post-processing, you can input a Station ID for the base GPS Antenna location. Otherwise just hit OK.

Reminder Pop-Box
You are reminded to connect the radio to the selected port.

Base's Lat/Lon/Hgt Position
The Lat/Long and Ellipsoid Height for the base position are displayed. This position will be used to calculate the correction that are sent to the rover. If your radio has a TX light, it should begin flashing.
Chapter 2. Field Module
Saving Base Settings to a File

It is always recommended to save the base position to a file if you are going to return to the same site survey again. You can setup on the same base position, recall the base REF file and enter the new antenna height. Then you can use the alignment file from the first day in the rover and not have to re-align.

When you save the base antenna position to a file it is stored with a REF extension denoting base reference file. By default, it goes in the Data directory. Input reference filename and pick Save and OK.

Configuring the Rover

After the base is configured, unplug the base receiver from the Carlson Field computer and plug in the rover receiver. In Equipment Setup, toggle the Station Type from Base to Rover. Then pick Exit. This will configure the receiver as a rover.

From the Field drop-down, pick the command Monitor GPS Position. The Status is reported as either Autonomous, Float or Fixed.

If the rover is Autonomous, it is not getting any corrections from the base.

If the status is Float, the rover is receiving corrections, but has not found the fixed solution. Once the solution becomes Fixed, the rover is locked on to the base corrections and is calculating an accurate position.
Align GPS To Local Coordinates

Function

Carlson Field reads a latitude, longitude and height position from the GPS rover receiver and converts these values to State Plane or UTM coordinates for the current zone as set in Configure Field. Using local coordinates and their corresponding GPS position, Align Local Coordinates applies a transformation to convert the state plane or UTM coordinate to the local. Carlson Field can operate in three different modes depending on the Align Local Coordinate settings:

1) No points - No Adjustment
2) One point - Translation Only
3) Two or more points - Translate, Rotate and Scale
Without any alignment points set, Carlson Field will operate with no alignment which directly uses the state plane or UTM coordinates. In order for the coordinates to be the true state plane coordinates in this alignment mode, the GPS base receiver must be set up over a known point and the true Lat/Long for the point must be entered in the base as the base position. Otherwise, if the base is set over an arbitrary point, then the coordinates will not be true state plane.

In one point alignment mode, one pair of GPS and local coordinates is specified. The differences between the GPS and local northing, easting and elevation for these points are used as the translation distances in the transformation. The rotation will use either the state plane grid or the geodetic as north. No scale is applied in this transformation.

A two or more point alignment is used to align to an existing local coordinate system. At least two pairs of local and GPS coordinates must be entered.

In addition to the northing and easting transformation, SurvStar will also translate the elevation from the GPS system to the local. The elevation difference between the two systems is modeled by a best-fit plane.

An alignment is only valid if the base receiver setup has not changed since the alignment points were recorded. In order to use an alignment when returning to a site, you must set up the base receiver in the same position and enter the same LAT/LONG coordinates for the base.

The Align GPS to Local Crds menu item brings up the alignment dialog box. There is more information than to fit in one window, so use the View button to switch between viewing the local coordinates and the GPS Lat/Lon.

Each line in the box represents one alignment point. Each point in an alignment file relates a specific Lat/Lon/Elv to a specific Northing/Easting/Elevation for your local coordinate system.
Carlson Field will use the current alignment file every time that the GPS is read. It provides the necessary adjustment to properly convert that position to your coordinate system.

In the local points view, the HRes column shows the horizontal residual and the VRes column shows the vertical residual. The residual is the difference between the actual point and the point calculated using the alignment transformation. In GPS points view, the HRMS and VRMS columns show the horizontal and vertical RMS values when that point was recorded.

The On/Off buttons allow you to switch whether the highlighted point is used for the horizontal and/or vertical alignment. The HV column shows a ‘Y’ if this point is used in the calculations. Otherwise it shows an ‘N’. The H column represents horizontal control and the V column vertical control. For example, you may wish to use 2 points for horizontal alignment and one for vertical.

The Optimize button will find the combination of turning alignment points on/off for horizontal and vertical such that the horizontal and vertical residuals are minimized.

The Desc field shows an optional description of the alignment points.

The scale factor and average horizontal and vertical residuals appear at the top of the window. These values serve as a check that the alignment is valid. The scale factor factor should be closed to 1.0 (in range of 0.9 to 1.1). The average residuals should be less than 0.2.

XY On/Off toggles the highlighted alignment point horizontal component off or on. Alignment points with the horizontal component toggled off will not use the northing and easting of that point for adjustment calculations.

Z On/Off toggles the highlighted alignment point vertical component off or on. Alignment points with the vertical component toggled off will not use the elevation of that point for adjustment calculations.

Note: When you toggle either the XY or Z component off or on for any alignment point the scale factor and Horiz/Vert residuals are recalculated automatically. Briefly toggling XY or Z components off or on and reviewing the scale factor and residuals changes is a quick approach to finding the best alignment points. Carlson Field can handle an unlimited number of alignment points.

Highlight an existing alignment point entry and pick Delete to delete that alignment point.

Pick the Add button to create an alignment point. The Add Alignment Point dialog box appears. There are two ways to enter the local coordinate points: by entering the N, E, Z, or by using an existing point number stored in the current coordinate CRD file. The GPS values can also be specified by two methods: by entering in the Latitude, Longitude and Height or by occupying the control point with the rover and taking a GPS reading at this location. Manually entering the Lat/Lon can only be done when the base is setup on a known location using a true lat/lon position. Otherwise Carlson Field needs to use the Read GPS method. For this method, the base can be
setup with a lat/lon that only needs to be close (within 100 feet) of the actual lat/lon. This type of position can be read from an autonomous GPS position. With the base setup on this approximate lat/lon, go with the rover to the control points and use the Read GPS option in the Add function. The rover GPS solution must be in "fixed" status when the alignment point is added. By reading the rover GPS position for the alignment points, the alignment will transform the coordinates from the GPS system of the current base setup to your local coordinate system.

Load allows you to open an existing alignment file. Only one alignment file can be open at a time. Alignment files have a DAT extension and stored in the Data directory by default.

Save stores alignment files (DAT extensions) to a file. Files are by default stored to the Data subdirectory.

The OK button will set the current alignment to the settings in the dialog.

Alignment Methods

Carlson Field can operate by the following Alignment methods:

Alignment Method 1) - No alignment points
Alignment Method 2) - One point alignment
Alignment Method 3) - Two or more alignment points

Alignment Method 1

With no alignment of the rover, Carlson Field will report Northing and Easting as State Plane or UTM coordinates. In order for this method to give accurate State Plane or UTM coordinate values, the GPS base receiver must be set up over a known point and configured using the true Lat/Long/Hgt or true State Plane coordinates. If the base is set over an arbitrary point, configured by reading the GPS, the RTK GPS stored coordinates will be translated up to a 200 feet but accurate in relation to each other.

When using this method, you can skip Align GPS to Local Crds and start surveying immediately once the base is configured and transmitting its position and the rover is fixed.

In most cases, you cannot use Method 1 because you will not have setup the base on a point whose exact true position you know. Therefore the base corrections are going to be off a certain distance north/south and a certain distance east/west. This is why you want to do an alignment. You are showing Carlson Field how to correct for the north/south and east/west offsets. Any points surveyed with the alignment file active will be translated to their proper position.

To gather alignment points, you put the GPS antenna over a point with known coordinates and Carlson Field records the GPS Lat/Lon/Elv and the Northing/Easting/Elevation you give it. This point can be a local coordinate, for example a stake you are calling 5000,5000. It can also be a true State Plane point. Using one or more State Plane points will give you an alignment to true State Plane (even if your base is not using its own true position.)
Alignment Method 2

This method uses one alignment point to translate the GPS coordinates to local or true State Plane coordinates.

Remember that if the base is set up over an arbitrary point, the GPS coordinates can be off from true state plane by up to 200 feet. This alignment method can be used to correct for this by translating the system onto true state plane coordinates.

You can choose if you want the coordinate system North to be Geodetic North or State Plane Grid North under Configure Field > GPS Settings. If you specify a scale factor in that dialog box, it will be applied to all points recorded.

One point alignment is useful for data collection on a new site. In this case you can set up the GPS base receiver anywhere convenient. Then position the rover over a point (preferably one you can find again) and add this point as your one alignment point by reading the GPS point and entering a local coordinate like 5000,5000,100. Now the local coordinate system is set around this first point at 5000,5000,100.

This method is commonly used for small topo or stockpile RTK GPS surveys. When collecting or staking data at distances greater than 2 miles from the base, both the horizontal and vertical errors will begin to increase gradually. Therefore, you should use a multiple point alignment for large projects.

Alignment Method 3

This method is useful if you are arriving on a job which has already been surveyed. It assures that your survey is in the same coordinate system as the original survey.

Using control points, this method transforms the GPS coordinates to an existing local coordinate system. This method takes pairs of GPS coordinates and the corresponding local coordinates to define the translation, rotation and scale of the alignment.

In Configure Field > GPS Settings, there is a choice for the transformation as Plane Similarity or Rigid Body. Plane Similarity will apply a scale factor to the transformation. The scale factor will be based on the alignment points and should always be very near 1.0 to be correct. The Rigid Body option will align by translate and rotate but no scale. Any difference in scale between the GPS and local coordinate systems will be distributed equally between the two alignment points. These differences will appear as horizontal residuals in the Alignment dialog.

Two pairs of points are sufficient to define the translation, rotation and scale for the transformation. But adding more alignment points yields the most accurate results for aligning to existing coordinate systems. Since two pairs of coordinates are sufficient to define the transformation, there is extra data when there are three or more pairs. The program uses a least-squares best-fit routine to find the transformation that minimizes the residuals. This one best-fit transformation is
used to convert from the GPS to the local coordinate system for all the points. The residuals are the differences between the transformed GPS coordinates and the actual local coordinates.

A multiple point alignment is especially helpful on a survey which covers a large area. The error in raw GPS coordinates increases as you get farther from the base. Taking alignment points around the perimeter of your job site as alignment points will give you the best geometry for the alignment.

**Point Store**

**Function**

This function creates points by reading from GPS or total station equipment. The new points are stored in the current coordinate and simultaneously drawn in the drawing. The measurement data is also stored to the current raw file which has the same name as the coordinate file except with a .RW5 instead of .CRD file extension.

The Point Store dialog docks on the side of the drawing window. This allows you to see the drawing view as you collect points. You can use the arrow keys to pan the drawing and the Page Up/Page Down keys to zoom out and in. There are also icons for the pan and zoom functions at the top of the dialog. Also, besides clicking the function buttons, most buttons have an associated function key such as F1 that you can use to run the routine.

Before taking measurements, make sure that the rod height is correct.

To take a measurement from the survey equipment, pick the Read button. The calculated northing, easting and elevation will be displayed in the dialog and a temporary icon will be shown in the drawing at the point location.
Before storing the point, make sure that the point number and description are set in the Point Number and Description fields. The point number is a required field for storing to the current
coordinate file. If the point number specified already exists in the coordinate file, then a dialog will pop-up with options to overwrite the existing point number, to use another point number or to cancel storing the new point. The **Point Number** field will automatically increment after storing the point.

The **Description** is an optional field for identifying the point. The maximum length of the description is 32 characters. Besides naming the point, the description can also be used to with Field-To-Finish to draw linework and to determine the symbol of the point in the drawing. When the Field-To-Finish option is set on in **Options**, the program will lookup the description in the current code table. If the description matches one of the codes, then the code can determine the symbol, layer, format of the point when it is drawn. Otherwise the defaults in the Point Setting section of the Options dialog are used for the point symbol, layer and format.

To store the new point to the coordinate file and draw the point, pick the Store button. At the time that Store is applied, the program uses the point number, description, linework options and special options currently set in the dialog.

You can also use the **Read & Store** button to do both functions in one step. With this method, the program will take a measurement and if the measurement is successful, then the point will be stored immediately.

The **Code** button brings up a list of point descriptions from the current Field-To-Finish code table. You can select a code from the list to set this code as the current point description. This function also shows a list of all the descriptions of currently active linework. You can end a currently active linework by highlighty the linework description from the Active Linework list and pressing the End Linework button.

Many of the options for storing points can be set in the Configure Field>Point Settings command. The **Options** button in this dialog is a shortcut to these point settings.

If you want lines or polylines to connect the points that you are about to record, select the **Start** button under **Linework**. After the first point, the **Linework** selection will change itself to **Cont** meaning continue. Leave this selected while you are recording points in the same line. Before shooting the last point in your line, change it to **End**. If you want the line to close itself onto its first point, check the **Close** button.

The Field-To-Finish Linework option is an automatic way to start linework. The program will lookup the point description in the code table. If the description matches a code and the code is defined to create linework, then the **Start** toggle in the **Linework** options is turned on. Otherwise you can begin new linework by toggling on **Start** manually.
When a point is stored and Start is on, Carlson Field pops-up a dialog for choosing between a line, 2D polyline or 3D polyline. A 3D polyline can contain points with different elevations, but a 2D polyline always has an elevation of zero. The Smooth Polyline option will create Bezier smooth polyline through the points.

Carlson Field can keep track of several lines being drawn at once. Each line corresponds to a set of points with a different description. Let's say you are shooting a line of points called "fence" and you want to shoot some points on a curb, but you're not finished with the fence. You change the Desc box from "fence" to "curb". Carlson Field lets the fence line go for now. It changes the Linework selection to No. You want a line for your curb, so you select Start. The points you shoot now will form a new curb line. To go back to recording fence points, change the description back to "fence". The fence line you were working on will continue to include any new fence points you shoot. If you want to end this fence line, select End under Linework and Carlson Field will
not connect any future "fence" points to this line. If you start a new linework with a description that already has linework, then Carlson Field pops-up a dialog with three options as shown. The *Continue Existing Code* option is the same as using Cont instead of Start. The *End Existing and Start New* option will end the active linework and start the new linework with the same description. The *Use New Description* option will keep the existing linework and start another linework with another description. For example if you are surveying two edge of pavement lines, you can have one with the description "EP" and the other with "EP2".

The *PC* and *PT* options are for drawing curves. If you want to plot a curve, check the *PC* box before recording the first point on your curve. Shoot as many points along the curve as you need. Carlson Field can handle compound curves as well as simple curves with this function. Before shooting the last point on the curve, check the *PT* button. If you don't specify a PT, Carlson Field will assume a three point curve.

The *Undo* button will remove the last point number created. The point is removed from both the coordinate file and the drawing.

For GPS and tracking total stations, there is a *Start Continuous* button which makes Carlson Field continuously read from the instrument. The coordinates are displayed in the dialog and your position is shown with an arrow icon in the graphics view. To store a point, you can use the Store button without using the Read button first. Once continuous reading is active, the button changes to *Stop Continuous* which will put you back in standard reading mode.

![Linework Code Dialog](image)

**Point Store with GPS**

When using GPS equipment, Carlson Field will also report the RMS values and solution status when you take a reading. If Carlson Field gives you a message that your RMS values are too high when you try to read a point, you can click on the *Monitor* button to bring up the *Monitor* window which will give you information on how accurately your position is determined and how many satellites you are tracking. The *Skyplot* button will bring up the window showing you where in
the sky the satellites are.

For points that are hard to reach directly by GPS, you can use the Offset option. This option can be used in areas of limited satellite communication such as high walls or under a tree. This allows you to setup the rover in a clear area and read the coordinate. The point that is actually stored is offset from the rover position. To create an offset point, turn on the Offset toggle and then choose Read. The offset direction can be entered as left, right or azimuth. The left and right offset is relative to the rover position at the previous read. The offset distance is entered in the dialog. A Vertical Offset can also be specified. Choose Store to store this point after the offset is done.

Offsets can also be done with laser guns when the laser option is setup in Configure Field>GPS Settings. There are two methods for taking laser offsets. One method is to use the Offset toggle and the Read button. In the Offset dialog, there is button for Read Laser for using the laser measurement for the offset distance and/or angle. This method creates a single offset point.

The other method is to use the Laser button which can create many offset points. This method brings up another dialog. The Setup button can be used to set the Laser Alignment Azimuth. This alignment applies to laser guns that use a magnetic compass for the horizontal angle. The magnetic north can vary from the north of your coordinate system. The Laser Alignment Azimuth is added to the measured laser azimuth to adjust for the difference. To set the alignment azimuth, specify a reference backsight direction by either entering an azimuth or by point number. Then choose the Read Laser For Alignment button and take a laser shot towards the backsight. The program will compare the azimuth from the laser with the reference backsight to figure the alignment azimuth. When the alignment azimuth is set, pick the Go button. Carlson Field then listens for measurements on the laser gun port. To take a shot, sight the target point and press the laser trigger. Carlson Field will read the laser measurement and read the GPS position. The laser angle and distance are combined with the GPS position for the new point coordinates. To return to regular GPS Point Store, choose the Exit button.
Point Store with Total Stations

Before taking measurements with total stations, you need to specify the occupied point coordinates of the instrument, the backsight and the height of the instrument. This current setup data is shown in the "OC:# BK:# HI:#" line in the dialog. Also icons are drawn to show the occupied point and backsight direction in the drawing view.

The Setup button at the top of this dialog brings up the Total Station Setup dialog, where you can change your occupied point, backsight and instrument height.

For robotic total stations, there is also a Joystick button to turn the instrument, search for the prism and set tracking or standby mode.

Carlson Field can shoot points with offsets. To shoot a point with an offset, check the Offset button on the Point Store dialog box. Click Read or press F1. A window appears to let you choose the type of offset to shoot. The choices are Distance/Angle and Enter Offset Distances. The Offset Vertical option will prompt for an elevation difference to apply to the point.

To do a Distance/Angle offset, you first take a distance shot and then angle shot. For the distance measurement, have the rodman stand to the side of the point. The prism and the point should both be the same distance from the total station. Carlson Field takes the first shot and gets the distance from it. It then prompts you to read the angle. Turn the gun so that it is aimed at the point. The prism is not needed for this step. Click OK and Carlson Field reads the horizontal angle from the gun and combines this with the distance from earlier to calculate the coordinates of the point. Also for combining these shots, there is an option whether to use the vertical angle from the distance or from the angle shot.
With the Enter Offset Distances method, you can supply both a left/right offset and an in/out offset. To do a In/Out offset, have the rodman stand a measured distance in front of or behind the point. The total station will take the shot and then Carlson Field will ask you how to move the point: in or out and the distance. If the prism is in front of the point, choose out. If it's behind the point, choose in. To do a Left/Right offset, have the rodman stand a measured distance to one side of the point. After taking the shot, Carlson Field will ask whether to offset right or left. If
you are at the total station, looking at the prism, and the point you are after is to the right of the prism as you're looking at them, choose right offset. Otherwise, choose left offset.

Choose *Store* to store this point after the offset is done.

The *D&R* option stands for Direct and Reverse. When this box is checked, Carlson Field will take sets of four shots to determine the coordinates of the next point. Two shots are taken for both the backsight point and the foresight point: one direct shot, one shot with the total station reversed. This yields a more accurate reading. Two options are available for the order of shots when doing a D&R. The first is Backsight Direct, Backsight Reverse, Foresight Reverse, Foresight Direct. The other option is Backsight Direct, Foresight Direct, Foresight Reverse, Backsight Reverse. Carlson Field also offers the option of shooting multiple sets of Direct & Reverse for even greater accuracy. The *Shoot Distances For Reverse Shots* option determines whether to take distance measurements on the foresight reverse and backsight reverse shots. When this option is off, the program will still use the reverse shots to mean the angles. Otherwise the program will also use the reverse shots to mean the distances. The *Use Robotics To Auto Flip Instrument* option applies to robotic total stations to have the program automatically turn the instrument for reverse shots.

To shoot a point as a Direct & Reverse, check the *D&R* box and click on *Read*. A dialog box appears, offering the choice of orders for the shots. Before each shot, Carlson Field tells you what kind of shot is being taken. After each shot, Carlson Field reports the measurements and allows you to confirm the measurement or to re-shoot. After all four shots are taken, Carlson Field does the math and reports the accuracy of each part of the measurement.
Choose *Store* after completing the Direct & Reverse to store this new point.

**StakeOut**

**Function**

The Stakeout function is used to find a specific point in the field. Once you tell Carlson Field the point that you are looking for, pick *Start* and the program draws an X-marks-the-spot bullseye on that point in the drawing. Carlson Field also draws a triangle on the drawing for where you are currently standing. These icons help to guide to to the target point graphically. Carlson Field also reports in the dialog box how far you need to move to reach the point.

There are several options for Stakeout defined in Configure Field>Stakeout Settings. These op-
tions should be set by Configure Field before running Stakeout. See the Configure Field section of the manual for a description of the stakeout options.

There are four ways to define the target point for stakeout. The first method is to specify a point number from the current coordinate file. To do this, click on the **Point Number** button and type in the point number in the dialog. The second method is to give a station and an offset from a centerline. The program will prompt for a centerline file (.CL) and then the station and offset. You can also specify the station interval for automatically incrementing to the next stakeout point. See the Roads section of this manual for how to create centerline files. The third method is to graphically pick the point from the drawing. Select **Pick Point** and a dialog box allows you to pick different snaps: endpoint, midpoint, center, node (point), or intersection. This will help you pick your desired point more accurately. For example, you can select endpoint and then pick on a polyline corner to stakeout that the polyline endpoint. See the Object Snap command in this manual for more on snaps. The fourth method is to simply type in the target point coordinates in the **Northing**, **Easting** and **Elevation** fields.

Once the stakeout point is set, click the **Start** button and Carlson Field begins the stakeout routine. The format of the stakeout screen that appears depends on whether you are using total stations or GPS as described below.

When you reach the target point, click the **Store** button. Carlson Field reports the difference between your current position and your target position. At this point you can choose to store this staked-out point as a new point in the coordinate file.
When the target stakeout point has an elevation, Carlson Field also reports the elevation difference between the target and current elevations. This cut/fill is also in an edit box that allows you to change the value for labeling. For example, you may want to round the cut/fill number to an even number to label on the stake with a mark to indicate where this even number occurs. When you change the cut/fill label from the original value, Carlson Field will report the offset for this mark. For total stations, Carlson Field will also report the zenith angle for locating this mark. There are also fields in the report dialog for entering vertical offsets to get additional cut/fill values. For example, if the target point is for the road surface and you want to also get the cut/fill to an 18 inch subgrade, then enter -1.5 as the vertical offset.

**GPS Stakeout**

After you click *Start* to begin staking the point, Carlson Field changes the dialog box to the one shown below. The dialog shows the target point, the current position northing, easting and elevation and the GPS HRMS/VRMS. The distance, azimuth and cut/fill from the current position to the target are also reported. Carlson Field also breaks down this distance into how far north/south and how far east/west to go. Finally based on your current heading, the program tells you whether to turn right, turn left or that you are on-line.

In the graphics view, the large "X" shows the point being staked-out and the triangle represents...
your position. A temporary line is drawn between your current position and the target. In Configure Field>Stakeout Settings, there is an option to auto zoom in as you approach the target point. Otherwise you can use the arrow keys to pan the display and the Page Up/Down keys to zoom out and in.

![Stakeout Report](image)

**Total Station Stakeout**

Before starting the stakeout, be sure that the instrument is setup with correct occupied point, backsight and instrument height. This setup data is displayed in the third line of the dialog. You can pick the Setup button to change the instrument setup.

After you click *Start* to begin staking the point, Carlson Field changes the dialog box to the one shown below. The dialog shows the angle to turn the gun and the horizontal distance to the target. Turn the instrument to this angle and position the rodman at this angle and distance. Then pick...
the Read button to take a measurement. Carlson Field will then report the horizontal distance and cut/fill from the current position to the target. This distance is also reported as how far north/south and how far east/west to go and as how far in/out and left/right to go. To in/out and left/right distances are relative to the rodman facing the instrument. Keep moving the rodman and picking the Read button until you reach the point. Then pick the Store button.

For robotic total stations operating remotely, there is a Continuous button that puts the instrument in tracking mode with continuous measurements.

In the graphics view, the large "X" shows the point being staked-out and the triangle represents your position. Also the location of the instrument is shown with an icon and the backsight is shown as temporary line.
Auto Points at Interval

Function

This command stores a point whenever the distance or time from the previous point exceeds the user-specified interval. This command only applies to GPS and robotic total stations. If you will be collecting a large number of points at once, Auto Points at Interval can be a useful tool. For example, you may want to plot the edge of a road. Once you start Auto Points, you can walk along that edge of road and let Carlson Field record your position automatically.

The Auto Points at Interval dialog box resembles the Point Store dialog box with the addition that you can set the interval to record points. You can set it to store a point every time you move a certain distance by selecting Distance and entering the distance you choose in the Interval box. The distance will be taken in feet if your project is using English units, or meters if your project is in Metric. If you select Time, the number in the Interval box will refer to the number of seconds between creating points.

Check the Draw Linework box to have your points connected by a line or polyline. You can enter a description or choose from the code table just like in Point Store.

The Offset toggle will apply an offset to the calculated coordinates. The horizontal offset is applied perpendicular either left or right to the direction of movement. There is also an option for a vertical offset.

Pick Start to begin storing points. Carlson Field will take a reading and store the first point. Then
Carlson Field will continuously read the GPS or total station. For distance interval method, as each point is read the distance from the last point is calculated. When the distance is greater than the specified interval, a point is created and the point number is displayed in the dialog. In practice, the actual distance between stored points will be greater than the distance interval. For example, if the distance interval is 10 and the current distance is 8.9, then no point is stored. Then you keep moving and the next distance is 11.4 which will store a point.

For time interval point storing, after reading and storing the first point, Carlson Field will wait for the interval time to pass, then read and store again.
The new points are both stored to the current coordinate file and drawn in the drawing.

When using GPS, if the RMS values of the position read are above the tolerance set in Configure Field, then the point will not be stored.

Carlson Field will continue to record points until you click on Stop.

**Track Position**

**Function**

This command shows the coordinates of your current position in a dialog and draws an arrow icon in the drawing view. This command only applies to GPS and robotic total stations. As you move along, the arrow icon will move through your drawing showing your position in real-time. If the arrow icon gets near the edge of the screen, Carlson Field will automatically pan over.

A dialog box also appears in Track mode. The dialog shows your current northing, easting and elevation. For GPS mode, the dialog displays the HRMS and VRMS values and solution status. There are buttons to take you to Monitor and Skyplot. There is also a Store button which will store your current location as a point and plot it, similar to the Point Store function.
Satellite SkyPlot

Function

When using GPS, it is important to know how many satellites you are tracking and their position in the sky. *Satellite Skyplot*'s visual and graphical screen aids in identifying when satellites are being masked by surrounding structures, trees and mountains. Satellites close to the horizon, under fifteen degrees, are less helpful resolving the rover position because of extra atmospheric interference. If there are too few satellites present, the receiver will be unable to resolve its position. Typically, five satellites are required to resolve position and four are needed to maintain a locked solution. *Satellite Skyplot* can be an invaluable tool to help you monitor the current satellite configuration.

The skyplot screen appears at left. The top half of this window displays the visible satellite information in chart form. *PRN* is the satellite identification number. *Azi* is an abbreviation for azimuth; the horizontal angle from due north, in degrees measured clockwise, to the satellite position (0 to 360 degrees). *Elv* is an abbreviation for elevation; the vertical angle above the horizon where the satellite can be found (0 to 90 degrees). One entry appears for each satellite that the receiver is tracking.

The image on the lower half of the window displays the same information graphically. It shows a map of the sky with North at the top, East to the right. The centerpoint, where the lines cross, is straight up. Each satellite appears as a symbol resembling an "H". As you can see, most of the visible satellites were in the Northeast when this image was captured. The inner circle represents an elevation of sixty degrees. The outer circle is the horizon. Roughly speaking, any "H" touching this circle is too low in the sky to be of much use. For GPS receivers that support GLONAS satellites, Skyplot will show these satellites with a "G" symbol.

For some types of GPS receivers, the receiver will report which satellites are being used for calculating the position and which are only being tracked. A satellite might be only tracked and not part of the solution if the satellite is too low on the horizon or when the signal is not clear. The skyplot will highlight the satellites that are part of the calculations.
Monitor GPS Position

Function

This command reports the current GPS Lat/Lon, local coordinates and GPS solution status. The latitude and longitude are reported in the DD.MMSSSSSSS format. In this example, the latitude is 42 degrees, 21 minutes, 46.4414 seconds north. The longitude is 71 degrees, 8 minutes and 31.5699 seconds west. Negative longitudes indicate longitudes west.

The next three items are state plane or local coordinates depending on the transformation in the Align Local Coordinates command. The HRMS and VRMS are measures of the reliability of the position that the receiver has calculated. They correspond to the position horizontally and vertically, respectively. If the receiver is autonomous, not receiving corrections from a base, the RMS can be up to a few hundred feet. If this rover is computing a "Fixed" position, the RMS values should be less than one foot, probably close to a tenth of a foot. If the receiver loses the fix and becomes "Float", the RMS values will jump to between one and ten feet.

Depending on the type of GPS receiver, the Monitor screen will also show more values like radio link status and receiver battery status.

The Skyplot button will jump you to that window so you can see the satellites the receiver is using.
Benchmark

Function

This command takes a measurement to a benchmark point with a known elevation in order to calculate the elevation at the occupied point. This command only applies to total stations.

In the Benchmark dialog, fill in the instrument and rod heights. The benchmark elevation is specified in the Target Elevation field. This field can be filled out by entering a target point number which reads the elevation from the current coordinate file for the specified point. Or you can simply type in the target elevation directly. There is a choice between calculating the occupied point elevation or the instrument height. For calculating the instrument height, you need to enter the occupied point elevation. When calculating the occupied point elevation, there is an option to store this elevation to the coordinate file for the occupied point number. When all the options are set and the target benchmark is sighted, pick the Read button to take a measurement. After the reading, the program will display in the dialog the calculated occupied point elevation or instrument height depending on the calculation mode.
Resection

Function

This command calculates point coordinates given the angle and distance measurements to two or three reference points. This command only applies to total stations. The reference points are specified by point number. These reference points need to be stored in the current CRD file before running this command. This function can be used when setting up the Total Station on an unknown point.

The command starts with a dialog to choose between using two or three reference points. Then for each reference point, there is a dialog to enter the reference point number, instrument height and target height. After entering this data and sighting the target, pick the Read Gun button to take a measurement. The measurement angles and distance are reported for a check in a dialog. When all the measurements are done, the results are show in a dialog. The results include the calculated coordinates and the residuals. The program calculates the coordinate by averaging the distance-distance and angle-angle solutions. Since there is redundant data, the final calculated coordinate will differ slightly from the individual measurements. For example in a 3-point resection, there are two different distance-distance solutions (between the first-second point and between the second-third points). The program reports the difference between the final coordinate and the individual solutions as the residuals which act as an indicator whether the data is good. High residuals suggest a problem with the input data. In the dialog that displays the final coordinates and residuals, there is a button to store the coordinates to the current CRD file with a
specified point number.

Building Face Surface

**Function**

Used to project all points onto a surface or plane. Upon executing the function, a menu will open,
prompting the selection of three points to define the plane/surface. Note that there must already be three points along the plane in the CRD file in order for this function to work properly. After selecting the three points (the "List" buttons will bring up a list of available points), select "OK" to proceed. A screen similar to the Store Points dialog will now open. Every point which is read will be plotted along the plane defined by the three points selected, even if it is at a different distance. When finished, simply exit out of the menu as with any other function.

**Pattern Point Survey**

**Function**

Used with a reflectorless Total Station. This function is used to shoot a regular, rectangular "pattern" of points across an area. It is useful when periodic measurements of an area are required. Upon starting the function, a query box will ask for two points defining a rectangle, the lower left corner and the upper right corner. For each of these, aim the gun at the corner of the area to be scanned and click "Read". After reading both points, a menu will prompt for several other parameters. Enter the first point number to shoot, as well as any desired description for the points, and both horizontal and vertical increments. These are angle increments, given in seconds. Once all of the above is entered, select "OK" to begin the survey. The total station will now begin turning automatically to the bottom-left corner of the area, and will begin shooting points. Upon reaching the right-hand limit of the area, it will begin a new row of points, starting at the left.

**Point Check By Robotics**

**Function**

This command works with robotic Total Stations made by Leica, Topcon or Geodimeter. This function is used to shoot and record a series of known target points. Before running this command, the instrument setup must be set (occupy point, backsight) with the Equipment Setup commands. After selecting Point Check by Robotics you will be prompted with a dialog box. Choose the points you want to check and click process. The Total Station will then go from point to point and take new measurements. When it is all done, a report will be given with the new measurements and any deviation.

**Carlson Field Icon Menu**

**Function**

*Chapter 2. Field Module*
The Carlson Field Icon Menu lets you select Carlson Field functions by pressing a function key F1-F10 or by picking the icon button. The set of commands that are available in this menu depends on the type of survey equipment that you are configured to. Before running these Carlson Field functions, you need to run Configure Field to set the equipment type and communication parameters.

There are two ways to bring up the icon menu. One way is by picking the Start Carlson Field icon. This start icon is displayed in the lower right of your screen when the Show Startup Icon option is on as set in Configure Field under General Settings. You can close the Start Carlson Field icon for the current drawing session by clicking the X in the icon title bar. To bring back the Start Carlson Field icon you can use the F11 key. The Start Carlson Field icon is only displayed when no commands are running. The other way to show the Carlson Field function menu is to pick the Menu(F11) button while running other Carlson Field commands. This method allows you to switch between Carlson Field functions without having to exit back to the CAD menu. For example, you can switch from Point Store directly to Stakeout.
Typical Alignment Scenarios

Scenario: New site. In this case, there are no established coordinates on the site.

Alignment: Choose a point on site and do a one point alignment. For the local alignment point, enter the coordinates that you would like to use (ie 5000,5000,100). Under Configure Field>GPS Settings, The One Pt Align Azimuth option chooses between using true north (geodetic) or state plane north (grid). To use real world ground distances, set the Project Scale Factor under Configure Field>GPS Settings. Otherwise the default scale factor of 1.0 will collect points on state plane distances.

Scenario: One known state plane coordinate and you want to work in the state plane coordinate system.

Alignment: Either setup the base over the known state plane coordinate or do a one point alignment on this known state plane point. In Configure Field>GPS Settings, set the One Point Align Azimuth to Grid and set the scale factor to 1.0.

Scenario: Multiple known control points.

Alignment: Choose two or more control points to align to. It is best to use control points around the perimeter of the site. Use as many control points as are available or enough to envelope the site. In Configure Field>GPS Settings, set the Transformation to Plane Similarity to fit the GPS points onto the control points and set the Project Scale Factor to 1.0. After making the alignment, stake out another control point (ideally one the is not used in the alignment) to make sure the
Surface Menu

The Surface pull-down menu has Elevation Difference and surface creation commands that are described in the Civil Design manual.

Elevation Difference

Function

This command reports the cut or fill between your current position and a design surface. The design surface can be one flat elevation, a grid file, a triangulation file, a road design file, or a section file.

The type of design surface is set in the dialog shown. The Vertical Offset in this dialog can be used to modify the design surface by adding this value to the design surface. For example, if you have a design surface for the top of a road and you want to get cut/fill values to a 1.5 subgrade, then enter -1.5 in the Vertical Offset field. The Use Centerline For Station-Offset option will report the station-offset of your current position in addition to the cut/fill. When this option is active, the program will prompt you for the centerline file (.CL) to reference. For GPS and robotic total stations, the Auto Store Points At Interval will creates points whenever your position moves by more than the specified distance or time interval. This option is similar to the Auto Points At Interval command with the addition that the default description will include to cut/fill to the
design surface. When all the options are set, pick OK and the program will then prompt you for a grid file or triangulation file if you have selected these types of design surface.

**Elevation Difference with GPS**

Carlson Field will continually read your current position from the GPS receiver. A dialog box appears displaying your current position. Carlson Field finds the design elevation for this point and compares it to the elevation being reported by the GPS receiver. It then tells you how much cut or fill is required to reach the design elevation from your current position. An arrow icon will appear on the drawing showing your location. You can move around the site while in *Elevation Difference* mode and Carlson Field will report the necessary cut or fill in real-time. If you move off the area covered by the design surface, then the program will stop reporting cut/fill and instead will report "Off Surface".

![Elevation Difference dialog box](image-url)
The Store button will create a point at the current position. The default description will include the current cut/fill. When Store is selected, a dialog box will appear for entering the point number and description.

**Elevation Difference with Total Stations**

Elevation Difference uses a dialog box that is very similar to the *Point Store* dialog. Under the Setup button, make sure that the occupied point, backsight and instrument height are set. Then have your rodman set the prism over the point you are interested in. Pick *Read(F1)* or *Read & Store(F5)* and the total station will take a shot.

After the shot is taken, the dialog box looks like the one at right. Carlson Field found the design elevation for this point (557.535) and compared it to the actual current elevation (530.0). Based on the current and design elevations, Carlson Field reports to how much cut or fill is required to get to design elevation. In this case, it is fill 27.535. The cut/fill also appears in the *Desc* box. If you click *Store*, Carlson Field will record this point and plot it on the drawing, including the *Desc* as a label.
Pull down Menu Location: DTM
Prerequisite: None
Keyboard Command: elevdiff
File Name: \lsp\gpsutil.arx

Prepare Story Stake

Function

This command creates points with cut/fill information stored in the note fields for the points. Beginning at a point and facing a specified direction, the cut/fill information describes a design surface that is defined by contours and 3D polylines in the drawing. The program prompts you to pick the starting point followed by a direction point. Then the intersections for all the contours and 3D polylines between these two points are calculated and the resulting horizontal distances and slopes are shown in a dialog. In this dialog, you can edit, add or remove these slopes descriptions. The Point Description can also be specified. When OK is clicked, a point in the coordinate file is created at the starting point with this information stored in the note file. An offset point is also created at the specified offset distance back from the starting point. At the end of Prepare Story Stake, a report of all the created points and the corresponding cut/fill data is shown if the Create Report option was set. Prepare Story Stake does not draw the points in the drawing. These points
can be drawn using the Draw-Locate Points command.

The cut/fill information in the note file can be used in the Stakeout routine. In Configure Field > Stakeout Settings there is an option to Display Point Notes in Stakeout Report. With this option active, the cut/fill data in the note file will be displayed when the point is staked out.

Prepare Story Stake is not a prerequisite for Story Stake By Points or Story Stake By Polyline. In fact, working in combination with Stakeout, Prepare Story Stake is an alternative to these other routines.
**Story Stake By Points**

**Function**

This command creates a report of cut/fill slopes and distances of a design surface from a starting point. First you move to the starting point and then take a reading from the instrument to get the starting point coordinates. This starting position is shown in the drawing. Next you pick a point in the drawing to define the direction. The drawing should contain the design surface entities. The program will then calculate all the intersections with contours and 3D polylines between these two points. The resulting horizontal distances and slopes are shown in a report dialog. From this dialog, there is an option to stakeout one or two offset points set back from the starting point at the specified offsets.
Story Stake Along Polyline

Function

This command creates a report of cut/fill slopes and distances of a line across a design surface. The line is defined as perpendicular from a polyline starting at a specified station and going a specified distance. The drawing should contain design surface entities. The program will calculate all the intersections with drawing contours and 3D polylines along the line. For example, the polyline could be a toe of slope and this routine would be used to create story stakes at an interval along this polyline.

The routine starts by selecting a 3D polyline from the drawing. Then there is a dialog to specify the settings. The Station is the distance along the polyline for the starting point of the story stake. The Next Interval is used to increment the station for the next default station. The Story Offset is the length of the story stake line from the starting point. To have the story stake line go perpendicular right from the polyline, enter a positive offset value. To go left, enter a negative offset. The Read Current Position button will take a measurement from the instrument to find the station of your current position. This current station is put in the Station field. The Pick Point button will prompt you to pick a point in the drawing view. The station of this point is used to fill out the Station field.

After specifying the stakeout station and story offset, then program runs the stakeout routine to
guide you to that station on the polyline. When that point is staked, the program calculates the story stake and the resulting horizontal distances, cut/fill and slopes are shown in a report dialog. From this dialog, there is an option to stakeout one or two offset points set back from the starting point at the specified offsets.
Roads Menu

The Roads pull-down menu has commands for road stakeout and preparing the road design files. Most of these commands are described in the Civil Design manual.

<table>
<thead>
<tr>
<th>Roads</th>
<th>GIS</th>
<th>Window</th>
<th>Help</th>
</tr>
</thead>
<tbody>
<tr>
<td>Centerline Position</td>
<td>Offset Stakeout</td>
<td>Slope Staking</td>
<td>Slope Inspector</td>
</tr>
<tr>
<td></td>
<td>Input/Edit Centerline File</td>
<td>Polyline to Centerline File</td>
<td>Draw Centerline File</td>
</tr>
<tr>
<td></td>
<td>Centerline Conversion</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Input/Edit Profile File</td>
<td>Draw Profile</td>
<td>Profile Conversion</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Design Template</td>
<td>Draw Typical Template</td>
<td>Template Transition</td>
</tr>
<tr>
<td></td>
<td>Input/Edit Super Elevation</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Input/Edit Section File</td>
<td>Draw Section File</td>
<td>Section Conversion</td>
</tr>
</tbody>
</table>

Centerline Position

Function

This function determines the station and offset relative to a centerline for a point. The centerline can be defined by a centerline file (.CL), a points, or by a polyline. The centerline file can be created with commands in the Roads menu. One advantage of centerline files is that it allows you to use profile files which can report cut/fills. For the points method, you can either give two points or a starting point and azimuth. The points can be defined by a point number from the current coordinate file or by simply entering the northing and easting. The first dialog for Centerline Position has the choice for centerline file or points method. With the centerline file option, the dialog shows the last centerline file name used. If this file is correct, then click OK. Otherwise use the Select button to choose the centerline file name.

Light bars are useful for left-right guidance. To enable the light bar option go to Configure Field,
then to Centerline Position Settings and pick Use Light Bar.

Centerline Position with GPS

Carlson Field will continually read your current position from the GPS receiver. A dialog box appears displaying your current position. Carlson Field finds and displays in the dialog the station/offset for this point.

Chapter 2. Field Module
An arrow icon will appear on the drawing showing your location. You can move along the centerline and Carlson Field will report the station/offset in real-time. If you move beyond the ends of the centerline, then the program will stop reporting station/offset and instead will report "Off CL".

The Store button will create a point at the current position. The default description will include the current station/offset. When Store is selected, a dialog box will appear for entering the point number and description.

**Centerline Position with Total Stations**

Centerline Position uses a dialog box that is very similar to the *Point Store* dialog. Under the Setup button, make sure that the occupied point, backsight and instrument height are set. Then have your rodman set the prism over the point you are interested in. Pick *Read(F1)* or *Read & Store(F5)* and the total station will take a shot.

After the shot is taken, the dialog box looks like the one at right. Carlson Field reports the current coordinates and the station/offset. The station/offset also appears in the *Desc* box. If you click *Store*, Carlson Field will record this point and plot it on the drawing, including the *Desc* as a label.
Pulldown Menu Location: Roads
Prerequisite: None

Offset Stakeout

Function

This function stakeouts a point at a given station and offset of a centerline and reports the cut or fill to a design elevation. The centerline and design elevation can be defined by four methods as set in the dialog show. The Design Files method uses a centerline file (.CL) for the horizontal alignment and a profile file (.PRO) for the vertical alignment. A template file (.TPL) for the design cross section is optional for the cross slope. Without a template file, the program will use the elevation of the profile along the centerline. A superelevation file (.SUP) and a template transition file are optional. These design files can be created with the routines in the Roads menu. The Section File method uses a centerline file for the horizontal alignment and a section file (.SCT) for the design elevation. The section file consists of cross sections of offset/elevation points for a series of stations. Section files can be used instead of the Design Files method when a road design is too complicated to model using design files. For example, if the road contains special ditches at various offsets and varying lane widths, then it may be easier to enter a final section file than to define the template and template transitions. The Points method uses two points to define both the horizontal alignment and design elevations. The design elevation is linearly interpolated between the points. The points to used are specified in the next dialog by entering point numbers from the current coordinate file or by directly entering the coordinates.
The 3D Polyline method uses a 3D polyline for both the horizontal and vertical alignment. With this method, the program will prompt you to select the 3D polyline from the drawing. For both the Points and 3D Polyline methods, you can specify the starting station of the horizontal alignment. When using the Design Files and Section File methods, the horizontal alignment starting station comes from the centerline file.
After specifying the offset stakeout method, Carlson Field prompts for the station and offset to stakeout as shown in the dialog. The station should be entered as a number without the "+" symbol. The Next Interval field is used to increment the stakeout station for the next stakeout point. In addition to incrementing to the next interval, Carlson Field will also pick up special profile or centerline points between intervals. Centerline special points include: start point, end point, curve (PC, PT) and spiral (TS, SC, CS, ST). Profile special points include: start, end, vertical curve (VC, VT), high points and low points. For example, if the current station is 100 and the interval is 50 and there is a centerline PC at 112.4, then the next station after 100 will be 112.5 followed by 150. The Station List button brings up a list of all the station intervals and special stations. You can select a station to stakeout by selecting the station from the list and pressing...
OK.

There are two offsets to allow for separate offsets for the design elevation and stake location. The Side For Stakeout toggle selects between left and right offsets. The Design Offset is where the stake point elevation is calculated. The Stake Offset determines X,Y position of the stakeout point by finding this offset at the stakeout station along the horizontal alignment. Having Design and Stake offsets applies, for example, to staking the back of a curb, where the Design Offset is 12, but the stake offset is 17 (5' behind the back of curb, with the elevation reference to the actual back of curb design elevation). The Stake Offset can be specified either as an offset from the design point or as an offset from the centerline. There is also an optional vertical offset that applies to the elevation of the design point. With the Design Files and Section File methods, the vertical offset works as an offset from the template or cross section surface. For example, a vertical offset of -0.5 could be used to stakeout the bottom of a 0.5 subgrade. With the 3D Polyline and Points methods, the vertical offset adjusts the elevation from the along the centerline at the stakeout station.

The Read Current Position button will take a measurement from the GPS or total station to find the station of your current position. This current station is put in the Station field.

The Pick Point button will prompt you to pick a point in the drawing view. The station and offset of this point are used to fill out the Station and Offset fields.

After specifying the stakeout stations and offsets, Carlson Field uses the same stakeout function as used in the Stakeout command. This stakeout function guides you to the stakeout point and reports the cut/fill to the design elevation. You can store the stakeout point. When the stakeout is done, the station/offset dialog appears for staking the next point. Either enter the next station/offset or pick Exit to end Template Stakeout.

For total stations, you should run the Equipment Setup command before Template Stakeout to set the occupied point, backsight and instrument height.
Stakeout dialog for GPS

Northing: 5053.087
Easting: 6011.998
Elevation: 1001.103
HRMS: 0.020 VRMS: 0.050

Distance: 1.016
Azimuth: 52.2
CUT: 458.257
N: 0.622
E: 0.803
RIGHT >>>>

Store (F2)  Exit

Stakeout dialog for total stations

H: 354°59'36"  V: 90°00'00"  SD: 33.000
Clock hour: 11:46

Turn to AR: 354°59'36"
ZenithAng: 88°18'41"
Horiz Distance: 33.000
South: 0.703  East: 0.062
Dist: 0.706 Angle: -0°00'00"
Current Elv: 542.200

In 0.706
Right 0.000
Fill 0.952

Rod Height: 0.000  Exit

Read (F1)  Store (F2)

Pulldown Menu Location: Roads

Chapter 2. Field Module
Prerequisite: None

Chapter 2. Field Module
For example, when staking the right side, the right most offset will be used as the pivot point. The section file can optionally contain additional offsets such as centerline and edge of pavement. The program can then report the horizontal and vertical distances from the catch point to these additional offsets. The section pivot offsets can also be assigned a description which the program reports before starting the slope staking. For example, a pivot offset could be "2:1 from flat bottom ditch" which is reported to the operator. When using section files, a centerline file is also required to establish the horizontal alignment. Any station along the centerline can be slope staked because the program will interpolate between entered section stations. The cut/fill slopes from the section can be either User-Entered or Continue Last Slope. The User-Entered option will use the cut/fill slope ratios as entered in the dialog. The Continue Last Slope option will use the last two points in the section file as the cut/fill slope. This Continue Last Slope option applies to section files that contain pivot point to ground segments whereas the User-Entered option is for section files that end at the pivot points.

User entry is a simple method for slope staking that only requires a centerline file. With this method, the program prompts for the cut/fill slopes and the pivot offset and elevation. The program finds this offset-elevation for the stake station along the centerline and begins the cut/fill slope from this point.

The 3D Polyline method uses a 3D polyline for both the horizontal and vertical alignments. The
program will prompt you to select the 3D polyline from the drawing. There are two polyline methods. The Station Along Polyline method does slope staking perpendicular to the polyline like the other slope staking methods. The Endpoint Projection is a special method that slope stakes from the selected end of the polyline. This method is described at the end of this section.

The first dialog in Slope Staking chooses the design method. For Design Files method, the files are specified in this dialog. For the other methods, the cut and fill slope ratios are also defined in this dialog.

The next dialog sets the station to slope stake. The station should be entered as a number without the "+" symbol. For the 3D Polyline method, the starting station of the polyline is specified in the first dialog. For all the other methods, the starting station of the alignment is set in the centerline file. The Next Interval field is used to increment the stakeout station for the next stakeout point. The Read Current Position button will take a measurement from the GPS or total station to find the station of your current position. This current station is put in the Station field. The Pick Point button will prompt you to pick a point in the drawing view. The station of this point is used to fill out the Station field. For the User-Defined method, this dialog also contains the offset and elevation of the pivot point. For the 3D Polyline method, this dialog also contains the pivot point offset and vertical offset from the 3D polyline to the pivot point.
For the design file method, the centerline elevation at the stakeout station is calculated using the design profile and then the template is applied to calculate the pivot point. For the section file method, the pivot offset is interpolated from the section file. For example, if the stakeout station is 75 with offset right and the section file has offset-elevation of 18.0 right, 100.0 elevation at station 50 and has 20 right, 102.0 elevation at station 100, then the pivot offset for station 75 would be 19.0 right, 101.0 elevation. For the user entry and 3D polyline methods, the pivot point is specified Station For Slope Stake dialog.

After the slope stake station and pivot point are specified, Carlson Field begins to read the GPS receiver or total station to get the current position. The existing surface to tie into is defined by the elevations from these current position coordinates. The point where the cut or fill slope from the pivot point intersects the existing ground is called the catch point. As each coordinate is read, an existing surface cross section is built and the catch point is calculated. Carlson Field will automatically determine whether to find the catch point on the right side or left side of the centerline depending on the side of your current position. The program displays, in real-time as you move, the northing-easting and station-offset-elevation of your current position and the offset of the catch point. The distance from the current position to the catch point is reported as the offset difference as either "IN" or "OUT". The OUT means you should move out from the centerline. The IN means that the catch point is closer to the centerline. Based on this offset difference, you move perpendicular to the centerline either towards or away from the centerline to reach a new offset from the centerline while maintaining approximately the same station. The difference between your current station and the stakeout station is reported as the "UP" or "DOWN" distance. The UP means that your current station is less than the stakeout station and you should move up the centerline. Likewise, the DOWN means that your current station is greater than the stakeout station.
station and you should move back down the centerline.

When the catch point is located, press the Store button to end the slope staking. A report dialog is then displayed. The Catch Pt is the actual station, offset and elevation of the target catch point. The Stake Pt is the as-staked station, offset and elevation of your current position. The dialog also reports the horizontal and vertical distances from the catch point to the pivot point and the other template points. The Store Catch Point option will record the as-staked coordinates of the catch point to the current coordinate file. The Stake Offset Point is an option to locate an offset point. The offset to stake can be entered as a distance from the catch point or as an offset from the centerline.

![Catch Point Report](image1)

To locate the offset point, the same stakeout function from the Stakeout command is used. This

![Offset Point Report](image2)

To locate the offset point, the same stakeout function from the Stakeout command is used. This

Chapter 2. Field Module
function will guide you to the offset point. When the offset is reached, pick the Store button. Then an Offset Point Report dialog pops up containing the station, offset and elevation of the offset point and the horizontal distance, vertical distance and slopes from the offset point to the catch point, from the catch point to the pivot point and from the pivot point to the template points.

After locating the offset point, the station to stakeout dialog appears. You can enter the next station to stakeout or pick the Exit button to end Slope Staking.

**Endpoint Projection**

This is a special case of the 3D Polyline method that slope stakes from the end of the polyline. The program will prompt to pick a polyline and the end to stake from is the end nearest to the pick position. The direction of slope staking is in the direction of the polyline as if extending the polyline. The program prompts for the elevation of the pivot point which defaults to the elevation at the polyline endpoint. There is also an option to offset the pivot point along the polyline back from the endpoint.

After the pivot point is specified, the program starts the stakeout routine to guide you to the catch point. Then there is a report to show the difference between the staked and the calculated catch point.
**Slope Inspector**

**Function**

This command reports the azimuth, distance and slope between your current position and a starting point. The command starts by prompting you to move to the starting point and take a reading. This sets the starting point.

**Slope Inspector with GPS**

Carlson Field will continually read your current position from the GPS receiver. A dialog box appears displaying your current position and the azimuth, distance and slope from the starting point to your current position. An arrow icon will appear on the drawing showing your location. Pick the New Start button to set a new starting point.

**Slope Inspector with Total Stations**

Before running Slope Inspector, make sure that the occupied point, backsight and instrument height are set correctly in the Equipment Setup command. Then start Slope Inspector and read a measurement for the starting point. A dialog box appears displaying your current position. Pick the Read button to take another measurement. Carlson Field will calculate the new point and report the azimuth, distance and slope from the starting point to the new point. An arrow icon will appear on the drawing showing your location. Pick the New Start button to set a new starting point.
GIS Menu

The GIS menu commands for Field are described in the GIS manual.
Equipment Menu

Apache Lightbar

Carlson Field can use an external Light Bar for determining elevation differences and centerline offsets.

Light Bars can indicate whether your current position is in cut, fill or on-grade when set vertically. When set horizontally, Light Bars can give centerline left/right offsets. Currently Carlson Field supports a light bar made by Apache, as well as by Mikrofyn, that has arrows for up/down, or left/right, and a row of lights for on-grade. The Light Bar must be connected to a separate serial port than the GPS.

CSI GBX Pro

Hardware Setup

1. Connect the receiver to the antenna by coaxial antenna cable if it is not already connected, and ensure that that the receiver has ample power.

2. Ensure that the antenna is tracking corrections from an MSK Radio Beacon. The easiest way to do this is to use the antenna's automatic frequency scanning when first powering on the receiver.
a) To do this, enter the [SETUP] menu, and select the option [AUTO BX SEARCH]

Note: The beacon automatically selected by this scan will be saved to the receiver's memory and used automatically in the future, until either the scan is executed again, or until a new beacon is specified manually. Thus, it is not necessary to scan each time the beacon is used, provided it is still operating in the same general area.

b) A scan can be performed again in the event that the beacon is lost to scan for the next nearest beacon.

3. Enter the [Setup] menu, then select [Options] then [NMEA ON/OFF]. This menu allows the enabling or disabling of various NMEA messages. The only ones which are necessary are the GGA, GSV and GSA messages. All others should be disabled.

Software Setup

4. In Carlson Field, no further setup is necessary to make use of the CSI GBX Pro. Simply use the other Carlson Field functions as normal. Note however, that the elevations reported by the CSI GBX Pro are MSL (Mean Sea Level).

Depth Sounder

Function

Carlson Field data collection can be used in conjunction with a depth sounder to survey the beds of rivers and lakes. Carlson Field takes input from both a GPS receiver and a depth sounder to determine and record the elevation of the terrain directly below the surveying boat or barge.

All of Carlson Field's routines work with the depth sounder to let you collect points on the underwater terrain. The elevation stored for each point is the elevation of the bed. Modeling of the bed surface works as easily as modeling any surface using Carlson Software. Carlson Field can be a powerful tool for marine surveying and construction.

Settings

To modify the Field depth sounder settings, go to the Field menu and select Configure Field. Choose the Depth Sounder Settings button.

The Depth Sounder Settings menu appears. At this point, Hydrotrac by Odom is the only equipment-specific depth sounder interface. Carlson Field works with other depth sounders that have NMEA standard interface. If you want to use Carlson Field without the depth sounder,
make sure the Model is set to None.

For the Hydrotrac model, the depth sounder should be set so it outputs message DESO25 I/O. This is done using the Hydrotrac software. Odom should be contacted with any problems involving setting this message (www.odomhydrographic.com or 225-769-3051). The draft setting on the Hydrotac should also be set. This will account for the height difference between the water surface and the working sensor of the Hydrotac.

On the next line appears a box labeled Store Depth in Notes. Carlson Field saves point data in a coordinate file and in a text note file. By checking this box, the note file will record the water depth at each reading along with the other information about that point. (Settings to control the rest of the information saved in this file can be found in the menu Configure Field > Point Settings.)

The window labeled Debug should be set to zero for normal use.

The row of buttons labeled Serial COM Port refer to the COM port on your computer where the depth sounder is plugged in. Carlson Field requires two serial points on the computer when working with a depth sounder (one for the GPS and the other for the depth sounder). The depth sounder serial port must be separate from the GPS serial point.

Starting Out

Before working with the depth sounder, we suggest that you make sure the GPS system is working properly with Carlson Field. De-activate the depth sounder by setting the Model to None in the Depth Sounder Settings dialog box. Set up the GPS system that you are using and plug the rover receiver into the COM port for the GPS. Go to Monitor GPS Position under the Field menu. Check that the information being output is correct: Are the latitude and longitude readings what they should be? Are the north and east coordinates aligned to your job coordinate grid? Are the HRMS and VRMS low enough (less than one)? Is the status fixed? If it's autonomous or
float, this rover could be having trouble receiving the radio corrections the base receiver should be broadcasting. If everything is working properly, exit the monitor screen and start the depth sounder setup.

Measure the vertical distance from the GPS antenna to the surface of the water. This distance will be called the rod height. Go to the Configure Field>General Settings window and enter this measurement in the Rod Height box.

Plug the depth sounder into the depth sounder COM port on your computer. Go to the Configure Field>Depth Sounder Settings window and set the depth sounder Model. Set the rest of these settings as you want them and click OK.

Go back to Monitor GPS Position. Everything should appear as before, except there should be a new entry called Depth and Elevation should have changed to Bottom Elv. The correct depth should be showing and the Bottom Elv should be showing the elevation of the bed.

The usual Carlson Field functions will all work with the depth sounder active. The windows for Monitor, Point Store and Auto Points at Interval will display the depth when the depth sounder is set as active.

Geodimeter

Geodimeter 600 For Remote Mode

Note: Firmware version 696-03.xx or higher is required on the instrument. To check the version, pick MNU-5-4-1.

SET-UP

1. Connect the instrument to the battery pack. There is no need to connect the keyboard to the battery if it is going to be turned off, or attached to the unit.
2. Connect the prism to the top port of GeoRadio.
3. Connect the bottom port of the GeoRadio to Carlson Field. Then turn on the radio.
4. Turn on the Geodimeter. The Geodimeter starts with the screen for leveling the instrument. When the instrument is leveled press [ENT] key to continue to the next step. Now the instrument starts compensator calibration. You can wait for calibration to finish or turn it off. To turn calibration off press on [F] 22, enter 0 for comp. This needs to be done when the instrument is turned on and before [ENT] is pressed.
5. Next Geodimeter will ask for different values for pressure, offset, etc. They can either be left like they are by pressing on [ENT] or they can be changed.
6. Press [F] 79, it is the End of Transfer character, which should be set to 4.
7. To set radio, and station channels, press [MNU], and enter 1 for "Set". After set press 5, which will give the user opportunity to change channel, station, and remote address.
Note: The channel, station and remote address on the Geodimeter should match the channel, sta-
tion and remote address in Carlson Field.

8. To set the Geodimeter for remote mode, press on RPU, then 3 for remote and 1 for ok, you can
answer [NO] to "Define Window?" If [ENT] is pressed, the instrument will ask "Aim to A Press
Ent", for which the user have aim to upper/lower left boundary and press [ENT], for "Aim to B
Press Ent", aim to the upper/lower right boundary and press [ENT]. For "Measure ref obj?" press
[ENT] if you want a reference object, otherwise press [NO]. Than the instrument is going to say
remove keyboard however the keyboard can stay on.

9. After Geodimeter display screen turns itself off, it's ready for Carlson Field.

CARLSON FIELD

1. In Configure Field, under equipment type there should be Geodimeter. In Communication
Settings Baud Rate should be set to 9600.

2. After Configure Field go to Equipment Setup and make sure GeoRadio is checked, and the
channel, station and remote address is the same as it is in the total station.

Note: We recommend using channel 3.

3. If calibration box is checked the instrument will calibrate, to turn of calibration the box should
be unmarked.

4. In setup there is also an option to turn on/off-tracking lights.

**Geodimeter 600 For Direct Connection**

1. Connect the instrument to the battery pack, and the control unit to Carlson Field.

2. Under Field go to Configure Field and place Geodimeter in Equipment type.

3. Click on General Settings make sure that the baud rate is set to 9600.

4. Exit Configure Field.

5. Go to Equipment Setup and check Connect to Station and click OK.

Now you are ready.
InnerSpace Tech depth sounder

The communication settings for the InnerSpace Tech depth sounder are 9600-N-8-1.

Laser Atlanta

To setup Laser Atlanta select Menu on the instrument, then Serial, and set Baud rate to the same as Carlson Field's and Format to Laser Atlanta.
Leica Disto

The communication settings for the Leica Disto are 9600-E-7-1.

Leica GPS System 500

Setting Up a 500 Series Receiver

1. Connect the antenna cable to the ANT Port on the front of the receiver, and to the antenna.

2. If you are using the PacCrest radio module, screw it in place over Port 1 on the receiver and attach its antenna cable. Otherwise, connect any radio being used to Port 1, 2 or 3.

3. If an external power source is being used, be sure to plug it into the PWR Port on the front of the receiver.

4. If external power is not being used, ensure that there are batteries in one or both of the battery slots on the bottom of the receiver.

5. Plug the 9 pin serial connection cable into the serial port of the computer running Carlson Field and into the Terminal Port on the front of the receiver.

Configuring Carlson Field for Use With a 500 Series Receiver

1. Select "Configure Field" from the Field pull-down menu. This will open a new window with several buttons on it, as well as a pulldown list labeled "Equipment Type." Select "Leica 500 Series" in the Equipment Type menu, then select "Communication Settings."

2. Ensure that the COM port is set to the one that the serial cable is plugged into, and that the Baud Rate is 9600, the Char Length 8, the Stop Bits 1, and the Parity None. Close this menu and the Configure Field menu.

3. In the Field pull-down menu, select "Equipment Setup." This will open another menu with several selectable options and several buttons.

4. Use the radio buttons on the top right to select whether the receiver will be a rover or a base station. Also be sure to select the antenna types being used from the pulldown menu at left.

5. Enter the desired Satellite Elevation Cutoff in the text box above the column of buttons. All satellites with elevations less than this number will not be used in position calculation(receiver default is 15).

6. Select the "Radio Settings" button. This will open another window with several selectable settings. Select the Port number the radio is attached to on the front of the receiver, the baud rate of the radio, number of radio stop bits and radio parity. These last three settings should be listed in the documentation for the radio being used. Also, select the desired format to use for sending and receiving messages from the bottommost option. Exit this menu.
7. If the GPS receiver is being configured as a base station, select the "Configure Base Station" button from the Equipment Setup menu, and proceed with step 8. Otherwise, the receiver is ready for use.

8. There will now be a menu with a few buttons to select a method of determining the base station's present location. The options are:

Read From GPS- Read one or more position readings from the GPS and use this position or the average of several positions for the base station corrections.

Enter Lat/Lon- This option will bring up a menu to enter the exact Latitude, Longitude, and elevation of the receiver's position by hand.

Enter State Plane Coord- This option will bring up a menu to enter the coordinates of the position of the base station according to the state plane coordinate system.

Read from File- This option will read a coordinate set from a file already saved to the computer.

Select whichever method will be used, and enter any necessary data. The receiver is now configured and ready for use.

Other Buttons In the Setup Menu:

1. Power Cycle Receiver will shut the receiver down and then power it up again. Used to clear the receiver's memory.

2. Power off Receiver shuts the receiver down. Note that if this button is pressed, any settings changes made while in this menu will not be saved to the receiver.

3. Send command to receiver allows for sending messages to the receiver. The user must enter the message by hand. This feature is only intended for use in conjunction with the technical support provided with Carlson Field.

Troubleshooting the Leica 500 Series in Carlson Field

Several possible errors can occur in the course of using a 500 Series Leica receiver with Carlson Field. Carlson Field will use all its standard error messages to report usual types of error messages, such as an inability to communicate with the satellites that are being tracked. In addition, the Leica 500 Series of receivers will have their own set of error messages unique to themselves. This type of error message is reported if there is an error during the transmission of various configuration messages to the receiver to set up the base station settings. Such messages will say "Set Port Message Rejected", or "Set Base Antenna Message Rejected" or "Set Antenna Height Message Rejected" or "Set RTK Message Rejected." Each indicates which particular facet of the configuration failed. If one of these messages is rejected, it is likely a momentary transmission error. If, on the other hand, several (or all) are rejected, it is possible there is a problem in the communication line between the computer and the receiver, which should be checked.
Leica TC Series

Leica TC Series Instrument Setup

On the instrument, make sure that the communication settings have CR/LF for the terminator.

Remote Mode

1. Turn on Leica

2. Connect Leica to rover radio, and connect the radio to the larger battery.

3. Connect the base radio to Carlson Field, and the smaller battery.

4. In the Field menu go to Configure Field, and under equipment type put Leica TC

5. To make sure the baud rate matches, under the Field menu go to Configure Field and click on Communication Settings and check if the baud rate is 19200. When Leica is turned on under Main Menu enter 5 for "Configuration", and 2 for "Communication Mode", then enter 1 for "Gsi parameters", and check if the baud rate is also set 19200.

6. Line Terminator in "Gsi parameters" should be set to CR/LF


8. When back in Communication Mode screen enter 5 for "RCS (Remote) ON/OFF" and make sure it's NOT set for remote mode.


10. In the Field menu go to Equipment Setup and for Connection Mode check remote.

11. When done click on OK.

To put Leica in Tracking: On Gun press "FNC" then ATR+ and LOCK+

TCA 1800

1. Turn on Leica

2. Connect Leica to Carlson Field

3. In Field go to Configure Field, and under equipment type put Leica TC

4. To make sure the baud rate matches, under the Field menu go to Configure Field and click on Communication Settings and check the baud rate. When Leica is turned on press [F3] for "conf", then enter 3. The baud rate can be changed by pressing [F6] for "list", when done enter [CONT]. In addition to baud rate parity, char length, and stop bits should also match.

Note: Default in Carlson Field is not the same as default in Leica.
Leica 1100

To set up the Leica 1100 total station select the following commands on the instrument: Main Menu > Configuration > communication mode > GSI parameters. In the GSI parameters command copy the following settings: (baud=to Carlson Field's, protocol=GSI, parity=to Carlson Field's, Terminator=CR/LF, Data Bits=to Carlson Field's, Stop Bit=1), Geocom Param (baud=Field's) RCS Param (baud=Field's). Also, make sure RCS mode is OFF.

Leica 700

To set communication settings for Leica 705s, go Shift key then Prog (Menu) key and then to All Settings and last to PC Comm.

Manual Total Station

This method allows you to run Carlson Field in total station mode without being connected to equipment. The program will prompt you to enter the horizontal angle, zenith angle and slope distance. This method can be used for demonstration purposes or to work with total stations that cannot connect to Carlson Field. For these total stations, instead of the automatic connection, you can take a shot, read the instrument and then manually enter the data into Carlson Field.

As with other total stations, the first step is to run Equipment Setup to establish the occupied point, backsight and instrument/rod heights before running Carlson Field functions. Then
in Carlson Field functions, when you pick the Read button, the program will bring up a dialog for entering the angles and distance. The angles should be entered in dd.mmss format (degrees.minutes.seconds).

**Mikrofyn Lightbar**

Carlson Field can use an external Light Bar for determining elevation differences and centerline offsets. Light Bars can indicate whether your current position is in cut, fill or on-grade when set vertically. When set horizontally, Light Bars can give centerline left/right offsets. Currently Carlson Field supports a light bar made by Mikrofyn named RD-4 1137551, as well as by Apache, that has arrows for up/down, or left/right, and a row of lights for on-grade. The Light Bar must be connected to a separate serial port than the GPS.
Navcom Configuration Guide

This guide will walk you through the setup process for your Navcom units. It covers individual unit setup as well as base/rover setup under the simplest possible configuration. If you want to customize the configuration, consult the reference manual.

A) Preliminary setup steps

Perform the following preliminary steps to initialize your computer for communication with your Navcom units:

1) Startup the Carlson product you intend to use.
2) Select CONFIGURE FIELD, from the FIELD Menu.
3) Select NA VCOM from the EQUIPMENT TYPE dropdown
4) Click the COMMUNICATION SETTINGS button. Verify that the SERIAL COM PORT is set to the port you intend to use to communicate with your Navcom unit. (usually COM1).
5) In the COM PORT SETTINGS box, click DEFAULT, and verify that baud rate=19200, parity=NONE, char length=8, and stop bits=1.
6) Click OK, and then click the GPS SETTINGS button.
7) Under GPS SETTINGS, set your HRMS and VRMS tolerance. For single-unit setup, these numbers should be at least 10. For base/rover configuration, they should be around 0.01.
8) Under PROJECTION TYPE, select the coordinate plane you wish to use. For state plane, make sure you choose the proper ZONE.
9) Click OK and then click EXIT

B) Single-unit setup (no corrections)

Before attempting a multi-unit setup, it is recommended that you first try setting up your Navcom unit to output an uncorrected position. The steps to do so are explained here:

1) Mount your GPS Antenna on a tripod in a place where its view of the sky is not obstructed.
2) If your antenna is separate from your receiver, connect your antenna to your receiver's AN TENNA port. (This step can be skipped for the RT-3010S, and other all-in-one models)
3) Plug your receiver into a power supply, or insert fully charged batteries into the battery ports. (Not all units have battery ports).
4) Turn your receiver on by holding down the power button for a few seconds, or until the status lights flash on.
5) Use the serial port cable to connect your computer to port A of your Navcom unit. Make sure the port on your computer that you use corresponds to the one you chose during preliminary setup.
6) Under the FIELD menu, choose EQUIPMENT SETUP.
7) If a PORT SETUP window pops up, set CONTROL PORT to PORT A, and RTK DATA PORT to RADIO PORT.
8) Setup is now complete. Steps that follow are optional.
9) Click the NAVIGATION STATUS button. From here you can monitor the progress of your Navcom unit as it calculates its position. Click AUTOREFRESH to view continuously updated status reports.

10) It may take a few minutes for the unit to calculate its position, if the unit was reset, or recently turned on. When the calculation is complete, VALID NAVIGATION will read YES. When this occurs, the Navcom unit is ready for use. Click CLOSE, then click CANCEL WITHOUT SAVING.

11) To monitor your position, choose MONITOR GPS POSITION from the FIELD menu, and you will see your current position. All Carlson Field GPS functions should now work.

**C) Multi-unit setup (using corrections from a base)**

1) Base Setup
   a) Perform the preliminary and single-unit setup steps described above.
   b) Attach the radio antenna to the radio port of your base unit.
   c) Select EQUIPMENT SETUP from the FIELD menu.
   d) Select a CORRECTION TYPE. We recommend NCT RTK.
   e) CLICK the CONFIGURE BASE button.
   f) When prompted to enter a position, enter the exact position of the base unit. Note that the accuracy of your rover's calculation depends on this position being completely accurate.
   g) Enter a station ID of 0. The station ID is only used in RTCM mode.
   h) Verify that STATION TYPE now reads BASE.
   i) Base station setup is complete. Click SAVE SETTINGS AND EXIT.

2) Switch the device you're plugged in to:

   After configuring the base, unplug your serial cable from your base's port A, and plug it into your rover's port A.

   Note: Whenever you switch the device you're plugged into be sure to close the Equipment Setup window first.

3) Rover Setup
   a) Perform the preliminary and single-unit setup steps described above.
   b) Attach the radio antenna to the radio port of your rover unit.
   c) Select EQUIPMENT SETUP from the FIELD menu.
   d) Select a CORRECTION TYPE. We recommend NCT RTK. Note that this selection must match the selection made during base setup.
   e) CLICK the CONFIGURE ROVER button.
   f) Verify that STATION TYPE now reads ROVER.
   g) Rover setup is complete. Steps that follow are optional.
   h) You can verify that the rover is receiving correction by clicking the MONITOR INCOMING...
CONNECTION, and then clicking AUTOREFRESH. The open window shows the time since each correction type was last received (delta time). In NCT RTK mode, the delta time of 5b, should stay around 1 second, and the delta time of message 5c should not go above 30 seconds. If these numbers are high, or if they read NEVER, try repeating the setup process or calling Carlson Software technical support.

i) Click SAVE SETTINGS AND EXIT, and then choose MONITOR GPS

POSITION from the FIELD menu. The STATUS display should eventually go to LOCK.

Troubleshooting Base/Rover Configuration:

If you've configured a base to output corrections, and you're rover does not appear to be receiving the corrections, try each of the following in order:

1) Verify that your BASE and ROVER are both set to the same correction type.
2) Under Configure Radio, check that your ROVER is set to slave, and that your BASE is set to master.
3) Under Configure Ports, check that both your base and your rover's RTK Data Ports are set to the proper value (Usually Radio Port).
4) Under the Edit Base Position, check that your BASE is set to a valid position. Note that if the given position is too far away from the position the BASE is reading, the BASE will not send corrections.
5) If you're trying to use RTCM, make sure the BASE and ROVER have the same station ID's.
6) Try increase the RTK Max Age constraint.
7) Under Navigation Status, verify that the Navigation is valid on both units. If either unit does not have a valid position solution, correction will not work.
8) Under Monitor Corrections, verify that the corrections you're using are arriving regularly. If they aren't you may need to reset both units.
9) Try configuring the BASE and ROVER again.
10) If all else fails, Soft Reset both units through the Reset Unit menu. After doing so, you will have to reconfigure the port settings of each device through the Configure Ports menu, and wait a few minutes for the devices to recalculate their position.
11) If none of these steps work, contact Carlson Software Technical Support.

Navcom GPS Setup

Carlson Field supports Navcom's NCT-2000D GPS message protocol, firmware versions 2.6 and later. If your Navcom unit has an earlier firmware version, contact Navcom for a free upgrade. Carlson Field has been tested extensively with Navcom models RT-3010S and RT-3020M.

From the Navcom GPS setup menu, or any of its submenus, the current device settings can be obtained by clicking the Retrieve Settings button. New settings can be saved by clicking the...
Save Settings or the Save Settings and Exit button. To cancel your changes, click Cancel without Saving.

By changing the SV Elevation Mask, you can prevent the Navcom Unit from using any satellite below a specified elevation angle (Range: 0-90).

By changing the PDOP Mask, you can prevent the Navcom Unit from using any GPS solution with a PDOP above a specified value (Range: 1-25).

By changing the RTK Max Age, you can prevent the Navcom Unit from using any RTK corrections older than a specified number of seconds. (Range: 0-1275, Multiple of 5).

By changing the Base Station ID on a base, you can provide your base with a unique identifying number so that rovers can specify which base they want to use for corrections. By changing this setting on a rover, you can specify which base unit you want to use. If 0 is specified, the rover will use any base station it can find. The base station ID only applies when using the RTCM correction format. (Range: 1-1023)

You can choose between 4 different Correction Types: NCT (Navcom Proprietary), CMR (Trimble's format), RTCM RTK (Messages 18-22), or RTCM DGPS (Message 1 and 9). When configured to BASE, changing the correction type changes the type being sent. When configured to ROVER, changing the correction type changes the type the unit is listening for. A ROVER will ignore all incoming correction messages except those of the type specified.

Configure Ports Submenu:

The Control Port should be configured to Port A or Port B, depending on which of the Navcom units' ports you are plugged into. Note that the Control Port refers to the number of the port on the Navcom unit, NOT the number of the COM port on your computer. If the Control Port is configured improperly, you will not be able to communicate with your Navcom unit.

The RTK Data Port refers to the device port out of which RTK corrections will be sent. This value should be set to Radio Port, unless you want to set up a non-wireless Base/Rover connection through Port A or Port B. The RTK Data Port cannot be the same as the Control Port.

Configure Radio Submenu:

The Radio ID is the value used to identify a unit on a wireless network of Navcom units. Make sure that no other Navcom unit in your vicinity shares the same Radio ID. By default, the Radio ID is the same as your Navcom unit's serial number. This value can be changed, although there isn't usually any need to do so.

The Local Radio Type can be set to either Master or Slave. Radio communication will only work between Masters and Slaves. Only one unit on your network should be set to Master. It makes sense to make the base unit a Master, and all rovers Slaves. These settings will be handled
automatically by the *Configure Base* and *Configure Rover* routines. So there generally isn't any reason to set the *Local Radio Type* manually.

The *Local Antenna Power Level* allows you to configure your radio to use more or less power. The less power the radio has, the less it will be able to communicate over longer distances. It may be useful to change the power level if you're rover is not traveling far from your base, and you're trying to conserve battery power.

Within the *Navcom Radio Setup* menu, you will be able to access the following status information for all visible Navcom units on the network:

- **External Power**: Indicates whether the unit is plugged into an external power source (On or Off).
- **Battery A**: Indicates whether a well charged battery is plugged into Battery Port A (On/Good or Off/Low)
- **Battery B**: Indicates whether a well charged battery is plugged into Battery Port B (On/Good or Off/Low)
- **Status**: Indicates whether the unit is sending out corrections. (BASE or ROVER)

If more than two units are present, you can access this information for the additional units by selecting the desired unit's radio ID from the *Remote Radio ID* dropdown menu.

*Configure RTCM* Submenu:

Note: To access this menu, first configure the unit as a BASE and set the *Correction Type* to either *RTCM RTK* or *RTCM DGPS*.

Choose *message 18/19* to make your RTCM RTK base broadcast RTCM message types 18/19.

Choose *message 20/21* to make your RTCM RTK base broadcast RTCM message types 20/21.

Choose message 1 to make your RTCM DGPS base broadcast RTCM message type 1.

Choose message 9 to make your RTCM DGPS base broadcast RTCM message type 9

*Edit Base Position* Submenu:

Note: To access this menu, first configure the unit as a BASE.

If your BASE already has a GPS position set, it will be shown here. (If you don't see it, trying pressing Retrieve.) To edit this value, change the displayed number and press the *Lock* button.

Click *Survey* to read a new GPS position from the Navcom unit.

Click *Empty*, to clear the GPS position from the unit.

*Reset Unit* Submenu:
Click *Soft Reset* to send a reset command to the Navcom unit. If the command is successful, all three status lights on the unit should go solid temporarily. After performing a soft reset, you will have to go to the *Configure Port* Submenu to reconfigure the control port.

Click *Factory Reset* to send an emergency reset command to the Navcom unit. However, in nearly all cases, it is only necessary to use the *Soft Reset* button. After performing a factory reset, you will have to go to the *Configure Port* Submenu to reconfigure the control port.

*View Firmware* Submenu:

This submenu displays the Navcom firmware version your unit is using, along with the hardware serial numbers and the hardware model name.

*Navigation Status* Submenu:

If *Valid Navigation* reads *Yes*, your unit has successfully solved its position. If it reads *No*, the unit's position has not yet been calculated, and an error message explaining why will be displayed in the *Error* field. A rover will not try to use RTK corrections unless its navigation is valid. Similarly, a base will not broadcast correction unless its navigation is valid.

*Navigation Status* will read *AUTONOMOUS* if it is not receiving the type of corrections it has been configured to use. It will read *FLOAT* if it is receiving the right kind of corrections, but hasn't finished using them to calculate its position. It will read *LOCK* when it is receiving corrections and has successfully used them to calculate its position.

*Navigation Mode* displays the specific type of correction that is currently being used.

*# of Satellites Used* shows the number of satellites the unit is able to use in its solution. All *DOP* values are also shown here (GDOP, PDOP, HDOP, VDOP, and TDOP).

Click *Refresh* to load the latest values from the device.

*Monitor Incoming Corrections* Submenu:

Note: To access this menu, the local unit must be configured as a ROVER.

This menu displays the number of seconds since the arrival of each RTK correction type. At the top, the correction type currently being used is displayed.

In *NCT Correction Mode*, the relevant messages are 5B (correction), which should be arriving every second, and 5C (base position), which should be arriving every 16 seconds.

In *CMR Correction Mode*, the relevant messages are cmr0 (correction), which should be arriving every second, and cmr1 (base position), which should be arriving every 30 seconds.

In *RTCM RTK Correction Mode*, the relevant messages are RTCM message 22, and either messages 18 and 19, or messages 20 and 21, depending on your base’s RTCM setup. Messages 18-21...
should be arriving every second. Message 22 should be arriving every 6 seconds.

In *RTCM DGPS Correction Mode*, the age of correction messages (1 and 9) cannot be monitored here.

Click *Refresh* to load the latest values from the device.

**Configure Base Submenu:**

Before clicking *Configure Base*, first choose the type of corrections you want to use. When you click *Configure Base*, all steps necessary to configuring a base will be performed. You will be prompted for a Base Position and a Radio ID. Upon completion, the unit status should read BASE. If it does not, or if an error occurs during base configuration, try again, or consult the Base/Rover configuration troubleshooting section below.

**Configure Rover Submenu:**

Before clicking *Configure Rover*, first choose the type of corrections you want to use. When you click *Configure Rover*, all steps necessary to configuring a base will be performed. Upon completion, the unit status should read ROVER. If it does not, or if an error occurs during rover configuration, try again, or consult the Base/Rover configuration troubleshooting section below.

**Switching the device you're plugged in to:**

Whenever you switch the device you're plugged into be sure to either close the Equipment Setup window, or click *Retrieve Settings* from the top level Equipment Setup menu.

**Troubleshooting Invalid Navigations:**

If the *Navigation Status* menu reports an invalid navigation, your unit has not yet been able to calculate its position. The unit may need more time, if less than 4 satellites are visible, or an error is reported. If you can't get a valid solution for a few minutes, try raising the PDOP mask, or lowering the Satellite elevation mask.

**Troubleshooting Base/Rover Configuration:** If you've configured a base to output corrections, and your rover does not appear to be receiving the corrections, try each of the following in order:

1. Verify that your BASE and ROVER are both set to the same correction type.
2. Under *Configure Radio*, check that your ROVER is set to slave, and that your BASE is set to master.
3. Under *Configure Ports*, check that both your base and your rover's RTK Data Ports are set to the proper value (Usually *Radio Port*).
4. Under the *Edit Base Position*, check that your BASE is set to a valid position. Note that if the given position is too far away from the position the BASE is reading, the BASE will not send corrections.
5. If you're trying to use RTCM, make sure the BASE and ROVER have the same station IDs.

6. Try increase the RTK Max Age constraint.

7. Under Navigation Status, verify that the Navigation is valid on both units. If either unit does not have a valid position solution, correction will not work.

8. Under Monitor Corrections, verify that the corrections you're using are arriving regularly. If they aren't you may need to reset both units.

9. Try configuring the BASE and ROVER again.

10. If all else fails, Soft Reset both units through the Reset Unit menu. After doing so, you will have to reconfigure the port settings of each device through the Configure Ports menu, and wait a few minutes for the devices to recalculate their position.

11. If none of these steps work, contact Carlson Software Technical Support.

**Troubleshooting when you cannot establish communication with the unit:**

If all of your commands in the Equipment Setup menu are failing, try opening the Configure Ports submenu, selecting the proper Control Port, and saving the new settings. Make sure that you're plugged into the port you have chosen to be the control port.

If this does not work, issue a soft reset command. If this fails, try a factory reset command. If even this fails, call Carlson Software Technical Support.

---

**Nikon Total Stations**

**Nikon A-Series**

Nikon A-Series includes the A5LG/A5, A10LG/A10 and A20LG/A20. Also the C-100 and D-50 have the same communication as the A-Series and should be used in the SET mode.

**Nikon 500 Setup**

1. Turn on Nikon

2. Turn it Horizontally and Vertically to set it.

3. Connect Nikon to Carlson Field

Note: 9-pin serial cable from Nikon to Carlson Field should be NGT type and not SOKTOP.

4. In Field go to Configure Field, and under equipment type put Nikon 300,400,500 series.

5. To make sure the baud rate matches, under the Field menu go to Configure Field and click on Communication Settings and check the baud rate. On Nikon press [MENU], then 3 for "sett", and 6 for "comm". The baud rate can be changed using the arrow keys.
6. Exit the Configure Field menu.

7. To check if units (Ft /M) matches for correct results, in Carlson Field under Settings go to Drawing Setup and select the appropriate button. On Nikon, press [MENU] and 3 for "sett" again, but now press 5 for "unit".

Nikon 310

Set the same baud rate in the Nikon 310 station as you did in Carlson Field and set the Nikon instrument to the record format by selecting on the instrument Fnc->5(Set)->6(other)->3rd screen.

OmniStar Otto

In Field go to Configure Field and under equipment type select CSI GBX/OmniStar Otto and in Communication Settings set the baud rate to 9600.

Simulation GPS

Simulation GPS mode is for demonstration purposes to show or practice Carlson Field functions. This mode allows you to run Carlson Field without being hooked up to any equipment. The program will automatically generate a position. This position is the first point in the alignment. If there is no alignment, then the starting point is 5000,5000,1000. There are keyboard commands to control the simulation position during continuous read commands such as Stakeout and Track.
Position.

Here are the keyboard commands:

L - Turn Left
R - Turn Right
F - Go Faster
S - Slow Down
U - Up
D - Down
W - Switch Direction

**Sokkia**

**Sokkia Radian IS**

Hardware Setup

1. Make sure that the Radian IS has fully charged batteries installed, as described in the receiver documentation.

2. Connect the Radian IS serial cable to "COM1" on the Radian IS, plugging the other end into the controlling computer's serial port.

3. If the Radian IS is to be used as a base, connect a PDL base radio to the "COM2" port of the receiver. If the IS is to be used as a rover, connect a PDL rover radio to the "COM2" port if the receiver.

4. Power the Radian IS on with its external power switch.

5. Once the receiver finishes its self-initialization (when all the lights on the side panel go out and then the battery light lights in just one position), it is ready for use with Carlson Field. However, positions will not be able to be logged until the receiver has acquired a few satellites. The receiver has enough satellites when the center light is at the second or higher level (when it is orange instead of red).

Software Setup

6. To configure the IS for use, select "Equipment Setup" from the Field pulldown menu. This will open a menu with several options:

a) Radio Baud Rate: This radio button sets the baud rate for COM2, the radio COM port. Make sure this number and the number the PDL's are set for is the same.

b) Station Type: This sets whether the Radian IS is to be configured as a base station or a rover.
c) Elevation Type: This allows selection of Geoid (MSL) or Ellipsoidal measures for height/altitude.
d) RTK Dynamics: This sets the dynamics mode of the receiver. In general, this setting should be set to "Dynamic/Kinematic".
e) Message Type: This sets what format of corrections this receiver will send/receive for RTK.
f) Motion Dynamics: This is used to set the receiver's calculations appropriate to the motion of the receiver.
g) Elevation Mask: This is the satellite elevation cutoff. No satellites with elevation less than this number will be used in corrections. This allows filtering out of satellites close to the horizon, which provide less accurate calculations for positions.
h) Send Command to Receiver: This allows a specific user-entered command to be sent to the receiver. Mostly used for troubleshooting with Technical support.
i) Configure Base: This configures the parameters of a base station for the receiver (Ex: Current position, etc.)
j) Power Cycle Receiver: This powers the receiver down and then turns it back on, clearing the main memory.
k) Save and Exit: This saves all settings changes and exit this menu.
l) Cancel: This restores original settings and exit this menu.

To set the Radian IS up as a Rover:
7. Select "Rover" for Station Type, and set the Radio Baud to match the PDL's which are being used. Also, set "RTK Dynamics" to "Dynamic/Kinematic", and set Motion Dynamics to the appropriate option.
8. Select "Exit and Save". The receiver is now ready for use as a rover.

To set the Radian IS up as a Base:
9. Select "Base" for station type, and set the Radio Baud to math the PDL's which are being used.
For most jobs, set RTK Dynamics to "Dynamic/Kinematic" (unless you are sure that static is more appropriate-even small fluctuations from wind on the pole can cause problems in Static mode). Set motion dynamics to Foot/Walking, and then select "Configure Base Station"
10. In the menu dialog that opens, there are a few buttons:
  a) Read from GPS: Read a position from the GPS and fix to that position
  b) Enter Lat/Lon: Fix to a manually entered Lat/Lon position
  c) Enter State Plane Coord: Fix to a manually entered State Plane Northing/Easting position
  d) Read From File: Fix to a position read from a *.ref file.
  e) Cancel: Cancel base setup

If Read From GPS is selected, the software will read once from the GPS receiver, and then fix to
that position. If Enter Lat/Lon is selected, a dialog box will open and a Latitude and Longitude must be input manually. If Enter State Plane Coord is selected, a dialog box will open allowing the input of a set of Northing/Easting coordinates by hand.

Read from File will open a File > Open dialog and ask for the file name of the file to open.

Regardless of which option is selected, after the position is determined, this position will be displayed, and dialog boxes will open to enter a station id and the measured base antenna height. Once these values are entered, base setup is complete and the "Exit and Save" button can be selected to exit the Equipment Setup menu.

**Sokkia 500 Series**

1. Turn on Sokkia
2. Turn it Horizontally and Vertically to set it.
3. Connect Sokkia to Carlson Field
4. In Field go to Configure Field, and under equipment type put Sokkia
5. To make sure the baud rate matches, under the Field menu go to Configure Field and click on Communication Settings and check the baud rate. On Sokkia press [ESC], then [CNFG]. Scroll down or enter 4 for "Comms setup." The baud rate can be changed using the arrow keys, when done press [ESC].

*Chapter 2. Field Module*
6. Exit the Configure Field menu.

7. To check if units (Ft /M) matches for correct results, in Carlson Field under Settings go to Drawing Setup and select the appropriate button. On Sokkia, in [CNFG] scroll to or enter 5 for "unit" and select appropriate unit using the arrow keys.

**Topcon Total Stations**

The Topcon instrument must have CR/LF (carriage return/linefeed) turned on for communication with Carlson Field.

**Topcon 200 Series**

To set CR/LF with 200 series:

1. Turn instrument off
2. Turn instrument on while holding F2 key
3. Choose F3 (Others set)
4. Press F4 (Page down)
5. Choose F3 (CR/LF) and set it on

To set this with 700 series:

1. Choose Parameter from the main screen
2. Scroll down until you find CR/LF and set it on

**Topcon ITS**

The command echo on the instrument must be turned off to work with Carlson Field.

**Topcon GTS-A4**

To setup the instrument hold down F-2 as you switch it on. This will bring up a parameters menu, press F-3 for Data Out. Hit Select to browse through the settings options, and make sure CR, LF: is ON and that Echo back: is OFF. Setup is complete.

**Topcon GTS-700**

To set the instrument to work with Carlson Field, press [F2] for "std" on the instrument.
Topcon 800-A Remote Setup

Topcon Setup:

Note: The instrument needs to be set to REC-A, not REC-B mode

1. Turn on the Topcon
2. Connect the Topcon to one of the radios, and the other radio connect to Carlson Field

3. Under Field menu go to Configure Field, and under equipment type select Topcon800A-Remote.

4. To set Topcon for external mode Press [F1] for "prog", then [F6] for "more". This will lead to more programs. Enter [F2] for "Ext.Link."

5. To select the radio channel, in External Link enter 2 for "settings" and 4 for "parameter (radio modem)", then 3 for "set channel". Using the arrow keys change the channel. When done press for [F1] for set, then press [ESC] until get back to External Link Menu.

   Note: Channel on the Topcon should match the channel set in Carlson Field.

6. After channel is set press 1 for "Execute"

7. Topcon is ready.

   Note: If the batteries are low either in Topcon or the radios, communication problems will arise.

Carlson Field Setup:

1. In Configure Field, under equipment type there should be Topcon800A-remote. In Communication Settings Baud Rate should be set to 9600.

2. After Configure Field go to Equipment Setup and make sure the radio channel or radio frequency matches the channel and frequency in Carlson Field. Press Ok when done.

**Topcon 800A Quick Lock**

1. Dismount the handle from the Topcon, and mount RC-2H. Secure it with the fixing screw.

2. Attach RC-2R to the prism, and turn it on.

3. Using the Y cable attach the RC-2H to the radio and Carlson Field.

4. In Joystick click on Quick Lock and Topcon will do angle turn until it finds a prism in which it will lock to, and will start tracking.

5. If RC-2H is not attached to the radio with Y cable, when Quick Lock is pressed the big yellow button on RC-2H needs to be pressed in order for the Topcon to search for the prism.
1. In order to properly configure the NT300D to work with Carlson Field, it must first be powered up in Setup mode (by holding down the [Setup] button on the front panel of the receiver while powering it on) so that the advanced setup options are available. Once the NT300D is powered up in this mode, bring up the Setup menu via the [Setup] button. Page down using the More menu option until the I/O menu item is available, and select it.

a) In the I/O menu, select whichever port is to be used to interface the receiver with the computer running Carlson Field (Port 1 by default). Next, set both the input and the output to transmit/receive in NMEA, at 9600 baud rate. The final option, Remote Select, should be set to Primary.

b) Now the NMEA sentences must be configured. From the I/O menu, enter the NMEA Sentences submenu. Disable all sentences, save for the GGA sentences and the GSA sentences. Ensure the Talker ID is GP. From here, Return to the I/O menu.

c) The NMEA Control menu item, reachable from the I/O menu, has three options. The Output Rate here should be set to 1 second, the Position Output Rate set to Output Rate, and the NMEA Output Version to 2.1.

Next, the GPS settings must be configured, and can be found in the GPS menu under the main Setup menu.

The GPS Mode should be set to 3D, and the DGPS mode set to Auto. The DGPS source should be
toggled to *Internal*, and the Pos/Vel Filter should be *Off*. Mask Values should be left at *Default*, and the SNR at *M*.

Finally, the Beacon Receiver configuration (under *Beacon Receiver* on the Setup Menu) needs to have its Search Mode set to *Auto-Dist* Mode. All other values in all menus ought to either be left at their default settings, or configured as necessary to the local conditions (in the case of antenna height, etc.).

2. The RMS value reported in Carlson Field is the RMS value of the standard deviation of the range inputs to the navigation process including pseudoranges and DGPS corrections.

The NT300D is now properly configured, and if connected to a computer running Carlson Field, will transmit position fix data to the computer automatically. Before using it, however, it is best to power it down and then turn it back on normally, as running it in Setup Mode is not recommended.

**Trimble 4000 Series**

**Hardware Setup**

1. Setup the antenna and GPS receiver as normal. The radio should be on I/O Port 2.
2. Connect the Computer that Carlson Field is running on to I/O Port 1 by the appropriate cable.

**Front-Panel Configuration:**

**Base Station:**

1. After powering on the receiver, press the [Control] Button. From the selections available, select **MORE**. This will bring up a second page of options. Select **MORE** again. The front panel screen should now be on **RECEIVER CONTROL** "3 of 7".
2. Select **BAUD RATE/FORMAT**, and from the menu that this creates, select **SERIAL PORT 1 SETTINGS**.
3. Ensure that the port is set to 38400 baud, 8-Odd-1 Format, with no flow control.
4. Similarly, make sure that the settings for I/O Port 2 agree with those of the type of radio being used (typically 9600 8-None-1).
5. Return to the **RECEIVER CONTROL** menu, and go to page 4 o 7. Select **REFERENCE POSITION**.
6. Enter the Lat/Lon of the position the base is located at. Alternately, select **HERE** to have the GPS unit read the current position and use that as the base reference point.
7. On page 1 of the **RECEIVER CONTROL** menu, select **RTK OUTPUT CONTROL**.
8. Set the **RTK OUTPUTS** to Port 2, and the **ANTENNA HEIGHT** to the measured height of the antenna.
9. Ensure that all other forms of output (Cycled Output, 1PPS output, Event Markers, etc.) are disable. These options may all be accessed with the submenus accessible through the [Control]
button.
10. Ensure that the Synch time of the Rover and Base are the same. This setting may be accessed by first pressing [Control] and then cycling through the menus until the MASKS/SYNCH TIME option is available.

Rover Station:

1. After powering on the receiver, press the [Control] Button. From the selections available, select MORE. This will bring up a second page of options. Select MORE again. The front panel screen should now be on RECEIVER CONTROL "3 of 7".
2. Select BAUD RATE/FORMAT, and from the menu that this creates, select SERIAL PORT 1 SETTINGS.
3. Ensure that the port is set to 38400 baud, 8-Odd-1 Format, with no flow control.
4. Similarly, make sure that the settings for I/O Port 2 agree with those of the type of radio being used (typically 9600 8-None-1).
5. Return to the RECEIVER CONTROL menus, and go to page 2.
6. Select RTK ROVER CONTROL.
7. Toggle the ENABLE setting to L1/L2.
8. Push the [Status] button, and select POSITION. There should now be an RTK option. Select it. This will bring up a screen displaying delta Northing/Easting, correction status, etc.
9. Ensure that the STATIC option appears at the right. This means you are in kinematic/rover mode. If instead the ROVE option is available, select it.
10. Ensure that all other forms of output (Cycled Output, 1PPS output, Event Markers, etc.) are disable. These options may all be accessed with the submenus accessible through the [Control] button.
11. Ensure that the Synch time of the Rover and Base are the same. This setting may be accessed by first pressing [Control] and then cycling through the menus until the MASKS/SYNCH TIME option is available.

**Trimble 4700/4800**

Hardware and Equipment:

1. Make sure that the computer's serial port is connected to the 4700/4800 in it's COM1 port (typically the port that a data collector is normally plugged into). Power should be supplied on COM2, and any radio used for RTK should be plugged into COM3.
2. All other equipment (antenna, wires, etc.) should be set up as normally directed by the manuals.

Software Configuration:

1. After selecting the Trimble 4700 equipment type from the "configure field" menu, open up "Equipment Setup." This should bring up a new window/dialog box with the following options:
   a. Receiver Type: Select whether you are using a 4700 or 4800 receiver.
b. Station Type: Choose what type of RTK station you are setting this receiver up as a base or rover.

c. RTK Correction type: Select the type of Corrections you would like a base station to transmit. Note that CMR messages should be used for most precision applications, as RTCM is only capable of producing less-accurate floating precision positions.

d. Radio Baud Rate: The baud rate of the radio port. This should be left at the default setting of 9600 in general.

e. Satellite Elevation Cutoff: All satellites with elevation from the horizon of less than this number will not be used in calculating a position. This allows less accurate low elevation satellite to be factored out of a position.

f. Configure Base Station: Will configure the receiver to act as a base. See "Configuring Base Station" below.

g. Cancel without saving: Will exit this menu without saving any changes that have been made.

h. Save and Exit: Will save these settings to the receiver and to Carlson Field's setup and exit out of this menu.

Configuring Rover:

No real configuration is necessary, aside from setting up the equipment and setting the appropriate Receiver Type, Station Type, and Satellite Elevation Cutoff.

Configuring Base Station:

1. After selecting all the appropriate settings in "Configure GPS," click on the "Configure Base Station" button.
2. In the menu dialog that opens, there are a few buttons:
   a) Read from GPS: Read a position from the GPS and fix to that position
   b) Enter Lat/Lon: Fix to a manually entered Lat/Lon position
   c) Enter State Plane Coord: Fix to a manually entered State Plane Northing/Easting position
   d) Read From File: Fix to a position read from a *.ref file.
   e) Cancel: Cancel base setup

If Read From GPS is selected, the software will read once from the GPS receiver, and then fix to that position. If Enter Lat/Lon is selected, a dialog box will open and a Latitude and Longitude must be input manually. If Enter State Plane Coord is selected, a dialog box will open allowing the input of a set of Northing/Easting coordinates by hand. Read from File will open a File > Open dialog and ask for a file name of a reference file (*.REF) to open for use in corrections.

Regardless of which option is selected, after the position is determined, this position will be displayed, and dialog boxes will open to enter a station id (used by the base to identify itself to
the rover(s)) and the measured base antenna height. Once these values are entered, base setup is complete and the "Exit and Save" button can be selected to exit the Equipment Setup menu. At this point, whenever looking at a menu that displays the connection status, "REFERENCE" will be displayed, instead of Float, Fixed, or Autonomous.

**Trimble 5800**

Carlson Field Configuration:

In Configure Field, set the Equipment Type to Trimble Generic.

In Equipment Setup, be sure to set the Data Type to match your receiver setup. This Data Type is the port on the receiver that communicates with Carlson Field. Typically, the Data Type should be set to 2 for the serial connection and to 4 for Bluetooth.

When configuring the Base receiver, use a base station id number in the range from 1 to 32.
Index

2D to 3D By Surface Model, 75
2D to 3D By Text With Leader, 80
2D to 3D Polyline by Points, 77
2D to 3D Polyline by Start-End Elevations, 83
2D to 3D Polyline-By Text, 78
3D Drive Simulation, 47, 48
3D Polyline, 268, 270, 272–274, 277
Adjust Overexcavate Surface, 25
Align GPS To Local Coordinates, 226
Apache Lightbar, 280
Areas Of Interest, 10, 41
ASE, 292–295, 297, 298
Assign Contour Elevation - From Contour Labels, 93
Assign Contour Elevation - Multiple in Series, 91
Assign Contour Elevation - Single Elevation Group, 94
Auto Points at Interval, 245, 283
Backfill, 161
Backsight, 199, 200, 239
Benchmark, 250
Boundary, 8, 9, 18, 114
Boundary Polyline, 8, 9, 21, 23, 162
Building Face Surface, 252
Calculate Total Volumes, 32, 39, 137
Calculate Volumes Inside Perimeter, 38
Carlson Field Icon Menu, 253, 254
Centerline, 58, 60, 74, 75, 256, 269
Centerline Position, 264–266
Change Elevations, 72
Change Layer, 73
Clear Overexcavate Surface, 27
Clear Strata Surface, 137
CL File, 60
COGO Menu, 183
Configuration, 289, 293, 297, 307, 308, 310
Contours, 30, 116, 117, 175, 176, 178
Convert LDD-AEC Contours, 65
Convert Spot Elev To Points, 88
Coordinate File, 195
Create Trench Network Structure, 143
CSI GBX Pro, 280, 281
Custom, 33, 41, 128, 135, 136
Cut/Fill Centroids, 60
Cut/Fill Color Map, 171, 172, 180
Cut/Fill Contours, 169
Cut/Fill Labels, 170
Cut/Fill Map Legend, 63
Cutsheet, 184, 195
Cutsheet Spreadsheet Editor, 184
Define Layer Target/Material/Subgrade, 2, 7, 12, 20, 29, 30, 39, 43, 106, 107, 164, 167, 173
Define Materials, 6, 42
Depth Contours, 138, 139
Depth Sounder, 281, 282
Design Contours, 167, 168
Design Drawing, 166, 167
Design Surface, 168, 169, 180
Design Surface 3D Viewer, 51
Design Surface Vertical Offset, 31
Digitize 2D Polyline, 110
Digitize 3D Polyline, 112
Digitize Areas, 114
Digitize Contour Polyline, 115
Digitize Design, 107
Digitize End Areas, 121
Digitize Existing, 106
Digitize Menu, 99
Digitize Other, 107
Digitize Perimeter, 113
Digitize Point, 107, 108
Digitizer Settings, 104, 106
Digitizer Setup, 101
Digitize Sections, 118
Digitize Spot Elevation, 108
Display Menu, 164
Display Options, 173
Dozer, 66
Drape 3D Polyline On Surface, 95
Draw Building Envelope Polyline, 83
Drawing Setup, 32, 299, 303
Draw Overexcavate Cut Color Map, 27
Draw Overexcavate Surface 3D Faces, 26
Draw Profile, 149, 153
Draw Strata Cut Color Map, 139, 140
Draw Strata Cut Depth Contours, 137, 138
Draw Strata Surface, 140, 141, 152
Draw Subgrade Hatch Legend, 14
Draw Trench Network - Plan, 151
Draw Trench Network - Profile, 152
Draw Typical Trench Template, 158
Drillhole Import, 127
Drillhole Menu, 124
Drillhole Reports, 135
Drillhole Strata Settings, 124
Duplicate Strata, 130

Earth Curvature, 196
Edit-Assign Polyline Elevations, 74
Edit Contours, 97
Edit Drillhole, 132
Edit Polyline Vertex, 96
Edit Selected Layer, 7
Edit Trench Network Structure, 146
Elevate Menu, 72
Elevation Difference, 198, 256–258
Equipment Menu, 280
Equipment Setup, 199, 203, 205, 213, 219,
225, 253, 270, 284, 286, 288, 289,
292, 297, 298, 300, 302, 305, 310
Erase Overexcavate Cut Color Map, 27
Erase Overexcavate Surface 3D Faces, 26
Erase Strata Cut Color Map, 139
Erase Strata Cut Depth Contours, 138
Erase Strata Surface, 140
Erase Subgrade Hatches, 14
Erase Subgrade Labels, 15
Existing Contours, 164, 165
Existing Drawing, 164
Existing Surface, 165, 166
Existing Surface 3D Viewer, 50
Existing Surface Vertical Offset, 31
Export Polyline File, 66
Export Topcon TIN File, 68
Export Trench Network Data, 148
Extrapolate, 206

Field Menu, 185
Field Module, 182
Field to Finish, 193, 194
Find Trench Network Structure, 148
FlyOver Along 3D Polyline, 53

Geodimeter, 200, 201, 283, 284
GIS File, 198
GIS Menu, 279
Grid File Utilities, 66
Hatch Subgrade Areas, 12, 14
Import/Export Carlson Triangulation Files, 67
InnerSpace Tech depth sounder, 285
Input-Edit Trench Template, 156, 158
Input Trench From Polyline, 141
Intersections, 21
Join, 62, 63, 94
Label Contours, 177
Label Elevations, 194
Label Subgrade Areas, 15
Laser Atlanta, 285
Latitude and Longitude, 217
Leica, 201, 207, 209, 253, 286–289
Leica Disto, 286
Leica GPS System 500, 286
Leica TC Series, 288
Linear, 4
List Points, 195
Load Tablet Calibration, 104
Locate on Real Z Axis, 194
Make Design Surface, 23, 30, 143, 145–147, 156, 162
Make Existing Ground Surface, 20, 21, 30
Make Overexcave Surface From Existing/Design Surfaces, 25
Make Overexcave Surface From Screen Entities, 24
Make Overexcave Surface From Strata, 24
Make Strata Surface, 137–140
Make User Defined Surface, 29
Manhole, 142, 143, 154, 162
Manual Total Station, 289
Material Quantities, 39
Merge Existing With Design, 31
MicroStation, 66
Mikrofyn Lightbar, 290
Monitor GPS Position, 249, 282, 283
National Geodetic Survey, 190
Navcom Configuration Guide, 291
Navcom GPS Setup, 293
Nikon, 298, 299
Nikon Total Stations, 298
Note File, 195, 198
Offset Stakeout, 267
OmniStar Otto, 299
Other Drawing, 172–174
Pad Polyline By Interior Text, 85
Pattern Point Survey, 253
Perimeter Polylines Properties, 64
Pipe Size, 144
Place Drillhole, 127, 130, 137
Plain View Label Settings, 149
Point Check By Robotics, 253
Point Description, 108, 198, 259
Point Notes, 192, 193, 260
Point Store, 231, 232, 235–237, 245, 247, 254, 258, 266
Polyline File, 67
Prepare Story Stake, 259, 260
PRO, 60, 267, 291
Property lines, 71
Quick Profile, 30, 58
Reduce Polyline Vertices, 67
Reduce Vertices, 176, 177
Remove Trench Network Structure, 147
repackaged, vii
Report Trench Network, 162
Resection, 251
Roads Menu, 264
Satellite SkyPlot, 248
Save Tablet Calibration, 103
Section File, 119, 267, 268, 272
Set Active Surfaces, 30
Set Layer For Design, 7
Set Layer For Existing, 7
Set Layer For Other, 8
Set Polyline to Elevation, 73
Sewer, 141, 153
Sewer Network, 152
Sewer Structure, 143, 145–147, 157
Simulation GPS, 299
Sketch, 115–117
Slope Inspector, 278
Slope Staking, 272, 274, 277
Smooth Contours, 175
Smoothing, 65, 176
Smooth Polyline, 234
Snap Tolerance, 85
Sokkia, 300, 302, 303
Sokkia G, 211, 212
StakeOut, 240
Standard deviation, 120, 122
State Plane, 189, 191, 196, 214, 215, 219,
    229, 230, 287, 301, 302, 309
Station-Offset, 256
Story Stake Along Polyline, 262
Story Stake By Points, 261
Subgrades, 6, 118, 159
Superelevation, 272
Surface Inspector, 30, 56
Surface Manager, 28
Surface Menu, 256
Surface Model, 55, 56
Surface Report, 56, 57
SurvStar, 227
Symbol Size, 23, 125

Tablet Calibrate, 99
Tablet Calibration, 100, 101
Tablet Off, 99, 104, 113
Tablet On, 99, 104
TakeOff Menu, 2
Takeoff Module, 1, 65
Tape Baseline, 183
Technical Support, 106, 293, 298
Template Database, 199
Text Explode To Polylines, 66
Text Size, 178, 180
Tools Menu, 47
Topcon Total Stations, 303
Topsoil Removal, 18, 19, 57, 58
Topsoil Removal/Replacement, 20, 23, 64
Topsoil Removal and Replacement, 16
Track Position, 247, 299
Trench Menu, 141
Trench Network File Backup, 149
Trench Network Quantities, 160
Trench Subgrade Areas, 159
Triangulation, 67, 69
Trim, 85
Trimble, 203, 210, 212, 213, 294, 306–308,
    310
Twist, 187
Typical Alignment Scenarios, 255
Update Colors For Set Elevations, 64
View, 32, 33, 49, 59, 227, 296
Volume, 8, 30, 32, 36, 160

Zone Polylines, 67

Index 314