

Introduction

The DTM-Contour module calculates surface models as either 3D grid mesh or triangulation networks. Contour maps, volumes and other applications use these surface models. Any point, insert, line, or polyline with elevation may be used to define the surface.

Understanding the concept of triangulation is helpful in working with the DTM-Contour module since nearly every routine in this module references triangulation. Even 3D grid mesh uses triangulation internally.

Triangulation creates a triangular network from points where points are connected by edges with each edge forming a segment of one triangle or a common edge of two triangles (see Figure 5-1). The edges are used to interpolate elevations between the points. Generally, points are connected to their nearest neighbors because points that are close together are more likely to be related. Sometimes making the nearest connection is not what you want. For instance, there might be two points that define a segment of a road, wall, or ridge. Even though these points might be far apart, you want them connected in the triangulation because you want to interpolate along this segment and you don't want any edge crossing this path. This can be achieved by drawing a line or 3D polyline between the points. All the vertices in lines and polylines are included as points in the triangulation, and all the segments become edges in the triangulation. In this way, lines and polylines are treated as barrier lines. In fact, 3D polylines can effectively used to create surface models. Carlson supplies several routines for creating and modifying 3D polylines in the DTM drop-down.

As another example, consider two points high up on opposite sides of a valley. If these two points are connected by triangulation, their high elevations would be interpolated across the valley creating a virtual dam in the valley where none really exists. This incorrect triangulation can be prevented by creating a barrier line through the bottom of the valley and between the high points on the opposite sides.

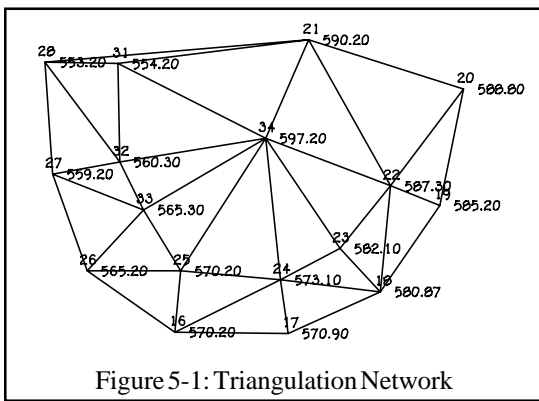


Figure 5-1: Triangulation Network

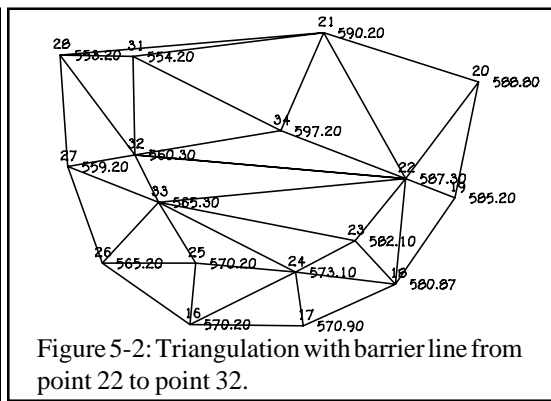


Figure 5-2: Triangulation with barrier line from point 22 to point 32.

3D Polyline Commands

3D Polylines are used in surface modeling to represent barrier or break lines which force interpolation along the segments of the polyline. For example a road centerline could be represented as a 3D polyline with a 2 percent slope. The surface model would interpolate along this polyline and hold the 2 percent slope. The Offset, Trim, Extend and Join commands are described in the Modify Commands section of the manual.

3Dpoly	DTM	Contour	Site	Misc
Draw 3D Polyline				
Draw 3D Polyline with Follow				
Slope Line (3D polyline)				
2D to 3D Polyline by Surface Model				
2D to 3D Polyline by Points				
3D Polyline by Slope on Surface				
Mesh from 3D Polyline				
Highlight Crossing Barriers				
Report 3D Polyline Station-Elev				
Offset 3D Polyline				
Fillet 3D Polyline				
Trim 3D Polyline				
Extend 3D Polyline				
Join 3D Polyline				
Break 3D Polyline				
Break 3D Polyline by Surface				
Tag Hard Barrier Polylines				
Untag Hard Barrier Polylines				

Draw 3D Polyline

Function

This command sets a user specified layer and plots a 3DPOLY which can be used to define barrier lines and features that better define surface models. You can pick points from the graphics screen or use point numbers from the current Coordinate File (.CRD file). Ranges of point numbers can also be specified. For example entering "3-1,5,10" would draw 3D Line segments from point number 3 to 2 to 1 to 5 to 10.

There is an option to use a surface model that is stored in a file which can be either a grid file (.grd) or triangulation file (.flt). When a point is picked or a point number is entered, the program uses the northing and easting of this point and calculates the elevation from the surface model.

The *Prompt for elevations (.XY filter)* option determines whether the elevation of the specified point is used automatically. With the XY filter active, the user is asked to enter the elevation either by directly typing the elevation or by specifying the percent slope or slope ratio from the previous point.

Warning: When creating a 3D polyline by picking points from the screen, be careful not to miss a point and get a zero elevation. This would create a 3D polyline that drops to zero at one point. As points are picked, Draw 3D Polyline reports the coordinate so that you can check that the

correct elevation was used. If you miss a point and see a zero elevation, then you can type Undo at the next prompt.

Prompts

Layer Name for 3DPoly <BARRIER>: *[Enter]*

Prompt for elevations (.XY filter) (Yes/<No>)? *Y*

Using the .XY filter allows the user to pick the X and Y coordinate from the screen and type in the elevation. If you use the N response then the z coordinate of the point picked will be applied.

Use surface model from file (Yes/<No>)? When a point is picked, this option calculates the elevation of the picked point using a surface model. The surface model can be either a grid file (.grd) or triangulation file (.flt).

Pick point or point numbers: *(pick point)*

Percent slope/Ratio slope/Elevation <99.15>: *99.5*

Undo/Pick point or point numbers: *15* This is a point number from the current coordinate file.

Percent slope/Ratio slope/Elevation <98.75>: *P* For Percent slope.

Ratio slope/Elevation/Percent slope <-2.53>: *-2* Sets the elevation of this point to make a 2 percent slope from the previous point.

Close/Undo/Pick point or point numbers: *Press Enter*

Pressing [Enter] ends the command.

Keyboard Command: 3dpline

Pull-Down Menu Location: 3DPoly

Prerequisite: None

File Name: \lsp\3dpline.lsp

3D Polyline with Follow

Function

In addition to allowing you to draw a polyline in the same way as the AutoCad pline command, this command can also add sections of existing polylines into the new polyline. To include part of an existing polyline, enter *S* for the Select existing polyline option. Then choose the existing polyline and pick an enter point and an exit point on the polyline. The new polyline will "follow" the existing polyline between these two points. If the existing polyline is closed, then there are two possible ways to go between the enter and exit points. By default the points are connected in the direction of the polyline. The command will highlight the default way. To connect in the other direction, enter *No* to the *Is this correct* question. There is also a 2D Polyline with Follow command in the Draw menu.

Prompts

Enter polyline layer name <0>: *Press Enter*
Select existing polyline/End/Pick point: *Pick a point*
Select existing polyline/Undo/End/Close/Pick point: *S (For Select existing polyline)*
Select the polyline to follow: *Pick a polyline*
Pick entrance point on polyline: *Pick a point on the polyline*
Pick exit point on polyline: *Pick a point on the polyline*
Select existing polyline/Undo/End/Close/Pick point: *E (To End)*

Pull-Down Menu Location: 3DPoly

Keyboard Command: follow3d

Prerequisite: None.

File Names: \lsp\follow & \lsp\poly3d.arx

Slope Line

Function

This command draws *3DPOLY* lines by specifying a slope ratio and a distance or elevation to plot to. This command can be useful in defining pads, excavations, drainage ditches and slopes from proposed design features to meet existing site conditions. For example, if you have plotted a 3dpoly defining a parking lot with the endpoints representing the proposed elevations. Now use the *Slope Line* command to define the slopes that run off the proposed to meet the existing ground (You could also use this command to plot the parking lot). Slope lines (3D polylines) can be used in conjunction with *Triangular & Contour* (as barrier lines), *Make 3D GRiD File* and the *Sections from Surface Model* commands to help define a surface model for contouring and/or volumetrics.

Prompts

Slope ratio + for uphill - for downhill, Start point should be 3D.

Slope Ratio (? : 1) <2.0>: 2

Start Point: *(pick point)*

Direction Point: *(pick point)*

Enter Hz Distance, [P]lot to Direction Point, or to [E]levation <P>: E

At this prompt you can enter a distance. Enter P to have the slope line plotted to the direction point you selected or as in this example, enter E and be prompted for the elevation to end the line at.

Elevation to plot to: 100

Horizontal Dist: 50.00

Start Point Elevation: 0.0

End Point Elevation: 100.0

Difference in elevation: 100.0

The command then plots the specified 3DPoly line.

2D to 3D Polyline 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 barrier lines. For example, a ridge barrier line 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 barrier line.

In addition to using entities in the drawing, the 2D polyline can be converted to 3D using a surface model stored in a grid (.grd) file or triangulation (.flt) file. A grid file is created with the Make Grid File command and a triangulation file is created with the Triangulation File option in Triangulate & Contour.

Prompts

Source of Surface Model (File/<Screen>)? *Press Enter*

Select surface line and polylines.

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

Reading points ... 1200

Select polylines to convert.

Select objects: *pick one or more polylines*

Pull-Down Menu Location: 3DPoly

Keyboard Command: 2dto3dp

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

File Names: \lsp\2dto3dp.lsp & \lsp\makegrid.arx

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 points or nodes (AutoCad POINT entities). This routine can be useful if the linework is created in 2D at zero elevation with commands like Design Polyline, 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 barrier line in surface modeling, this command can convert the polyline into a 3D polyline.

Prompts

Select points and polylines.

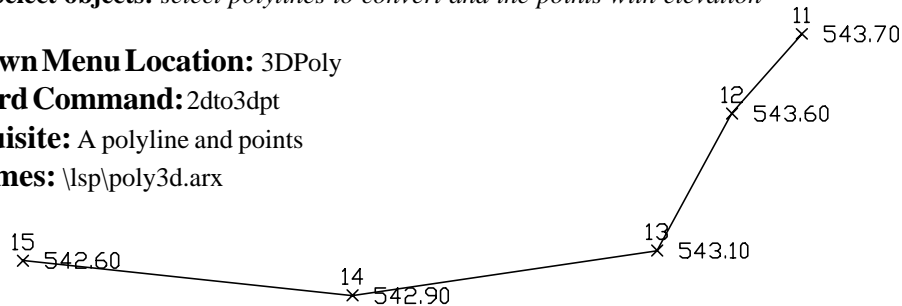
Select objects: *select polylines to convert and the points with elevation*

Pull-Down Menu Location: 3DPoly

Keyboard Command: 2dto3dpt

Prerequisite: A polyline and points

File Names: \lsp\poly3d.arx



3D Polyline by Slope on Surface

Function

This command creates a 3D polyline at a user-specified slope. The user picks the starting point and then the polyline continues along the surface at the slope until it reaches a point where the maximum slope at the point is less than the design slope. The surface is defined by a grid file (.GRD) which must be created before calling this routine. Applications for this command include designing haul roads or ditches.

Prompts

Enter the polyline layer <SLOPE_ROAD>: *Press Enter*

Select the Grid File dialog

Reading row > 51

Extrapolate grid to full grid size (Yes/<No>)? *Y* (For yes)

Pick origin point of 3D polyline: *Pick a starting point*

Direction of 3D Polyline (<Up>/<Down>)? *Press Enter*. The slope must go either uphill or downhill.

Direction of 3D Polyline facing up slope (<Left>/<Right>)? *R* (For right). Imagine facing uphill. Do you want the polyline to go to the left or right?

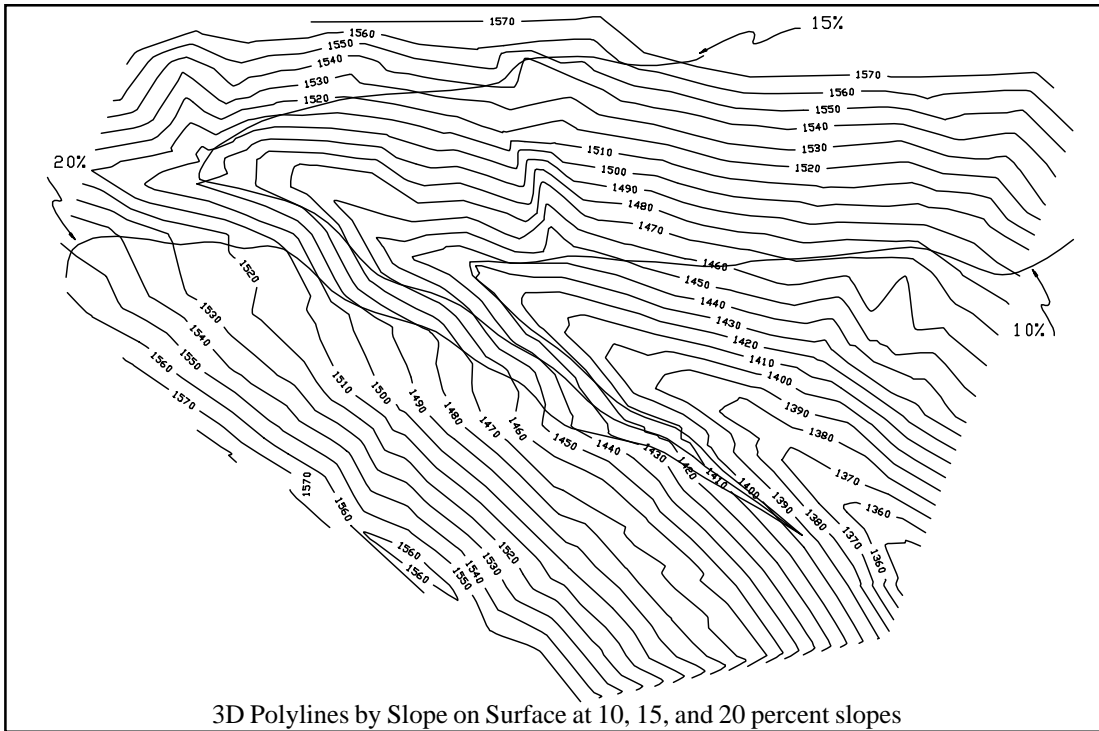
Enter the design slope: *10* (This value is in percent slope)

Pull-Down Menu Location: 3DPoly

Keyboard Command: surfpl

Prerequisite: None.

File Names: \lsp\slope_rd.lsp & \lsp\cntr_grd.arx



Mesh from 3D Polyline

Function

This command plots a mesh defined by a series of polylines. This mesh can be used to visualize a design feature such as a road or site. The polylines used for selection could be generated by commands like *Sections to 3D Polylines*.

Prompts

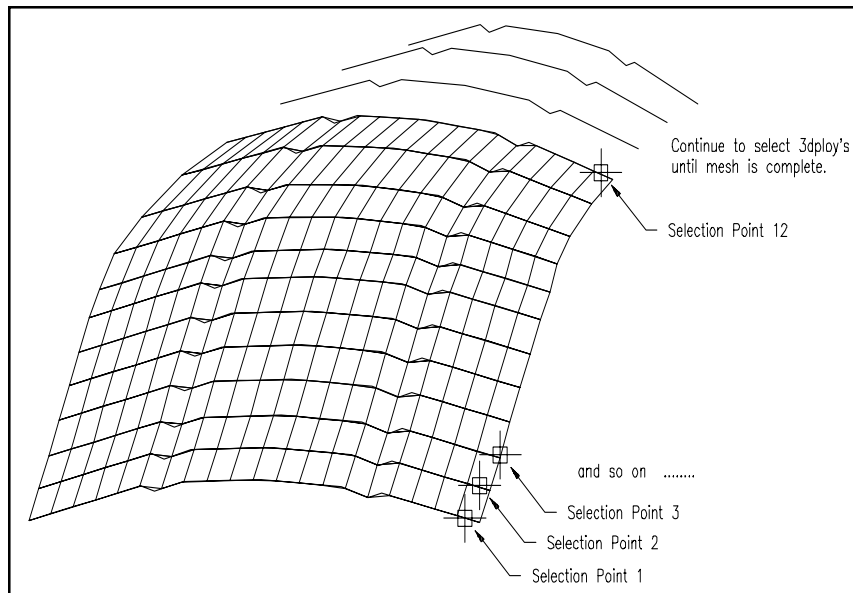
Layer Name for Mesh <MESH>: *ROADMESH*
Surface Resolution (surftab1) <40>: *20*
Select polyline ([Enter] to End): *(select polyline)*
Select polyline ([Enter] to End): *(select polyline)*
Select polyline ([Enter] to End): *[Enter]*
 Pressing [Enter] ends the command.

Keyboard Command: meshf3dp

Pull-Down Menu Location: 3DPoly

Prerequisite: Plot the *3DPoly* lines to use for selection.

File Name: \lsp\meshf3dp.lsp



Break 3D Polyline by Surface

Function

This command breaks 3D polylines against a surface. The surface is defined by a 3D grid surface model which can be selected from a .grd file or the user can select 3D entities on the screen from which program internally calculates a 3D grid. This routine is one step in 3D polyline design. In this example, a valley fill is designed using 3D polylines as follows:

1. Draw 3D Polyline - draw a 3D polyline at elevation 1450 across valley
2. Offset 3D Polyline - offset the 3D polyline at 2:1 slope and 10:1 for benches and top
3. Break 3D Polyline by Surface - trims the 3D polylines by the valley sides
4. Draw 3D Polyline - with OSNAP endpoint, draw 3D polyline perimeter around 3D polylines ends
5. Make 3D Grid File - create 3D grid file of fill surface
6. Plot 3D Grid - draw the 3D grid using 3D polyline perimeter as inclusion polyline
7. Viewpoint 3D - creates the last figure view

Prompts

Source of surface model (File/<Screen>)? *Press Enter.* The File option allows you to choose the .grd file that models the site. Otherwise a grid will be calculated by picking the grid location and selecting surface entities on screen (ie contour polylines). Using the File option can be quicker because the surface is already calculated. Also the .grd file can be drawn to preview the existing

surface. Use Make 3D Grid File to create a .grd file.

Pick Lower Left limit of surface area: *pick a point*

Pick Upper Right limit of surface area: *pick a point*

Make GRiD Setting Dialog Click OK. See Figure 5-5.

Select polylines to clip.

Select objects: *pick the 3D polylines*

Select surface entities.

Select objects: *select objects that define the surface.*

Erase polyline below surface (<Yes>/<No>)? *Press Enter.* If you answer yes, the segments of the polylines below the surface will be erased from the intersection, if any, of the polyline with the surface. Otherwise the polylines will only be broken into separate polylines at the intersection.

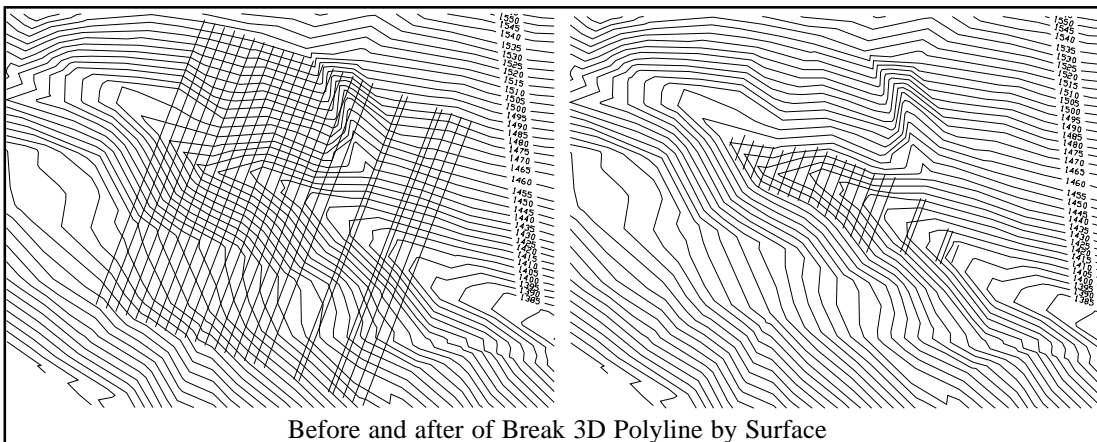
Specify layer names (Yes/<No>)? *Press Enter.* If you answer yes, you will have the option to place the broken polylines into different layers.

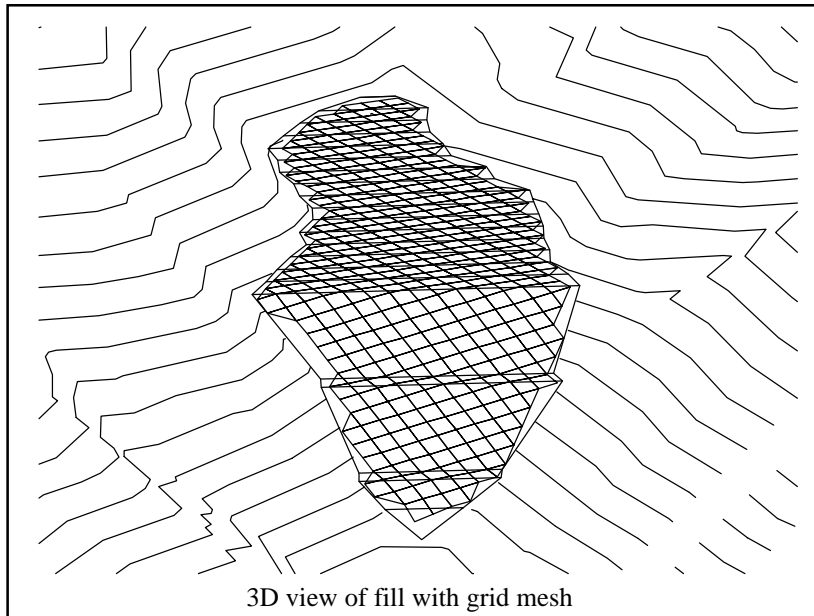
Pull-Down Menu Location: 3DPoly

Prerequisite: Plot the 3D Polylines to use for selection and entities that define a surface.

Keyboard Command: surfbreak

File Name: \lsp\flatpond.lsp & \lsp\pond.arx





Highlight Crossing Barriers

Function

Barriers are lines or polylines that are used in surface modeling to represent a break line such as a ridge, stream or curb. Barrier lines force triangulation between the two endpoints of the barrier line which prevents any other triangulation line from crossing the barrier line. This forces interpolation along the barrier line. It is important to avoid crossing barrier line in the surface model because the program cannot force triangulation along both since holding one will cross the other. Also the program cannot hold the interpolation along both because the elevation at the intersection point can be different for the two barriers. For example, consider a barrier line going from 101 to 105 and another going from 107 to 109. If these lines had an intersection at the midpoint, the elevation for one would be 103 and the other 108. This is an error that needs to be fixed by the user because those two barriers are holding different elevations at the same point.

This command checks for intersections between the selected barriers and then identifies any crossing barriers by highlighting them. You can then edit these crossing barriers before doing surface modeling such as Make 3D Grid or Triangulate & Contour.

Prompts

Ignore zero elevations (<Yes>/No)? *Press Enter*

Select surface entities to check.

Select objects: *select polylines and lines*

Finding points on barriers.

2 crossing barriers are highlighted

OR

Found no crossing barriers

Pull-Down Menu Location: 3DPoly

Keyboard Command: xbar

Prerequisite: Polylines.

File Names: \lsp\makegrid.arx

Report 3D Polyline Station/Elevation

Function

This command reports station or elevation information on a 3Dpolyline. If the station information is entered the program will return the elevation at the entered station. If the elevations are entered the program will determine and report the station at which the entered elevation occurred. If the elevation entered occurs at more than one location along the 3Dpolyline, all occurrences of the elevation are reported.

A prompt is provided allowing you to designate a starting station, or accept the default value of <0.0>. All entry and values are recorded and are displayed in the Carlson Standard Report Viewer upon completion.

Prompts

Select 3D polyline to report: *pick a polyline*

Starting Station <0.0>: *Press Enter*

Enter elevation or station (<Elevation>/Station)? *S*

Enter Station to calculate elevation: *100*

Station: 1+00.000 **Elevation:** 1052.262

Enter Station to calculate elevation (Enter to end): *Press Enter*

Select 3D polyline to report: *pick a polyline*

Starting Station <0.0>: *Press Enter*

Enter elevation or station (<Elevation>/Station)? *E*

Enter Elevation to find stations: *1210*

Station: 3+27.509 **Elevation:** 1210.000

Station: 3+56.246 **Elevation:** 1210.000

Station: 5+21.339 **Elevation:** 1210.000

Enter Elevation to find stations (Enter to end): *Press Enter*

Pulldown Menu Location: 3Dpoly

Keyboard Command: plreport3

Prerequisite: A 3D Polyline

File Names: \lsp\poly3d.arx

Tag Hard Barrier Polylines

Function

This command tags polylines with a description so that Triangulate & Contour can identify these polylines as hard barrier lines. The tag is invisible and doesn't change the polyline. Triangulate & Contour will not smooth the contours as they cross these hard barrier lines. For example you could tag 3D polylines that represent a wall so that the contours go straight across the wall without smoothing curves.

Prompts

Select hard barrier lines.

Select objects: *select polylines*

Pull-Down Menu Location: 3DPoly

Keyboard Command: hardbrk

Prerequisite: Polylines.

File Names: \lsp\hardbrk.lsp

Untag Hard Barrier Polylines

Function

This command removes description tags from polylines. These tags are used by Triangulate & Contour to identify polylines as hard barrier lines. Contours are not smoothed as they cross these hard barrier lines. This routine untags polylines so that contours are smoothed across them.

Prompts

Select polylines to remove hard barrier tag from.

Select objects: *select polylines*

Pull-Down Menu Location: 3DPoly

Keyboard Command: softbrk

Prerequisite: Polylines with hard barrier tag.

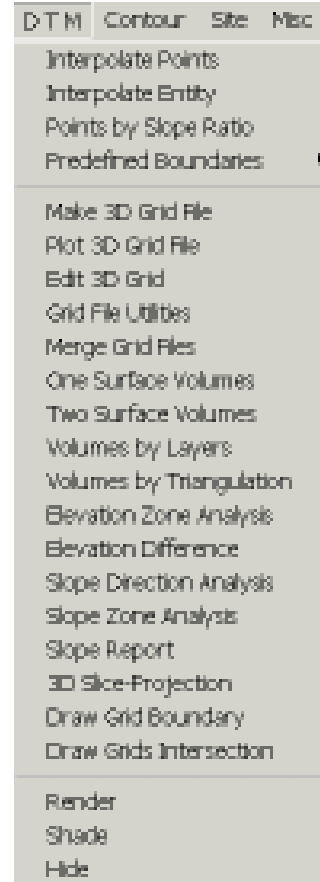
File Names: \lsp\softbrk.lsp

DTM Commands

In Carlson, a DTM (Digital Terrain Model) is represented by a grid where the grid corners are given elevations. This grid is equivalent to a 3D rectangular mesh.

Tag Predefined Boundaries
Untag Predefined Boundaries

The D.T.M. pull-down menu is shown below with its two cascading sub-menus. The top section contains commands for placing points and 3D polylines that define the surface. The second section includes Make 3D GRiD File which is the central routine in the DTM commands. It calculates a surface model from 3D entities and stores the model as a 3D grid in a .GRD file. Below the Make 3D GRiD File routine are commands that apply the .GRD files. The final section has AutoCad viewing commands that may be useful after drawing a DTM.



Interpolate Points

Function

This command divides the distance between two points and inserts one of the point symbols at the specified distances. It can also interpolate elevations (To interpolate elevations the points picked must be at their real z axis elevation).

Prompts

Interpolate Elevations <Y>: *[Enter]*

Selected points must be 3D points (must be at an acad elevation)

Point w/elevation to calculate from?

Pick point or point number: *3*

2nd Point w/elevation?

Pick point or point number: *(pick point)*

Number of Segments/Divisions: *5*

The command then locates 4 points.

Pull-Down Menu Location: DTM

Prerequisite: Execute *Drawing Setup* to set defaults. Locate two points to divide between and if you want to interpolate elevation they should have a real Z axis elevation.

Keyboard Command: divlin

File Name: \lsp\divlin.lsp

Interpolate Entity

Function

This command divides the distance of a *LINE*, *ARC* or *PolyLINE* and locates points at the computed distances. It also interpolates elevations (To interpolate elevations the points picked must have an AutoCad or real Z/elevation). The figure below shows a graphical example.

Prompts

Interpolate Elevations <Y>: [Enter]

Point w/elevation to calculate from?

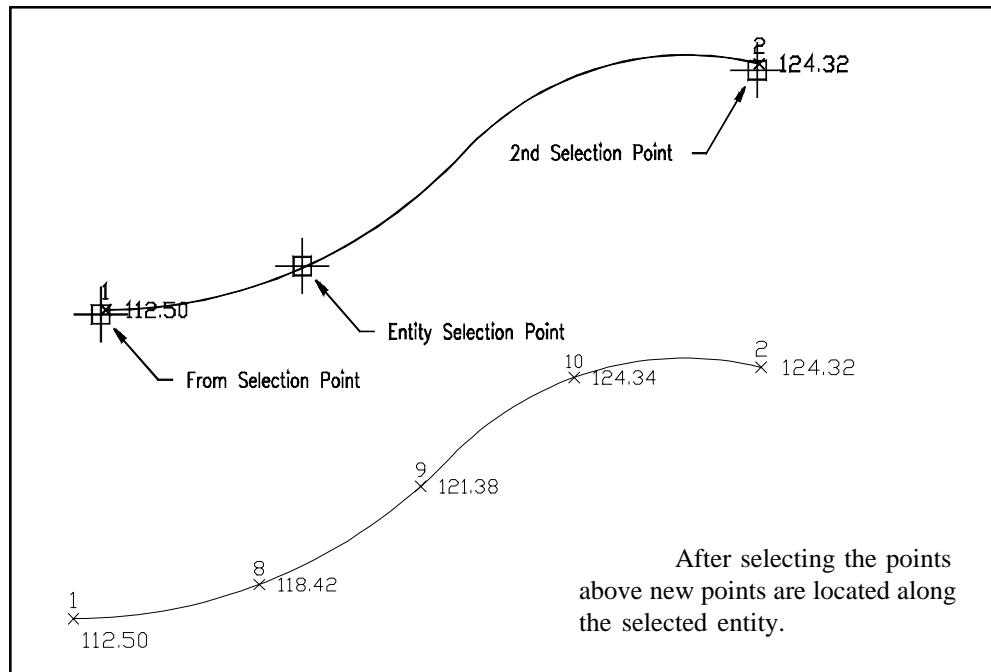
Pick point or point number: 1

2nd Point w/elevation?

Pick point or point number: 2

These points don't have to be on the entity selected to divide.

Select Entity to Divide: (pick point on entity)



Number of Segments/Divisions: 4

The command then locates 3 points along the selected entity.

Keyboard Command: divent

Pull-Down Menu Location: DTM

Prerequisite: Two elevation points and the entity to divide between.

File Name: \lsp\divent.lsp

Tag Predefined Boundaries

Function

This command allows you to identify closed polylines to be used as inclusion or exclusion boundaries. These boundaries are applied in DTM commands such as Two Surface Volumes and Triangulate & Contour. Inclusion polylines limit processing to inside the polyline. For example an inclusion polyline for volumes would be the limit of disturbed area. Exclusion polylines prevent processing inside the polyline. For example a building perimeter could be an exclusion polyline for contouring. Tag Predefined Boundaries assigns a site name to polylines and whether the polyline is for inclusion or exclusion.

Many DTM commands will prompt for inclusion and exclusion polylines. The advantage to Predefined Boundaries is that you don't have to select the boundary polylines each time that you run the DTM command. Instead the program will recognize that the boundary is already specified and will prompt *Use predefined boundary Area 1 (<Yes>/No)?* This lets you just press enter to use your predefined boundary. If you want to pick the boundaries, you can type *N* for No. When you have more than one set of boundaries, the program lets you choose from a list of the boundary names (as shown in dialog).

Prompts

Boundary name <Site 1>: *Area 1*

Select Inclusion perimeter polylines.

Select objects: *pick the closed polylines or press Enter for none*

Select Exclusion perimeter polylines.

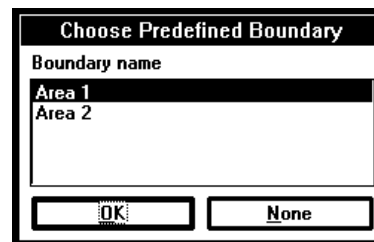
Select objects: *pick the closed polylines or press Enter for none*

Keyboard Command: plzone

Pull-Down Menu Location: DTM

Prerequisite: Closed polyline

File Name: \lsp\plzone.lsp



Untag Predefined Boundaries

Function

This command removes the predefined boundary names from the selected polylines. These polylines will no longer be automatically recognized as boundary polylines.

Prompts

Select polylines to remove boundary tag from.

Select objects: *pick the boundary polylines*

Keyboard Command: nozone

Pull-Down Menu Location: DTM

Prerequisite: predefined boundary polylines

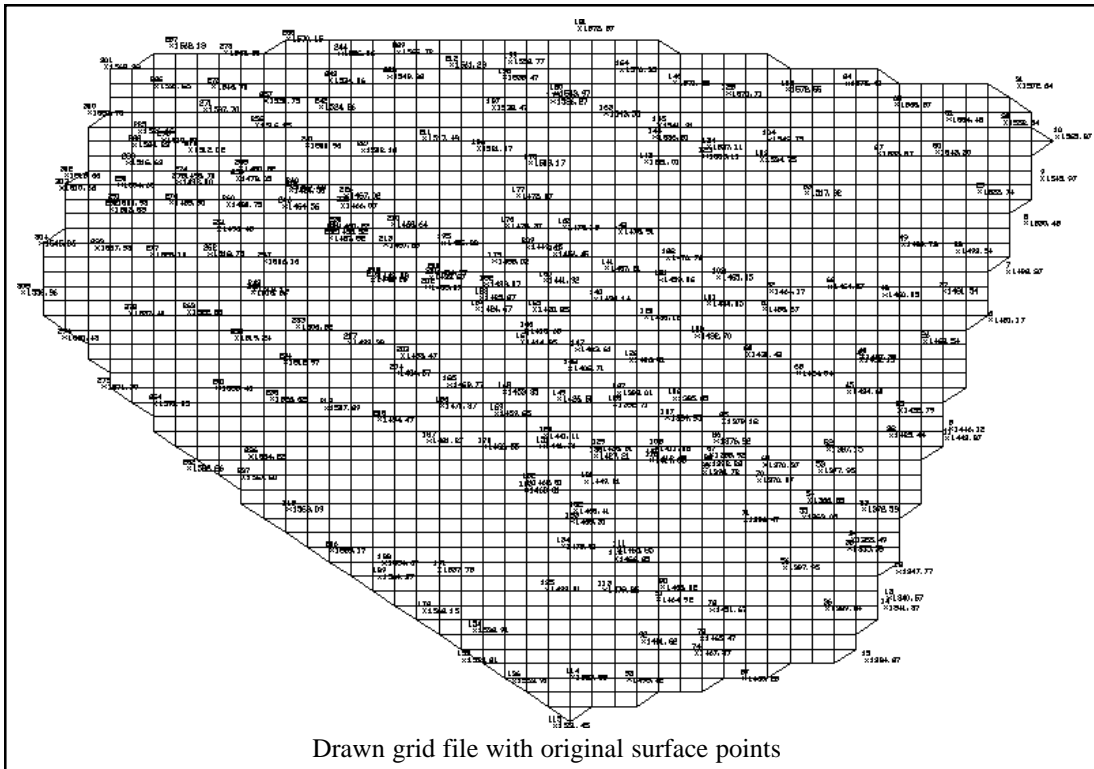
File Name: \lsp\dtm1.lsp

Make 3D Grid File

Function

This command creates a .GRD file which serves as a surface model and is a prerequisite to most of the other DTM routines. The program internally makes a triangular network of the data points and then interpolates the elevation values of a rectangular grid at the specified grid resolution. Data points can be either points, inserts, lines, or polylines. Lines and polylines are treated as barrier lines in the triangulation.

The grid location is specified by first picking a lower left corner and then an upper right corner. The screen cannot be twisted when this is done because grids always run north-south and east-west. Then a dialog box sets the range of elevations to process, modeling method and grid resolution. Entities with elevations outside the range to process are ignored. The modeling method almost always should be triangulation. Polynomial, inverse distance, kriging and linear least squares apply to random data points for surfaces like underground features. The grid resolution is specified by either the number of grid cells or by the size for each grid cell. While the program can handle really huge grids, a general rule of thumb is to keep the total number of grids cells under 250,000 (500 by 500). The grid location and resolution can also be specified by using the settings from an existing grid file. In this case, the location and resolution of the new grid will match those of the selected grid file which is useful for routines such as Elevation Difference and Two Surface Volumes that require two grid files with identical locations and resolutions.

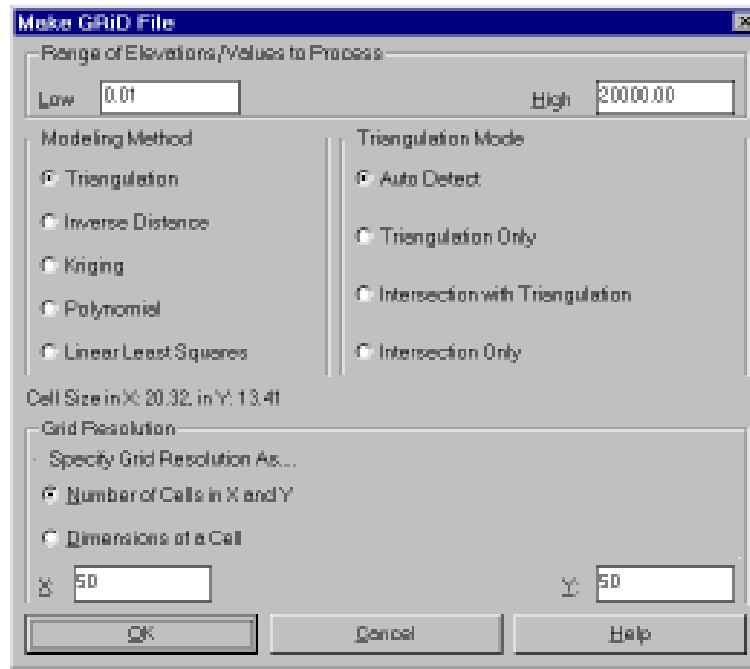


There are four triangulation modes: AutoDetect, Triangulation Only, Intersection with Triangulation and Intersection Only. The Triangulation Only method builds a triangulation surface out of all the selected points, lines and polylines. All lines and polylines are treated as barrier lines. Grid node elevations are calculated based on the triangulation.

The Intersection with Triangulation will triangulate all the selected points and the only end points of the lines and polylines. Then a "Steepest Intersection" method is used to assign the grid node elevations from the linework of the triangulation lines and the selected lines and polylines. This "Steepest Intersection" method is described in the Grid from Contour Maps section.

The Auto Detect method automatically chooses between the Triangulation Only and Intersection with Triangulation methods. If the selected surface entities are primarily made of polylines, then the Intersection with Triangulation method is used. Otherwise the Triangulation Only method is used.

The Intersection Only method goes directly to the "Steepest Intersection" method using the selected lines and polylines. The triangulation step is skipped and any selected point data is not used. This method can be used for making grids out of polylines such as a contour map as long as the surface is defined just by contour polylines without needing spot elevation points. Skipping the triangulation step makes this method a lot faster especially for large files.



No elevations are calculated on grid cells that extend beyond the extent of the data. The figure shows an example of how the grid is calculated to the limits of the data points. Extrapolation can be used to calculate elevations for the grid cells that are beyond the data limits. When there are grid cells with no elevation in a .grd file, many routines will prompt *Extrapolate grid to full grid size?* Extrapolation fills in all the grid cells. The method to extrapolate uses a safe calculation that tends average out or level the extrapolated values. So extrapolated grid areas are not as accurate as grid areas within the limits of the data. The Grid File Utilities command can be used to apply and save extrapolation to a grid file. The Plot 3D Grid command can then draw the grid file so that you can see the extrapolation.

The .GRD file has the following format:

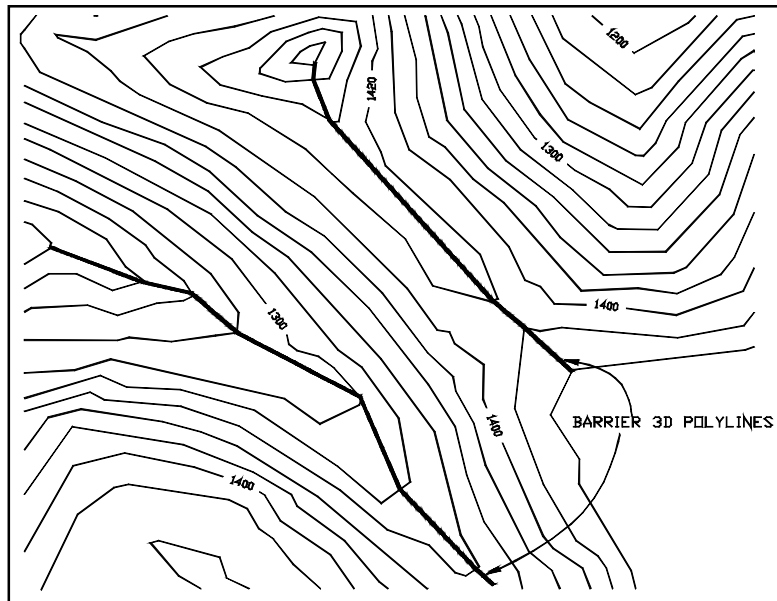
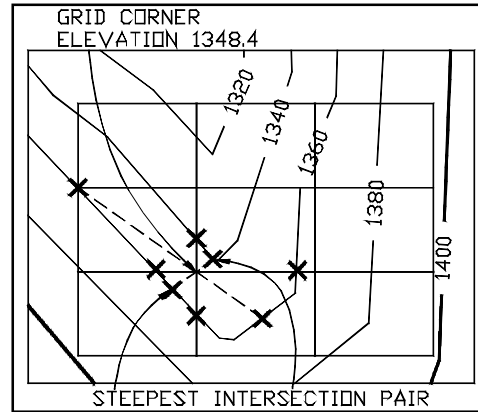
- Line 1 is the lower left Y coordinate
- Line 2 is the lower left X coordinate
- Line 3 is the upper right Y coordinate
- Line 4 is the upper right X coordinate
- Line 5 is the X direction grid resolution
- Line 6 is the Y direction grid resolution

The rest of the lines are the Z values of the grid intersects starting from the lower left moving in the left to right direction and ending at the upper right. If the intersect has no value, the letter 'N' is saved instead of the Z value.

Grids from Contour Maps

A grid file can be created from contours represented as polylines with elevation. The program calculates the elevation of each grid corner by looking for contour intersections in eight directions (N, S, E, W, NE, SE, SW, NW) and then interpolating the elevation between the two steepest intersections.

To accurately model the surface, it might be necessary to add entities in addition to the contour polylines. For one, spot elevation points can be added for the high and low points. Otherwise the grid model will plateau at the last contour. Also 3D barrier polylines need to be added on long narrow ridge and valley contours because in these areas the program will find the same contour when it looks for intersections in the eight directions. When all eight intersections are the same contour, the interpolated grid elevation equals the contour elevation instead of rising up the ridge or dipping in the valley. The 3D barrier polylines force interpolation along the ridge or valley. To draw these polylines, set the OSNAP to Nearest and run 3D Polyline under Draw. Then draw the polyline by picking the contour polylines where the barrier polyline crosses them. Another way to quickly create barrier polylines is to first draw the barrier polylines as 2D polylines. Then convert these polylines into 3D polylines with the *Screen* option in the 2D to 3D Polyline by Surface Model command under 3DPoly.



Prompts

GRiD File to Create File Selection Dialog Enter a name for the grid file. The default directory is the Carlson data directory.

Use position from another file or pick grid position (File/<Pick>)? Press Enter Using the position from another file will apply the lower left and upper right corners and the grid resolution of the another grid file to the current one.

Pick the Lower Left grid corner: *pick a point for the lower left limit of the grid.*

Pick the Upper Right grid corner: *pick a point*

Make GRiD File dialog box

In this dialog, you specify the grid resolution and whether or not to include data points with zero elevations. You can specify the resolution by entering the number of grid cells in the X and Y directions. By the Dimensions option, you to set the X and Y size for each grid cell.

Select the points, lines, and polylines to grid from.

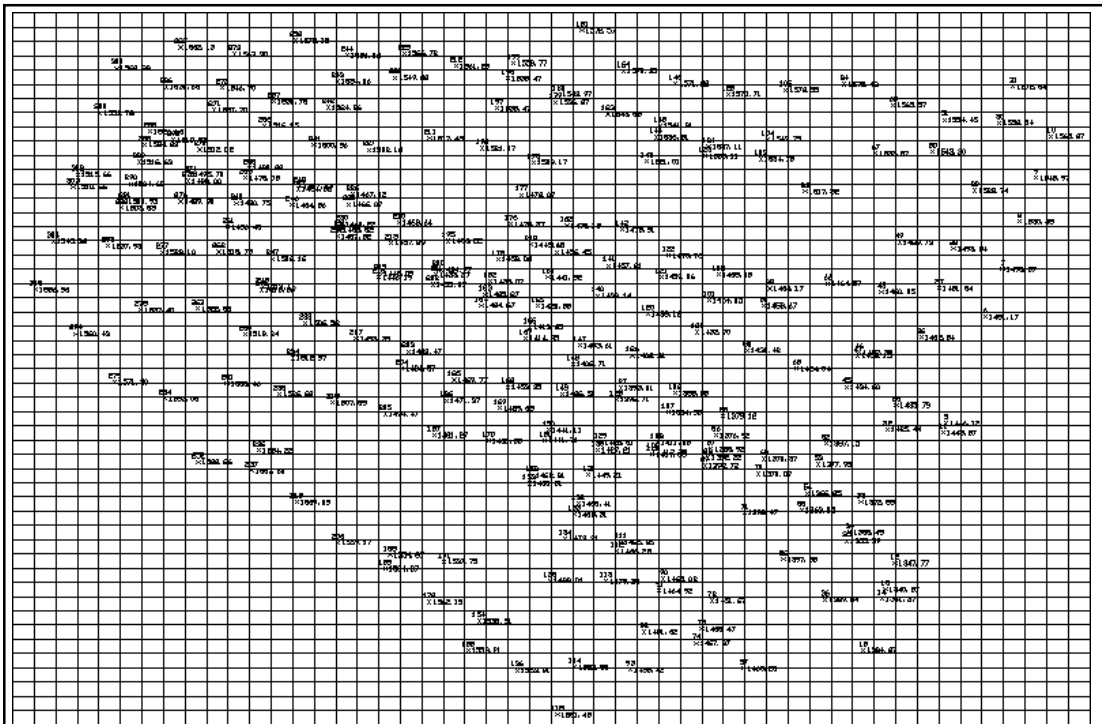
Select objects: *pick the objects that will define the surface model*

Pull-Down Menu Location: DTM

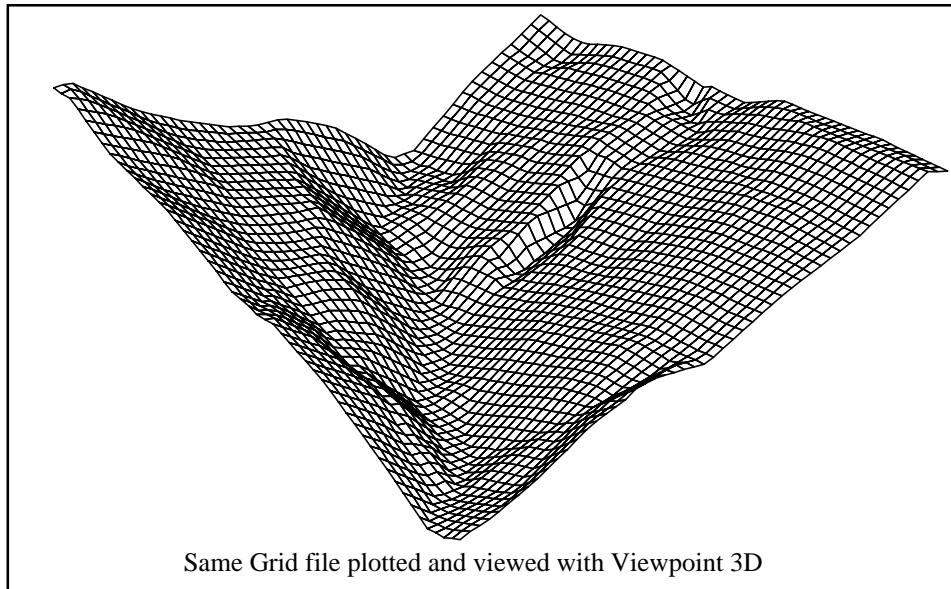
Prerequisite: Entities that define the surface

Keyboard Command: mkgrid

File Name: \lsp\dtmmkgrid.lsp, \lsp\makegrid.dcl, \lsp\makegrid.arx



Extrapolated and drawn grid file in plan view



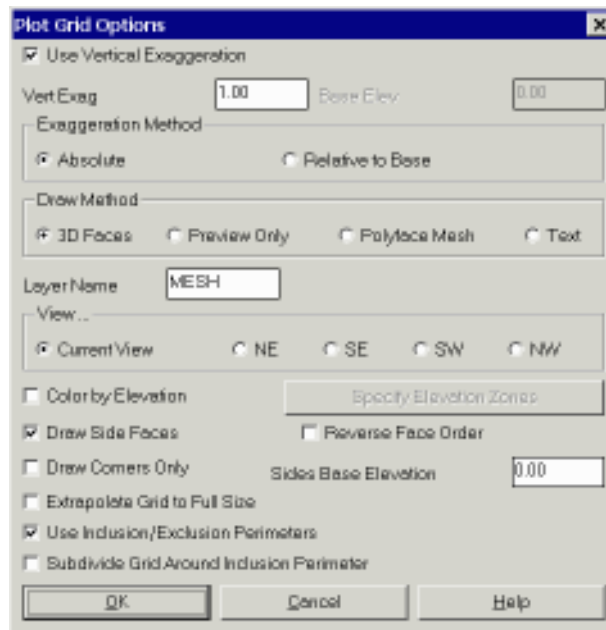
Plot 3D Grid File

Function

This command draws the 3D grid mesh of the chosen .GRD file. Each grid cell can be drawn as a 3D Face entity, Polymesh, Text or temporary lines. 3D Faces and Polymesh can be viewed from different directions using the Viewpoint 3D command in the Display menu. AutoCad's Hide and Shade commands can also be applied to 3D Faces and Polymesh. Also 3D Faces are used by other routines like Slope Zone Analysis.

The Text option will label the grid elevation at the grid corner. The text is placed center justified over the grid corner. To reduce clutter, there is an option to skip rows and columns.

The Preview Only option draws the grid as temporary "ghost" lines. The advantage with this method is that it is a much faster way to view the grid. However as soon as you do a



Edit 3D Grid

Function

This command edits the elevation of a grid node by graphically picking the grid corner and entering a new elevation. The grid is a surface model that is represented by a rectangular mesh of grid cells. Each grid cell has four corners with elevation. This command modifies the elevation of one of these grid corners. After picking the grid node to edit, the program draws a temporary X marker on the selected point and shows the current elevation for the point. Before running this routine, a grid file (.grd) must be created with the Make 3D Grid File command. Also the grid must be drawn on the screen using the Plot 3D Grid File command. Besides updating the elevation of the grid in the drawing, the grid file can also be updated. Whether to update the grid file is specified at the first prompt in the program.

Prompts

Update drawing only (Yes/<No>)? *Press Enter.* Choose between modifying the grid drawing or both the grid drawing and file.

Select Grid node to edit: *pick a grid cell corner*

Enter new Grid node elevation <305.519>: *Press Enter*

Select Grid node to edit: *Press Enter to end*

Pull-Down Menu Location: DTM

Prerequisite: a .GRD File and drawn grid 3D Faces

Keyboard Command: editgrid

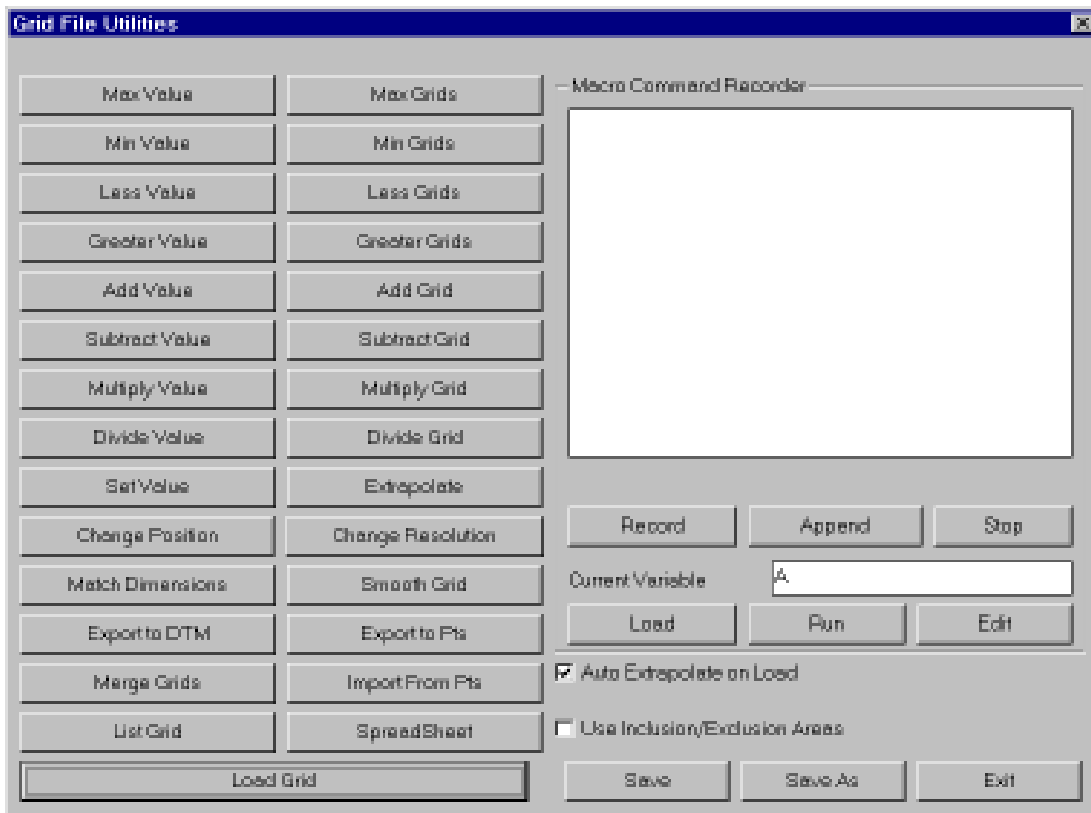
File Names: \lsp\poly3d.arx

Grid File Utilities

Function

This command has many functions to modify grid files. The modifications can be done manually and in a batch mode using the grid macros. To modify manually, start by picking the Load Grid button. The **Auto Extrapolate on Load** option will extrapolate values for any null values in the grid as the grid is loaded. There is another option to use inclusion and exclusion polylines. With this option active, the program will prompt for inclusion and exclusion polylines when a grid action is started. Only grid cells inside the inclusion polylines will be modified. Grid cells inside the exclusion polylines will not be modified. If no inclusion and exclusion polylines are selected, then the entire grid will be modified.

List Grid displays a list of the northing, eastings and elevation of each grid corner. A grid corner will have no elevation, listed as None, if the grid corner was outside the limits of the data during Make 3D Grid File.



Spreadsheet displays the grid elevations in a row and column spreadsheet that is in the same layout as the grid file. Grid elevations can be edited in this spreadsheet.

Smooth Grid applies a quadratic smoothing algorithm to the grid by using neighboring nodes to adjust each grid node. This routine can be used to refine a grid so that the contours from the Contour from Grid routine appear smoother. Typically this adjustment is relatively small. To get more smoothing, run the routine more times.

Change Position lets you change the lower left and upper right corners of the grid file. For example, you can use this routine to localize a grid file if you have a large grid for the entire site but are currently working on a smaller area. If the new position covers area outside the original position, any grid cells in this area will be assigned a null value. Otherwise the program uses the original grid values for the new grid position.

Change Resolution changes the grid resolution (number of grid cells). The program uses the original grid values for calculating the grid values at the new resolution.

Match Dimensions sets the grid position and resolution to match another grid file. The program will prompt for a grid file to get the position from.

Export to DTM writes the current grid file to a DTM format text file. The format of this file is the following:

DTM 1.0	Header Line
test.dtm	Name of file
51	Number of cells in X direction
51	Number of cells in Y direction
79442.4697	Lower left grid corner Y coordinate
14899.0326	Lower left grid corner X coordinate
0.0	Lower left grid corner Z coordinate
11.5618	Dimension of cell in X direction
7.0639	Dimension of cell in Y direction
1581.2612	Grid cell values starting from lower left, moving from left to right
1580.8879	
1580.3257	
etc...	

Export to Points stores the northing, easting and elevation of grid nodes to a CRD file. There are options to skip rows and columns from the grid. For example, you can enter skip=1 to create points for every other row and column.

Import from Points creates a grid file from X Y Z data in a text file. There does not need to be a current grid file loaded since this routine will create a grid file. The text file should consist of one X Y Z coordinate per row with the first coordinate being the lower left grid corner and the last coordinate as the upper right grid corner. There are options for space or comma separated coordinates and for the order of the coordinates as either row (left to right) or column (bottom to top).

Extrapolate can be used to assign elevations to all grid corners by any of four methods. Global Trend finds the average slope and slope direction from the existing grid elevations and applies this slope to calculating the missing elevations. Average method calculates a grid elevation as the average of its nearest neighbors. Projected method extends the trend at the edge. Combined method uses both Average and Projected. The Set Value button assigns the grid elevations to the user-specified value.

Merge Grids creates a grid file by merging together two existing grid files, grid1 and grid2. The current grid is grid1 and the program will prompt for a second grid. These two grids must overlap with the same location and resolution. The inclusion and exclusion perimeters apply to grid2 such that the merged grid will consist of grid2 cells within the inclusion perimeters and outside the exclusion perimeters and grid1 cells everywhere else. The result is stored in the current grid.

Set Value assigns the grid elevations to the user-specified value. For example by using Set Value with the inclusion perimeter option, you could set the grid values to 0.0 within the inclusion

polyline for a strata thickness grid. You can also use Set Value to assign grid cells to NULL (no value).

Add Value, Subtract Value, Multiply Value and **Divide Value** perform their function on each grid cell using a user-specified value. For example, add 5 to the grid.

Add Grid, Subtract Grid, Multiply Grid and **Divide Grid** apply their function using another grid file. This second grid file must have the same location and resolution. For example, you can use Add Grid to add a strata thickness grid to a strata bottom elevation grid in order to create a top elevation grid.

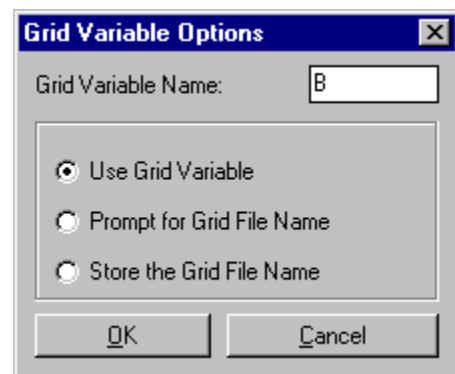
Min Grids compares the each grid elevation of the current grid file with a second grid file and takes the minimum value. Likewise **Max Grids** compares two grids and takes the maximum value.

Min Value compares the each grid elevation of the current grid file with a user-specified value and takes the minimum value. Likewise **Max Value** compares the grid with a value and takes the maximum value.

Less Value asks for a value to compare and a value to assign and uses the following logic:
if grid < compare_value then grid =assign_value, otherwise no change

Greater Value is the same as Less Value except the < is replaced with a > . **Less and Greater Grids** use the same logic using grids to compare and assign instead of single values.

The Macro Command Recorder allows you to store the grid manipulation steps to a .GFU file. The macro can be recalled to rerun the steps. The Record button will prompt for a macro file name to create. Then start choosing grid action buttons and each grid file function will be stored to this file. Each grid file in the script is represented as a variable name such as A or B. The current grid file that is being modified is specified in the Current Variable edit box. When recording a step that involves another grid file, there are three options for storing this grid into the script. Use Grid Variable will use the grid assigned to the Grid Variable Name. Prompt for Grid File Name will bring up a grid file selection dialog each time the macro is run. Store the Grid File Name will save the specific grid file name into the macro.



Pull-Down Menu Location: DTM

Prerequisite: Make a .GRD file with the *Make 3D GRiD File* command.

Keyboard Command: GFU

File Name: \\sp\makegrid.arx

One Surface Volumes

Function

One Surface Volumes calculates the cut and fill volumes between the surface modelled by one .GRD file and a constant elevation. This is the same as Two Surface Volumes except that the second surface is a flat plane at a constant elevation instead of a 3D grid surface. If the grid contains grid cells that have no elevations, you have the option to extrapolate elevations from the grid cells with elevations. When you choose not to extrapolate, no volume is calculated for these grid cells. There are also options to specify inclusion and exclusion areas. When inclusion areas are specified, only the volume within this inclusion area is calculated. Volumes within an exclusion area are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to calling One Surface Volumes.

Prompts

Specify GRiD File File Selection Dialog

Choose a .GRD file to process.

Extrapolate grid to full grid size (Yes/<No>)? *Press Enter.* If you enter Yes to this prompt, surface elevations will be computed for any grid cells that have null elevations.

Enter the base elevation: 570

This defines the second surface.

Select the Inclusion perimeter polylines or ENTER for none:

Select objects: *Press Enter*

Select the Exclusion perimeter polylines or ENTER for none:

Select objects: *Press Enter*

Preprocessing grid cells ...

Processing cell 2500

Volume Report Options Dialog

Pull-Down Menu Location:

DTM

Prerequisite: A .GRD file

Keyboard Command: volcalc1

File Name: \lsp\volcalc.lsp &
\lsp\volcalc.arx

<p>Volume report Comparing GRiD file: D:/SC12/DATA/FINAL.GRD Lower left grid corner : 15965.00,12657.00 Upper right grid corner: 16275.00,12917.00 X grid resolution: 31, Y grid resolution: 26 X grid cell size: 10.00, Y grid cell size: 10.00 Total inclusion area: 56357.81 sq ft, 1.29380 acres Cut to Fill ratio: 0.95 Cut (C.Y) / Area (acres): 9781.46 Fill (C.Y) / Area (acres): 10303.79 Cut volume is 341691.014 cubic ft, 12655.22274 cubic yards Fill volume is 359937.370 cubic ft, 13331.01369 cubic yards Sample One Surface Volumes</p>
--

Two Surface Volumes

Function

Two Surface Volumes calculates the cut and fill volumes between two surfaces modelled by .GRD files. These two grid files must have the same location and resolution. To create the grid files, use the Make 3D Grid File routine. When creating the second grid file, choose *Use position of another file* and select the first grid file. Using the position of the first grid file sets the location and resolution of second grid to match the first.

There are several other routines that calculate volumes based on grid files. Grid based volumes can be calculated by One Surface Volumes, Volumes by Layer, Stockpile Volumes, and Pond/Pit Volumes. These routines have special prompting and calculate the grid surfaces and volume in one step.

Volumes by Two Surface Volumes has three steps:

1. Creating the first grid file with Make 3D Grid File
2. Creating the second grid file with Make 3D Grid File
3. Running Two Surface Volumes

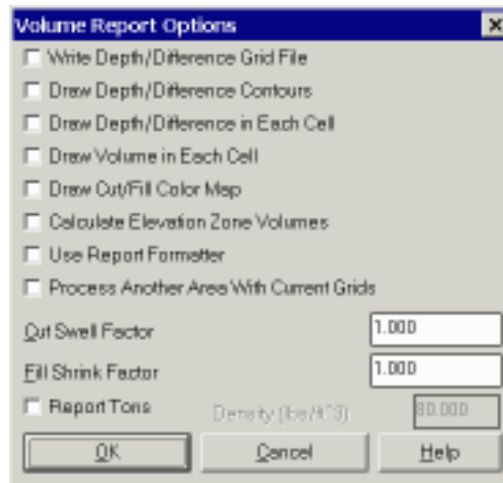
One advantage to Two Surface Volumes is that you have more control in the creation of the grid files. Also you can draw the grid files with the Plot 3D GRiD File routine so that you can view exactly what is being compared. Another way to verify that the grid files model the surfaces correctly is to run the Contour From Grid routine and review the contours. Two Surface Volumes also has more output options to check volumes.

Besides grid based volumes, volumes can also be calculated between triangulation surfaces using the Volumes by Triangulation command. Cross section end area is another volume method that is used by the Calculate Sections Volume command in the Section-Profile module.

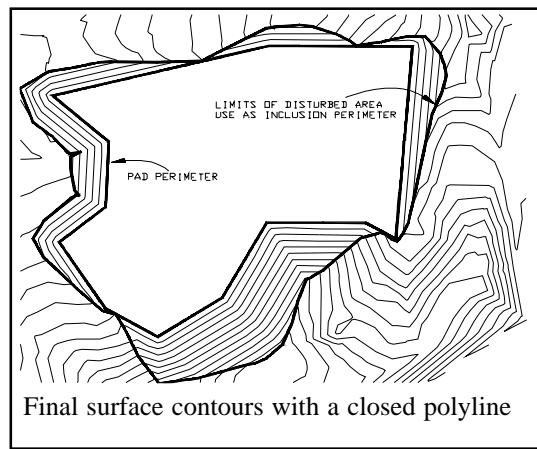
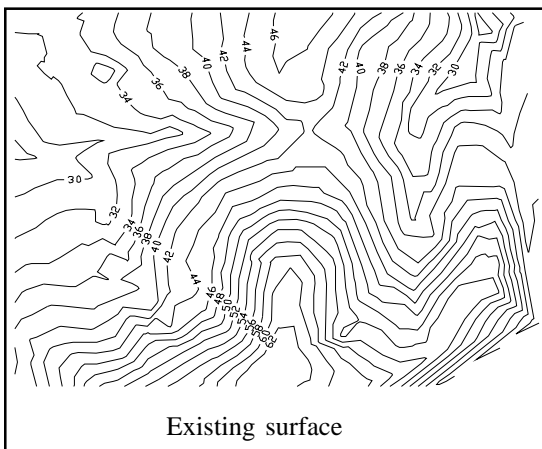
There are also options to specify inclusion and exclusion areas. When inclusion areas are specified, only the volume within this inclusion area is calculated. **Important:** whenever possible you should use a polyline that represents the limits of disturbed area as the inclusion perimeter. Volumes within an exclusion area are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to calling this routine.

If the grid contains grid cells that have no elevations, you have the option to extrapolate elevations from the grid cells with elevations. When you choose not to extrapolate, no volume is calculated for the grid cells left without elevations. In general, extrapolation is not very accurate and should be avoided whenever possible. Sometimes you may get small amounts of cut in stockpiles that should only be fill, or small amounts of fill in pits that should only be cut. These extraneous quantities are due to extrapolation at the border and should be small enough to be ignored. When inclusion or exclusion polylines are used, the program will automatically extrapolate the grids.

In addition to writing a volume report to the file, printer or screen, there are several volume report options. **Write Difference Grid File** will create a .GRD file of the elevation difference of the two grid files. **Draw Difference Contours** will create a contour map of the difference or depth between the two grid files. **Draw Elevation Difference in Each Cell** will plot the elevation difference at the grid corners which is the same as the Elevation Difference routine. **Draw Volume in Each Cell** will plot the calculated volume for each grid cell and is an excellent way to verify the volume calculation. If a cell contains both cut and fill, both values will be plotted. **Calculate**



Elevation Zone Volumes calculates the cut and fill between different elevation ranges. The **Draw Cut/Fill Color Map** routine will solid fill color each grid cell with different shades based on the average cut or fill in the cell. Red shades are used for cut and blue for fill. There is an option to draw a color legend. The **Use Report Formatter** allows you to customize the report by choosing the fields to report and their order. Also the report formatter can be used to output the report data to Excel or Access. The **Process Another Area with Current Grids** option will run Two Surface Volumes again using the same grid files but different inclusion/exclusion polylines. This option saves the step of reloading the grid files to calculate volumes from the same grids for multiple areas. The **Cut Swell Factor** is multiplied by the cut volume in the report. Likewise the **Fill Swell Factor** is multiplied by the fill volume. The **Report Tons** option allows you to enter the material density and the program will report the cut and fill tons in addition to volume.



Volume Report

Comparing GRiD: C:/SC14/DATA/SIMO.GRD
and GRiD: C:/SC14/DATA/BASE.GRD

Lower left grid corner : 186551.67,57624.98

Upper right grid corner: 186828.81,57897.09

X grid resolution: 75, Y grid resolution: 75

X grid cell size: 3.70, Y grid cell size: 3.63

Total inclusion area: 37016.71 sq ft, 0.850 acres

Cut to Fill ratio: 1.14

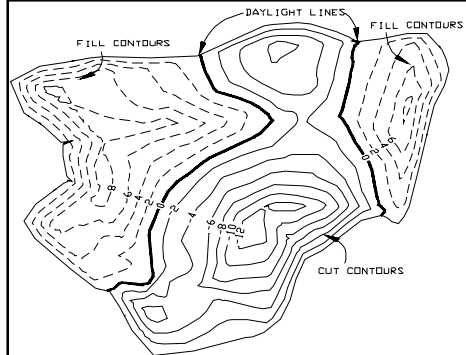
Cut (C.Y) / Area (acres): 3642.35

Fill (C.Y) / Area (acres): 3182.70

Cut vol: 83570.89 cubic ft, 3095.22 cubic yards

Fill vol: 73024.56 cubic ft, 2704.61 cubic yards

Sample Two Surface Volumes report



Contours from the Draw Depth/Difference Contours option. This is a good way to check that both surfaces are modeled correctly and to verify the volumes. The cut contours are red, fill contours are blue, and

Given two good .grd files, this routine will calculate accurate volumes. So to verify the volume calculation, it is a good idea to check the .grd files either by drawing them with *Plot 3D Grid File* and viewing them with *Viewpoint 3D* or by contouring the grids with the *Contour Grid File* command.

Prompts

Select the Inclusion perimeter polylines or ENTER for none:

Select objects: *Pick a closed polyline for the limits of disturbed area*

Select objects: *Press Enter*

Select the Exclusion perimeter polylines or

Report from the Calculate Elevation Zone Volumes option which calculates the cut and fill in different elevation ranges at a user-specified interval and beginning at a user-specified starting elevation.

Volumes by elevation zone

Zone 20.00 to 30.00

Cut volume : 0.30 cubic ft, 0.01 cubic yards

Fill volume: 107.90 cubic ft, 4.00 cubic yards

Zone 30.00 to 40.00

Cut volume : 4.88 cubic ft, 0.18 cubic yards

Fill volume: 73021.14 cubic ft, 2704.49 cubic yards

Running total:

Cut volume : 5.18 cubic ft, 0.19 cubic yards

Fill volume: 73129.05 cubic ft, 2708.48 cubic yards

Zone 40.00 to 50.00

Cut volume : 65044.26 cubic ft, 2409.05 cubic yards

Fill volume: 0.25 cubic ft, 0.01 cubic yards

Running total:

Cut volume : 65049.44 cubic ft, 2409.24 cubic yards

Fill volume: 73129.29 cubic ft, 2708.49 cubic yards

Zone 50.00 to 60.00

Cut volume : 17786.85 cubic ft, 658.77 cubic yards

Fill volume: 0.00 cubic ft, 0.00 cubic yards

Running total:

Cut volume : 82836.29 cubic ft, 3068.01 cubic yards

ENTER for none:

Select objects: *Press Enter*

Specify Base Grid File Selection Dialog

Choose a .GRD file to process.

Extrapolate grid to full grid size (Yes/<No>)? *Press Enter.* If you enter Yes to this prompt, surface elevations will be computed for any grid cells that have null elevations.

Specify Final Grid File Selection Dialog

Choose a .GRD file to process.

Extrapolate grid to full grid size (Yes/<No>)? *Press Enter.*

Volume Report Options

Pull-Down Menu Location: DTM

Prerequisite: Two grid files

Keyboard Command: volcalc2

File Name: \lsp\volcalc.lsp & \lsp\volcalc.arx

-83.8	-82.1	-63.2	-31.8	-7.5 +0.4	+15.1	+37.2	+59.4
-81.2	-78.8	-61.2	-33.8	-11.6	-0.1 +10.6	+32.7	+54.9
-77.2	-73.9	-60.4	-38.3	-16.1	-0.6 +6.7	+28.2	+50.4
-71.7	-65.6	-53.3	-36.9	-18.7	-2.1 +3.6	+23.7	+45.8
-64.1	-53.8	-40.3	-26.7	-12.7	-1.1 +4.7	+21.8	+41.6
-53.2	-40.3	-26.8	-13.3	-2.1 +2.5	+14.3	+28.5	+44.4

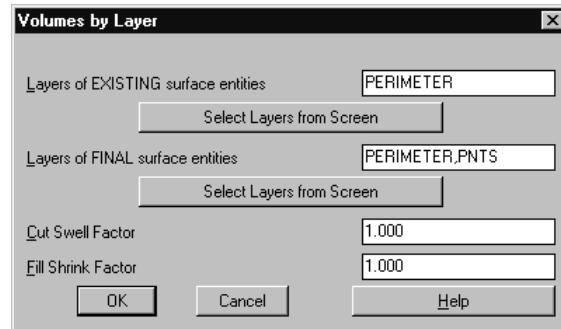
This shows a grid drawn by Plot 3D Grid File and volume values drawn by the Draw Volume in Each Cell option of the Two Surface Volumes routine. Cut appears as negative and fill as positive. Notice that cells bordering cut and fill regions contain a little of both.

Volumes By Layer

Function

Volumes By Layer is the easiest yet still equally accurate method for calculating volumes. For this command, volumes are calculated in one step by a simple window of the area, selecting the items, and calculate.

First you must specify the grid location and resolution. The grid location should enclose the area for volume calculations. Next the program asks for the layer names of the entities for the base and final surfaces. These entities, for use in modelling the surfaces, can be points, lines (such as triangulation lines), 2D polylines (such as contours), and 3D polylines (such as barrier lines). Inclusion and exclusion perimeters may optionally be specified to limit the volume calculation area on the grid. An inclusion perimeter should be used if there is a closed polyline for the limit of the disturbed area. Then the program internally generates grids of the surfaces from the entities on the corresponding layers and then calculates and reports the volume. The main disadvantage to this routine is that it doesn't have the special output options of Two Surface Volumes such as Depth Contours.



Prompts

Pick Lower Left limit of surface area: *Pick a point*

Pick Upper Right limit of surface area: *Pick a point to window the entire area*

Make Grid File Dialog Choose a grid resolution.

Volumes by Layer Dialog Specify layer names. Click Select Layers to get layers of picked entities on screen.

Select surface entities on corresponding layers.

Select objects: *select all the objects on both the EXISTING and FINAL layer sets for the site.*
In this example, select all entities on the perimeter, ctr, and pnt layers.

Select the Inclusion perimeter polylines or ENTER for none:

Select objects: *Press Enter*

Select the Exclusion perimeter polylines or ENTER for none:

Select objects: *Press Enter*

Processing cell 2500

Write report to file (Yes/<No>)? *Press Enter*

Write report to printer (Yes/<No>)? *Press Enter*

Volume Report *See Figure 5-10*

Pull-Down Menu Location: DTM

Prerequisite: Entities that define both the base and final surfaces.

Keyboard Command: layervol

File Name: \lsp\layervol.lsp & \lsp\makegrid.arx

Volumes By Triangulation

Function

Volumes By Triangulation is an alternative volume method that compares two triangulation networks. This method is different from the grid based volume routines (*Volumes By Layer, One Surface Volumes, Two Surface Volumes, Stockpile Volumes, etc.*) and the cross section volume routine (*Calculate Section Volume*). Volumes by Triangulation calculates faster in most cases than the other methods, and it is the most accurate because it models every edge on the site. This added accuracy in general is very small. The grid resolution is usually sufficient to model the surface for the grid based volumes. The Volume By Triangulation accuracy applies well when there is a feature like a 5 foot wide ditch. Then the grid resolution would need to be less than 5 foot to model the ditch which might be difficult on a large site.

The disadvantage to this routine is that it lacks the output options that help the analysis of the volume such as Difference Contours. Also Volumes by Triangulation does no extrapolation and stops calculating volume at the perimeter of the smaller of the two triangulation networks. Volumes By Triangulation is better when used with point data instead of contour data because contour data requires triangulating all the contour polylines as barrier lines which creates a large triangulation network and is slower.

The triangulation networks to compare are defined in .flt files that are created by *Triangulate & Contour* with the Write Triangulation File option. Before Volumes By Triangulation, run *Triangulate & Contour* twice to create an .flt file for each surface.

The volume calculation is limited by either the extent of the triangulation networks or by an inclusion perimeter which is a closed polyline. An inclusion polyline should be used when there is limit of disturbed area perimeter.

Prompts

Select EXISTING surface Tmesh File

Choose a .flt file

Select FINAL surface Tmesh File

Choose another .flt file

Pick inclusion perimeter polyline (ENTER for none): *pick a closed polyline perimeter*

Calculating ...

Write report to file (Yes/<No>)? *Press Enter*

Write report to printer (Yes/<No>)? *Press Enter*

Comparing Triangulation files: C:\SC124\DATA\TRI1.FLT

and C:\SC124\DATA\TRI2.FLT

Cut volume: 66891.35 C.F., 2477.46 C.Y.

Fill volume: 43458.01 C.F., 1609.56 C.Y.

Pull-Down Menu Location: DTM

Prerequisite: Two .flt files.

Keyboard Command: trivol

File Name: \lsp\contour4.arx

Elevation Zone Analysis

Function

This command can be used to calculate the surface area of a surface in different elevation zone ranges. The program requires *3D Face* entities that can be generated by the *Plot 3D GRid File* or *3D Slice* commands. The Draw Triangulation Faces option in Triangulate & Contour can also be used to create triangular 3D Faces. For each elevation zone, the 3D Faces can be hatched with an AutoCad hatching pattern, solid filled with the "SOLID" pattern, or left empty with the "NONE" pattern. The 3D Faces are also placed in a layer for each zone.

Elevation	Range	Color	Pattern	Scale	Layer	Page: 1
540.00	< 540.00	Color...	SOLID	50.00	zone1	
550.00	540.00 to 550.0	Color...	SOLID	50.00	zone2	
560.00	550.00 to 560.0	Color...	SOLID	10.00	zone3	
570.00	560.00 to 570.0	Color...	SOLID	10.00	zone4	
580.00	570.00 to 580.0	Color...	SOLID	10.00	zone5	
590.00	580.00 to 590.0	Color...	SOLID	50.00	zone6	
	> 590.00	Color...	SOLID	50.00	zone7	
		Color...	SOLID	50.00	zone8	
		Color...	SOLID	50.00	zone9	
		Color...	SOLID	50.00	zone10	
		Color...	SOLID	50.00	zone11	
		Color...	SOLID	50.00	zone12	
		Color...	ANGLE	50.00	zone13	
		Color...	ANGLE	50.00	zone14	

OK Cancel

There are also options to specify inclusion and exclusion areas. When inclusion areas are specified, only the area within the inclusion polyline is calculated. Areas within an exclusion polyline are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to calling this routine. Without inclusion and exclusion polylines, all the area of each selected 3D Face is used.

Prompts

Elevation Zone dialog Specify the elevation ranges, colors and patterns.

Plot average elevation text in each grid cell (Yes/<No>): *Y*

Select Rectangular 3D Faces to Analyze...

Select objects: *C*

First corner: *(pick point)* **Other corner:** *(pick point)* **1200 found**

Select objects: *[Enter]*

Select the Inclusion perimeter polylines or ENTER for none:

Select objects: *Pick a closed polyline for the limits of disturbed area*

Select objects: *Press Enter*

Select the Exclusion perimeter polylines or ENTER for none:

Select objects: *Press Enter*

Zone	Range	Surface Area	%of Total
1	<= 100.00	26822.41	27.25
2	> 100.00	27439.15	27.88
3	> 110.00	16727.98	17.00
4	> 120.00	12357.28	12.55
5	> 130.00	11700.36	11.89
6	> 140.00	3380.98	3.43

Total Surface Area> 98428.16

Average Elevation > 110.83

Minimum Slope% > 0.00 **Maximum Slope%>** 252.97

Minimum Elevation > 90.00 **Maximum Elevation>** 149.53

If you have the printer/device set to ON in the *Configure SurvCADD* the above report is sent to it.

Select point for color legend: *(pick point)*

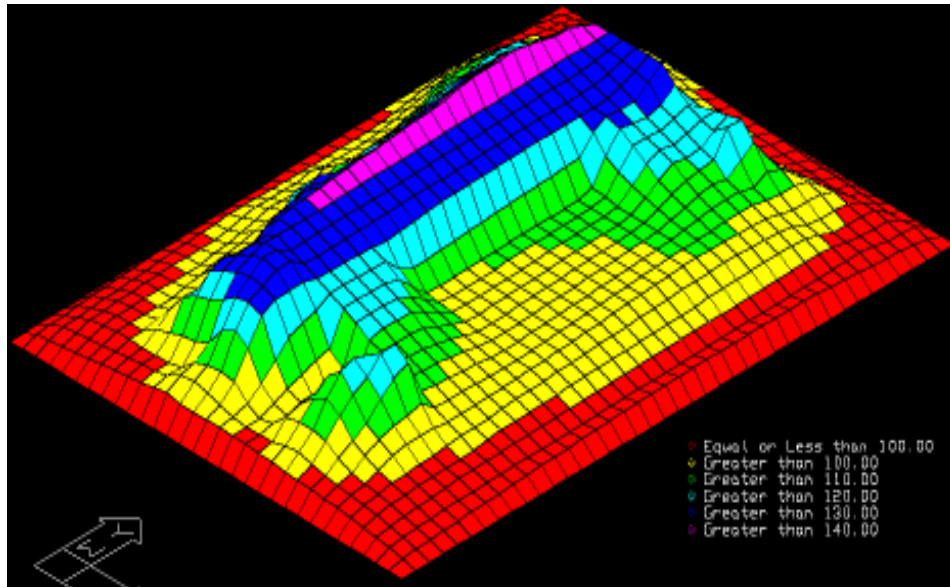
If a point is picked, a legend showing the color of each range is drawn. The legend is drawn aligned to the current view *UCS*. For this reason it is best to have the mesh at the *Vpoint* at which it will be plotted before executing the analysis program.

Pull-Down Menu Location: DTM

Prerequisite: 3D Face entities plotted usually created by the *Make 3D GRiD File* command.

Keyboard Command: elvzone

File Name: \lsp\elanal.lsp & \lsp\contour4.arx



Result of Elevation Zone Analysis viewed in 3D with the Viewpoint 3D command and shade command.

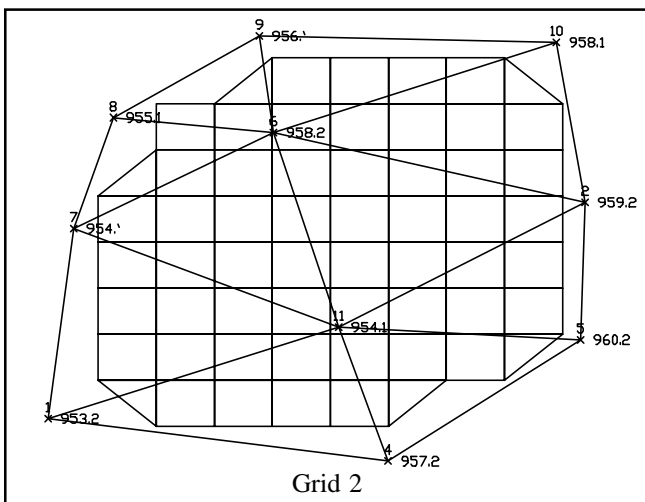
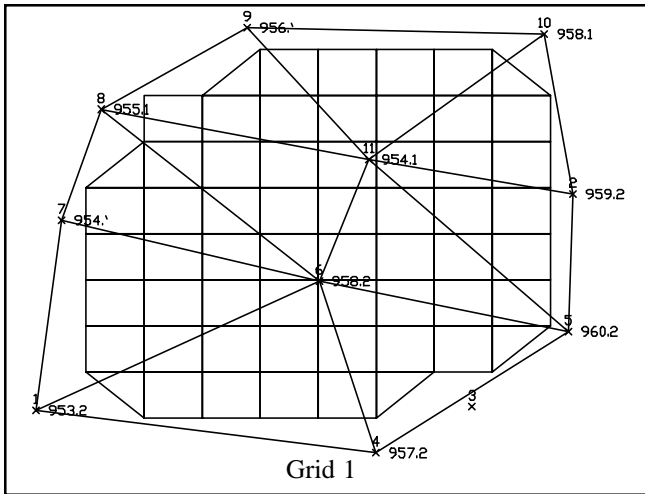
Elevation Difference

Function

This routine calculates the elevation difference either between two grid files at each grid corner or between points and a grid or triangulation file. Grid file model surfaces are created with the *Make 3D GRiD File* command and triangulation files are created with the *Triangulate & Contour* command.

When comparing two grid files, the elevation difference is drawn as text, and the + or - sign is placed where the measurement is taken at the grid corner. The two grids must have the same resolution and location. To get the grids to match, either pick the exact same lower and upper grid points, or use the Use position from another file option of *Make 3D GRiD Files* when making the second grid. Elevation Difference can be used to obtain a map of the elevation differences between two surfaces. The elevation difference labels can be drawn in different layers for cut, fill and zero difference (daylight). The default is to have cut in red, fill in blue and daylight in green. You can also specify the text size, decimal places and the number of rows and columns to skip. For example, on a large dense grid, you may want to skip every other row and column so that not too much text is drawn.

Elevation Difference also has an option to compare points stored in a coordinate file (.crd file) with a grid file or triangulation file. The routine generates a report of the point coordinates, surface



elevation and elevation difference. The report formatter is used to select the fields to report. The report can also be output to Excel or Access. There are also options to add the elevation difference to the point descriptions in the .CRD file. Another option draws text below the point for the elevation of the point, the elevation of the grid at the point, and then the elevation difference.

Prompts

For elevation difference between two grids:

Calculate difference at points or between grids (Points/<Grid>)?

Press Enter

Select the Grid 1 File Name Dialog

Select the Grid 2 File Name Dialog

The default layering method is to put cut in red layer CUT, fill in blue layer FILL, and zero elevation difference in green layer DAYLIGHT.

Layer name for the Cut text

<CUT>: *Press Enter*

Layer name for the Fill text

<FILL>: *Press Enter*

Highlight zero elevation differences (<Yes>/<No>)? *Press Enter*

Layer name for zero elevation difference text <DAYLIGHT>:

Press Enter

Size of text <0.73>: *[Enter]*

Number of rows to skip between text <0>: *[Enter]*

Number of columns to skip between text <0>: *[Enter]*

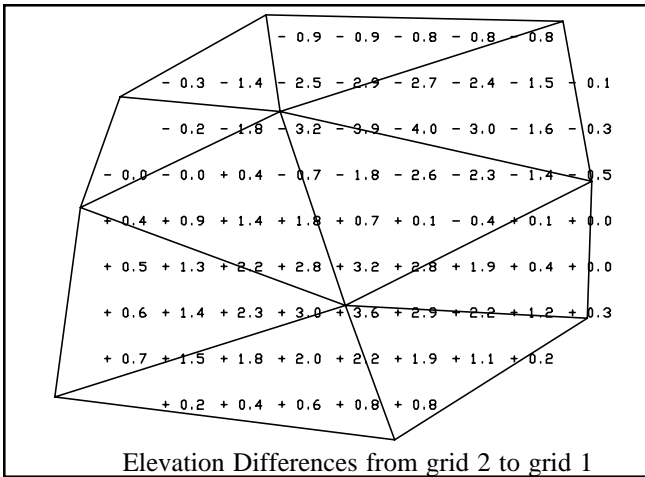
For elevation difference between points and grid:

Calculate difference at points or between grids (Points/<Grid>)? *P for points*

Select Coordinate File

Choose the .crd file of the point with elevations to compare with the grid surface.

Range of point numbers to use (A for All): *18-30*



Elevation Differences from grid 2 to grid 1

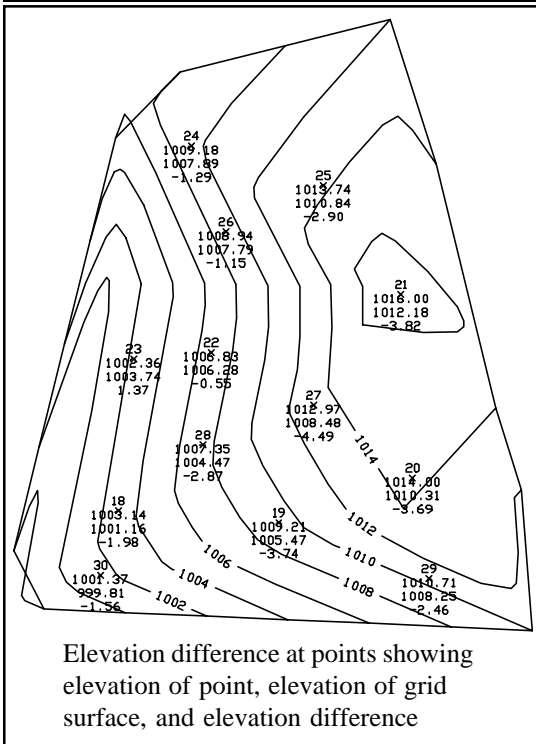
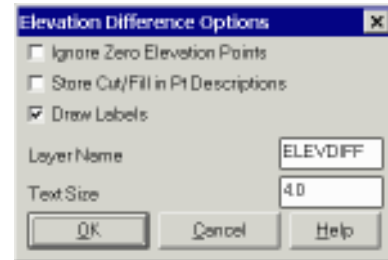
Wildcard match of point description

<*>: Press Enter

Select Grid File

Choose the .grid file that models the surface

Elevation Difference Options dialog



Elevation difference at points showing elevation of point, elevation of grid surface, and elevation difference

Pull-Down Menu Location: DTM

Prerequisite: Two .grid files or a .grid file and a .crd file.

Keyboard Command: elevdiff

File Name: \lsp\cntr_grid.arx

Point#	Northing	Easting	Point Z	Surface Z	Difference
23	5,043.336	5,041.267	582.100	582.091	-0.009
24	5,029.491	5,014.899	573.100	573.401	0.301
26	5,033.449	4,929.189	565.200	564.463	-0.737
27	5,076.303	4,914.027	559.200	558.824	-0.376

Report of Elevation Difference Between Points and Surface Model

Slope Direction Analysis

Function

This command categorizes the slope direction as either N, NE, E, SE, S, SW, W or NW. The program requires *3D Face* entities that can be generated by the *Plot 3D GRiD File* command or the *Draw Triangulation Faces* option of *Triangulate & Contour*. Each 3D Face is colored by the slope direction zone and a report of the area for each zone is generated. A pinwheel color legend can also be drawn.

There are options to specify inclusion and exclusion areas. When inclusion areas are specified, only the slope area within the inclusion polyline is calculated. Slope area within an exclusion polyline are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to calling this routine. Without inclusion and exclusion polylines, all the slope area of each selected 3D Face is used.

Prompts

Select 3D Faces to Analyze ...

Select objects: *pick the 3D faces*

Select the Inclusion perimeter polylines or ENTER for none:

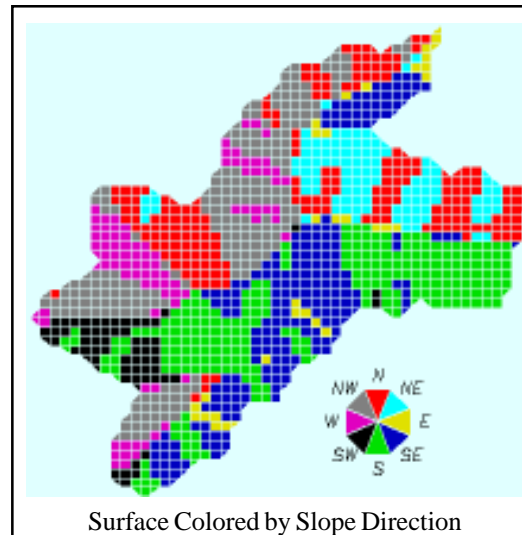
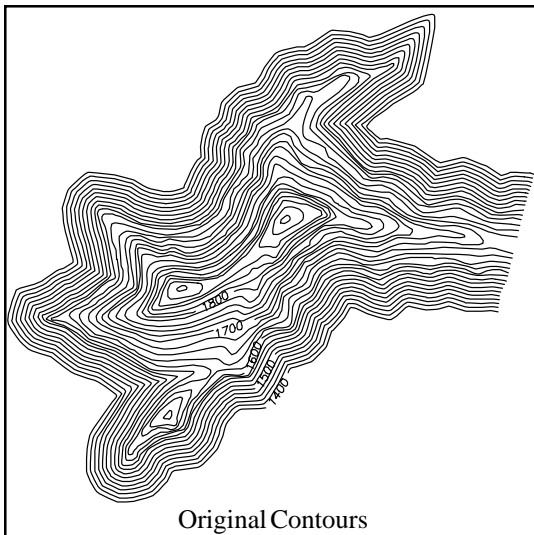
Select objects: *Pick a closed polyline if needed*

Slope Direction Report

Number of 3DFaces analyzed> 1180

Zone	Area S.F.	Acres	% of Total
N	1244327.38	28.57	17.24
NE	674141.92	15.48	9.34
E	198462.34	4.56	2.75
SE	1307331.30	30.01	18.11
S	1524694.82	35.00	21.13
SW	381173.70	8.75	5.28
W	548134.09	12.58	7.59
NW	1338833.26	30.74	18.55

Total Surface Area> 7217098.81



Select the Exclusion perimeter polylines or ENTER for none:

Select objects: *Pick a closed polyline if needed*

Select point for color legend: *pick a point*

Pull-Down Menu Location: DTM

Prerequisite: 3D Face entities

Keyboard Command: dzone

File Name: \lsp\contour4.arx

Slope Zone Analysis

Function

This command calculates the surface area of a site in different slope zone ranges. The program requires *3D Face* entities that can be generated by the *Plot 3D GRiD File* command or the *Draw Triangulation Faces* option of *Triangulate & Contour*. For each slope zone, the 3D Faces can be hatched with an AutoCad hatching pattern, solid filled with the "SOLID" pattern, or left empty with the "NONE" pattern.

This command can also generate contours of the slope zones based on the calculated slope at each point of the 3D Faces. The slopes can vary greatly between neighboring points. When contoured directly, these slope data points produce incoherent contours. Instead this routine applies a filtering algorithm that reduces the noise. There is another option to output a grid file of the slope values.

There are also options to specify inclusion and exclusion areas. When inclusion areas are specified, only the slope area within the inclusion polyline is calculated. Slope area within an exclusion polyline are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to calling this routine. Without inclusion and exclusion polylines, all the slope area of each selected 3D Face is used.

Prompts

Slope Zone dialog Specify the slope zones, colors and patterns from lowest to highest.

Draw contours of slope zones (Yes/<No>)? Yes

Output a grid file of the slope (Yes/<No>)? No

Apply hatch patterns to grid cells (Yes/<No>)? No. This option fills the 3D faces with the hatch pattern defined for the different zones. Without the hatch patterns the program will just modify the 3D faces color.

Freeze grid layer after processing (Yes/<No>)? No. This freezes the layer of the 3D faces so that the slope contours are easier to see.

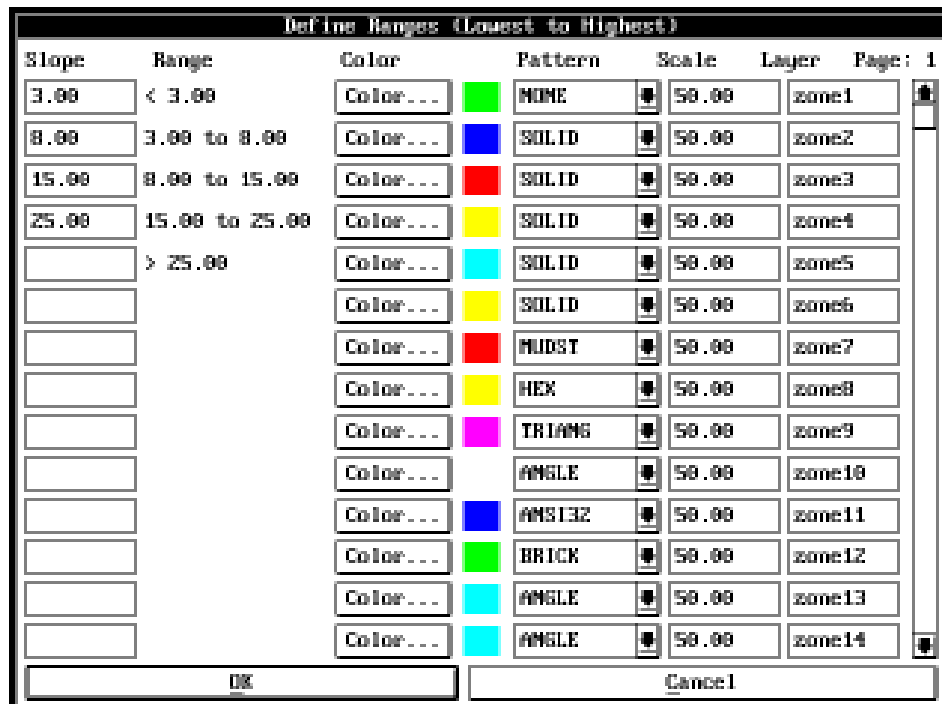
Select Rectangular 3D Faces to Analyze...

Select objects: C

First corner: *(pick point)* **Other corner:** *(pick point)* 1200 found

Select objects: *[Enter]*

Select the Inclusion perimeter polylines or ENTER for none:



Select objects: *Pick a closed polyline for the limits of disturbed area*

Select objects: *Press Enter*

Select the Exclusion perimeter polylines or ENTER for none:

Select objects: *Press Enter*

Analyzing Entity> 1000 of 1000

Select point for color legend: *(pick point)*

If a point is picked a legend showing the color of each range is drawn. The legend is drawn aligned to the current view *UCS*. For this reason it is best to have the mesh at the *Vpoint* at which it will be plotted before executing the analysis program.

Contouring Options Dialog This dialog appears if the Draw Slope Contours option is active. You may want to change the Contour Layer so it doesn't conflict with any existing contour layers. The Hatch Slope Zones applies the hatch patterns to the slope zones between the slope contours.

Reading points 1000 ...

Processing points 1000 ...

Triangulating points 1000 ...

Inserted 4205 contour vertices.

Pull-Down Menu Location: DTM

Prerequisite: 3D Face entities plotted usually created by the *Make 3D GRiD File* command.

Keyboard Command: szone

File Name: \lsp\elanal.lsp &\lsp\contour4.arx

Contouring Options

Contour Layer

Minimum Contour Line Length

Draw Index Contours

Label Contours

Smooth Contours Low High

Reduce Vertices Offset Distance

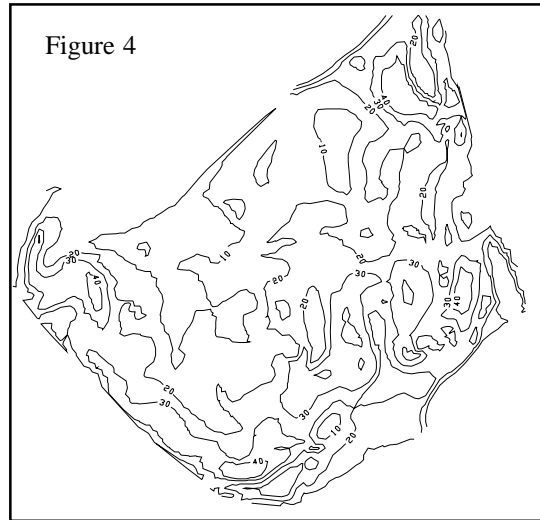
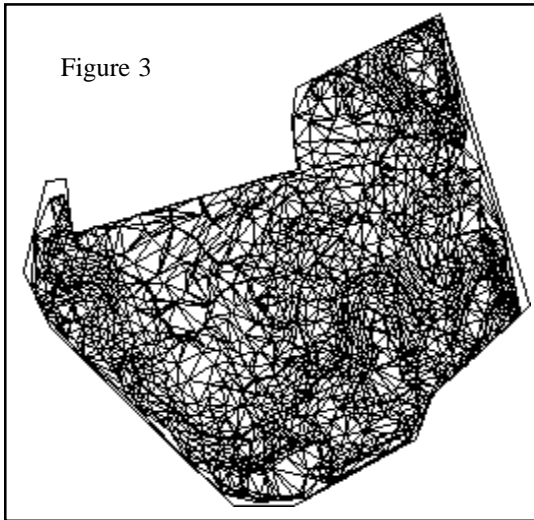
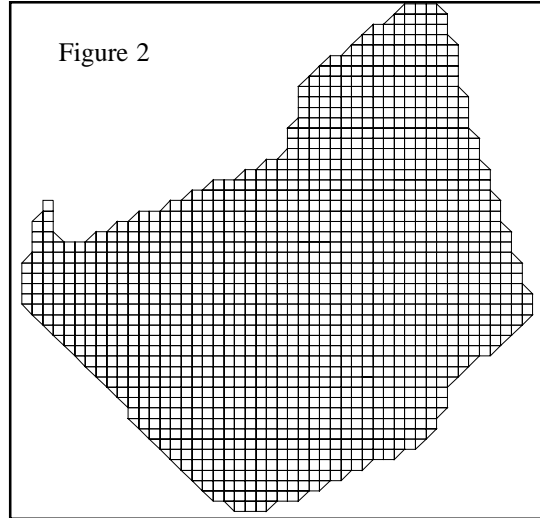
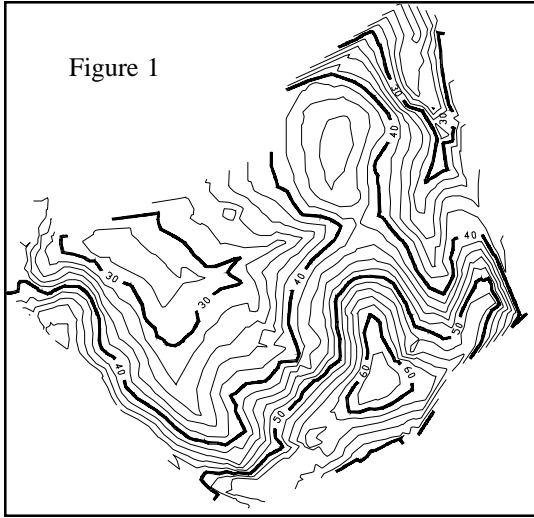
Hatch Slope Zones

Slope Report
Number of 3DFaces analyzed> 1334

Zone	Range	Surface Area S.F.	% of Acres	Total
1	<= 10.00	19902.43	0.46	16.37
2	> 10.00	38169.69	0.88	31.39
3	> 20.00	35833.72	0.82	29.47
4	> 30.00	19622.12	0.45	16.14
5	> 40.00	8082.44	0.19	6.65

Total Surface Area> 121610.41
Average Slope > 21.87
Minimum Slope%> 0.00 Maximum Slope%> 66.77

Slope Zone Analysis report



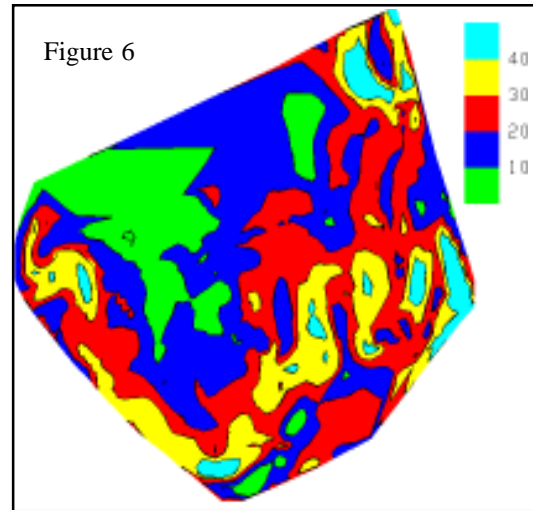
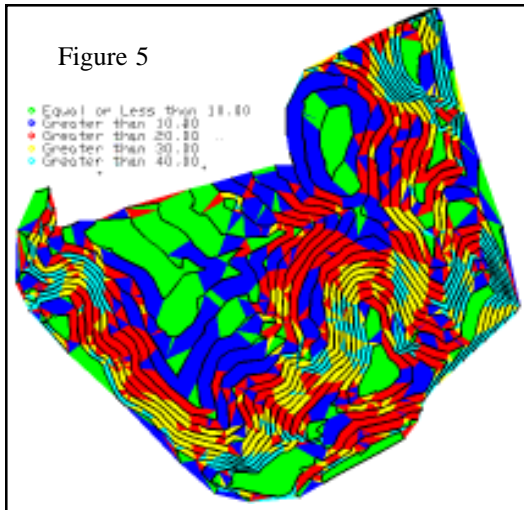


Figure 1: Surface contours

Figure 2: 3D Faces from a grid surface model created by Make 3D Grid File and drawn by Plot 3D Grid File.

Figure 3: 3D Faces created by Triangulate & Contour with the Draw Triangulation Faces option.

Figure 4: Slope zone contours created from the grid 3D Faces.

Figure 5: Slope zones that follow the surface contours using the triangulation 3D Faces.

Figure 6: Hatched slope zone contours created from the grid 3D Faces.

Slope Report

Function

This command calculates the sloped surface area, average slope and average elevation on a site. The surface can be defined by a grid (.GRD) file or generated from 3D entities on the screen.

For the screen method, the surface is modeled from the user-selected entities such as contour polylines. Besides the surface entities, a perimeter polyline is used as the inclusion area for the slope report. The perimeter polyline must be a closed polyline in the PERIMETER layer.

For the grid file option, there are options to specify inclusion and exclusion perimeters. When inclusion perimeters are specified, only the area within the inclusion perimeters is calculated. The area within exclusion perimeters is not included in the calculations. Inclusion and exclusion perimeters are represented by closed polylines and must be drawn prior to calling this routine.

While Slope Zone Analysis also reports slope area, this routine works from a .GRD file instead of 3D Face entities, allows both inclusion and exclusion areas, and is faster.

Prompts

Source of surface model (<File>/Screen)? S for Screen

For Screen method:

Ignore zero elevations (<Yes>/No)? *Press Enter*

Select surface entities and perimeter.

Select objects: *pick the objects*

For File method:

Select the GRiD file

Choose a .GRD file of the site.

Select the Inclusion perimeter polylines or ENTER for none: *Pick any inclusion polylines*

Select the Exclusion perimeter polylines or ENTER for none: *Pick any exclusion polylines*

Pull-Down Menu Location: DTM

Prerequisite: A .GRD file created by Make 3D Grid File

Keyboard Command: sarea

File Name: \lsp\sloparea.lsp & \lsp\volcalc.arx

```
AverageSlopeReport
Processing GRiD file: C:\SC14\DATA\FINAL.GRD
Lower left grid corner : 4942.02,4821.97
Upper right grid corner: 5212.70,5130.76
X grid resolution: 50, Y grid resolution: 50
X grid cell size: 5.41, Y grid cell size: 6.18
Horizontal Area: 55992.227 sq ft, 1.285 acres
Slope Area: 56315.725 sq ft, 1.293 acres
Average Elevation: 1056.252
Average Slope: 9.011
```

Slope Report

3D Slice-Projection

Function

To use this command first plot the strata meshes that you would like to take a slice of using the *Plot GRiD File* command. For best results each strata mesh should be plotted on a different layer (set to a different color). Then from *Plan* view start this command and select the corners (points) of the slice box-line. The command then trims the faces outside the slice box and draws-projects faces down from strata to strata. Shift the *VPOINT* to a 3D view and use the *SHADE* or *HIDE* commands to 'Solidify' the strata model. Figure 5-15 shows the selection points.

Prompts

Select 1st corner of slice box-line: (*select point*)

Select 2nd corner of slice box-line: (*select point*)

You can select any number of the rows and columns from the mesh. The command then processes the selected 3DFaces.

Pull-Down Menu Location: DTM

Prerequisite: Plot a least two .GRD files or 3D Rectangular Meshes.

Keyboard Command: 3dslice

File Name: \isp\mcsllice.lsp & \isp\contour4.arx

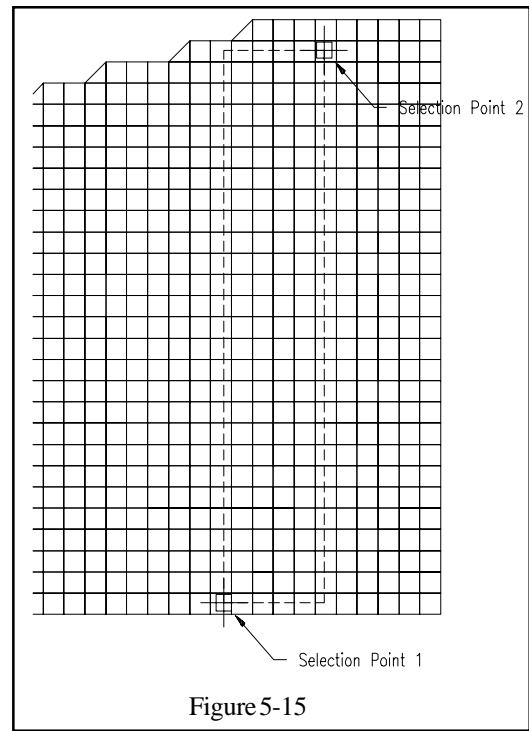
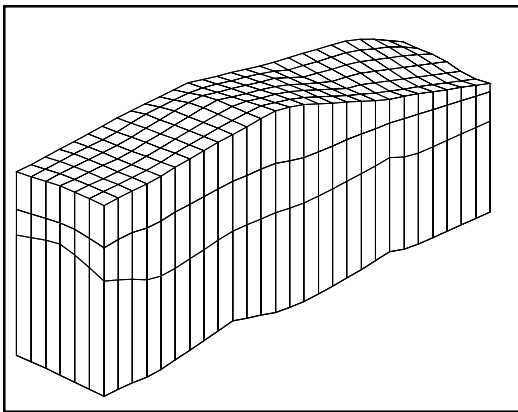


Figure 5-15

Draw Grid Boundary

Function

This command creates a polyline around the outside perimeter of the area defined by a grid file. This feature is useful to show the area that a grid covers without having to run the Plot 3D Grid command which draws all the grid cells and is much slower.

Prompts

Enter the polyline layer <GRID>: *Press Enter*
Select Grid File dialog
Reading cell> 10201
Done.

Pull-Down Menu Location: DTM

Prerequisite: A .GRD file

Keyboard Command: grdlimit

File Name: \lsp\cntr_grd.arx

Draw Grids Intersection

Function

This command compares two grid files and creates polylines where the grid intersect each other. For example, the two grids could represent existing and final surfaces and the intersection polylines would represent the daylight lines. Another example would be to have one grid for the ground surface and the other grid for a strata surface. Then the grids intersection would represent the strata outcrop. The grids should have the same X,Y location and resolution. The intersection polyline can be drawn in 3D at the elevation of the intersection or in 2D at 0 elevation. The Smooth Polyline and Reduce Vertices options are described in the Triangulate & Contour function.

Pull-Down Menu Location: DTM

Prerequisite: Two .GRD files

Keyboard Command: grdlimit

File Name: \lsp\grdcross.lsp, \lsp\tri4.dcl & \lsp\makegrid.arx



Contour Commands

Shown below is the Contour pull-down menu. The commands in the top section allow you to create contours. The middle section commands annotate, smooth, and highlight index contours. The final section has commands for digitizing contours.

Triangulate & Contour is the principle routine in this pull-down. In addition to triangulation and contouring, this command also can optionally do the contouring embellishments of the middle section all in one step.



Triangulate & Contour

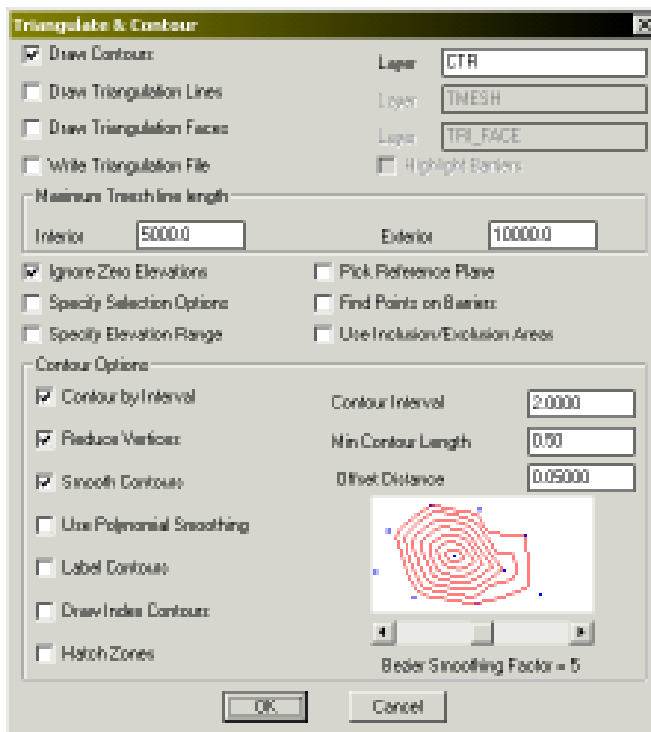
Function

Triangulate & Contour provides all the functionality related to contouring in one routine. Given data points that represent the surface, this command can create a final contour map with labeled, smoothed, and highlighted contours. The data points can be points, inserts, lines, polylines, and points from ASCII or .CRD files. In order to force Triangulate & Contour to interpolate elevations between two points that define a straight line in the surface such as points on a ridge, wall, or road, a barrier line must be between the points. A barrier line can be specified as a 3D polyline or line. In fact, all lines and polylines are treated as barrier lines.

Triangulate & Contour has many options which are specified in the dialog box shown in the dialog below. With this command, you can do any combination of drawing the triangulation network lines, drawing the contours, drawing triangulation network 3D Faces, and writing a triangulation file. The triangulation options are in the top section of the dialog while the contouring options are at the bottom. All the Triangulate & Contour options can be set in Configure SurvCADD as defaults. Configure SurvCADD also has an *Auto-run Contour* option which skips the dialogs when running Triangulate & Contour.

If Triangulate & Contour reports zero points found and fails to do anything when you're using

Carlson points, then those points are probably located at zero elevation. To fix this problem, you can either choose *Specify Selection Set Options* and then select *SurvCADD Point Inserts* which will read the elevation from the elevation attribute of the point. Or you can go to *Drawing Setup* and select *Locate on Real Z Axis* and then call *Locate Points* to replot the points.



Prompts

Triangulate & Contour Options Dialog Box

All the options for this command are controlled by this dialog.

Draw Contours

When this box is marked, the program will draw contour lines after triangulating. Otherwise, only the triangulation is performed.

Contour Layer

This setting is the name of the

layer in which the contour polylines will be drawn.

Draw Triangulation Lines

When this box is marked, the program will draw the triangulation as 3D lines.

Triangulation Lines Layer

This setting is the name of the layer in which the triangulation network lines will be drawn.

Draw Triangulation Faces

When this box is marked, the program will draw each triangle in the triangulation network as a 3D Face. These 3D Faces can then be used in AutoCAD's modelling routines such as HIDE and SHADE. 3D Faces are also used by the Slope Zone Analysis routine.

Triangulation Faces Layer

This setting is the name of the layer in which the triangulation network faces will be drawn.

Write Triangulation File

This option stores the triangulation surface model as an .flt file which is a text file of the edges in the triangulation network. This triangulation file can be used by several commands such as Volumes By Triangulation, Spot Elevations, and Profile from FLT File. The .flt file is in a Softdesk fault file format. To create a Softdesk TIN from the fault file, go to the TIN menu and choose Fault Data, From File. Give a surface name as usual and then choose the fault file from Carlson. Then choose Create TIN. Check only the Fault Data box and click OK.

Highlight Barriers

This option highlights barrier lines in the triangulation network by drawing the triangulation lines along barriers in yellow.

Max Triangulation Line Length

This value limits the length of the triangulation network lines. Any triangulation line that exceeds this limit will not be drawn or included in contouring. This allows you to avoid abnormally long triangulation lines where you have relatively too few data points and on the outskirts of your data points. The **Exterior** value applies to triangulation lines around the perimeter of the triangulation area and the **Interior** value applies all the other triangulation lines. Generally you would have the exterior value smaller than the interior.

Ignore Zero Elevations

When activated, this setting will filter out all data points at an elevation of zero from the data set.

Pick Reference Plane

The triangulation network is based on the x,y position of the points. This option allows you to contour an overhang or cliff by changing the reference plane to a side view. The reference plane can be specified by first using the Viewpoint 3D command and then using the View option, or you can specify three data points on the cliff (two along the bottom and one at the top).

Specify Selection Set Options

This toggle allows you to control what type of entities Triangulate & Contour uses. After you click OK on this dialog, the next dialog will be the Selection Set Options dialog box shown in Figure 5-18.

Find Points on Barriers

This option will attempt to locate intersection points on long barrier lines. For example, the intersection points of side slopes with a top of bank barrier line.

Specify Elevation Range

The program will automatically contour from the lowest elevation in the data set up to the highest at the increment specified in Contour Interval. If you would like to manually set the range over which to contour, select this option.

Use Inclusion/Exclusion Areas

When this box is activated, the program will later prompt you for inclusion and exclusion polylines which are used to trim the contours. The inclusion and exclusion polylines must be closed polylines and must be drawn before starting Triangulate & Contour. Only the parts of the contour lines that are within the inclusion polylines will be drawn. For example, an inclusion could be the perimeter of the site. The parts of contour lines that are inside the exclusion polylines are not drawn. Exclusion polylines can be used for areas where you don't want contours such as within buildings.

Contour by Interval

This option switches between contouring by interval (ie every 10 feet) or by value. The by value option allows you to contour specific values. For example, if you want just the 100ft contour, then do contour by value and enter 100. The default mode is by interval.

Contour Interval

This value determines the increment between contour lines. If Contour by interval is turned off, this value will correspond to the contour value as described above.

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

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.

Minimum Contour Line Length

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

Smooth Contours

This option activates smoothing on the contour lines. Bezier smoothing holds all the contour points calculated from the triangulation and only smooths between the calculated points. The smoothing factor described below affects the smoothing bulge.

Use Polynomial Smoothing

Polynomial smoothing applies a fifth degree polynomial for smooth transition between the triangulation faces.

Label Contours

This selection leads to the Labeling Options dialog box shown in Figure 5-20 after OK is selected on the current dialog box.

Draw Index Contours

This option allows you to highlight contours at a specified interval. The dialog box shown in Figure 5-19 will be activated.

Hatch Zones

This option will hatch the area between the contours sequentially.

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 has 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.

Choose Types to Triangulate Dialog

This dialog sets what types of entities are to be included in triangulation (see Figure 5-18). This dialog only appears if the Specify Selection Options in the main dialog is checked.

Points, 3D Polylines, 2D Polylines, Lines, Inserts

These are the standard AutoCad entities that contain elevations.

Carlson Inserts

This option will include Carlson point inserts such as the insert SRVPNO1 with the point number, elevation, and description attributes.

Bottom Elevation Inserts

This option will read the elevation in text entities that start with 'X'. These text entities are created by the Bottom Elevation routine in the Mine module.

From File

This option allows you to triangulate from the points in a .CRD or ASCII file.

Value to Process

This allows you to choose what value to contour from the coordinate file. The available keywords are Z for the coordinate elevation, DESC for the point description and NOTE for the point note fields. The NOTE keyword can be followed by a note line number. For example, to contour the third line from the note field, use NOTE3. Equations can also be entered in this field. For example, (Z+DESC)/2.0.

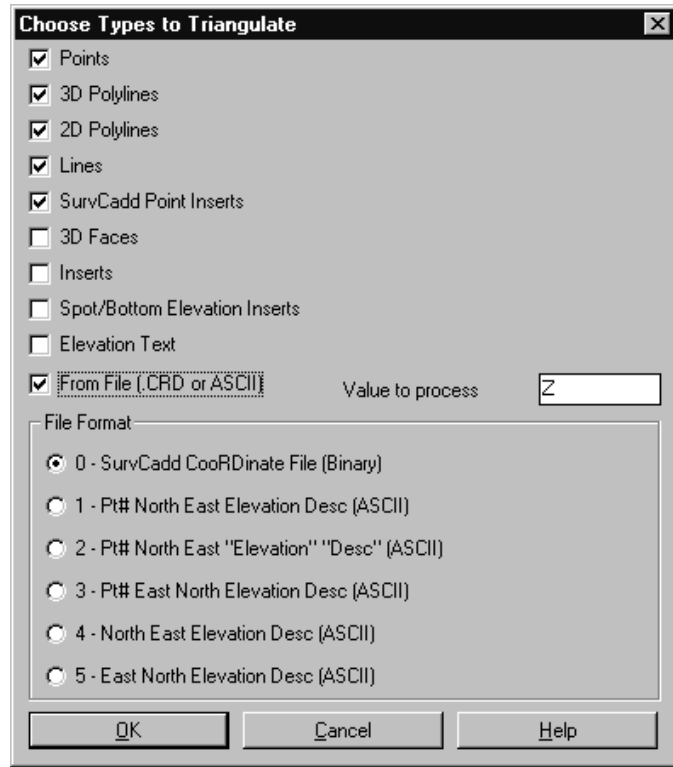
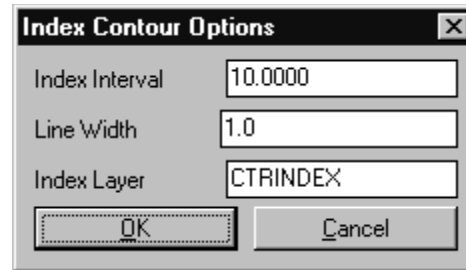


Figure 5-18

Figure 5-19



Index Contour Options

This dialog (see Figure 5-19) is activated if the Draw Index Contours option is set. The settings here specify the contour interval, line thickness, and layer name for the index contours.

Contour Label Options

This dialog appears if the Label Contours option is on (see Figure 5-20).

Pick the clip edge polyline: *pick a polyline.* This must be a closed polyline. It serves as the trim edge when drawing the contour lines. This prompt only appears if the Use Inclusion Perimeter option is set.

Select the point and barrier lines to Triangulate.

Select object: *select the objects that represent the surface*

Reading points 53 ...

Triangulating points 53 ...

Starting contour elevation

<542.0000>: *Press Enter.* This value is the elevation to begin contouring at. The default is the lowest elevation in the triangulation network. This prompt appears if the Specify Starting & Ending Elevations option is on.

Ending contour elevation

<597.2000>: *Press Enter.* This value is the highest elevation to contour.

Contouring elevation 597

Inserted 1459 contour vertices.

Pull-Down Menu Location:

Contour

Prerequisite: Data points of the surface

Keyboard Command: tri

File Name: \lsp\tri4.lsp, \lsp\tri4.dcl, \lsp\tri4.arx

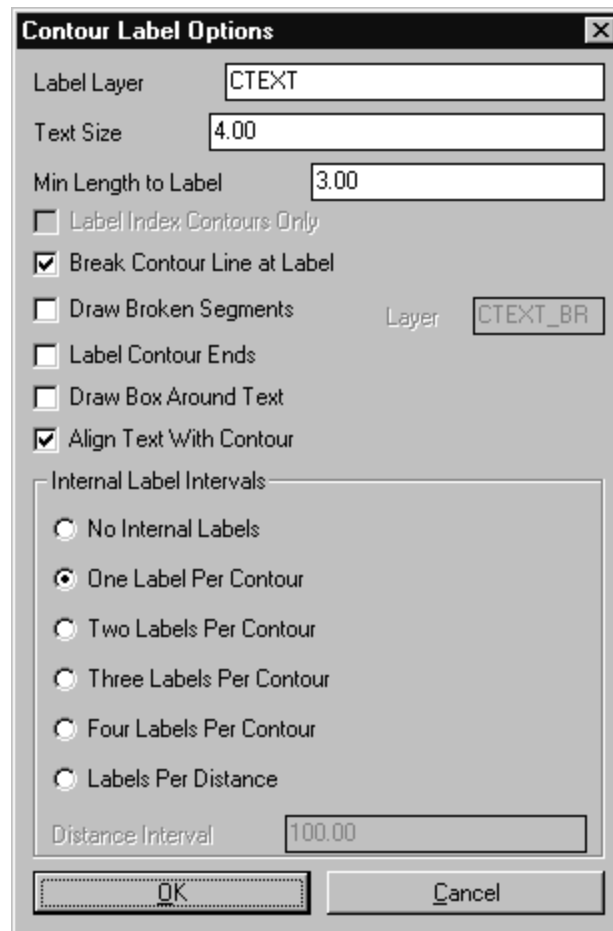
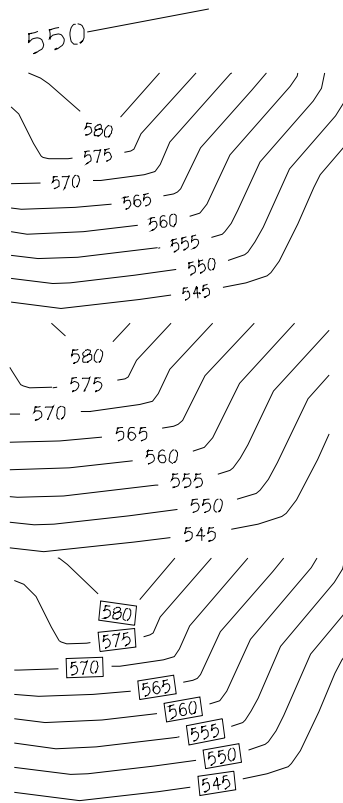


Figure 5-20

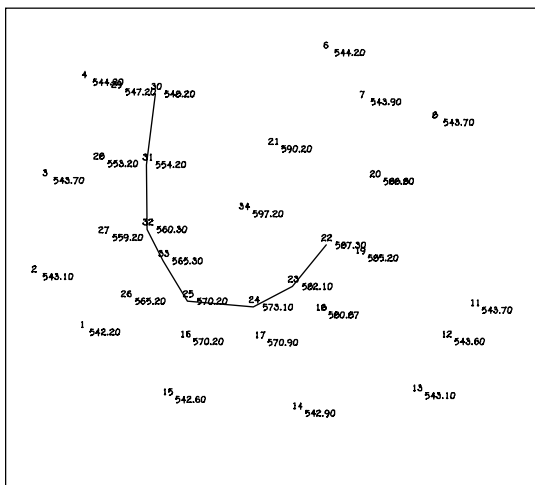


Label Contour Ends

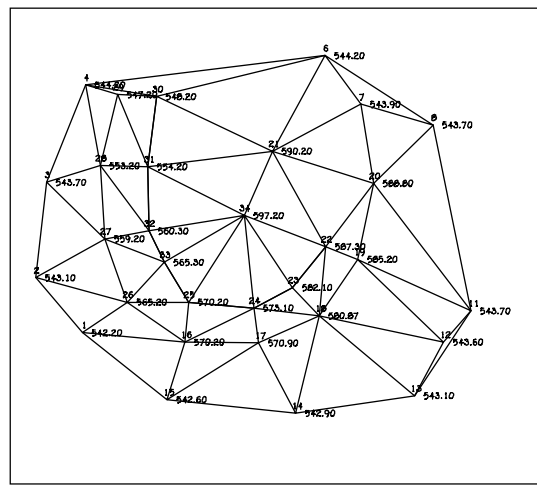
Align Text With Contour ON

Align Text With Contour OFF

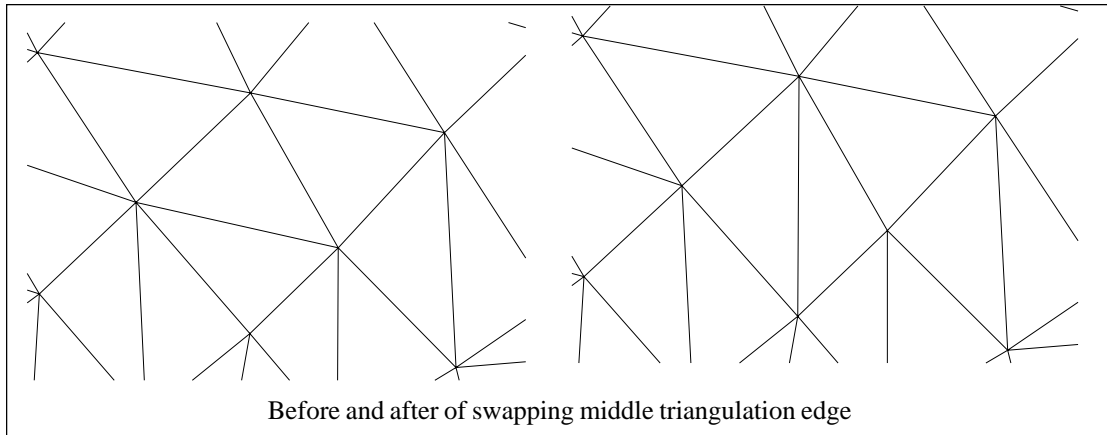
Draw Box Around Text



Original data points with one 3D polyline



Triangulation network without contouring



Draw Triangular Mesh

Function

This command draws 3D lines for a triangulation that is stored in a triangulation file (.flt). The triangulation file can be created by Triangulate & Contour.

Prompts

Select T_MESH File to Draw
Choose a triangulation file (.flt)
Layer name <T_MESH>: *Press Enter*
Drawing edges ... 2515

Pull-Down Menu Location: Contour

Prerequisite: A triangulation file (.flt)

Keyboard Command: drawtri

File Name: \lsp\contour4.arx

Contour from Triangular Mesh

Function

This command draws contour lines from a triangulation network drawn by Triangulate & Contour. There are not as many contouring options here as there are in Triangulate & Contour. Creating contours with this command makes contouring a two step process. First draw the triangulation network and then create contours from this triangular mesh. The main advantage to contouring this way is that you may modify the triangular mesh before contouring.

Prompts

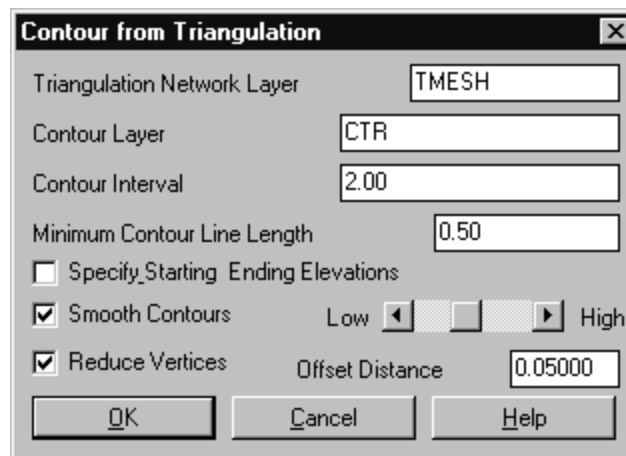
Contour from Triangulation dialog box
Select the triangulation lines.
Select objects: *pick the tmesh lines*
Contouring elevation 590
Inserted 900 contour vertices.

Pull-Down Menu Location: Contour

Prerequisite: A triangulation network drawn by Triangulate & Contour

Keyboard Command: contour

File Name: \lsp\contour4.lsp, \lsp\contour4.arx



Contours from Grid File

Function

This command draws contour polylines from a surface model defined by a grid file. Contouring from a grid employs a different method than from a triangulation network and generally produces contours that loop more. The grid has data points at a regular interval while the triangulation has edges for every point and break line in the surface. The smoothness of the contours depends on the grid resolution. The Smoothing Sub-Division option will internally subdivide the grid cells with a quadratic smoothing algorithm to help create smoother contours. The Hatch Zones option will fill in the specified elevation ranges with hatch patterns or solid color fills. The Create Polyline Topology will create closed polylines for each contour range and will draw a zone text label within each area. This polyline topology can be used in GIS routines such as Polygon Processor.

Prompts

Select the Inclusion perimeter polylines or ENTER for none.

Select objects: *pick a closed polyline for the contour boundary if any*

Select the Exclusion perimeter polylines or ENTER for none.

Select objects: *pick a closed polyline for the area to exclude*

Grid File to Process dialog *Select a .grd file*

Contour from Grid File options dialog

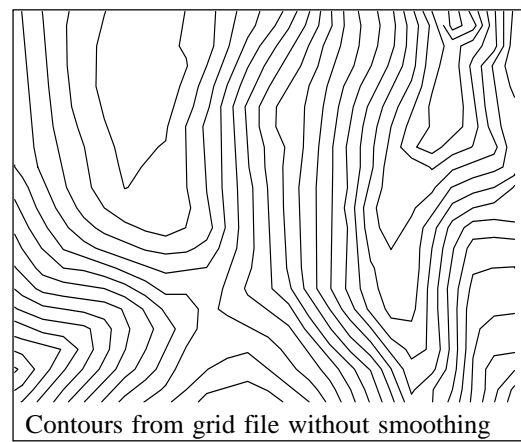
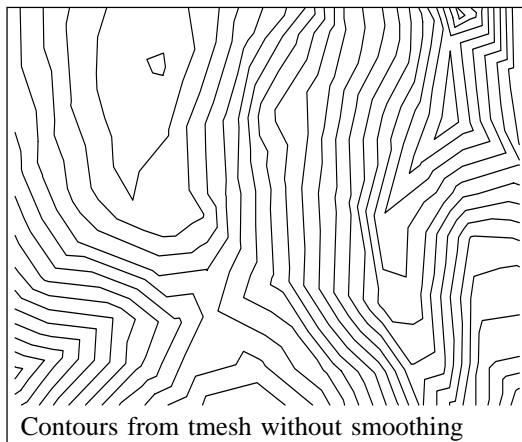
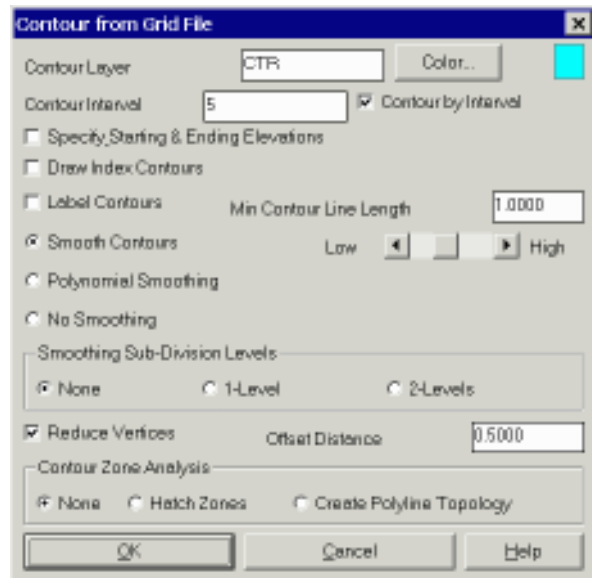
Extrapolate grid to full grid size (Yes/<No>)? *Press Enter.* This prompt appears if your grid extends beyond the limits of your data points in some areas.

Pull-Down Menu Location: Contour

Prerequisite: A grid file created by Make 3D GRiD File

Keyboard Command: cntgrd

File Name: \lsp\contour4.lsp,
\lsp\cntr_grd.arx



Smooth Contours

Function

This command has options for applying smoothing to polylines. Select the radio button for the smoothing option you want to apply. If you use quadratic smoothing or cubic spline type smoothing the spline segments setvar is relevant. The Curve fit option provides the least smoothing, and the Cubic spline option applies the most. Another effective way of smoothing is by creating the contours from rectangular meshes using various grid resolutions. Increase the smoothing by lowering the grid resolution and decrease by raising the grid resolution. The Bezier option provides an incremental type of smoothing but can not be decurved. The Linetype Generation option allows you to apply the pedit switch for Ltype Gen to a group of polylines. For more information on this option and the spline smoothing options see the AutoCad Reference Manual on the *PEDIT* command. After selecting the *OK* button the program prompts. Figure 5-29 shows contours on the left un-smoothed and on the right smoothed with the cubic spline method.

Prompts

First corner: *(pick point)* **Other corner:** *(pick point)*

Select the polylines that represent the contour lines.

Select objects: *[Enter]*

Select Plines to spline smooth.

Select objects: *C*

First corner: *(pick point)* **Other corner:** *(pick point)*

Select the polylines to smooth.

Select objects: *[Enter]*

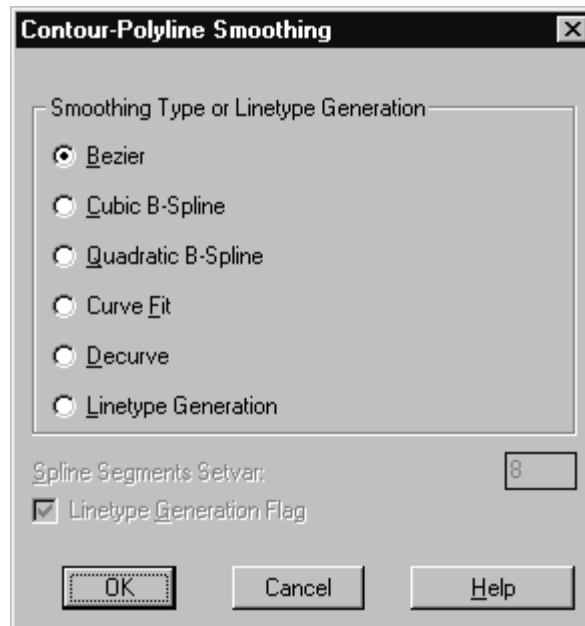
The program then applies the specified option to the selection set.

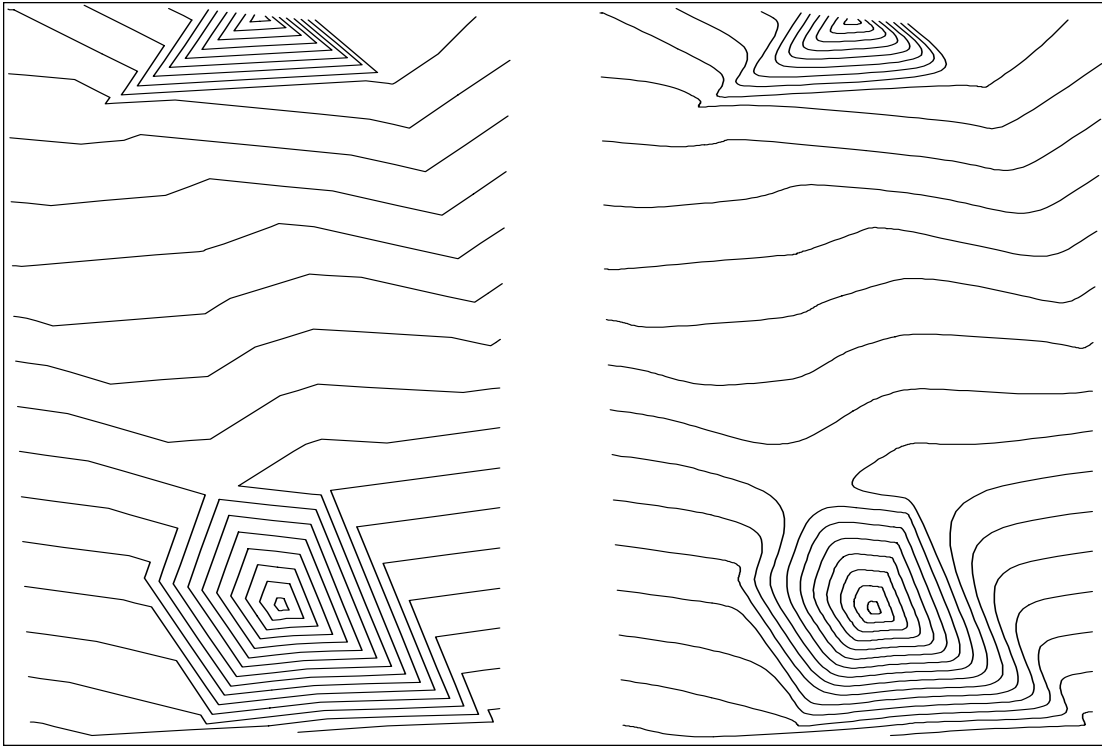
Keyboard Command: Smooth

Pull-Down Menu Location: Contour

Prerequisite: Create Contour lines to smooth.

File Names: \lsp\scadcntr.dcl,
\lsp\scontour.lsp, \lsp\tri4.arx





Cubic-smoothing result

Bezier Smoothing

Function

Besides running separately in the Smooth Contours routine, Bezier Smoothing is also embedded in many of the routine that create contours. Bezier Smoothing applies the Bezier smoothing algorithm to polylines. This smoothing technique has two advantages over Spline or Curve-Fit smoothing. One is that a Bezier smoothed polyline will pass through all of the vertices in the original polyline, while a Spline smoothed polyline only curves towards the original vertices and can pull away from vertices at sharp corners. Hitting all the original vertices can be an important feature in contour maps for maintaining the exact location of the contours. The other benefit of Bezier is the user's ability to control the looping and vertex factors. A higher looping factor increases the curving effect. Vertices reduction can also be applied along with the smoothing. This avoids having to create smoothed polylines with numerous vertices and then having to reduce these vertices in a second step. Be sure not to make the cutoff offset for reduction too high or you can negate or even reverse the smoothing effect (see Figures 5-30 - 5-33).

A drawback to Bezier is that it cannot be "decurved" like the other smoothing techniques. Also, be careful with the looping factor on contour lines because a high looping factor may cause close contour lines to cross.

Prompts

Enter the looping factor (1-10) <5>: *Press Enter.* This determines the extent of curving. 1 - least curvy, 10 - most curvy.

Enter the offset cutoff <0.05>: *Press Enter.* This value is the maximum shift distance for vertices reduction. A higher value removes more vertices.

Select polylines to smooth.

Select objects: *Pick polylines*

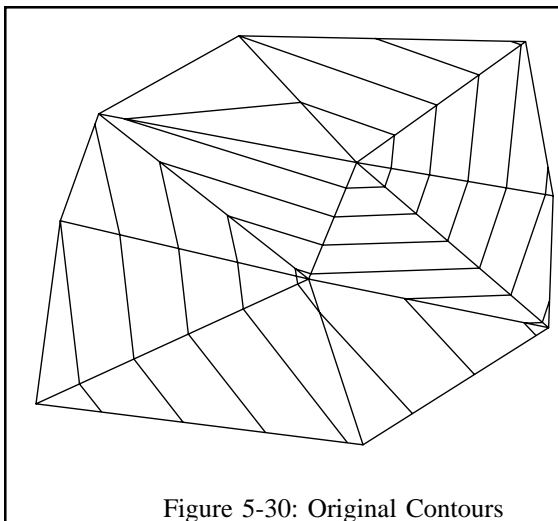


Figure 5-30: Original Contours

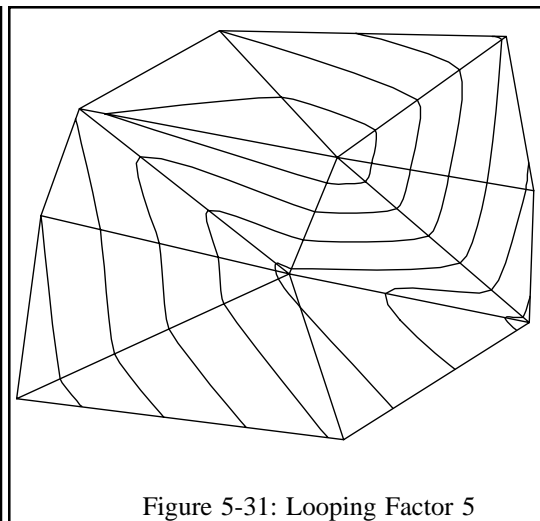


Figure 5-31: Looping Factor 5

Pull-Down Menu Location: Bezier Smoothing is accessed through the *Smooth Contours* command on the Contours pull-down in the DTM & Contour menu.

Prerequisite: Create Contour lines to smooth.

File Names: \lsp\scadentr.dcl, \lsp\scontour.lsp, & \lsp\tri4.arx

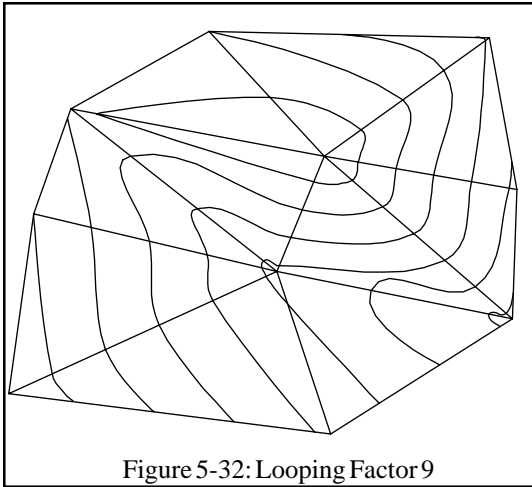


Figure 5-32: Looping Factor 9

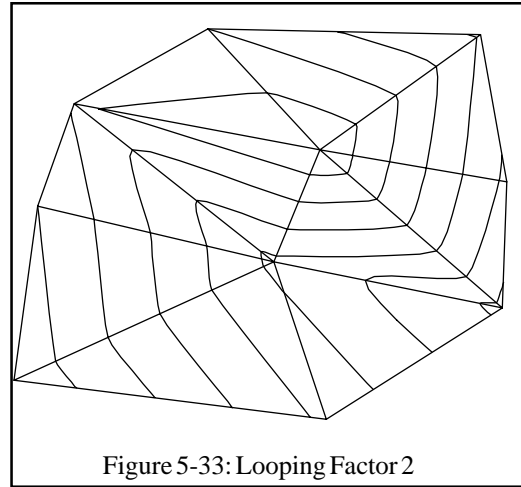


Figure 5-33: Looping Factor 2

Reduce Vertices

Function

Carlson's contouring and smoothing often creates an explosion in file size due to the many vertices it adds. Fortunately, many of these vertices are very close together and some can be removed with no visible effect on the contour line drawing. Reduce Contours removes these extra vertices which can reduce the total number of vertices by 30% to 90%. This has the benefits of a smaller drawing file, faster drawing loading, and faster regens.

Reduce Contours removes vertices in a polyline that are within a user specified offset cutoff. The algorithm looks at three vertices at a time, and calculates the distance between the second point and the line from the first to the third point (see Figure 5-34). If this distance is less than the user specified cutoff, the second point is removed. In theory, reducing the polyline vertices should not shift the polyline more than the user's cutoff distance. The default for this cutoff is one tenth of a foot. Increasing the cutoff will remove more vertices while decreasing it will more closely preserve

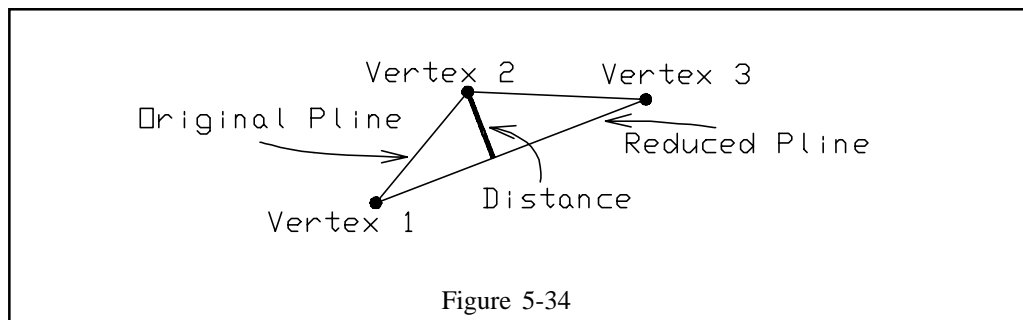
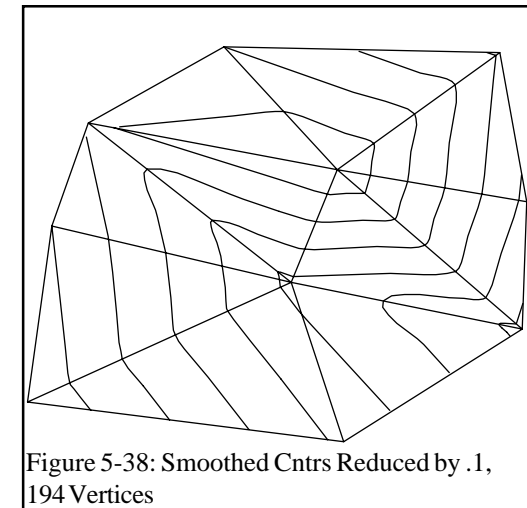
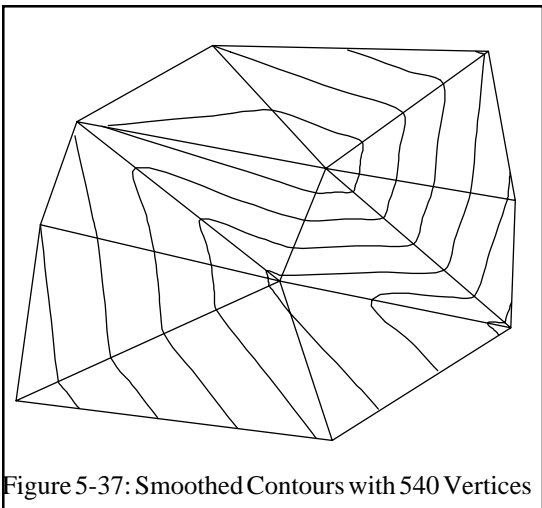
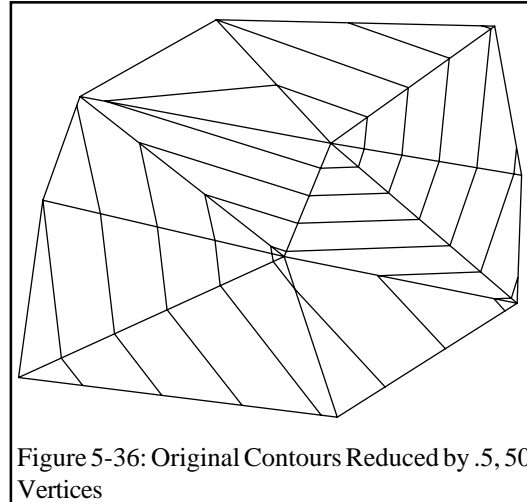
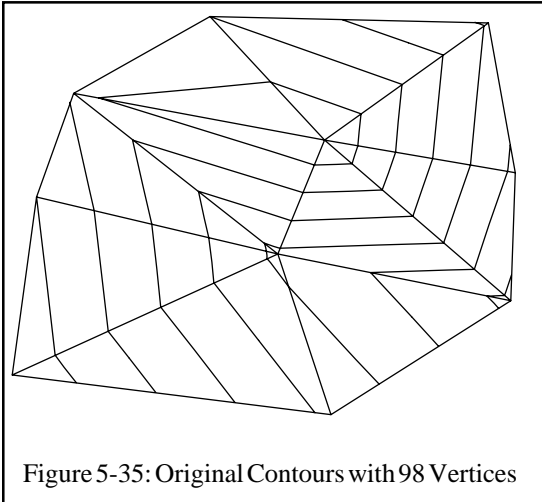
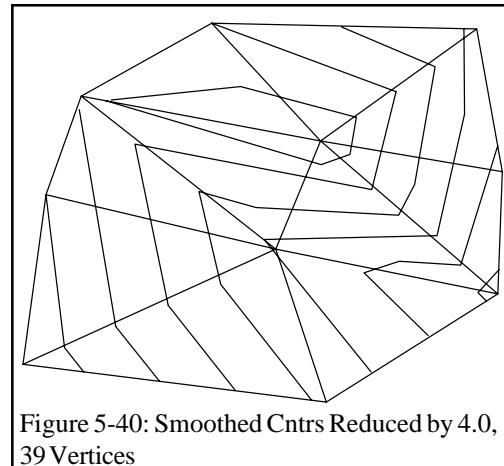
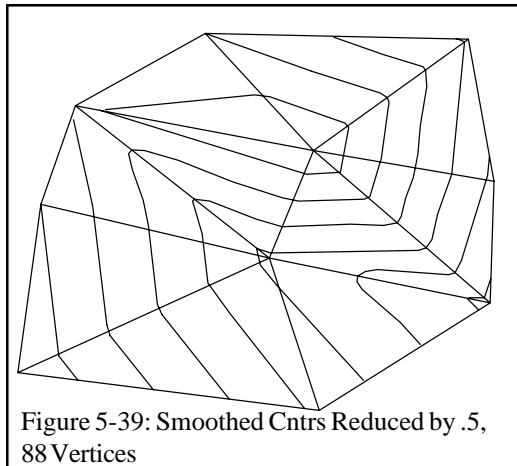


Figure 5-34

the original contour line. Figures 5-35 through 5-40 contain examples of different cutoff values. When combining vertex reduction with smoothing, it is suggested to smooth before reducing, although it can be done the other way around.





Pull-Down Menu Location: Contour

Prerequisite: Create Contour lines to reduce.

Keyboard Command: reduce

File Names: \isp\scadcntr.dcl, \isp\scontour.lsp, & \isp\reduce.arx

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*

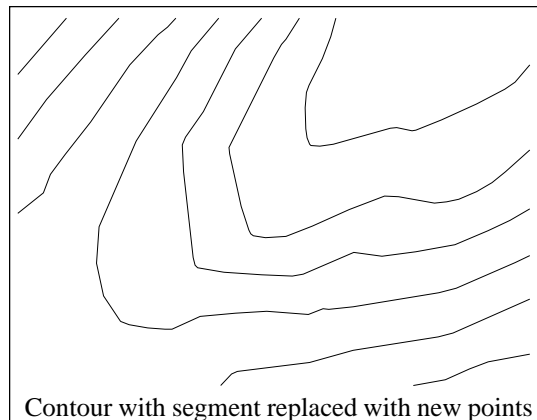
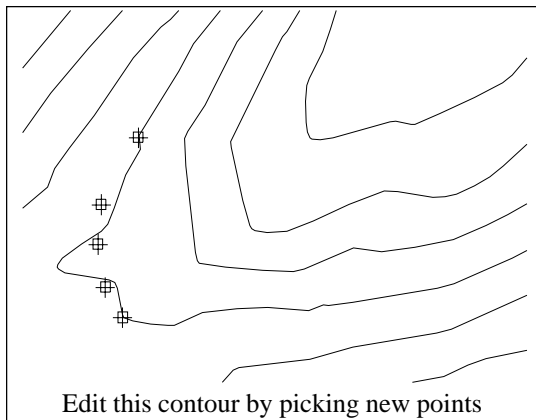
Pick reconnection point on contour: *pick the contour polyline at the place to join*

Keyboard Command: editctr

Pull-Down Menu Location: Contour

Prerequisite: polylines with elevation (contour polylines)

File Name: \isp\poly3d.arx



List Elevation

Function

This command displays the elevation of a polyline or line. With a 3D polyline, the elevation of the 3D polyline at the pick point is reported along with the elevation of each vertex.

Pull-Down Menu Location: Contour

Keyboard Command: LSTELEV

Prerequisite: a polyline or line

File Name: \lsp\lstelev.lsp

Contour Elevation Label

Function

This command can be used to label a group of contour lines with their elevation. To place the labels, pick two points across the contour polylines. The program will find all the contour polylines that intersect the line defined by the two picked points and the labels will be drawn to the intersection points. The text size is a scalar that will be multiplied by the horizontal scale to set the text size in AutoCad units.

Prompts

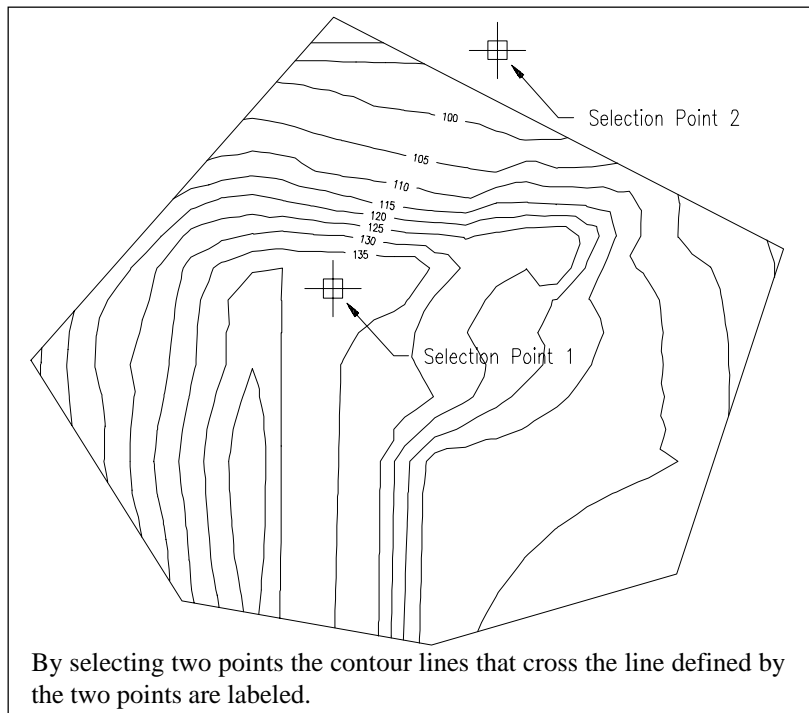
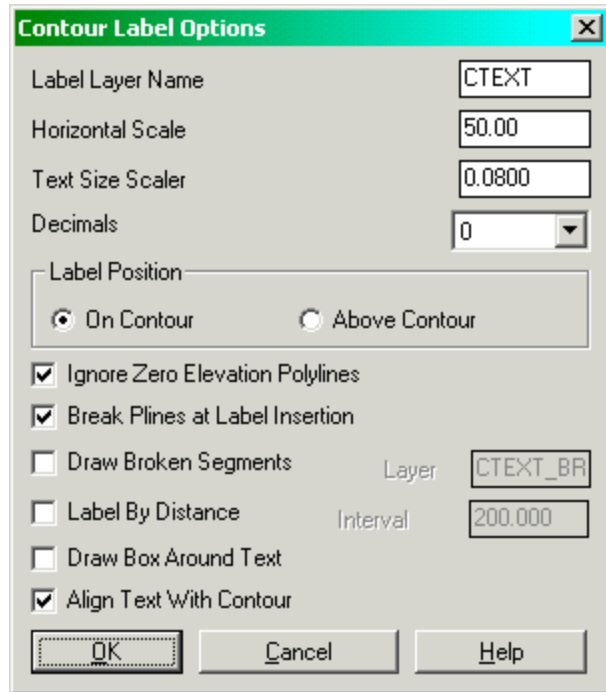
Global Elevation Dialog

Define a line which slices the contours at the place to label them.

Pick 1st point: (*pick point*)

Pick 2nd point: (*pick point*)

Keyboard Command: gclabel
Pull-Down Menu Location: Contour
Prerequisite: polylines with elevation
(contour polylines)
File Name: \isp\gclabel.lsp &
\isp\contour4.arx



Local Elevation Label

Function

This command allows the user to place elevation labels on contour lines or other entities with elevations. The command prompts for two points then centers the label in between. It then prompts for two break points which are used to erase/break the contour that runs through the elevation label. If a break is not required just press [Enter] at the break point prompts.

Pull-Down Menu Location: Contour

Keyboard Command: clabel

Prerequisite: None

File Name: \lsp\labcont.lsp

Flip Contour Labels-Text

Function

This command individually rotates each of the selected text entities by 180 degrees.

Pull-Down Menu Location: Contour

Keyboard Command: fliptext

Prerequisite: Text labels on the contours.

File Name: \lsp\fliptext.lsp

Highlight Index Contours

Function

This command will freeze a layer of contour lines and copy a specified interval to another layer. This allows the user to change the color or width of a certain interval.

Prompts

Layer name of existing contours <CTR>: [Enter]

Layer name for highlight contours <NCTR>: [Enter]

Select Contours to Highlight.

Select objects: W

This is AutoCad's standard selection set prompt, use Window or Crossing.

First corner: (pick point)

Second corner: (pick point)

Select objects: *[Enter]* Enter ends the selection process. The program then sorts and displays the High and Low interval of the selected contours.

Contour increment to highlight: 10

Starting Highlight at elevation <98.0>: 100

Ending Highlight at elevation <152.0>: 150

Assuming we had drawn 1 foot intervals, the above example would copy contours on elevations 100, 110, 120, 130, 140 and 150 to the layer NCTR and freeze the remaining lines on layer CTR.

Keyboard Command: indexctr

Pull-Down Menu Location: Contour

Prerequisite: Contours should be plotted and visible on the screen.

File Name: \lsp\ccolor.lsp

Highlight Depression Contours

Function

This command highlights depression contours by changing their layer, color, and adding tick marks. A depression contour is a closed contour line that leads to a local minimum such that there are no contour lines with a higher elevation within the contour. This routine finds the depression contours out of the selected polylines. The depression contours are highlighted, and the user selects which ones to label.

Prompts

Layer name of existing contours <CTR>: *[Enter]*

Layer name for depression contours <DCTR>: *[Enter]*

Width for depression contours <1.0>: *[Enter]*

Tick Interval for depression contours <50.00>: *[Enter]*

Tick Size for depression contours <6.0>: *[Enter]*

Select the existing contours.

Select objects: Select all the contour polylines, even the contours that aren't depression contours.

Select objects: *[Enter]* Enter ends the selection process. The program then sorts and displays the high and low elevations of the selected contours.

Reading the selection set ...

Locating the depression contours ...

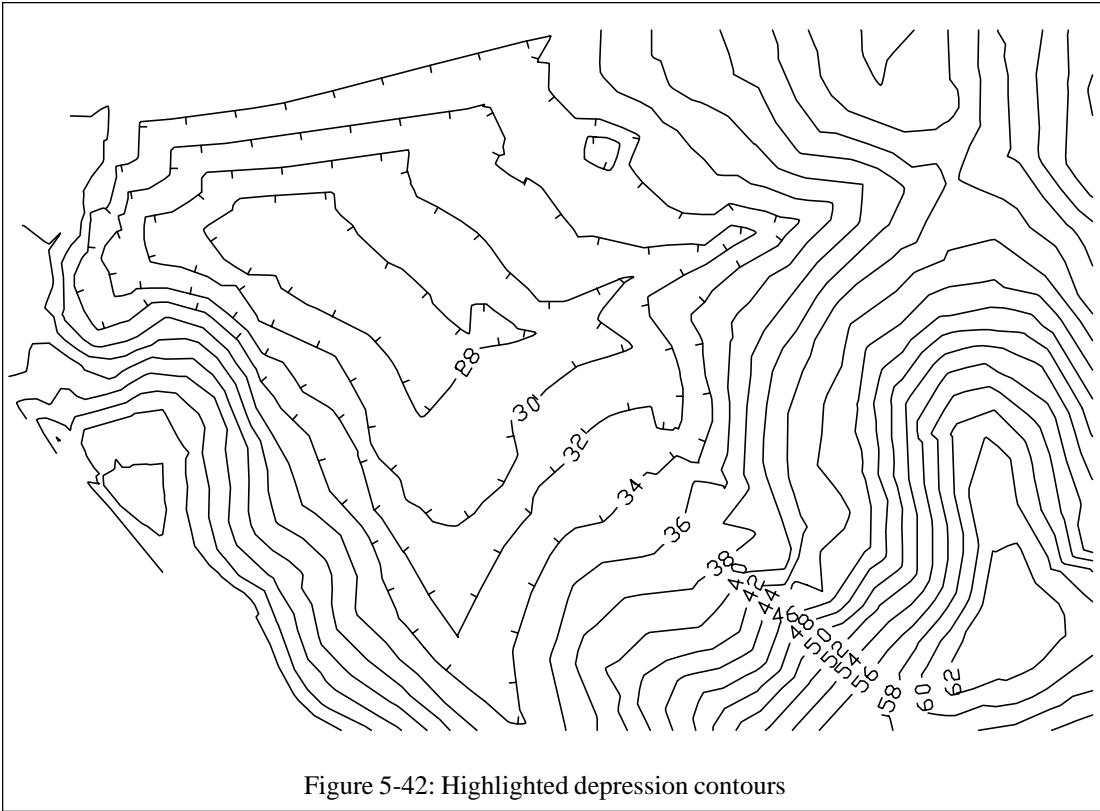
Drawing the depression contours ...

Pull-Down Menu Location: Contour

Prerequisite: Contours should be plotted and visible on the screen.

Keyboard Command: depress

File Name: \lsp\depress.lsp & \lsp\volcalc.arx



Change Contour-Plines Width

Function

This command allows the user to select a group of contours/plines and change their width for emphasis when plotting. For example, use the above command Highlight Contours to freeze unwanted intervals and make visible the contour intervals of interest. Then using this command change them to the desired width.

Keyboard Command: cwidth

Pull-Down Menu Location: Contour

Prerequisite: Contour polylines should be drawn and visible on the screen.

File Name: \lsp\cwidth.lsp

Trim Contour-Plines by Pline

Function

This command can be used to trim a group of contour lines or polyline entities that cross a perimeter defined by a 2D polyline. The trim can be executed on the inside or the outside of the perimeter.

Prompts

Select polyline which represents perimeter: (*select trim perimeter*)

Perimeter has 6 points.

Trim Inside/<Outside>: [*Enter*]

Use the I option to trim polylines inside the defined perimeter. This is useful for deleting contour lines that fall inside a building or some area that you want void of contours.

Select PolyLines and/or 3DPolys to Trim.

Select objects: *W*

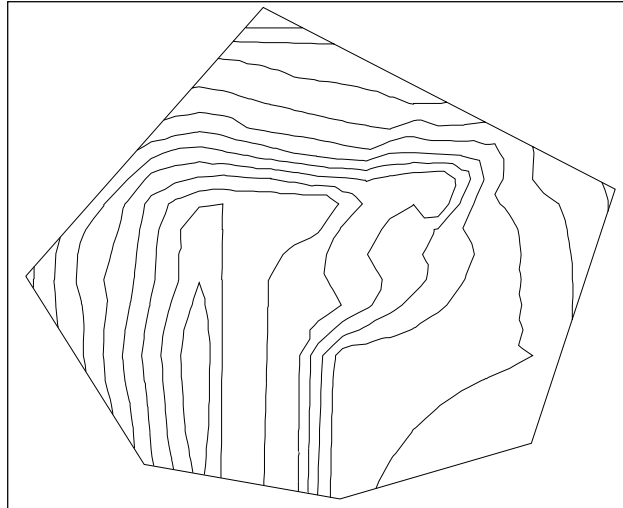
This is AutoCad's standard selection set prompt, use Window or Crossing.

First corner: (*pick point*)

Second corner: (*pick point*)

Select objects: [*Enter*]

[Enter] ends the selection process.



Keyboard Command: polytrim

Pull-Down Menu Location: Contour

Prerequisite: Draw a 2D closed polyline perimeter.

File Name: \lsp\polytrim.lsp

Color Contours by Elevation

Function

This command changes the color of the selected polylines based on the polyline elevation. The color to use is defined in elevation range table.

Prompts

Define Ranges Dialog

Select polylines to color:
pick the polylines

Pick point for color legend:
pick a point to a clear area of the drawing to place a legend or press Enter for no legend

Keyboard Command: ctrcolor

Pull-Down Menu Location: Contour

Prerequisite: Contours polylines

File Name: \lsp\contour4.arx



Assign Contour Elevations - 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*

Pull-Down Menu Location: Contour

Prerequisite: polylines

Keyboard Command: setcelev

File Name: \lsp\poly3d.arx

Group Change Elevation

Function

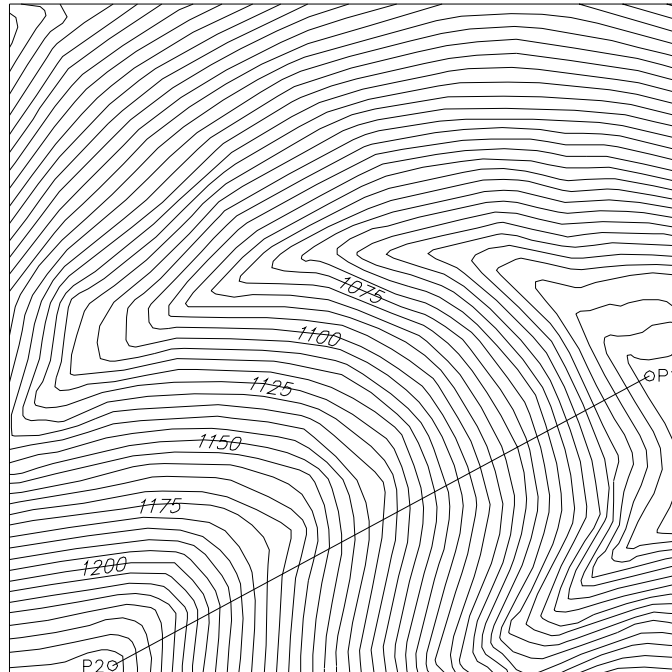
Group Change Elevation can be used to quickly and accurately assign the elevation of series of AutoCAD polylines that have been converted from raster or digitized without correct elevations. The routine will automatically assign elevations to the polylines crossing the fence line selected by two points. 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

- 1) Command: **GRPCELEV**
- 2) Settings/First Point: (*Press S to change settings or pick first point.*)
- 3) Second Point: (*Pick second point*)
- 4) Beginning Elevation <0.00>: **1020**
- 5) Increment Direction U/D <U>: (*enter*)

	Minor	Major
Interval:	5	25
Layer:	CTR	NCTR
Color:	BYLAYER	BYLAYER
Linetype:	BYLAYER	BYLAYER
Width:	0	1.5

OK Cancel Help



Tablet Calibrate

Function

Executes AutoCad's *TABLET* command to set the digitizer to a drawing's coordinate systems. Refer to the AutoCad Reference manual for further information.

Keyboard Command: tablet

Pull-Down Menu Location: Contour

Prerequisite: Affix a drawing to your digitizer tablet.

Auto Tablet On

Function

This command controls whether the commands *Digitize Contours*, *Digitize Sections*, and *Digitize End Areas* automatically turn the tablet to digitizer mode at the appropriate time. If this command is not executed the user is responsible for toggling the tablet on at the point picking prompts.

Prompts

[1] - Turn Tablet On Automatic **[0] - NO Auto Tablet On <0>: /**

Entering 1 enables automatic tablet on mode.

Keyboard Command: digdefs

Pull-Down Menu Location: Contour

Prerequisite: None

File Name: \lsp\digdefs.lsp

Digitize Contours

Function

This command allows you to digitize contours as polylines one at a time. The first time it prompts for a layer, an elevation, and an interval and then draws a polyline by picking points. The command can be repeated by pressing enter at the AutoCad command prompt. When repeated, the command defaults to the last elevation plus the interval and the previous layer. While picking points when you get near the edge of the screen, the program will automatically zoom the display to center around the last point.

Keyboard Command: digcont

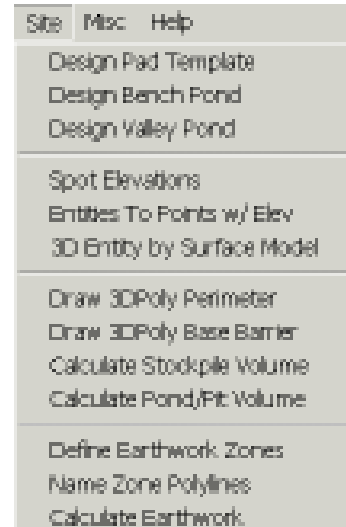
Pull-Down Menu Location: Contour

Prerequisite: Calibrate the tablet with the *Tablet Calibrate* menu option if contours are to be taken off an existing drawing.

File Name: \lsp\digcont.lsp & \lsp\contour4.arx

Site Commands

The Site menu offers commands that assist in site design, stake out, and volumetrics. The Site pull-down menu is shown to the right.

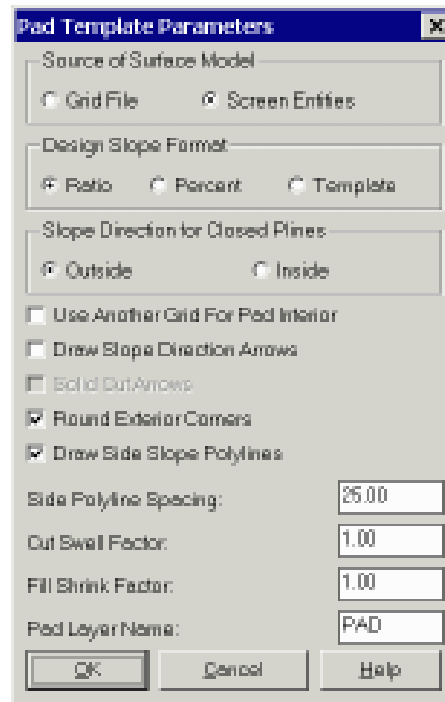


Design Pad Template

Function

This command creates design slopes from a perimeter polyline at specified cut/fill slopes to reach existing ground. This routine can be used to design building pads, pits, roads, ditches, stockpiles, etc. The design is drawn as 3D polylines for the cut/fill slopes and for the daylight perimeter where the design meets existing ground. Before beginning this routine, you must have drawn the polyline perimeter. The polyline perimeter can be either a 2D or 3D closed or unclosed polyline. For a 2D polyline, the program will prompt for an elevation for the pad perimeter. With a 3D polyline, the pad perimeter is set to the elevations of the 3D polyline. For an unclosed polyline, the program will prompt for the side for the design. With a closed polyline, the program designs the slopes either outward or inward depending on the setting in the dialog.

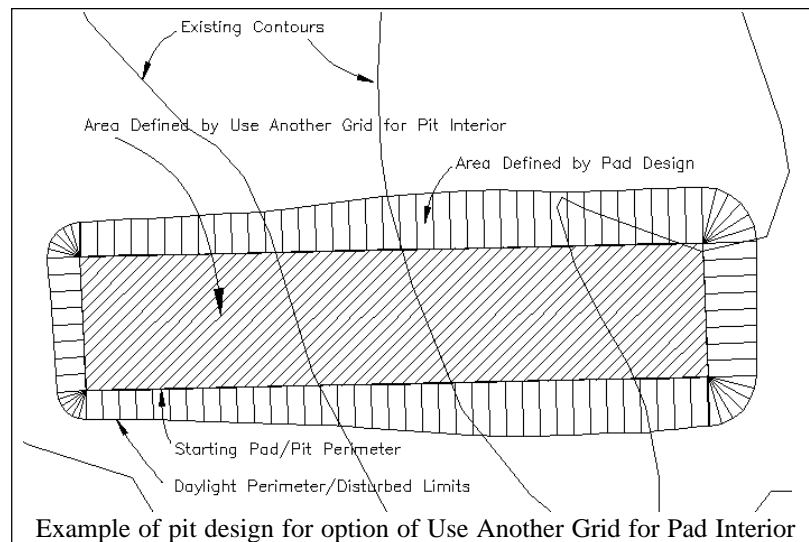
Several of the Design Pad Template options are set in the dialog shown here. The **Draw Slope Direction Arrows** option draws an arrow on the outslopes that points in the downhill direction. Arrows on fill slopes are drawn as solid filled. The **Solid Cut Arrows** chooses between drawing the

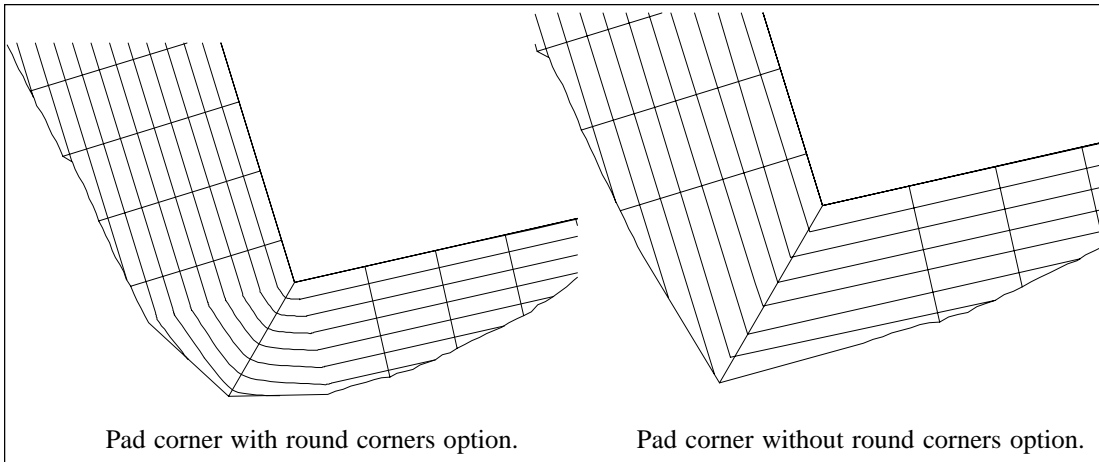


cut arrows also as solid filled or as wire frame. The **Pad Layer Name** specifies the layer name that the pad 3D polylines will be drawn in. The **Round Exterior Corners** option holds the outslopes around the corners. Otherwise the side outslopes stay straight until they meet at the corners as shown in the figure. The **Draw Side Slope Polylines** will draw 3D polylines perpendicular to the pad perimeter from the pad to the catch point. These side polylines are drawn at the interval set in the **Side Polyline Spacing** field. Besides at the interval, side slope polylines are also drawn at grid corners. In the Pad Design report, the cut volume is reported as the **Cut Swell Factor** multiplied by the original cut volume and the fill volume is reported as the **Fill Shrink Factor** multiplied by the original fill volume.

There is an option to calculate volumes for the pad design. The volumes are calculated by comparing the existing surface with the pad design. The inclusion perimeter for the volume calculation is the daylight perimeter polyline which represents the limits of disturbed area. The existing surface model is defined by the existing grid file or screen entities selected at the beginning of the command. The pad design surface is calculated by making a grid from the pad 3D polylines including the starting pad perimeter, the side polylines and the daylight perimeter. The **Use Another Grid for Pad Interior** option will prompt for another grid file to use for the design surface within the starting pad perimeter. Otherwise the program will model the pad interior by straight interpolation from the starting pad perimeter elevations. For example, if a building pad has a starting pad perimeter at a set elevation and the pad is supposed to be flat, then this option is not needed. This option is needed in a case where you are designing a pit and the starting pad perimeter is a 3D polyline that follows an undulating pit bottom surface. The pad design will model the pit side slopes. In order to model the undulating bottom of the pit, you need the Use Another Grid for Pad Interior option to select a grid that models the pit bottom.

Besides calculating the volumes in the Design Pad Template routine, you can also calculate the volumes with the Two Surface Volumes command. The design surface for Two Surface Volumes





can be the final output grid from Design Pad or you can create a design grid with Make 3D Grid File using the 3D polylines created in Design Pad. The reasons to use Two Surface Volumes are that this volume routine has more output options (cut/fill color maps, etc.) and you can check the volumes by plotting or contouring the grid files. Also you can combine several pads and other final surfaces by running Make Grid File and then calculate the overall site volumes with Two Surface Volumes.

For closed pad perimeters, there is an option to draw the slopes inward or outward from the perimeter. The outward method starts the slopes at the design elevation of the perimeter and projects out to intersect the existing surface. The inward method projects the slopes inside to reach the grid surface. The inward method is equivalent to starting from the disturbed area limits.

The surface is defined by a 3D rectangular grid mesh. The grid can be selected from a .grd file or calculated by the routine. When calculated, Design Pad Template internally creates a grid mesh from the surface entities found by doing a crossing selection of the grid limit. Make sure that the grid area covers the entire area for the pad including room for the cut/fill slopes.

The design is drawn as 3D polylines and the earthwork volumes are calculated. Before ending, the program allows you to adjust the design by changing the pad elevation, slopes and offset. The program can find the cut/fill balance by automatically adjusting the pad elevation. If adjustments are specified, the pad polylines are redrawn and the volumes recalculated. At the end, there is an option to trim existing contours inside the disturbed limits of the pad. Then there is an option to draw contours on the pad. If contouring is selected, a dialog lets you set the contouring options. Usually you should specify a new contour layer and turn off smoothing.

The cut and fill slopes can be defined by ratio, percent slope or template file. Using a template file lets you do road design without cross sections. Instead the existing surface is modeled by the grid. To do road design by cross section, use the Earthworks & Final Contours command in the Section-Profile module.

The template file (.tpl) is created in the Design Template routine in the Section - Profile module. When using a template, the pad perimeter represents the centerline. One way to create the pad perimeter for the template is to use the Profile to 3D Polyline command which converts a 2D centerline to a 3D polyline using a design profile. With a template, the program uses not only the cut and fill slopes from the template file but also draws all the template grade points such as edge of road, curb and ditch. The subgrade, superelevation and template transition options of the template file are not used in this command. These options are only applied in the Process Road Design command. The grade points are drawn as 3D polylines parallel with the centerline. Cross section 3D polylines that include the grade points are also drawn at the specified interval.

Prompts

Pick Lower Left limit of pad disturbed area: *Pick lower left* These prompts appear for the Screen Entities surface model method.

Pick Upper Right limit of pad disturbed area: *Pick upper right*

Be sure to pick these limits well beyond the area of the top of pad polyline in order to make room for the outslopes.

Make Grid File Dialog

After selecting the limits of the disturbed area the program will generate a 3D grid that represents the surface. Specify the grid resolution desired and select OK.

Pick the top of pad polyline: *Select perimeter polyline*

Enter the fill outslope ratio <2.0>: 2.5

Enter the cut outslope ratio <2.0>: 2.5

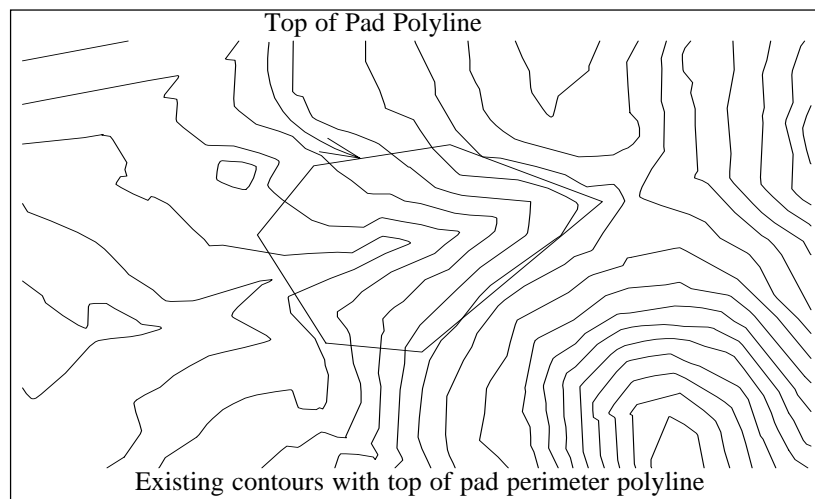
After entering outslopes slope ratios, a range of elevations along the pad top will be noted.

Enter the pad perimeter elevation <29.54>: 39

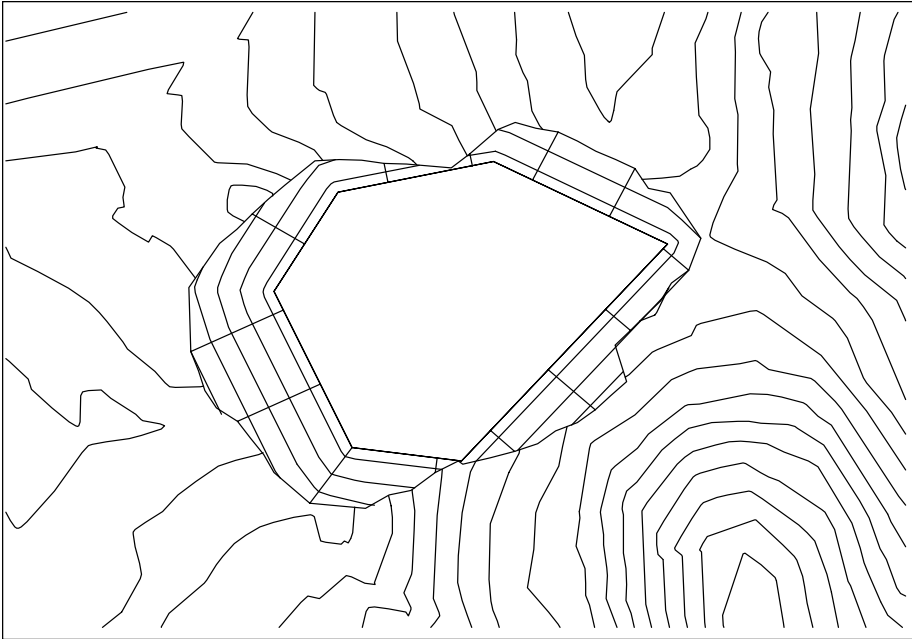
Calculate earthwork volumes (<Yes>/<No>)? *Press Enter*

Pad Report Viewer Reports cut/fill volume

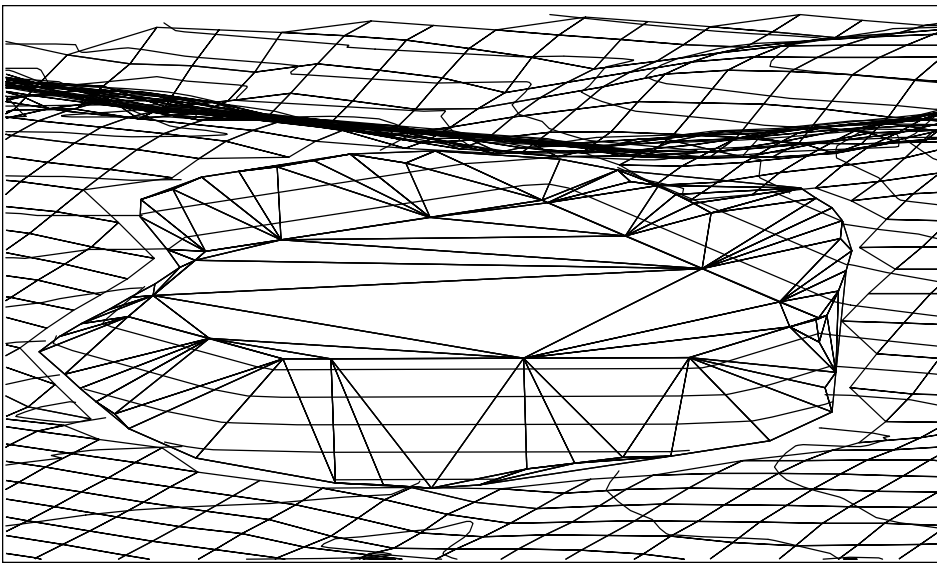
Write final surface to grid file (Yes/<No>)? *Press Enter.* This option will output a grid file using the elevations of the pad within the disturbed area polyline and using the original ground surface everywhere else.



Adjust parameters and redesign pond (Yes/<No>)? Press Enter
Trim existing contours inside pad perimeter (Yes/<No>)? Press Enter
Contour the pad (<Yes>/No)? Press Enter



Pad template
with contours



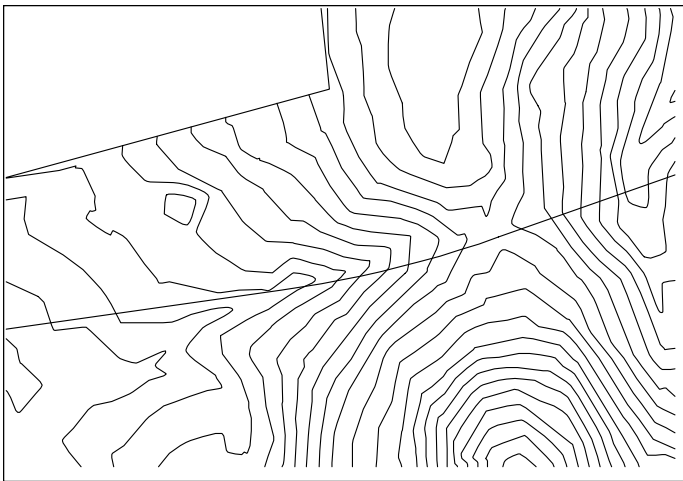
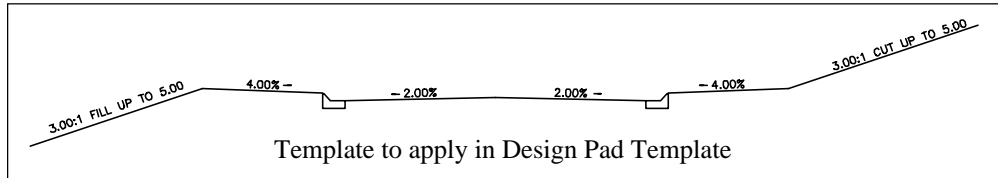
3D view of
pad with
DTM of
surface and
triangulation
faces of pad

Pull-Down Menu Location: Site

Prerequisite: A pad perimeter polyline and surface entities or .grd file

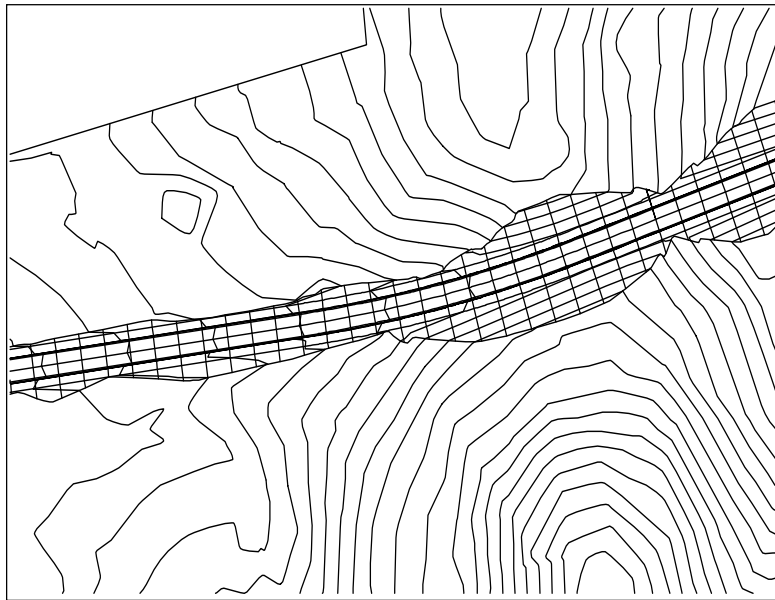
Keyboard Command: pad

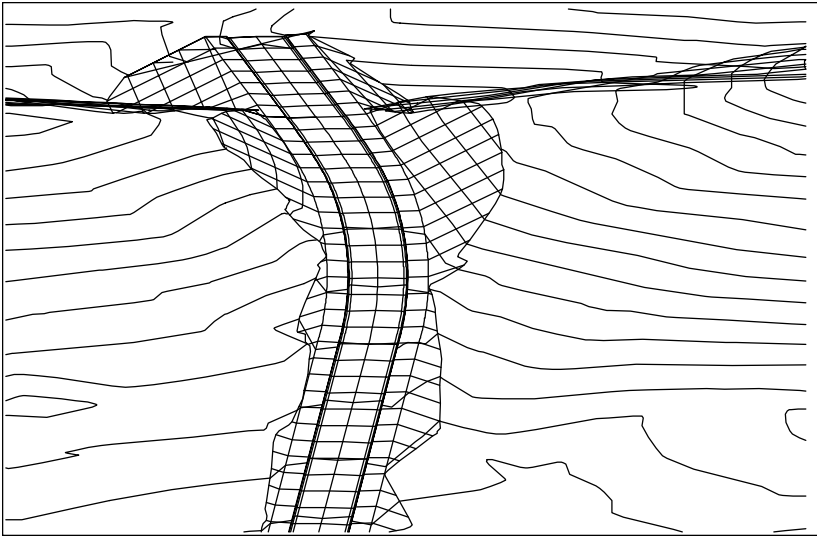
File Names: \lsp\flatpond.lsp & \lsp\pond.arx



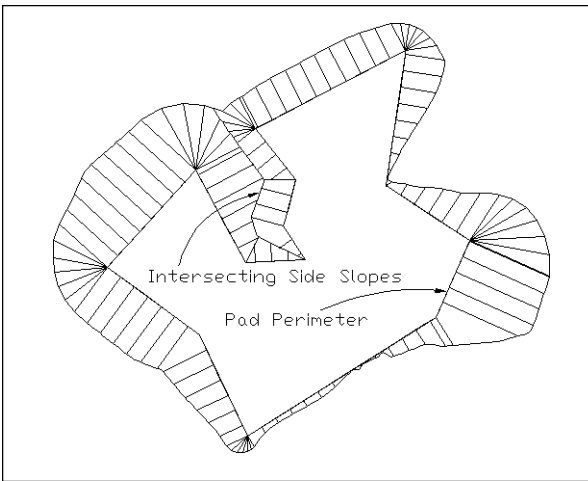
Existing surface with
3D polyline centerline

Result of Design Pad Template showing template grade polylines, cross section polylines, cut/fill slopes, and final contours.

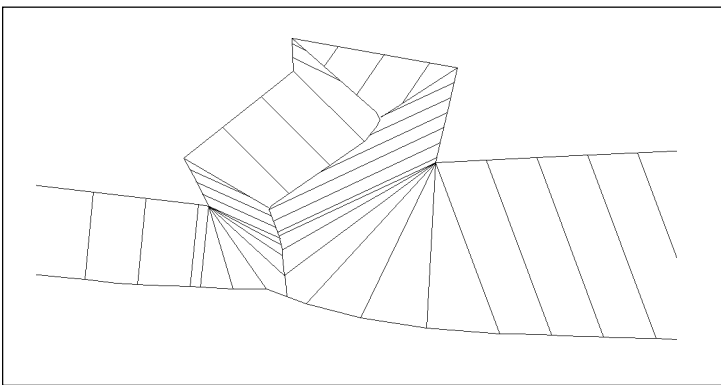




Viewpoint 3D view of Design Pad Template



Design Pad Template can also handle self-intersecting side slopes



Viewpoint 3D view of intersecting side slopes

Bench Pond Design

Function

This feature will design an on bench or hillside pond from a closed 2D polyline that defines the top dam perimeter of the pond and a surface model. Before beginning this routine, you must have the surface entities and the closed polyline as shown in Figure 5-45.

The surface is defined by a 3D rectangular grid mesh. The grid can be selected from a .grd file or calculated by the routine. When calculated, the program internally creates a grid mesh from the surface entities (Contours, Tri-Mesh, 3dPolys) found by doing a crossing selection of the grid limit.

The design is drawn as 3D polylines with an option to drawn contours on the pond, and the earthwork volumes and stage-storage volumes are calculated.

Prompts

Source of surface model (File/ <Screen>)? *Press Enter.* Use the File option to select a .grd file.

Pick Lower Left limit of pond disturbed area: *Pick lower left*

Pick Upper Right limit of pond disturbed area: *Pick upper right*

Be sure to pick these limits well beyond the area of the top of dam polyline in order to make room for the out slopes.

Make GRiD File Dialog

After selecting the limits of the disturbed area the program will generate a 3D grid that represents the surface. Specify the grid resolution desired and select OK.

Pick the top of dam polyline: *Select closed polyline (see Figure 5-45).*

Interior slopes to set elevation or original surface (<Elev>/Surface)? *Press Enter.* The interior of the pond can be created to one bottom elevation or the interior sides can stop where they intersect the original surface.

Enter slopes as percent grade or slope ratio (Percent/<Ratio>)? *Enter*

If Percent is desired press P, then Enter. In this example ratio will be used.

Enter the fill outslope ratio <2.0>: *2.5 Enter*

Enter the cut outslope ratio <2.0>: *2.5 Enter*

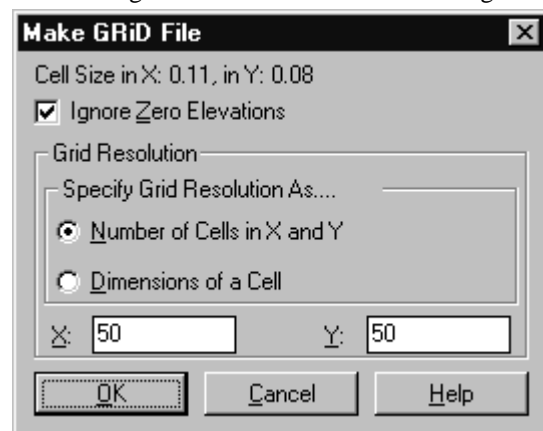
Enter the interior slope ratio <2.0>: *2.0 Enter*

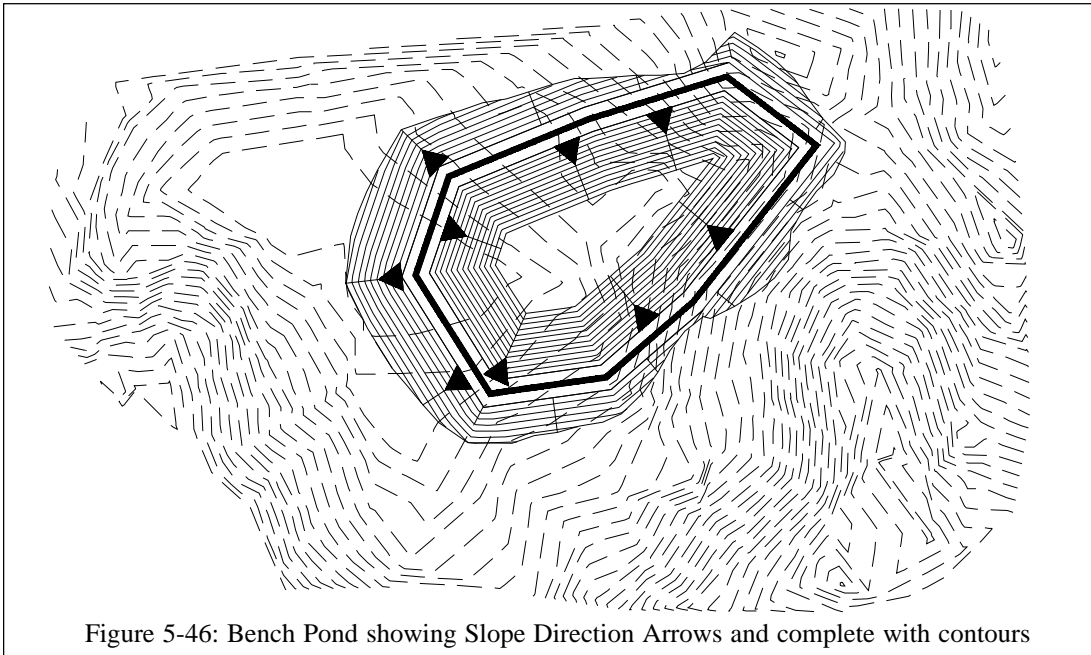
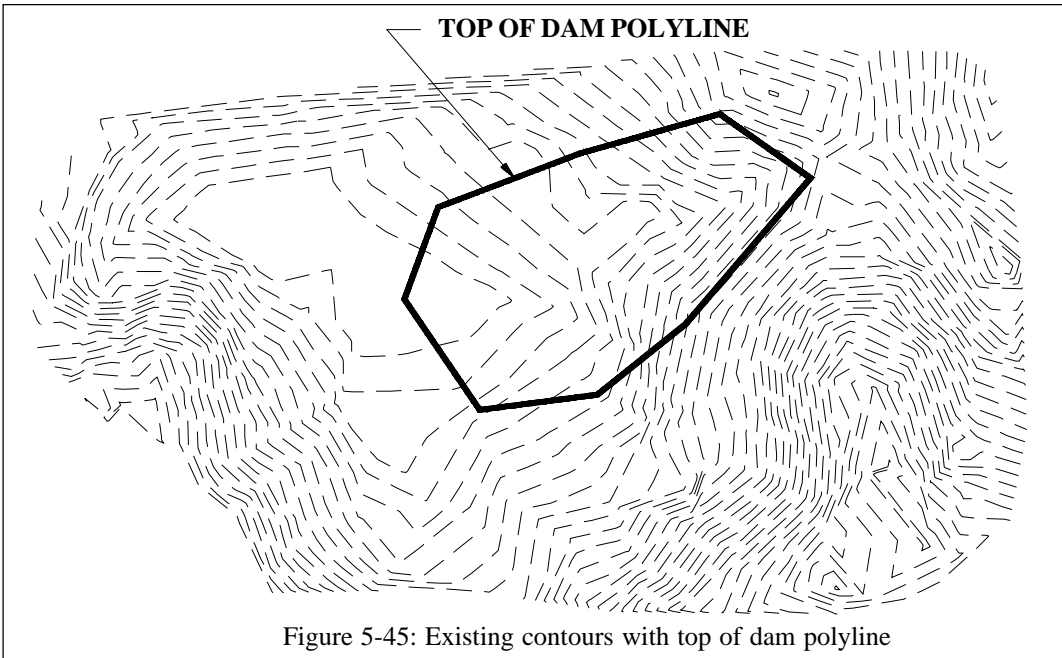
Enter the top of dam width: <10>: *10 Enter*

After entering out slopes and interior slope ratios a range of elevations along the dam top will be noted.

Enter the top of bank elevation <29.54>: *40 Enter*

Figure 5-44: Grid Resolution dialog





Enter the pond bottom elevation: 28 Enter

Draw slope direction arrows (Yes/<No>)? Yes Enter

Choose method to specify storage elevations (<Automatic>/Manual)? Enter

If manual is selected the user can specify the elevation(s) to calculate.

Write stage-storage to SEDCAD file (Yes/<No>)? Press Enter. This option creates a .cap file to use with Pond Stage Storage Curve which draws the stage-storage curve. The .cap file is also used in SEDCAD.

Write report to file (Yes/<No>)?

If yes, the user will be asked to enter a filename and the report will be written to disk (see Figure 5-48)

Write report to printer(Yes/<No>)?

Adjust parameters and redesign pond (Yes/<No>)?

Trim existing contours inside pond perimeter (Yes/<No>)?

Contour the pond (<Yes>/No)?

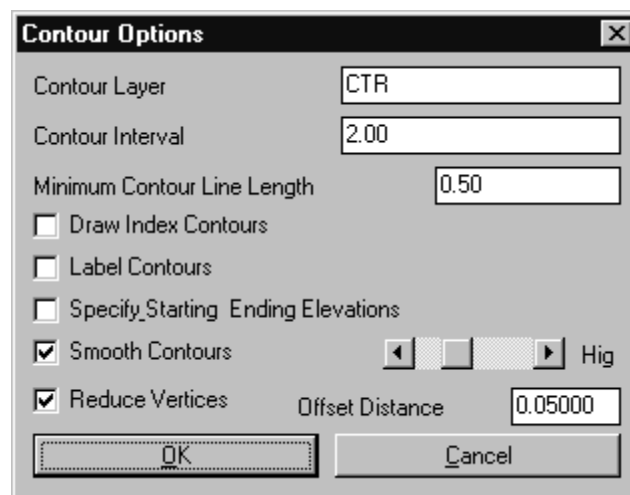


Figure 5-47: Contour Options dialog

Pull-Down Menu Location: Surface in Mining module, Site in DTM-Contour module, Pond in Hydrology module

Prerequisite: Closed polyline that defines top of dam and surface entities

Keyboard Command: bpond

File Name: \lsp\flatpond.lsp, \lsp\pond.arx

Bench Pond Design Report

Top of dam elevation: 40.0000
Bottom of pond elevation: 28.0000
Top of dam width: 10.0000
Cut slope percent grade: 40.00, slope ratio: 2.50
Fill slope percent grade: 40.00, slope ratio: 2.50
Interior slope percent grade: 50.00, slope ratio: 2.00

Lower left grid corner : 186395.20,57620.23
Upper right grid corner: 186803.82,57872.93
X grid resolution: 50, Y grid resolution: 50
X grid cell size: 8.17, Y grid cell size: 5.05

Pond EarthWork Volumes

Total fill: 2087.624 C.Y., 56365.84440 C.F.
Total cut: 772.791 C.Y., 20865.34961 C.F.

Pond Storage Volumes

Water Elev: 30.00, Pond Storage: 179.887 C.Y., 4856.94921 C.F.
Water Elev: 32.00, Pond Storage: 453.708 C.Y., 12250.12126 C.F.
Water Elev: 34.00, Pond Storage: 817.981 C.Y., 22085.48186 C.F.
Water Elev: 36.00, Pond Storage: 1321.035 C.Y., 35667.93750 C.F.
Water Elev: 38.00, Pond Storage: 2001.216 C.Y., 54032.83340 C.F.

Figure 5-48: Report generated by Write Report to File option

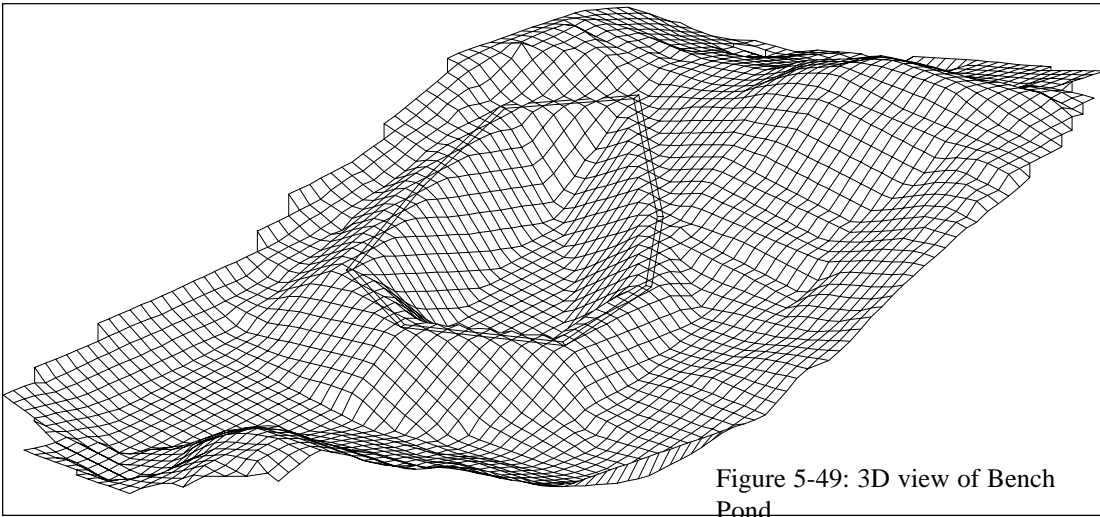


Figure 5-49: 3D view of Bench Pond

Valley Pond Design

Function

This feature will design a valley pond from a 2D polyline that defines the top of dam and a surface model. Before beginning this routine, you must have the surface entities and the polyline as shown in Figure 5-51.

The surface is defined by a 3D rectangular grid mesh. The grid can be selected from a .grd file or calculated by the routine. When calculated, the program internally creates a grid mesh from the surface entities (Contours, Tri-Mesh, 3dPolys) found by doing a crossing selection of the grid limit.

The design is drawn as 3D polylines with an option to drawn contours on the pond, and the earthwork volumes and stage-storage volumes are calculated.

Prompts

Source of surface model (File/ <Screen>)? *Press Enter.* Use the File option to select a .grd file.

Pick Lower Left limit of pond disturbed area: *Pick lower left*

Pick Upper Right limit of pond disturbed area: *Pick upper right*

Be sure to pick these limits well beyond the area of the top of dam polyline in order to make room for the outslopes.

Make GRiD File Dialog

After selecting the limits of the disturbed area the program will generate a 3D grid that represents the surface. Specify the grid resolution desired and select OK.

Pick the top of dam polyline: *Select closed 2d polyline (See Figure 5-51)*

Pick a point within the pond: *Pick a point*

Enter slopes as percent grade or slope ratio (Percent/<Ratio>)? *Enter*

If Percent is desired press P, then Enter. In this example slope ratio will be used.

Enter the fill outslope ratio <2.0>: *2.5 Enter*

Enter the cut outslope ratio <2.0>: *2.5 Enter*

Enter the top of dam width: <10>: *10 Enter*

After entering outslope ratios and dam width a range of elevations along the top of dam will be noted.

Enter the top of dam elevation: *90 Enter*

Cut pond interior (<Yes>/No)? *No Enter.* This option has two methods for cutting volume from

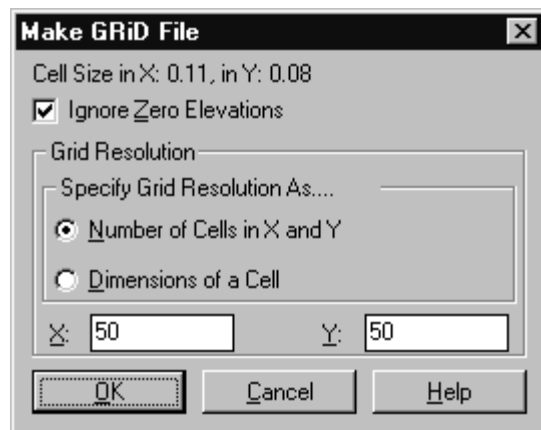
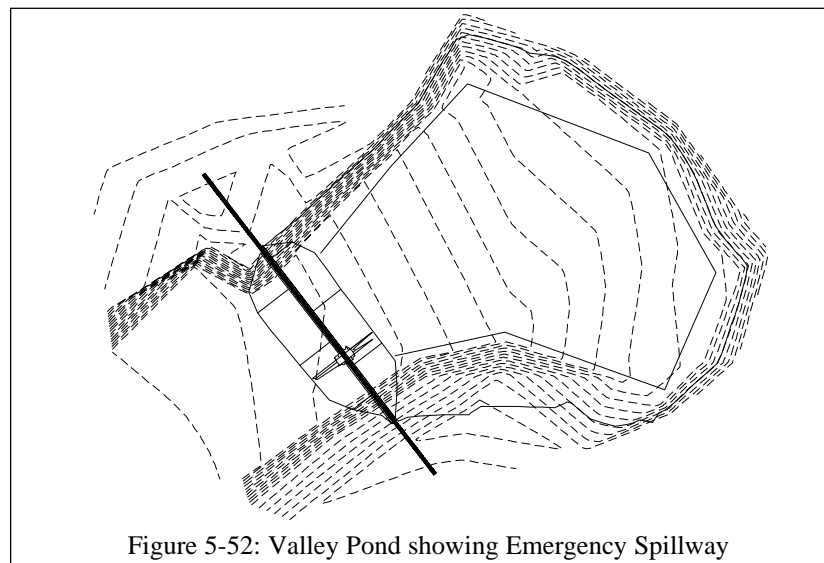
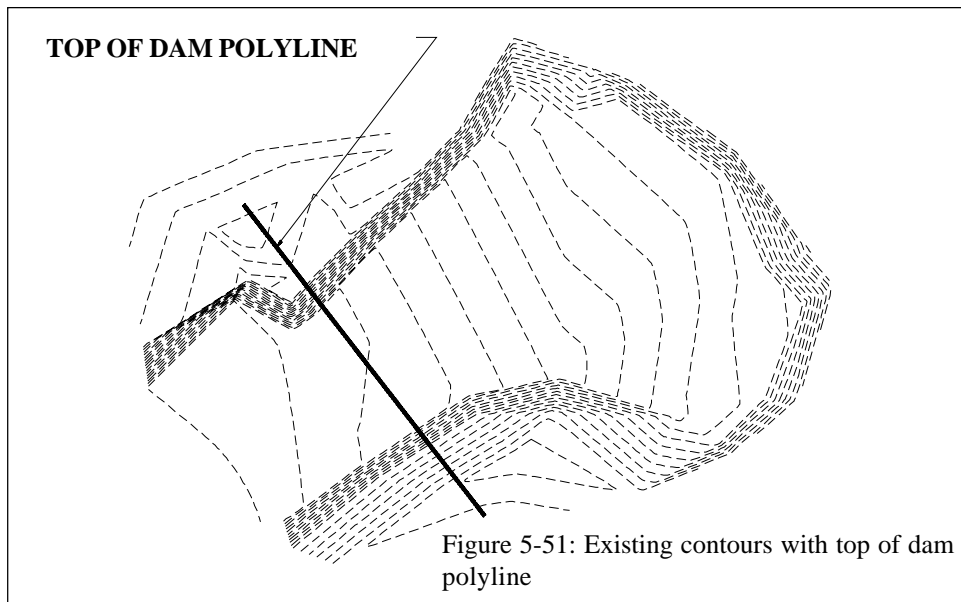


Figure 5-50: Grid Resolution dialog

the pond interior. This cut will create more water storage.

The trace method prompts you to define a 3D polyline by picking points starting at the dam and going around the pond counterclockwise. At each point you enter an elevation. The default is the current ground elevation and typically you would enter a lower elevation. Then you enter a cut slope and the program will cut from this perimeter polyline at the entered slope.

The polyline method requires a pre-drawn closed polyline inside the pond. The program will ask for a depth to cut and a cut slope. Polyline is set to the current ground elevation minus the cut depth. Then the program cuts out from the polyline to the original ground at the cut slope.



Choose method to specify storage elevations (<Automatic>/Manual)? *Enter*

If manual is selected the user can specify the elevation(s) to calculate.

Write report to file (Yes/<No>)? *Enter*

If yes, the user will be asked to enter a filename and the report will be written to disk.

Write report to printer(Yes/<No>)? *Enter*

Adjust parameters and redesign pond (Yes/<No>)? *Enter*

If yes, the user will be able to enter new slope ratios and dam widths etc...

Contour the pond (<Yes>/No)? *Enter*

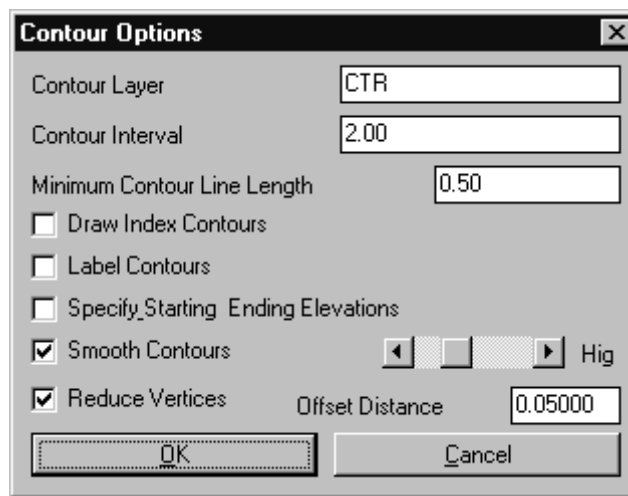


Figure 5-54: Contour Options dialog

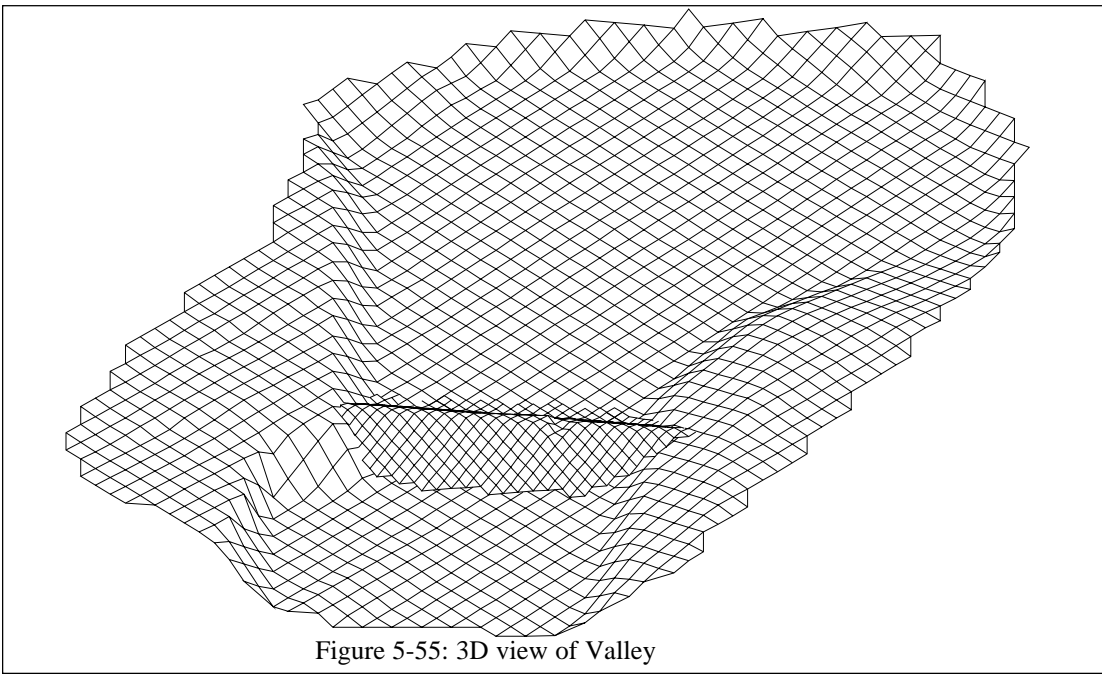
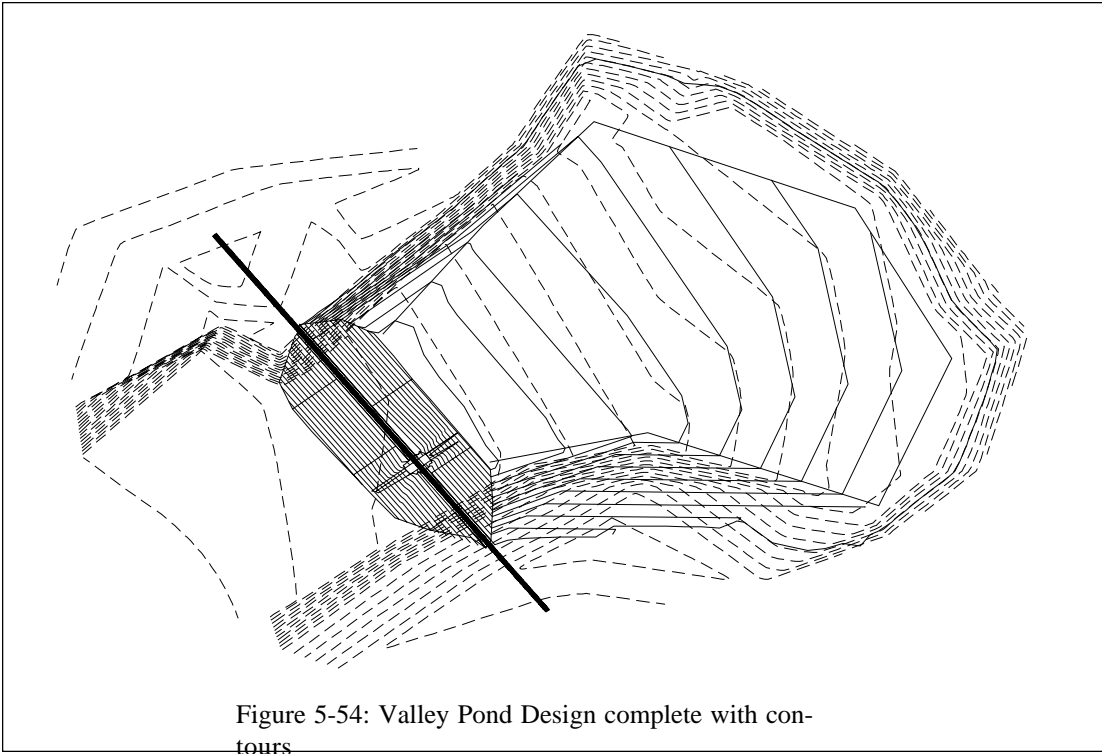
Pond

Pull-Down Menu Location: Surface in the Mining module and Site in the DTM-Contour module

Prerequisite: Polyline that defines top of dam

Keyboard Command: vpond

File Name: \lsp\flatpond.lsp, \lsp\pond.arx



Spot Elevations

Function

This command will calculate the z coordinate of any point that falls within the surface model. Use this command to calculate the elevations for points of a design for slope staking or for putting spot elevations on a topography map. The calculated points are stored in the current coordinate file. A surface model is either selected from a .grd or .flt file or internally calculated from the existing entities on the drawing.

Spot elevations can be calculated at various user-specified points or at a specific interval. For random spot elevations, the user picks or enters the x,y coordinates for each spot elevations. The elevation at the current position of the crosshairs is displayed in real-time at the bottom of the side bar. For interval spot elevations, the alignment for the intervals is defined by a polyline that must be created before starting this routine.

Prompts

Source of surface model (File/<Screen>)? *Press Enter.* Use the File option to select a .grd or .flt file.

Layer for points <POINTS>: *Press Enter*

Add spot points to Coordinate File (Yes/<No>)? *Yes.*
This option stores any points created in this routine to a .crd file and draws Carlson point entities.

Draw nodes only (Yes/<No>)? *Press Enter.* This prompt only appears if Add points to Coordinate File is off. This option either draws only AutoCad POINT entities or an X mark and elevation text.

Pick Lower Left limit of surface area: *pick a point at the lower left corner of your area of interest*

Pick Upper Right limit of surface area: *pick a point to window around your area of interest*

Make Grid File Dialog

Choose a grid resolution for modelling the surface.

For picked point spot elevations:

Random spot elevations or interval along pline (<Random>/Interval)? *[Enter]*

Enter or pick point (Enter to end): *pick a point*

Enter or pick point (Enter to end): *Press Enter*

For spot elevations along a polyline:

Random spot elevations or interval along pline (<Random>/Interval)? *Interval*

Pick the centerline polyline: *pick a polyline*

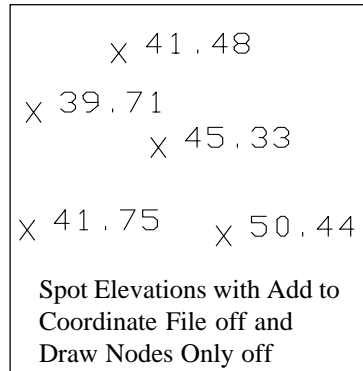
Interval along polyline <50.0>: *25*

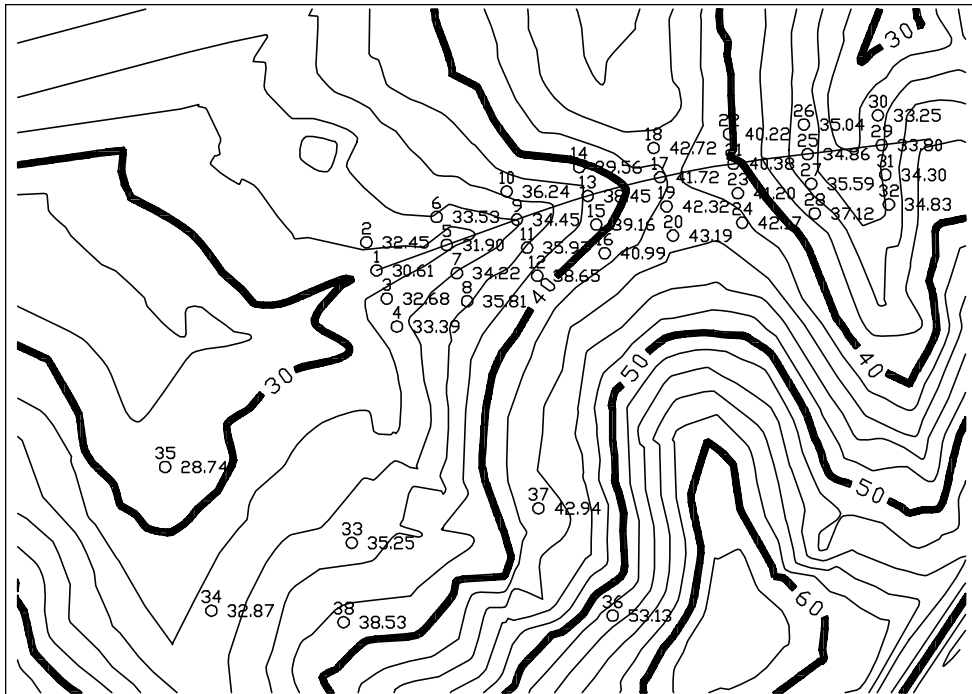
Number of left offsets <0>: *1*

Enter left offset interval <25.0>: *10*

Number of right offsets <0>: *2*

Enter right offset interval <10.0>: *Press Enter*





Interval spot elevations for points 1-32. "Random" spot elevations for points 33-37.

Pull-Down Menu Location: Site

Prerequisite: Surface entities or a .grd file

Keyboard Command: spotelv

File Names: \lsp\flatpond.lsp & \lsp\pond.arx

Entities to Points with Elevations

Function

This command works similar to the *Entities to SurvCADD Points* command explained in the Cogo Section except it has the added ability to compute an elevation from a surface model that is stored in a grid (.grd) file or triangulation (.flt) file.

The program uses the X,Y from the selected entities and calculates the Z from the surface model. For example, if you need the elevations of the design features for stakeout. Choose the lines and arcs of your design with this command to specify the PC's, PT's and endpoints. Then use the command *Spot Elevations* explained previously to compute additional points. The results are stored in the current .CRD file and could be transferred to an electronic field book.

Prompts

Entities to Points Dialog

Choose whether to prompt for descriptions, elevations or point numbers

Entities to Process Dialog Choose entities to process

Source of Surface Model

Choose a grid (.grd) or triangulation (.flt) file.

Select arcs, faces, line and/or polylines.

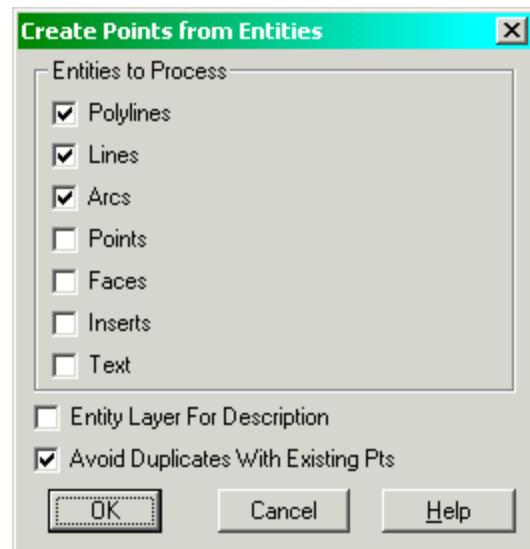
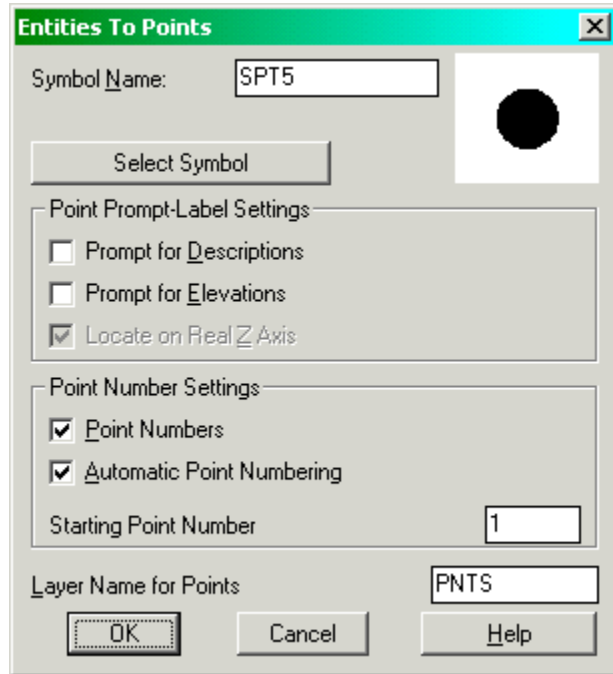
Select objects: *pick entities*

Keyboard Command: autoelev

Pull-Down Menu Location: Site

Prerequisite: Create a surface model (.grd or .flt) and entities to convert

File Name: \isp\autoelev.lsp & \isp\makegrid.arx



3D Entity By Surface Model

Function

This command assigns the elevations of the selected points, inserts, lines and polylines to the elevations defined by the specified grid (.GRD) or triangulation (.FLT) file. For lines and polylines, additional vertices are added to model the surface. 3D Polylines cannot have arcs. So any arcs in the original 2D polylines are converted to a series of chords. There is an option to retain the original polylines in case you need to keep the 2D polylines with the arcs.

Prompts

Choose Grid or Tmesh File to Process dialog
Select points, inserts, lines and polylines to convert.
Select objects: *pick the entities to convert*
Keep existing polylines [Yes/<No>]? *Press Enter*
Converting entities ...
Done.

Pull-Down Menu Location: Site

Prerequisite: entities to be converted

Keyboard Command: 3dconvert

File Name: \lsp\poly3d.arx

Draw 3D Polyline Perimeter

Function

This command draws a 3D polyline in the PERIMETER layer. This polyline is required by the Calculate Stockpile Volume and Calculate Pond/Pit Volume routines. In these routines, this polyline is used as the inclusion perimeter for volumes. You may want to run AutoCad's OSNAP routine and set your osnap prior to running this routine so that you can grab the elevations of existing points while creating the 3D polyline. Besides picking and entering the points, you can also use point numbers from the current Coordinate File (.CRD file).

Prompts

Ignore Z values of picked points (.XY filter) (Yes/<No>): *Y*
Using the .XY filter allows the user to pick the X and Y coordinate from the screen and type in the elevation. If you use the N response then the z coordinate of the point picked will be applied.
Number/<Pick point>: *(pick point)*
Number/Close/Undo/<Pick point>: *N*
Close/Undo/Pick point/<point Number>: *15* This is a point number from the current coordinate file.

Close/Undo/Pick point/<point Number>: *Press Enter*
Pressing [Enter] ends the command.

Pull-Down Menu Location: Site

Keyboard Command: 3dperim

Prerequisite: None

File Name: \lsp\3dpline.lsp

Draw 3D Polyline Base Barrier

Function

This command draws a 3D polyline in the BASE_BARRIER layer. This polyline is used by the Calculate Stockpile Volume and Calculate Pond/Pit Volume routines to model the base surface. You may want to run AutoCad's OSNAP routine and set your osnap prior to running this routine so that you can grab the elevations of existing points while creating the 3D polyline. Besides picking and entering the points, you can also use point numbers from the current Coordinate File (.CRD file).

Prompts

Ignore Z values of picked points (.XY filter) (Yes/<No>): *Y*

Using the .XY filter allows the user to pick the X and Y coordinate from the screen and type in the elevation. If you use the N response then the z coordinate of the point picked will be applied.

Number/<Pick point>: *(pick point)*

Number/Close/Undo/<Pick point>: *N*

Close/Undo/Pick point/<point Number>: *15* This is a point number from the current coordinate file.

Close/Undo/Pick point/<point Number>: *Press Enter*

Pressing [Enter] ends the command.

Pull-Down Menu Location: Site

Prerequisite: None

Keyboard Command: 3dbase

File Name: \lsp\3dpline.lsp

Calculate Stockpile Volume

Function

This command is the simplified method for calculating fill volumes and is the compliment of the Calculate Pond/Pit Volume routine. The program internally computes base and final grids that enclose the limits of your data points. You have the option of setting the resolution of the grids. The base grid is calculated from the 3D polyline in the PERIMETER layer, which must be drawn by Draw 3D Polyline Perimeter before using this routine, and from optional 3D polylines in the BASE_BARRIER layer as drawn by the Draw 3D Polyline Base Barrier routine. The 3D polyline perimeter is also used as the inclusion perimeter for the volume calculation. The final grid is calculated from the other selected surface entities such as points, line, inserts, and polylines along with the perimeter polyline but not including the BASE_BARRIER polylines.

The Make 3D GRiD File and Two Surface Volumes routines are an alternative to this command.

Prompts

Make GRiD File dialog

Set the resolution and then click OK. See Figure 5-50.

Select stockpile entities and perimeter.

Select object: *pick the objects that define the surface and the 3D polyline perimeter.*

Reading points ... 53

Inserting barriers 17

Triangulating points ... 53

Processing cell 2500

Volume report

Volume report

Lower left grid corner : 15965.45,12657.05

Upper right grid corner: 16269.40,12906.29

X grid resolution: 50, Y grid resolution: 50

X grid cell size : 6.08, Y grid cell size: 4.98

Stockpile volume : 1191674.87825 cubic ft,

44136.107 cubic yards

Figure 5-56: Sample volume report

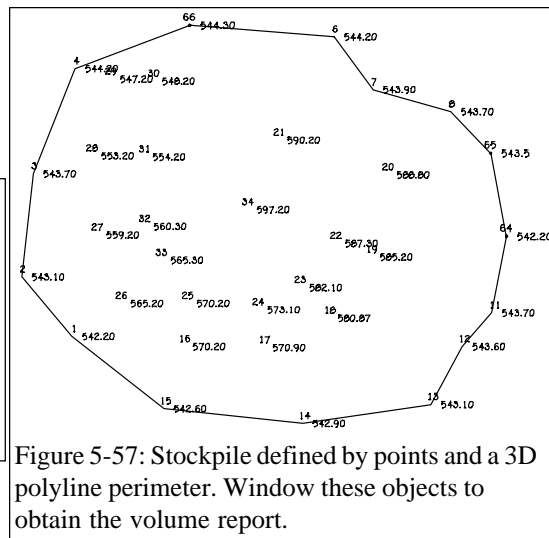


Figure 5-57: Stockpile defined by points and a 3D polyline perimeter. Window these objects to obtain the volume report.

Pull-Down Menu Location: Site

Keyboard Command: stockvol

Prerequisite: Data points of the surface and a 3D polyline in the PERIMETER layer.

File Name: \lsp\stockvol.lsp, \lsp\volcalc.arx

Calculate Pond/Pit Volume

Function

This command is the simplified method for calculating fill volumes and is the compliment of the Calculate Stockpile Volume routine. The program internally computes base and final grids that enclose the limits of your data points. You have the option of setting the resolution of the grids. The base grid is calculated from the 3D polyline in the PERIMETER layer, which must be drawn by Draw 3D Polyline Perimeter before using this routine, and from optional 3D polylines in the BASE_BARRIER layer as drawn by the Draw 3D Polyline Base Barrier routine. The 3D polyline perimeter is also used as the inclusion perimeter for the volume calculation. The final grid is calculated from the other selected surface entities such as points, line, inserts, and polylines along with the perimeter polyline but not including the BASE_BARRIER polylines.

The Make 3D GRiD File and Two Surface Volumes routines are alternatives to this command.

Prompts

Make GRiD File dialog

Set the resolution and then click OK. See Figure 5-50.

Select pond/pit entities and perimeter.

Select object: *pick the objects that define the surface and the 3D polyline perimeter.*

Reading points ... 53

Inserting barriers 17

Triangulating points ... 53

Processing cell 2500

Volume report

Pull-Down Menu Location: Site

Prerequisite: Data points of the surface and a 3D polyline in the PERIMETER layer.

Keyboard Command: pitvol

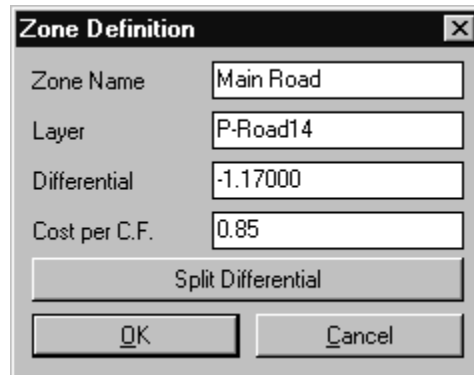
File Name: \lsp\stockvol.lsp, \lsp\volcalc.arx

Define Earthwork Zones

Function

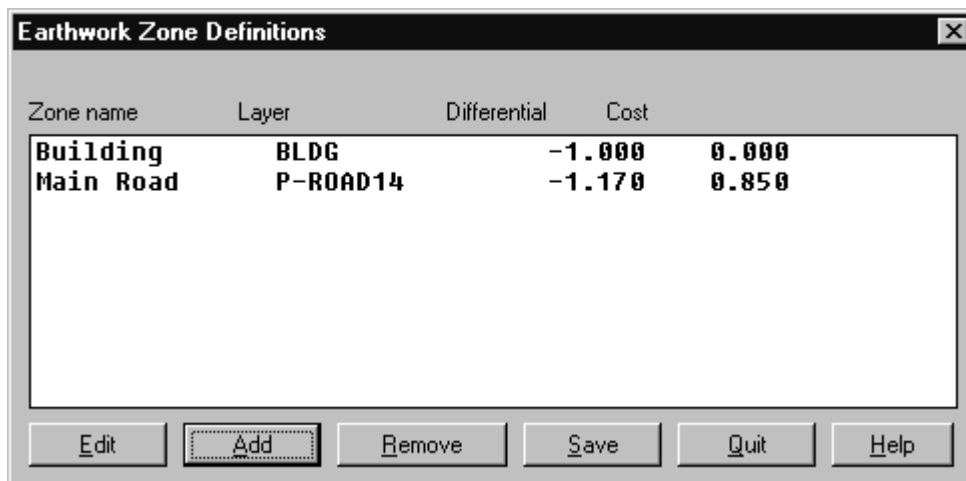
This command assigns a layer and differential to each zone name. Closed polylines for zones can be drawn in the corresponding layers. The Calculate Earthwork command matches the zone polylines by layer name with the definitions. The elevation of the final surface within the zone closed polyline is adjusted by the zone differential. If a zone has two or more layers, the Split Differential option can be used to specify the thickness and description for each layer. The zone definitions are stored in the file *zones.dta* in the Carlson EXEC directory.

Pull-Down Menu Location: Site
Prerequisite: None
Keyboard Command: zones
File Name: \lsp\zones.lsp, \lsp\contour4.arx



The Zone Definition dialog box contains the following fields and buttons:

- Zone Name: Main Road
- Layer: P-Road14
- Differential: -1.17000
- Cost per C.F.: 0.85
- Buttons: Split Differential, OK, Cancel



The Earthwork Zone Definitions dialog box displays a table with the following data:

Zone name	Layer	Differential	Cost
Building	BLDG	-1.000	0.000
Main Road	P-ROAD14	-1.170	0.850

Buttons at the bottom: Edit, Add, Remove, Save, Quit, Help

Name Zone Polylines

Function

Zone polylines are identified by their layer name. This command adds another identification to zone polylines to distinguish between zone polylines in the same layer. The Calculate Earthwork command will read these zone names and report volume individually for each name as well as the total volumes. For example, if you had several building zone polylines in the layer BLDG, you could assign them different names like BLDG 1 and BLDG 2. Then Calculate Earthwork will report the volumes separately for BLDG 1 and BLDG 2 instead of all the BLDG zones.

Prompts

Label area names (<Yes>/<No>)? Press Enter
Text height <4.0>: *Press Enter*
Area name <Area 1>: *BLDG 1*
Select area perimeter polylines.
Select objects: *pick the zone polyline for this name*
Specify another area (<Yes>/<No>)? *N*

Pull-Down Menu Location: Site

Prerequisite: zone polylines

Keyboard Command: plname

File Name: \lsp\plname.lsp

Calculate Earthwork

Function

This command calculates earthwork cut and fill between two grid files while adjusting for top soil removal/replacement and differentials in zones. Grids files representing the base and final surfaces must be created by Make 3D Grid File before starting this command. Also the zones must be predefined by the Define Earthwork Zones command which creates a zone definition file that specifies the zone differential and layer.

Top soil removal/replacement is applied to the base grid before calculating the remaining earthwork.

Zone areas are defined by a closed 2D polyline in the layer of the corresponding to the zone definition file. The zone differentials within each zone area are applied to the final grid. In the example on the following page, ROAD 1 is a closed polyline drawn in the layer P-ROAD14 which has a zone definition with a differential of -1.17. The final grid elevations within the ROAD 1 polyline will be lowered by 1.17. Roads 2-6 are in a layer P-ROAD8 with a differential of -0.67.

After adjusting the base and final grid surfaces, the earthwork is calculated between the grids. The volume within each zone is calculated and reported separately. The area that is not in a zone is reported as the Base volume.

Prompts

Select Base Grid File dialog
Choose the base surface .GRD file.
Select Final Grid File dialog

Choose the final surface .GRD file.

Select the Inclusion perimeter polylines or Enter for none:

Select objects: *pick inclusion polylines such as the limits of disturbed area.* These polylines apply to the overall site.

Select the Exclusion perimeter polylines or Enter for none:

Select objects: *pick exclusion polylines if any*

Fill shrinkage factor <1.0>: *Press Enter*

Cut swell factor <1.0>: *Press Enter*

Select the Zones polylines or Enter for none:

Select objects: *pick the zone closed polylines if any*

Enter the top soil removal depth in fill <0.0>: *0.5*

Enter the top soil removal depth in cut <0.5>: *0.5*

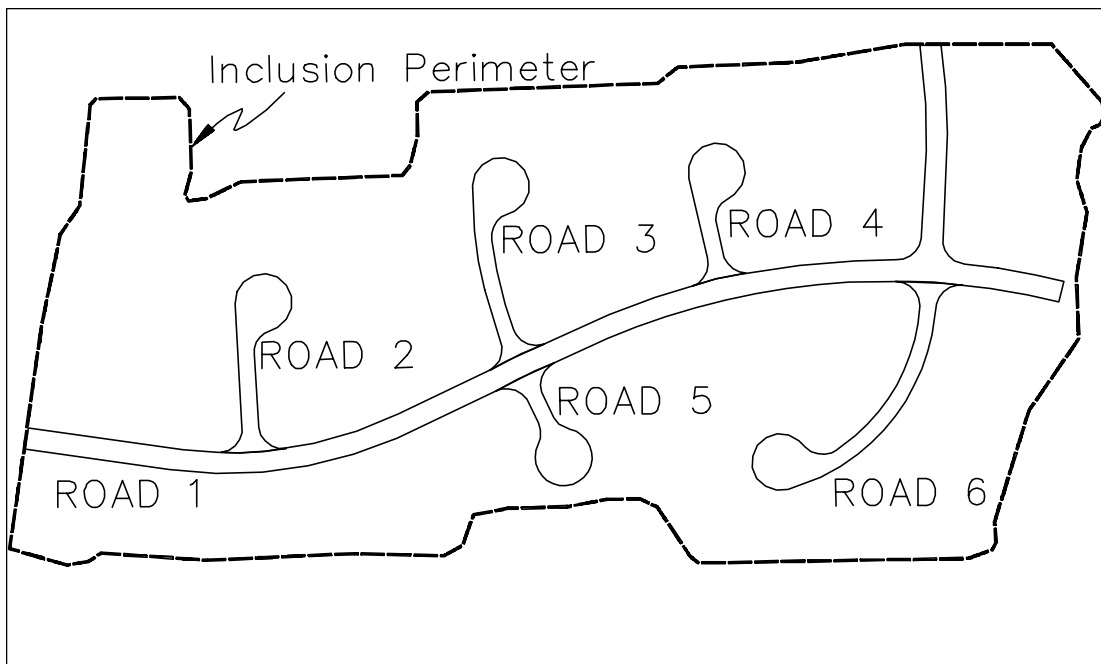
Apply top soil removal to entire site (<Yes>/<No>)? *Press Enter.* Answer no to choose inclusion and/or exclusion polylines for top soil removal.

Enter the top soil replacement depth in fill <0.0>: *Press Enter*

Enter the top soil replacement depth in cut <0.0>: *Press Enter*

Create tabluar report (<Yes>/<No>)? *Press Enter.* This option controls the format for the report.

Write zone modified grid files (Yes/<No>)? *Press Enter.* Use this option to create grid files that show the modifications. The base grid will have the top soil adjustments and the final grid will have the zone differentials.



Zone Volume Report						
Comparing GRiD file: C:/scdev/data/EX10-23.GRD						
and GRiD file: C:/scdev/data/PR10-23.GRD						
Lower left grid corner : 11771871.78,6850458.48						
Upper right grid corner: 11774321.78,6853448.48						
X grid resolution: 245, Y grid resolution: 299						
X grid cell size: 10.00, Y grid cell size: 10.00						
Top Soil Removal Depth: for fill is 0.500, for cut is 0.500						
Top Soil Removal Area is 1031385.35 S.F., Volume is 19099.73 C.Y.						
Name	Diff	Cut (C.F.) (C.Y.)	Fill (C.F.) (C.Y.)	Area (S.F.) (Acres)	Length	Cost
Main Road Road 1	-1.170	221204.30	45689.84	62721.48	3920.54	\$62376.52
		8192.75	1692.22	1.44		
Side Roads Road 2	-0.670	16661.99	1166.47	11489.26	748.06	\$6543.13
		617.11	43.20	0.26		
Side Roads Road 3	-0.670	7586.22	74932.54	12106.25	817.10	\$6894.51
		280.97	2775.28	0.28		
Side Roads Road 4	-0.670	48293.64	0.00	9597.66	591.72	\$5465.87
		1788.65	0.00	0.22		
Side Roads Road 5	-0.670	8541.28	34589.47	8817.97	561.53	\$5021.83
		316.34	1281.09	0.20		
Side Roads Road 6	-0.670	11209.97	18814.02	15500.00	1094.86	\$8827.25
		415.18	696.82	0.36		
Base		1035124.57	2077552.12	910833.59		
		38337.95	76946.37	20.91		
Total		1348621.96	2252744.45	1031066.21		
		49948.96	83434.98	23.67		
Total cost of zones: \$95129.11						

Pull-Down Menu Location: Site

Prerequisite: Zone definitions and closed polylines, and base and final grid files.

Keyboard Command: zonecalc

File Name: \lsp\zonecalc.lsp, \lsp\contour4.arx