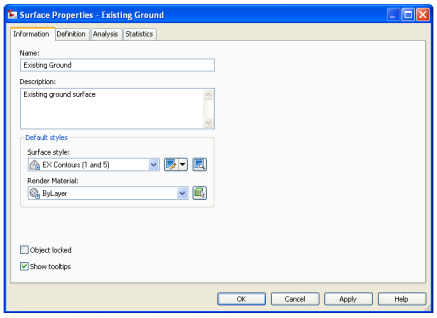
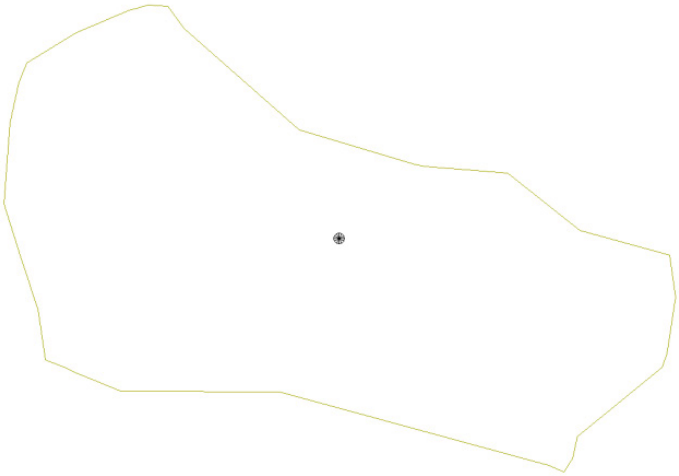
The visual display of a surface, such as contours and triangulation lines, is controlled through the use of surface styles. The Minnesota Civil 3D drawing template includes several default surface display styles, which are described in more detail below.



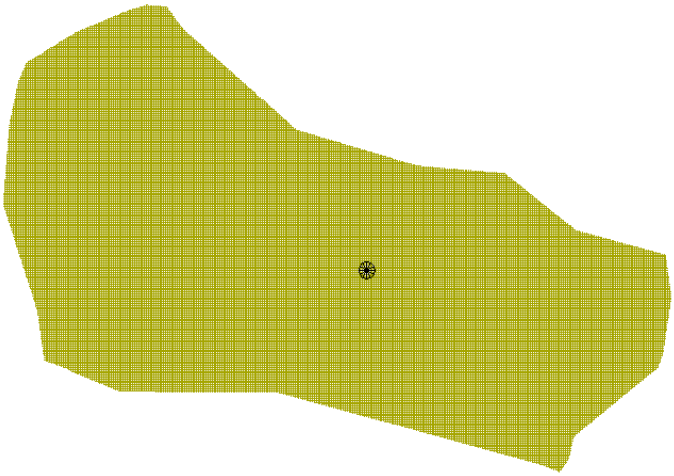
To change the style that is set for a surface, go to the *Prospector* tab in *Toolspace*, find the name of the surface you want to change the style for, right click on the surface name and select *Surface Properties*… The surface properties icon should also appear on a Tin Surface contextual tab that appears at the top of the window when you select the surface in the drawing.

The surface style is selected on the *Information* tab of the *Surface Properties* window. Click on the down arrow next to *Surface style*: and select a style from the drop-down menu.

Surface Style: _Border
Description: Displays only the surface's border
Example:

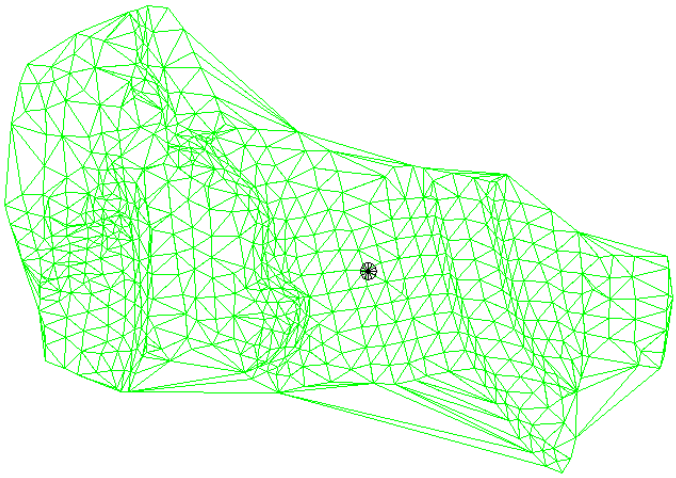


Surface Style: _Grid
Description: Displays the surface as grids
Example:

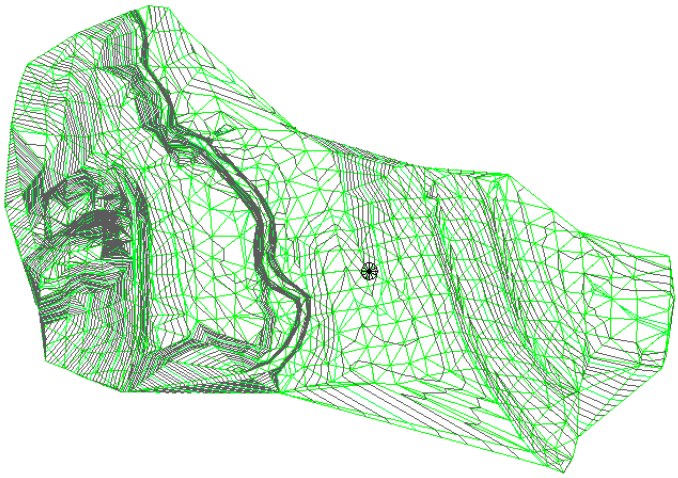


Surface Style: _No Display
Description: Suppresses the surface display

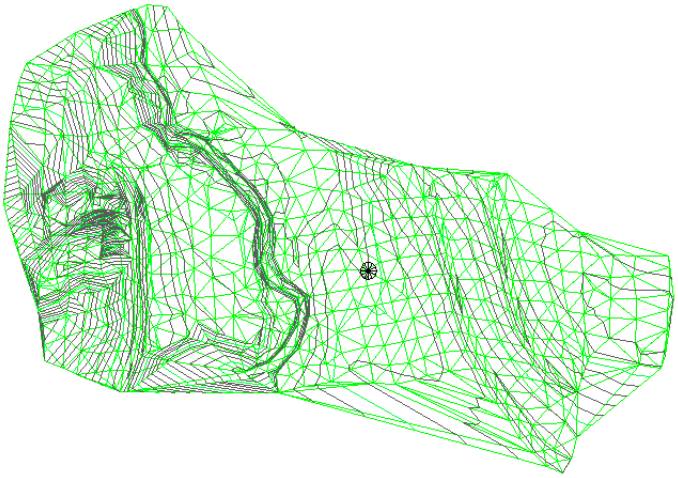
Surface Style: _Triangles
Description: Shows green triangulation lines (TIN)
Example:



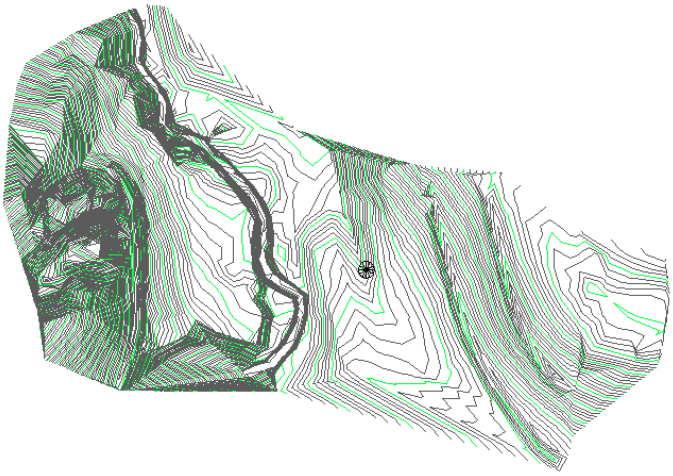
Surface Style: Contours (1 and 5) and Triangles
Description: Displays contours at 1 and 5 foot intervals and green TIN lines
Example:



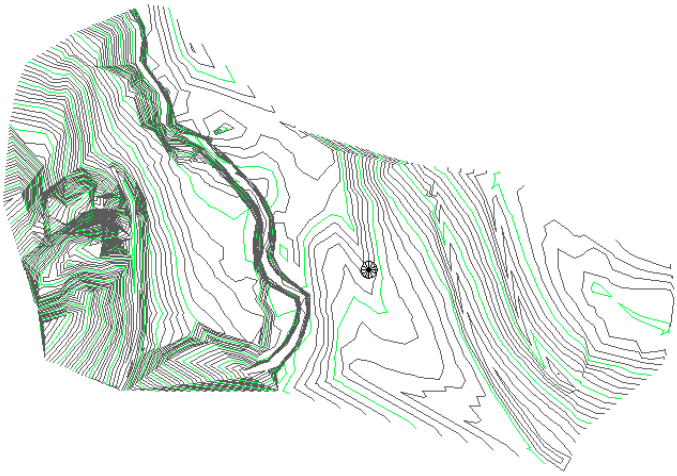
Surface Style: Contours (2 and 10) and Triangles
Description: Displays contours at 2 and 10 foot intervals and green TIN lines
Example:



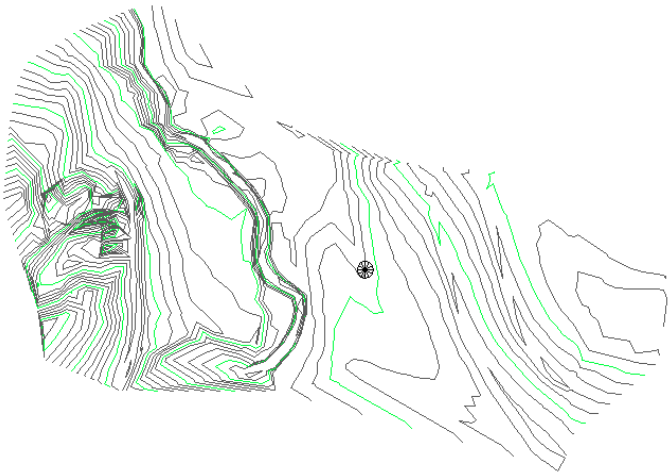
Surface Style: EX Contours (0.5 and 2.5)
Description: Displays existing ground contours at 0.5 and 2.5 foot intervals
Example:



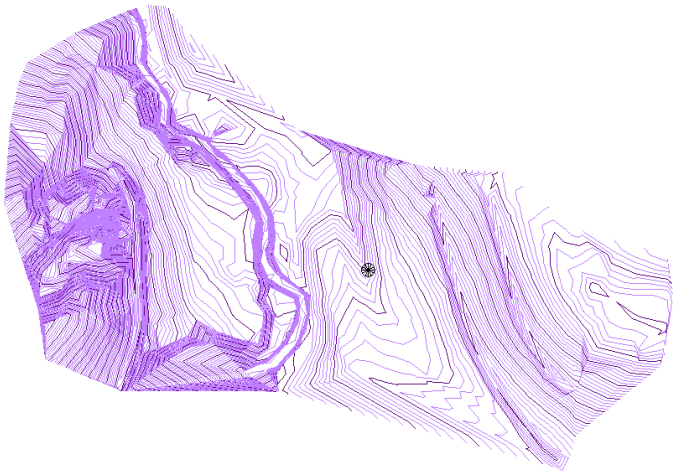
Surface Style: EX Contours (1 and 5)
Description: Displays existing ground contours at 1 and 5 foot intervals
Example:



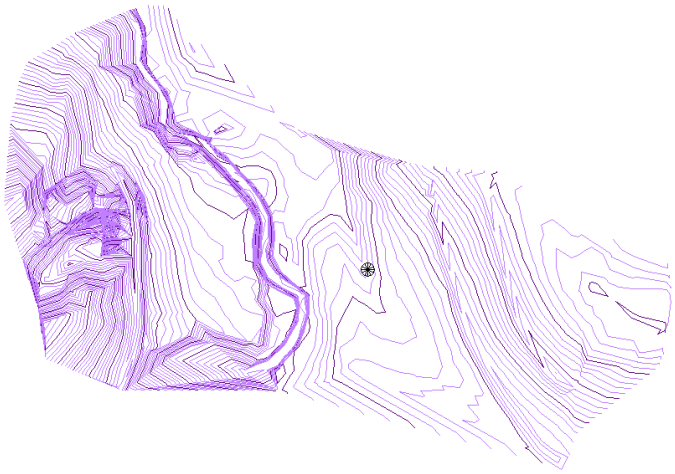
Surface Style: EX Contours (2 and 10)
Description: Displays existing ground contours at 2 and 10 foot intervals
Example:



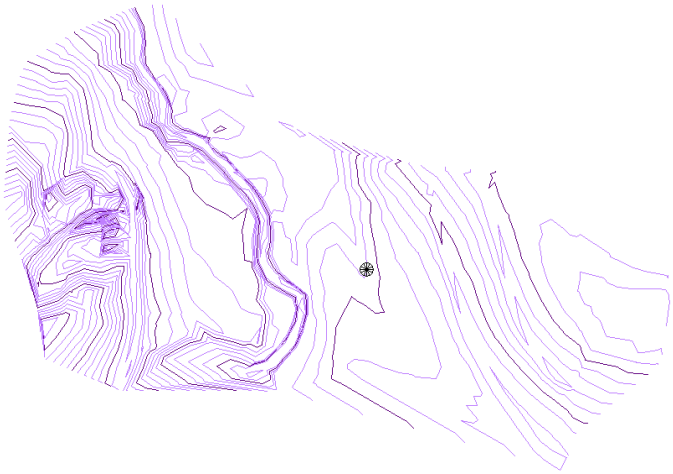
Surface Style: FG Contours (0.5 and 2.5)
Description: Displays finished ground contours at 0.5 and 2.5 foot intervals
Example:



Surface Style: FG Contours (1 and 5)
Description: Displays finished ground contours at 1 and 5 foot intervals
Example:



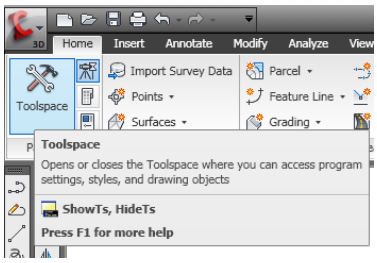
Surface Style: FG Contours (2 and 10)
Description: Displays finished ground contours at 2 and 10 foot intervals
Example:



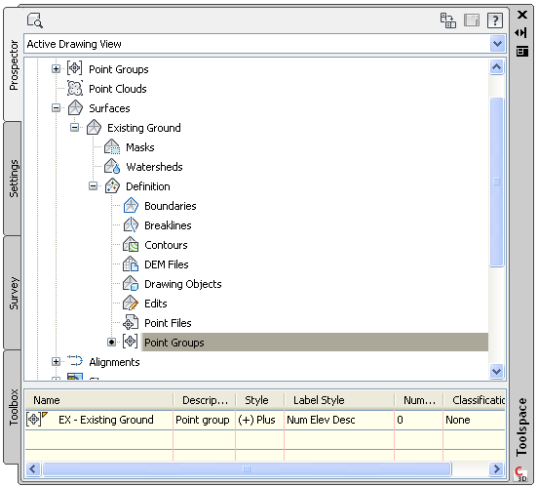
The Minnesota Civil 3D drawing template contains, by default, a terrain surface with the title *Existing Ground*. This surface is automatically created using survey points contained in the *EX – Existing Ground* point group. This point group is automatically populated with points representing the surveyed ground surface when point codes that follow the Minnesota standard survey point codes are used during the field survey. Refer to Quick Reference Guide *100.0 Minnesota Standard Survey Point Codes* for more information on the Minnesota standard survey point codes.

The points in this point group will automatically be added to a terrain surface with the title *Existing Ground* which models the topographic surface represented by the original ground survey points.

A. Defining the Surface



1. Surfaces are controlled through theToolspace window. To open the Toolspace window, click on the Toolspace icon on the Home ribbon.
2. Make sure you are on the *Prospector* tab in the *Toolspace* window.
3. Click on the plus sign next to *Surfaces* in the Prospector tab of the Toolbox to expand a listing of all of the surfaces in the drawing.
4. Find the surface with a title of *Existing Ground*.
5. Click on the plus sign next to *Existing Ground* to expand the listing of commands for the surface
6. Click on the plus sign next to *Definition* to access the list of data that you can use to define the *Existing Ground* surface.
7. If you click on the Point Groups listing under Definition, you will see in the window at the bottom of the Toolspace window that the EX – Existing Ground point group is already listed. By default, any point data in this point group will be used to define the Existing Ground surface.
8. You can always add more point groups to a surface definition by right clicking on the Point Groups listing and selecting Add… Select the point group that contains the data that you want to use to define the surface and click on the OK button.



B. Dynamic Updates

If a yellow shield with an exclamation point appears next to a surface or a component of a surface, this indicates that the surface needs to be rebuilt. To do this, right click on the surface name and select *Rebuild*.

You can also choose to automatically update a surface when it is modified by right clicking on the surface name and selecting *Rebuild – Automatic*.

C. Surface Display

The surface will be displayed in the drawing according to the surface style that is set for that surface. To change the surface style, right click on the *Existing Ground* title in the list of surfaces and select *Surface Properties…* The surface style is set on the *Information* tab of the *Surface Properties* window. The default surface style is *EX Contours (1 and 5)*, but this style can be changed by clicking on the down arrow next to the surface style and selecting a different style from the list of styles that are available.

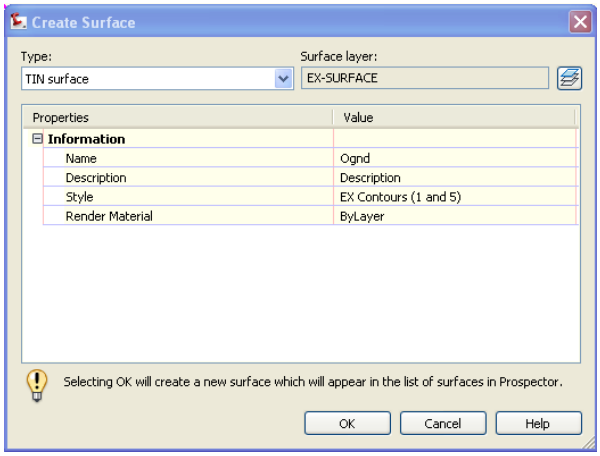
Refer Quick Reference Guide *400.0 Styles – Surface Styles* for more information on the surface styles available in the Minnesota drawing template.

(This Page Intentionally Left Blank)

The Minnesota drawing template contains a terrain surface with the title *Existing Ground* which models the topographic surface represented by the points located in the *PR – Existing Ground* point group. For more information on working with the *Existing Ground* surface, refer to Quick Reference Guide *410.0 Setting Up the Standard Existing Ground Surface.*

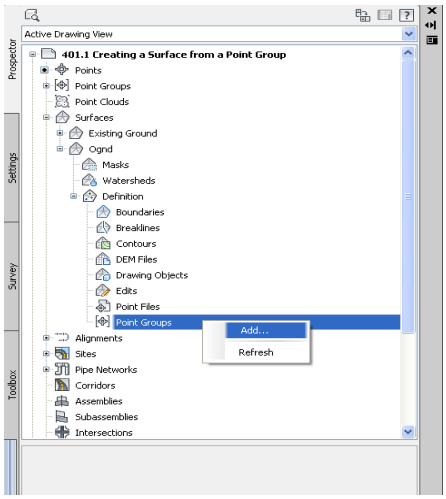
A. Add a new surface to the drawing

1. Surfaces are controlled through theToolspace window. To open the Toolspace window, click on the Toolspace icon on the Home ribbon.
2. Make sure you are on the *Prospector* tab in the *Toolspace* window.
3. Click on the plus sign next to *Surfaces* in the Prospector tab of the Toolbox to expand a listing of all of the surfaces in the drawing.
4. Right click on *Surfaces* and select *Create Surface…* from the shortcut menu.
5. In the *Create Surface* window, provide a name for the surface and select the display style that you want to use.



B. Define the surface

1. Click on the plus sign next to the name of the surface that you created in order to expand the listing of commands for that surface
2. Click on the plus sign next to *Definition* to access the list of data that you can use to define the surface.
3. Right click on the Point Groups definition option and select *Add…*
4. Select the point group that contains the data that you want to use to define the surface and click on the OK button.



C. Dynamic updates

If a yellow shield with an exclamation point appears next to a surface or a component of a surface, this indicates that the surface needs to be rebuilt. To do this, right click on the surface name and select *Rebuild*.

You can also choose to automatically update a surface when it is modified by right clicking on the surface name and selecting *Rebuild – Automatic*.

D. Surface display

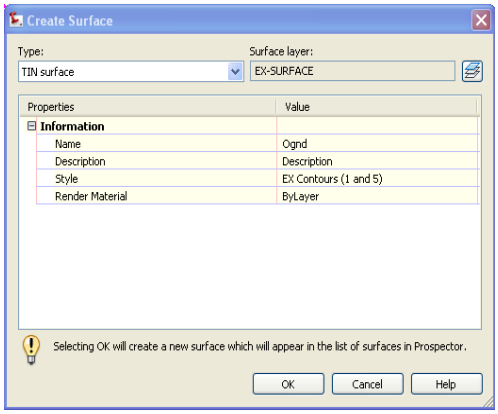
The surface will be displayed in the drawing according to the surface style that is set for that surface. To change the surface style, right click on the surface title in the list of surfaces and select *Surface Properties…* The surface style is set on the *Information* tab of the *Surface Properties* window. The default surface style is *EX Contours (1 and 5)*, but this style can be changed by clicking on the down arrow next to the surface style and selecting a different style from the list of styles that are available.

Refer Quick Reference Guide *400.0 Styles - Surface Styles* for more information on the surface styles available in the Minnesota drawing template.

(This Page Intentionally Left Blank)

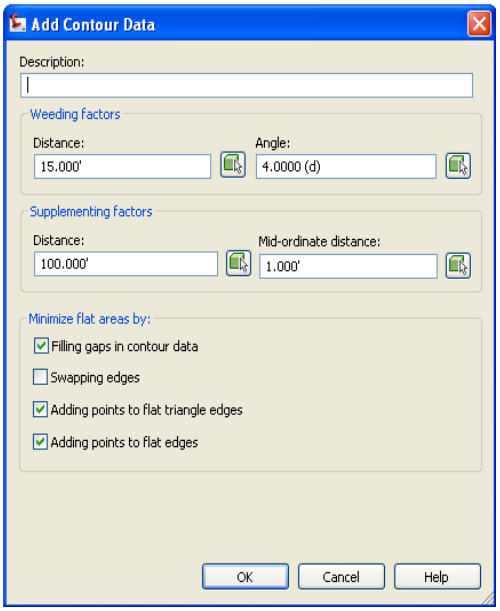
A terrain surface can be created using 2D polyline objects in a drawing. These polylines do not need to be closed polylines, but they do need to have elevations assigned to the polyline objects. The procedure below can be used to create a surface from 2D polylines.

A. Add a new surface to the drawing



1. Surfaces are controlled through theToolspace window. To open the Toolspace window, click on the Toolspace icon on the Home ribbon.
2. Make sure you are on the *Prospector* tab in the *Toolspace* window.
3. Click on the plus sign next to *Surfaces* in the Prospector tab of the Toolbox to expand a listing of all of the surfaces in the drawing.
4. Right click on *Surfaces* and select *Create Surface…* from the shortcut menu.
5. In the *Create Surface* window, provide a name for the surface and select the display style that you want to use.

B. Define the surface



1. Click on the plus sign next to the name of the surface that you created in order to expand the listing of commands for that surface
2. Click on the plus sign next to *Definition* to access the list of data that you can use to define the surface.
3. Right click on the *Contours* definition option and select *Add…*
4. The *Add Contour Data* window will appear. This window is used to provide a description for the contours, as well as other options for cleaning up the surface.

   You can specify weeding factors for the breakline(s). Weeding factors are used to simplify a complex breakline by removing vertices along the breakline. Weeding factors are not typically needed.

   You can also specify Supplementing factors for the breakline(s). Supplementing factors are used to add extra vertices on long sections of the breakline, but again are not usually needed.

   One common characteristic of a terrain surface that has been created using contours is that the surface will be flattened at high and low points in the surface, such as the bottom of channels and tops of hills. The Add Contour Data window contains several options that you can choose from to try to minimize flat areas in the surface.
5. Click on the *OK* button to exit from the *Add Contour Data* window.
6. You will return to the drawing where you will be prompted to select the contour objects to be used to define the surface.

C. Dynamic updates

If a yellow shield with an exclamation point appears next to a surface or a component of a surface, this indicates that the surface needs to be rebuilt. To do this, right click on the surface name and select *Rebuild*.

You can also choose to automatically update a surface when it is modified by right clicking on the surface name and selecting *Rebuild – Automatic*.

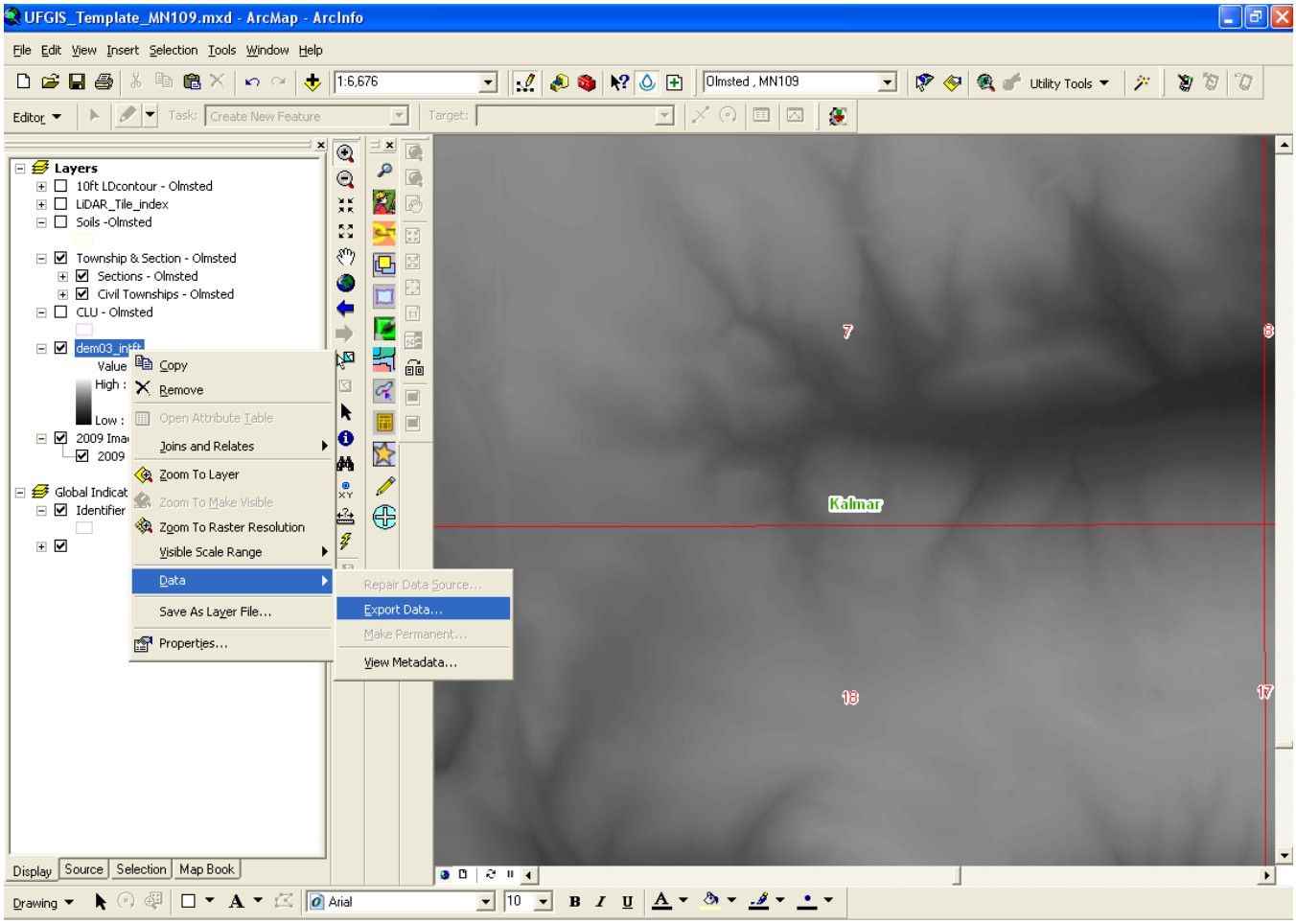
D. Surface display

The surface will be displayed in the drawing according to the surface style that is set for that surface. To change the surface style, right click on the surface title in the list of surfaces and select *Surface Properties…* The surface style is set on the *Information* tab of the *Surface Properties* window. The default surface style is *EX Contours (1 and 5)*, but this style can be changed by clicking on the down arrow next to the surface style and selecting a different style from the list of styles that are available.

Refer Quick Reference Guide *400.0 Styles - Surface Styles* for more information on the surface styles available in the Minnesota drawing template.

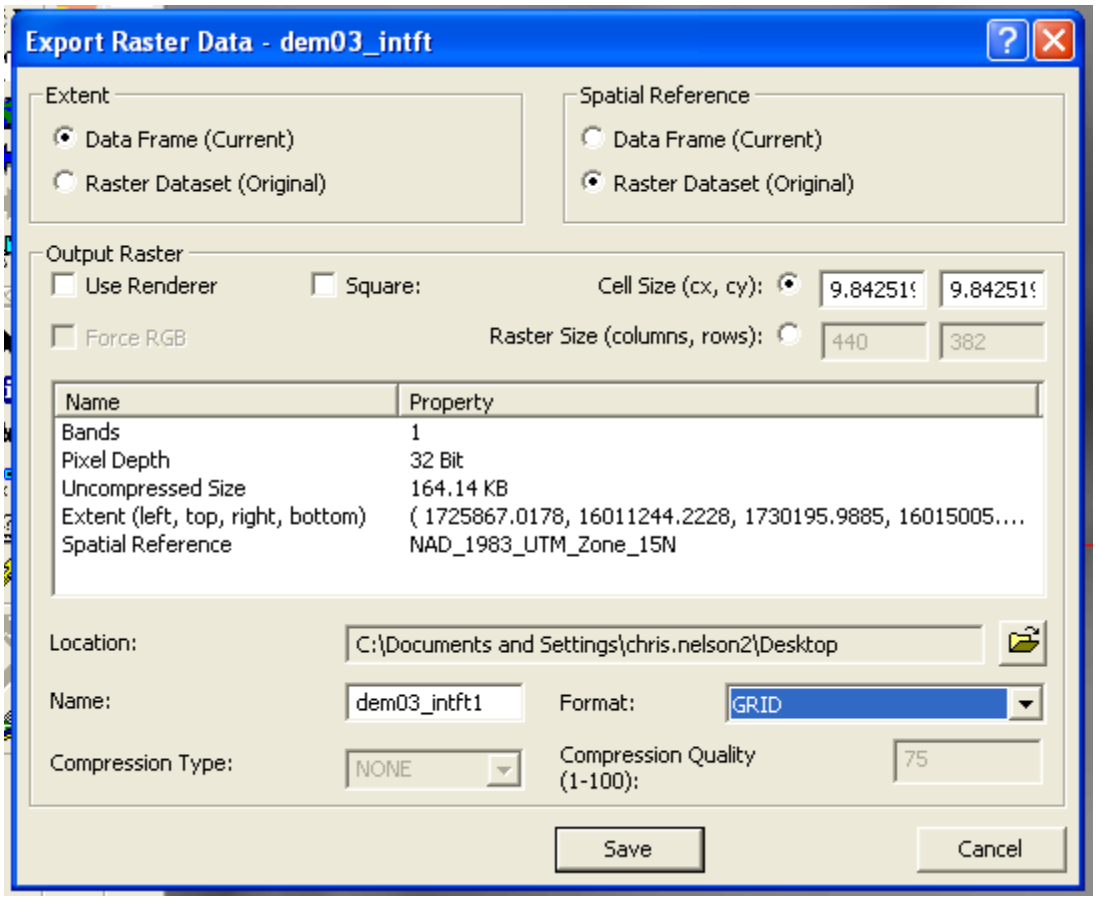
This guide covers the procedure of importing a LIDAR DEM into a Civil 3D surface. You must have a DEM converted to feet to be able to use it effectively in Civil 3D. The original DEM's can be exported right out of ArcMap into Civil 3D but the elevation that is originally in meters does not change. This could be changed by raising the surface by a conversion from meters to international or U.S. Survey feet. A tool can be developed to do this automatically in ArcMap.

Begin by opening ArcMap and starting with a county template or even a blank template.
-Add the dem03inft
-Zoom to the area of interest
(Include all the area that you want to use in Civil 3D in the data frame)
-Right-click on the dem03inft layer
-Data
-Export Data



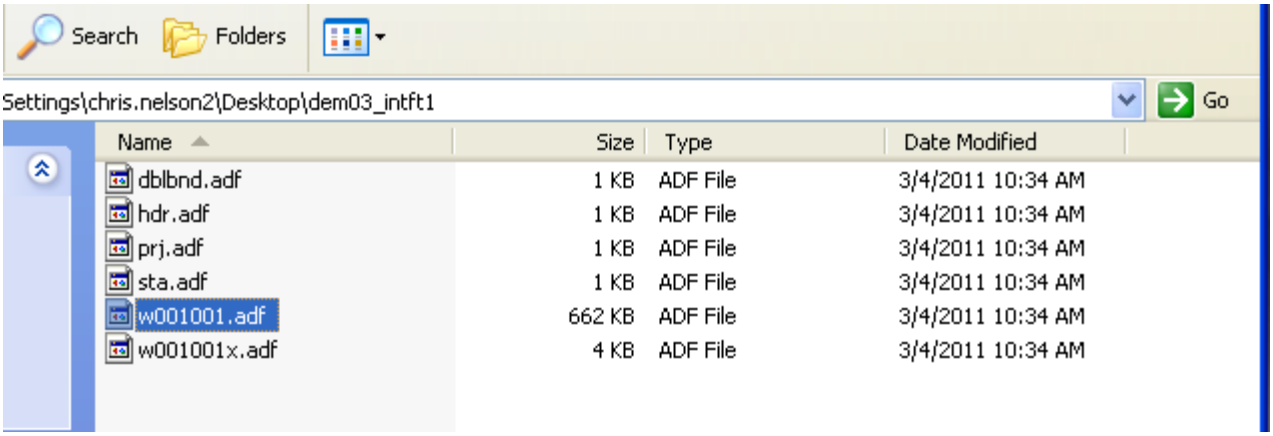
The following window will appear and check for the following settings:

-Extent
    -Data Frame (Current)
-Spatial Reference
    -Raster Dataset (Original)
-Location
    -Choose the folder where you want the files to be places
-Format
    -Grid
-Click **Save**
---You will be asked if you want to add the data to the map as a layer which you can say no because you won't need it in ArcMap.

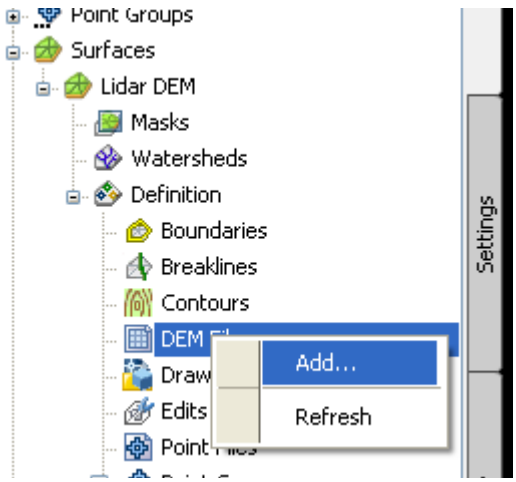


**All of the other settings should be the default settings that appear automatically.

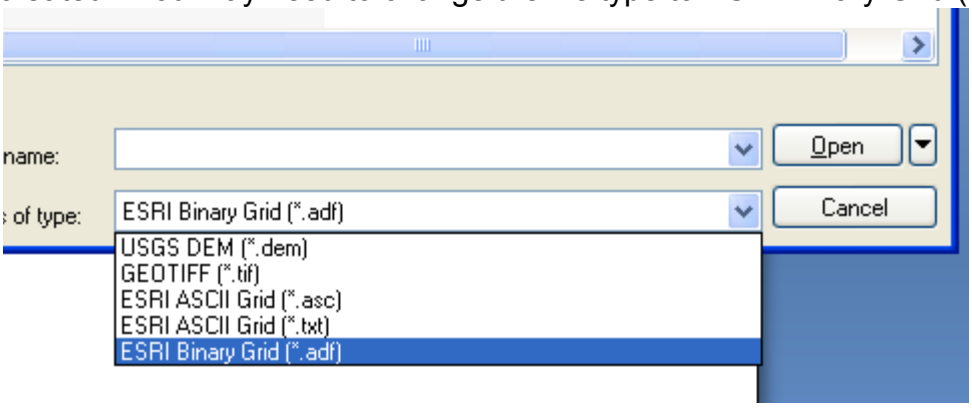
This process will develop a series of files located in the location that you selected that are in an **.adf** format. The file that you are looking for is the largest file in the folder.



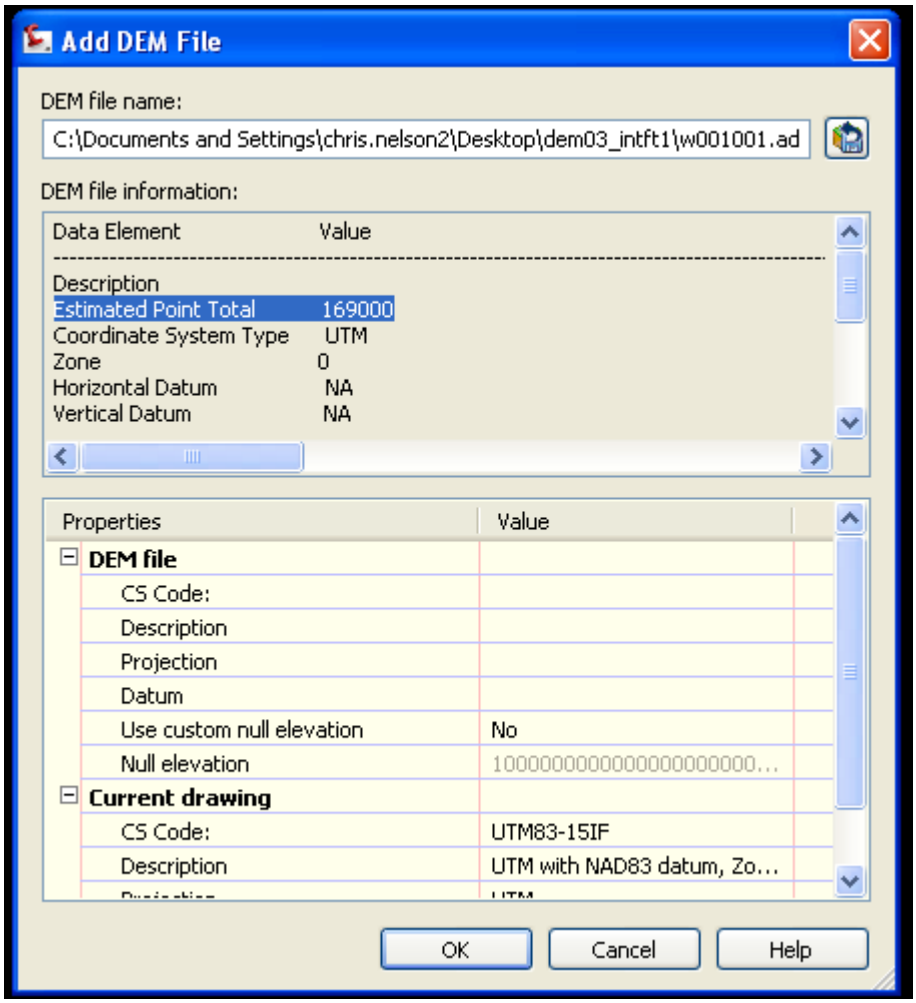
Open Civil 3D and go to the Toolspace and either create a new surface called Lidar Dem or use one that is already created. Expand the surface and navigate to the definition and right-click on DEM Files and select Add.



A window will appear that will allow you to navigate to the location of your DEM that you created. You may need to change the file type to ESRI Binary Grid (*.adf).

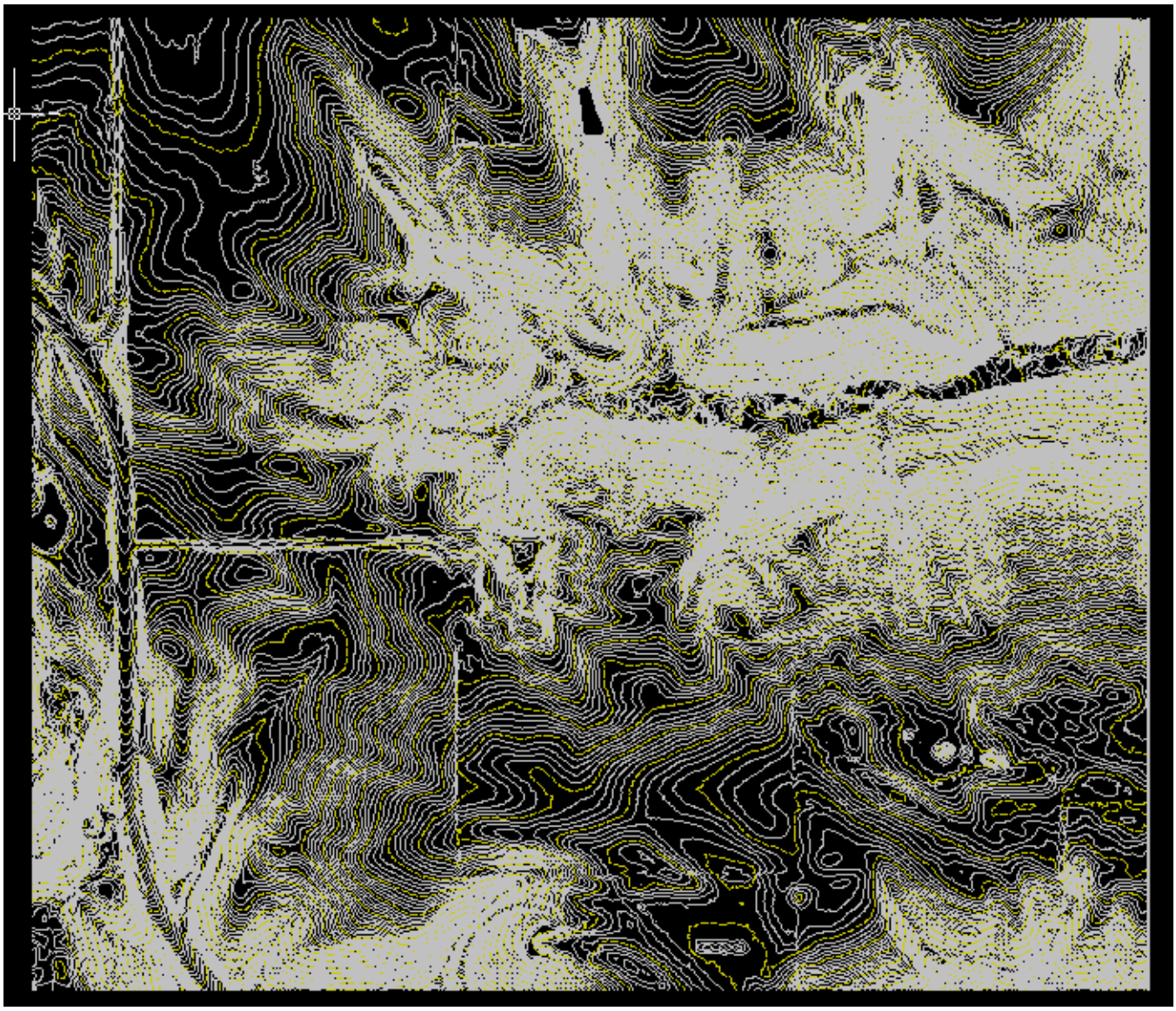


Choose the appropriate file and click open. This will populate the DEM file information in the previous window. You will be able to see the estimate point total listed in this window.



Click OK.

Zoom to the extents of the drawing to see that the surface was created. You will notice on the right side of the surface there will be an area that seems to be referencing a “zero” elevation. This can be remedied by adding a boundary to the surface.



Keep in mind that larger surfaces like this have a tendency to slow the functionality of the system and have caused the program to lock up at times. It seems that if the size of the DEM is kept around the 150kb size it is more manageable and that may be remedied by increasing the virtual memory on the computer.

(This Page Intentionally Left Blank)

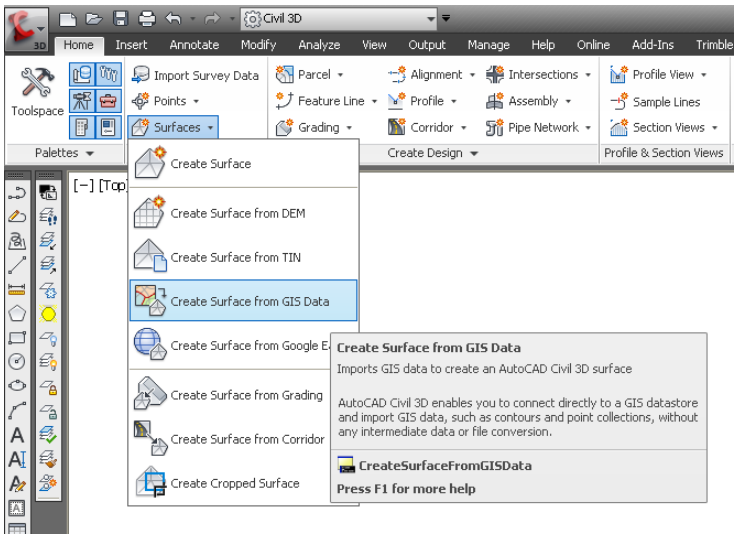
This guide covers the procedure for creating a terrain surface in Civil 3D using a shapefile which contains elevation data in the form of contours. The shapefile that is being used to create the surface is located on a local hard drive or shared server.

Make sure a coordinate system is assigned to the Drawing

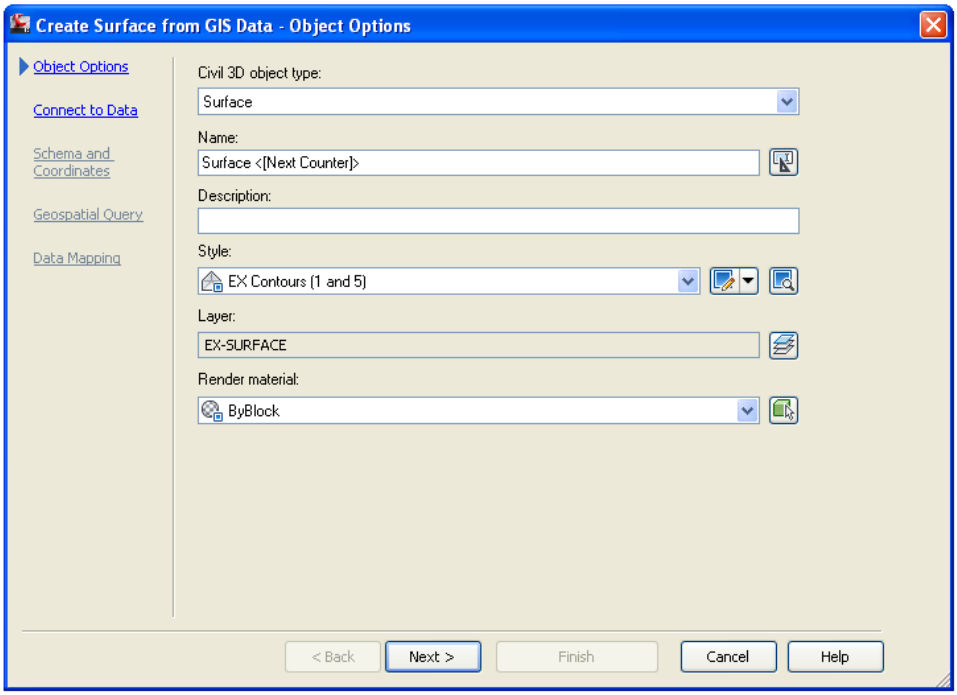
1. Go to the Settings tab on the Toolspace.
2. Right click on the drawing name and select *Edit Drawing Settings…*
3. In the *Drawing Settings* window, go to the *Units and Zone* tab.
4. Select the coordinate system for the drawing from the list of available coordinate systems in the *Zone* section.

Create a Surface from the Shapefile

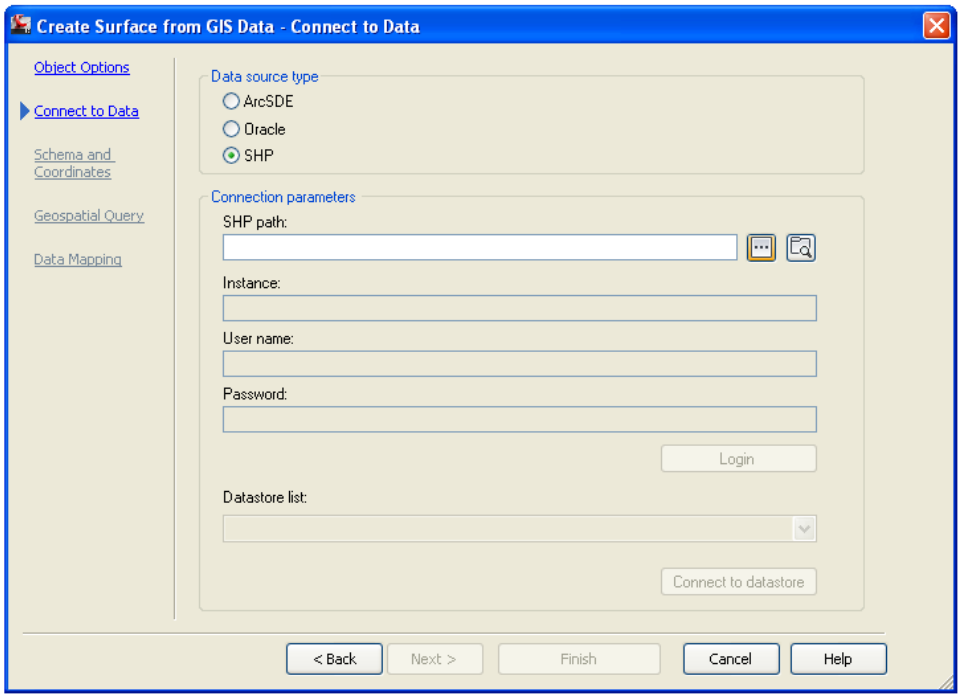
1. Click on the down arrow next to *Surfaces* on the *Create Ground Data* panel on the *Home* ribbon and select *Create Surface from GIS Data*.



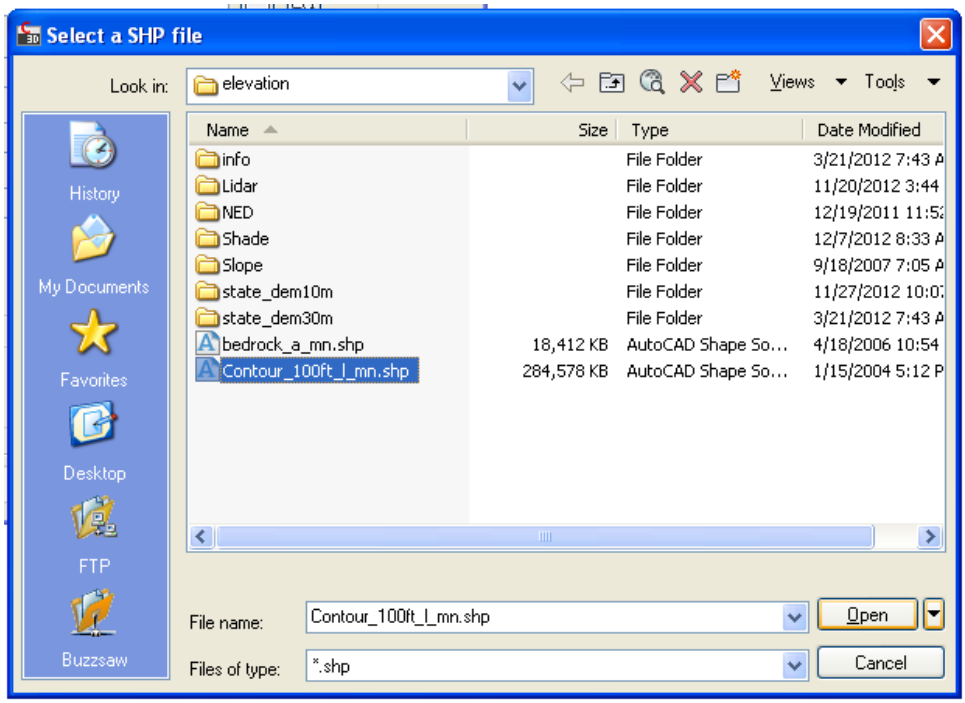
2. The Create Surface from GIS Data window will open. The Object Options tab is used to provide a name and optional description for the surface and to select a style, layer and rendering material.



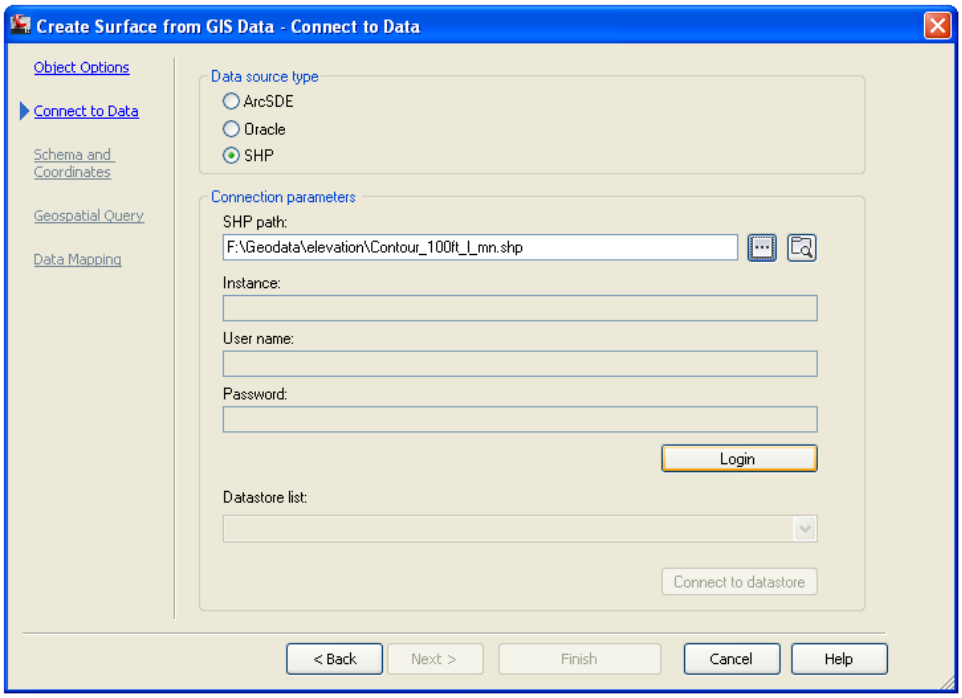
1. Click on the Next > button to open the Connect to Data window. There are three types of source data that can be used, ArcSDE, Oracle or SHP. For this exercise, select SHP.



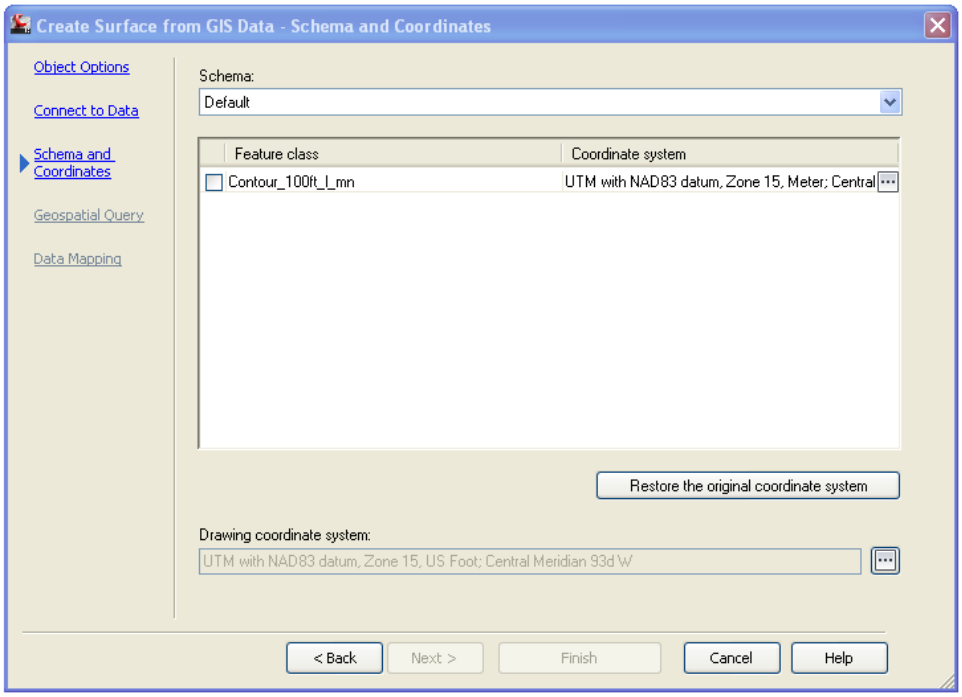
2. Select the shapefile by clicking on one of the icons next to *SHP path:* The icon on the left will allow you to browse and select an individual .shp file, while the icon on the right allows you to select all of the shapefiles in a folder. Browse and select the shapefile and click on the *Open* button.



3. Back in the *Create Surface from GIS Data – Connect to Data* window, click on the Login button.

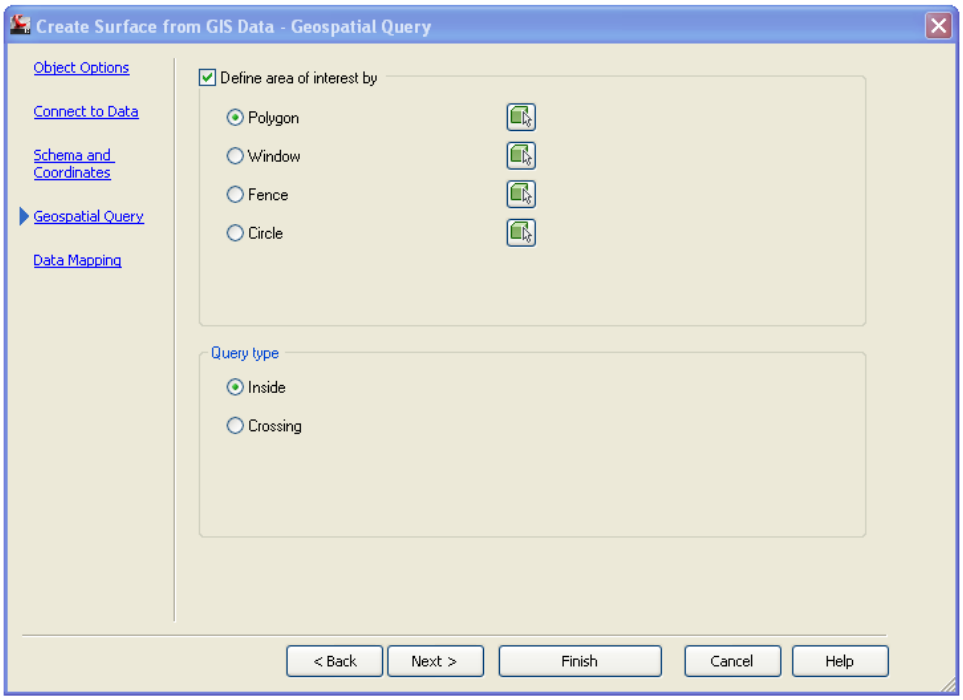


4. The Create Surface from GIS Data – Schema and Coordinates window will appear after the login completes (this may take a few seconds). The shapefile name and its coordinate system will appear in the center window, and the drawing's coordinate system will be listed at the bottom of the window.

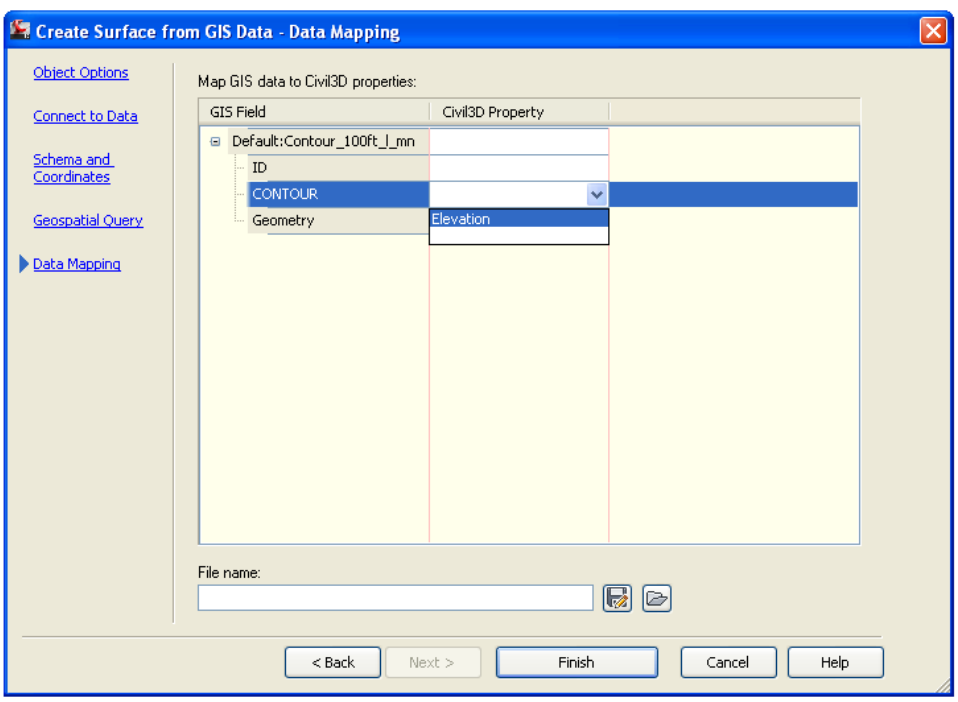


Check the box next to the shapefile name and click on the *Next* > button.

5. In the *Create Surface from GIS Data – Geospatial Query* window, you can define an area of interest in order to limit the area of the surface, and you can choose a query type. Uncheck the box next to *Define area of interest by* to use all of the contours in the shapefile to build the surface. Click on the *Next >* button.



6. In the *Create Surface from GIS Data – Data Mapping* window, click on the cell next to CONTOUR in the Civil3D Property column to assign elevation values from the shapefile contours to the Civil 3D surface.



7. Click on the Finish button to create the surface.

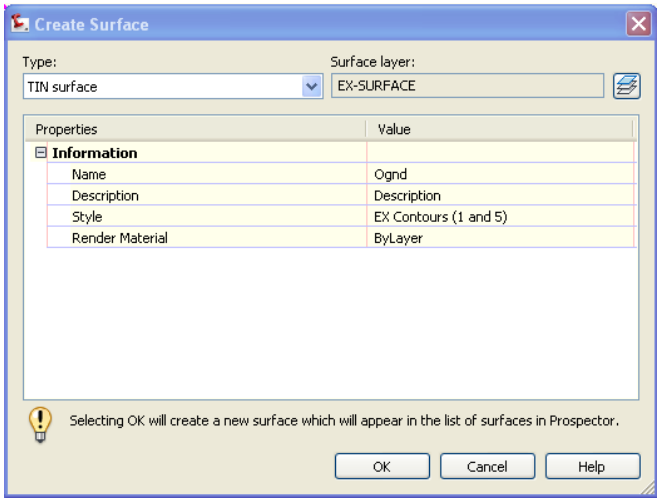
This guide covers the procedure for creating a terrain surface in Civil 3D using an .adf format GIS data file. The file that is being used to create the surface is located on a local hard drive or shared server.

A. Make sure a coordinate system is assigned to the Drawing

1. Go to the Settings tab on the Toolspace.
2. Right click on the drawing name and select *Edit Drawing Settings…*
3. In the *Drawing Settings* window, go to the *Units and Zone* tab.
4. Select the coordinate system for the drawing from the list of available coordinate systems in the *Zone* section.

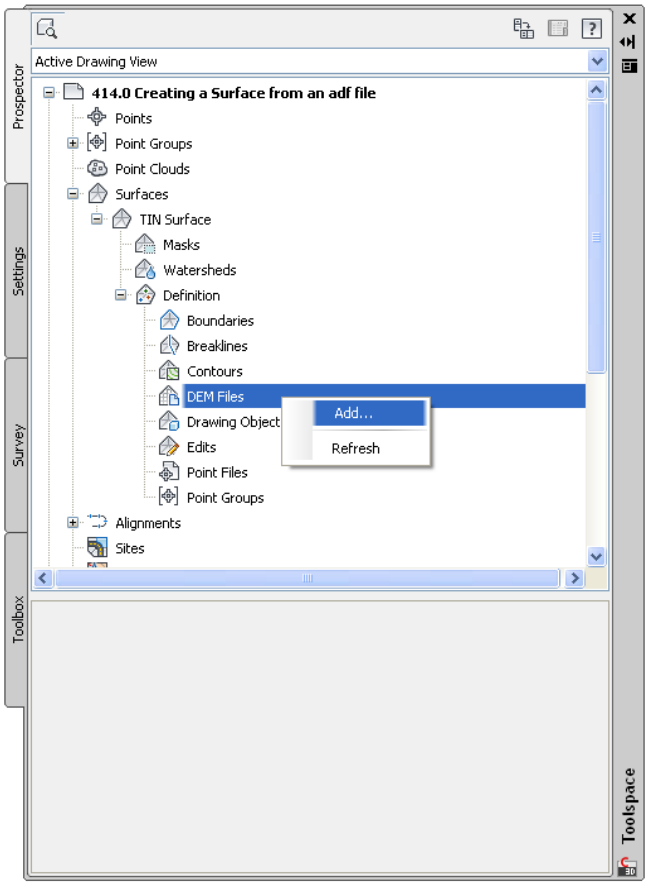
B. Add a new surface to the drawing

1. Surfaces are controlled through the Toolspace window. To open the Toolspace window, click on the Toolspace icon on the Home ribbon.
2. Make sure you are on the *Prospector* tab in the *Toolspace* window.
3. Click on the plus sign next to *Surfaces* in the Prospector tab of the Toolbox to expand a listing of all of the surfaces in the drawing.
4. Right click on *Surfaces* and select *Create Surface…* from the shortcut menu.
5. In the *Create Surface* window, provide a name for the surface and select the display style that you want to use. Because DEM datasets are usually very large, it can help speed up the process if you select a surface style of No Display or Border.

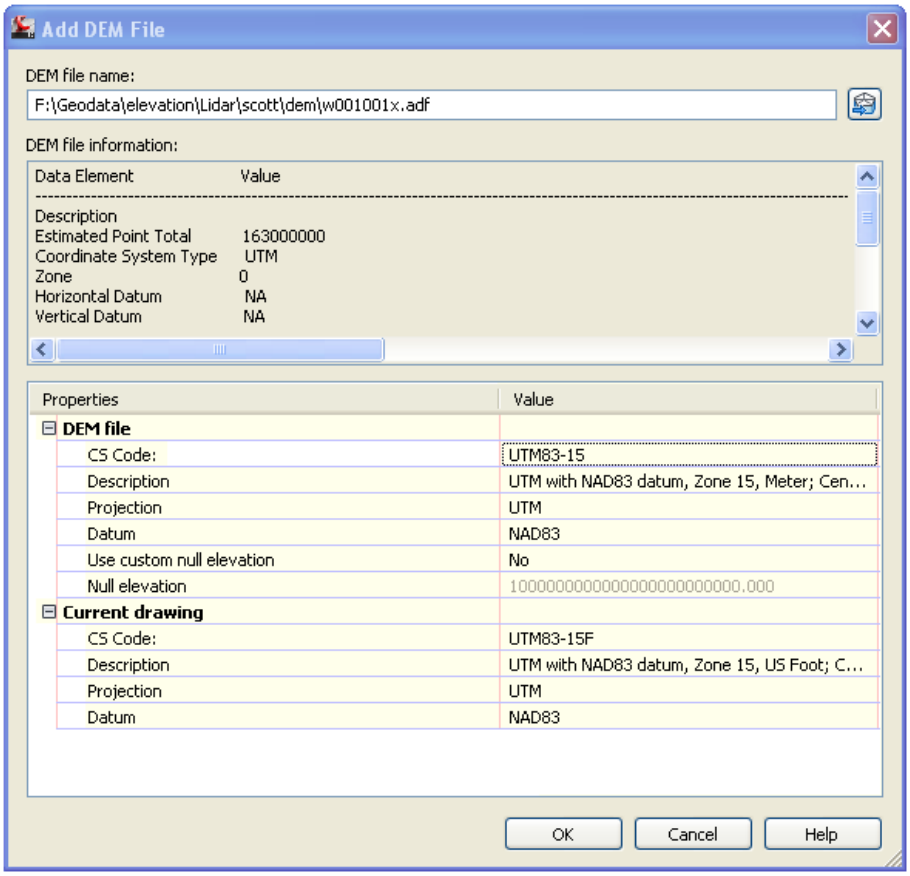


C. Define the surface

1. Click on the plus sign next to the name of the surface that you created in order to expand the listing of commands for that surface
2. Click on the plus sign next to *Definition* to access the list of data that you can use to define the surface.
3. Right click on the *DEM Files* definition option and select *Add…*

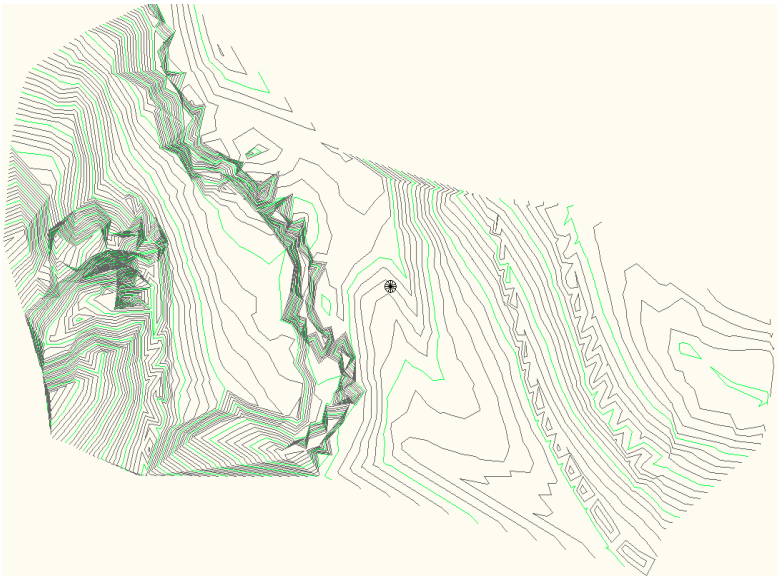


4. In the *Add DEM File* window, browse and select the .adf file that will be used to develop the terrain surface and click on the Open button to return to the *Add DEM File* window. If the coordinate system for the DEM file is not automatically set, you can manually specify its coordinate system by clicking in the box in the Value column next to CS Code:



5. Click on the OK button to add the DEM data to the surface.

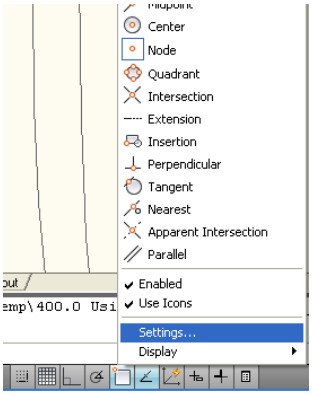
The appearance of a surface created from point data will often appear choppy in areas where there is a significant change in grade such as channels and the top and bottom of banks. This is illustrated in the graphic to the left. The surface can be refined in these areas by using breaklines to define these grade change locations. Breaklines force the surface model's triangulation lines to orient along the grade break rather than crisscrossing across breaks, which causes potholes in channel areas and choppy ridge lines along banks.

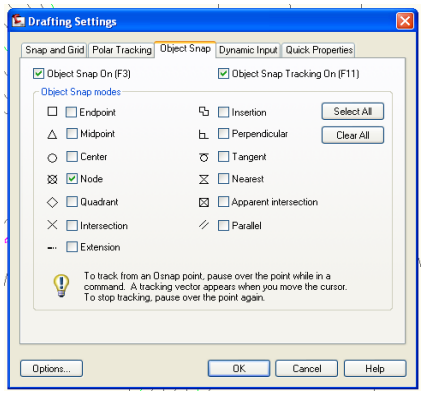


A breakline is a 3D polyline that has elevation values assigned to its vertices. The easiest and most accurate way to create a 3D polyline to be used as a breakline is to create a 3D polyline by snapping to survey points taken at grade breaks, top or bottom of bank locations, or other points where the terrain changes significantly.

The procedure below outlines the steps that can be followed to create a 3D polyline that will be used as a breakline.

1. Make sure the points that you want to use to create the 3D polyline are displayed in the drawing.
2. You will want to be able to snap to the nodes of the points in the drawing in order to pick up the points elevation, but you do not want to accidentally snap to any other objects in the drawing. To restrict the object snapping to selecting only point nodes, right click on the *Object Snap* icon on the *Application Status Bar* at the bottom right hand side of the window and click on *Settings*…



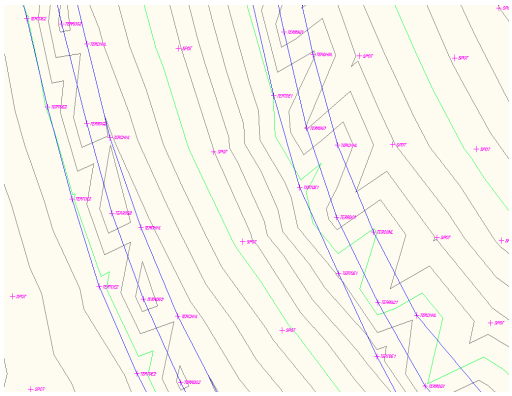


In the Drafting Settings window, click on the *Clear All* button and then select only *Node*. Click on the *OK* button.

3. Draw the 3D polyline using the 3D polyline command located on the *Draw* panel of the *Home* ribbon. You will need to click on the down arrow on the *Draw* panel to expand the command list and find the 3D polyline command. Click on points in the drawing to set the nodes for the 3D Polyline object. When you reach the end of the 3D polyline, right click and select *Enter*.

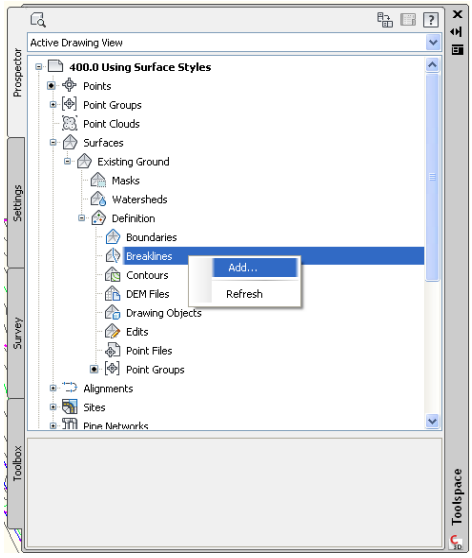
   If you click the wrong point you can back up by right clicking and selecting *Undo* from the shortcut menu.

   The graphic to the right shows 3D polyline breaklines that have been created along a set of terraces. These 3D polylines will be used to clean up the terrain model along these terraces.



Breaklines are added to a surface in the *Definition* area under the surface name in the Toolspace. The following commands are used to add breaklines to a surface definition.

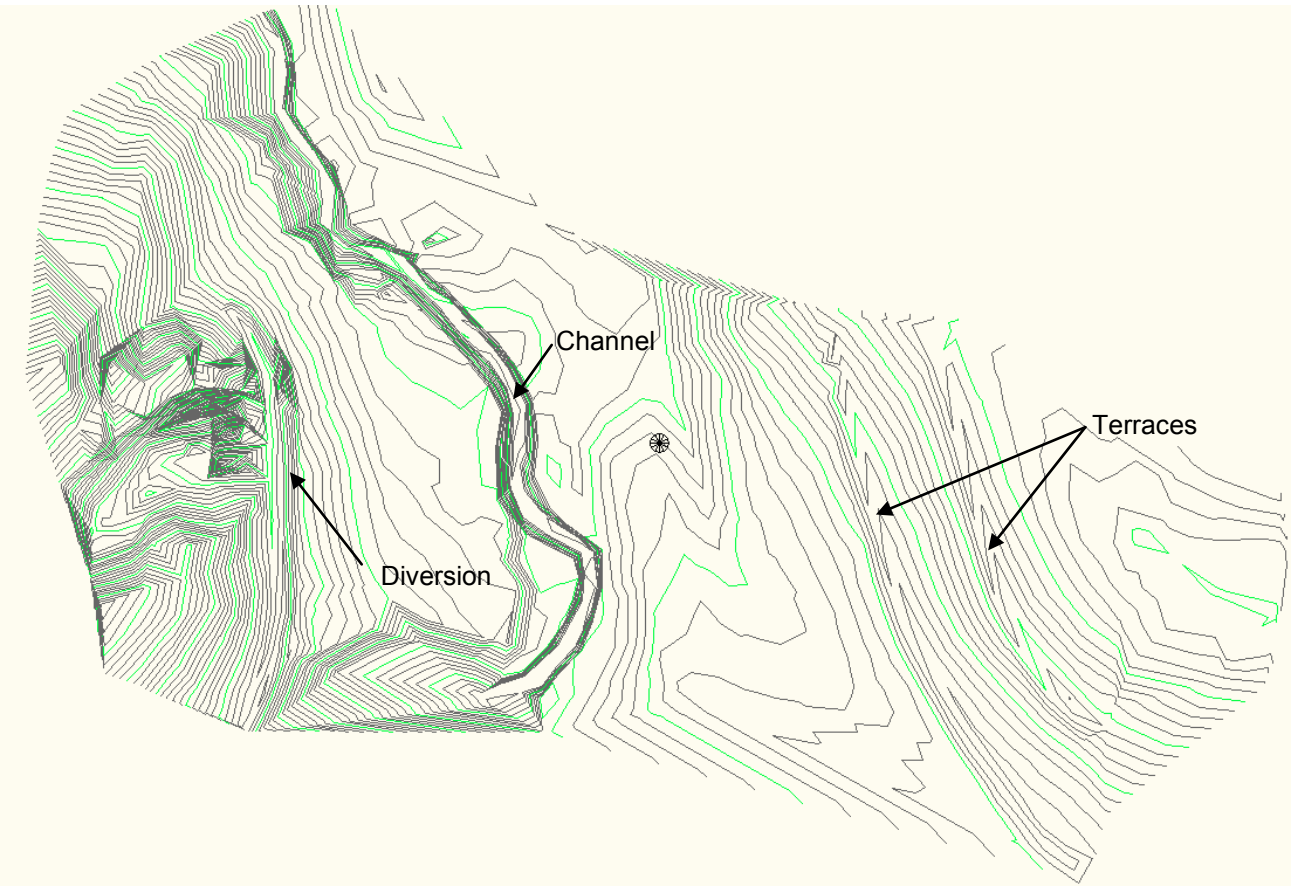
1. On the *Prospector* tab of *Toolspace*, go to the surface name and expand the list of commands by clicking on the + sign to the left of the surface name.



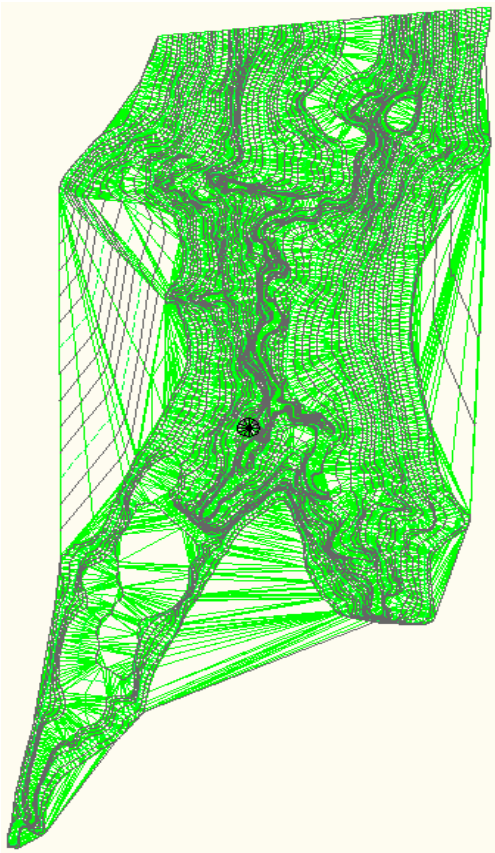
2. Expand the list under *Definition*, right click on *Breaklines* and select *Add…*
3. The *Add Breaklines* window will appear. The following selections can be made In this window
   a. You can provide a description for the breakline(s)
   b. Make sure the breakline type is set to *Standard*

      The most commonly used type of breakline is the Standard breakline. This is used for breaklines created from a 3D polyline that follows surveyed point data along a grade break, such as the top of a bank. Another type of breakline is the proximity breakline. This type of breakline will interpolate a three dimensional breakline along a 2D polyline based on the elevations of a surface.
   c. You can specify weeding factors for the breakline(s). Weeding factors are used to simplify a complex breakline by removing vertices along the breakline. Weeding factors are not typically needed.
   d. You can also specify Supplementing factors for the breakline(s). Supplementing factors are used to add extra vertices on long sections of the breakline, but again are not usually needed.

The graphic below shows the same surface that was shown on page 1, but it has been refined by adding breaklines along the channel, diversion and terraces.

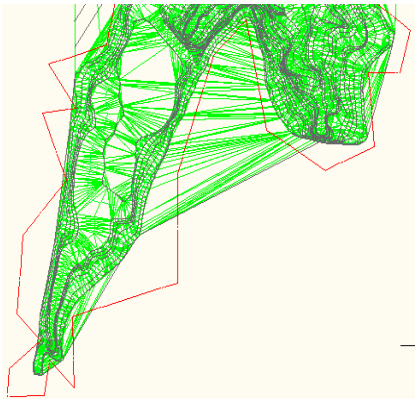


A boundary can be used to clean up the outer edges of a surface model where TIN lines may extend across space where survey data is not provided. This is illustrated in the figure below.

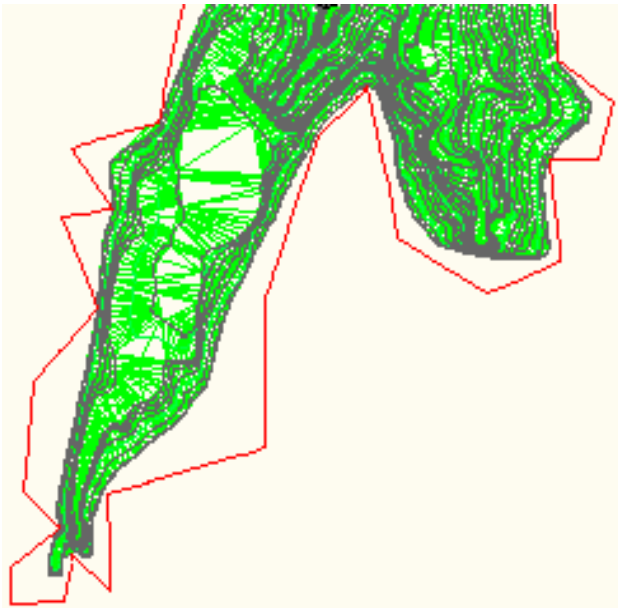


A boundary is created using a closed 2D polyline object which crosses over all TIN lines that you want to remove from your surface model. Start by changing the display of your surface to show only the triangles. It will also help to turn off object snapping and polar tracking while you are drawing the polyline so that the program is not continually trying to find nodes to snap to while you are drawing the polyline.

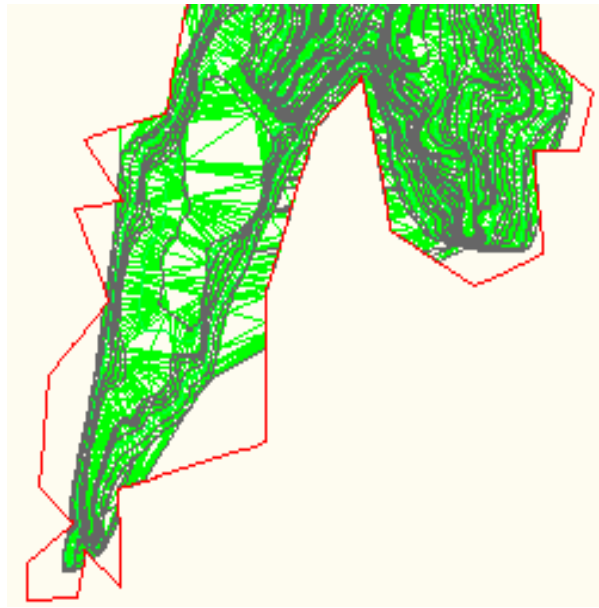
Start drawing the 2D polyline at any point outside of your surface and start drawing a polyline all the way around the surface back to the starting point, crossing over all TIN lines that you want to remove along the way, as shown in the figure to the right. When you return to the starting point of the polyline you can type *C* at the command line to close the polyline.



Once you are finished drawing the polyline, you will add it to the surface definition. In the *Definition* area under the surface name, right click on *Boundaries* and select *Add…* In the Add Boundaries window, you can provide a name for the boundary if you want to and make sure the type of boundary is set to *Outer.* Also make sure the box next to *Non-destructive breakline* is unchecked. Leaving this box checked will cause the TIN lines to be trimmed at the location where the boundary line crosses them rather than deleting the lines altogether. Examples of how the surface is modified when applying an outer boundary are shown below. Click on the OK button and rebuild the surface if necessary to see the changes.

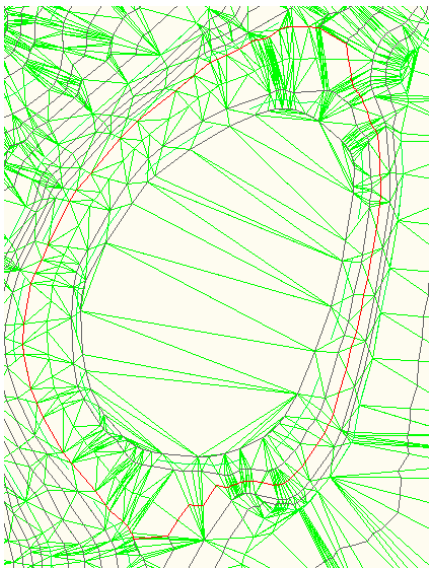


Non-destructive breakline box unchecked



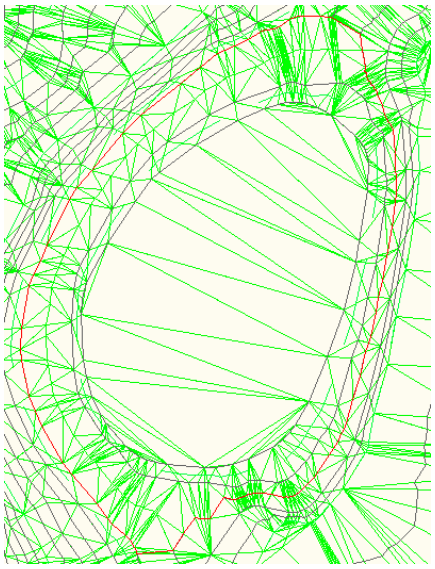
Non-destructive breakline box checked

Two other types of boundaries that are commonly used to define flat or hidden areas inside the terrain surface, such as the water surfaces in a reservoir, are the *Hide* and *Show* boundaries. A *Show* boundary creates a flattened area in the surface defined by the boundary polyline. A *Hide* boundary is similar to the *Show* boundary, but it cuts out the area defined by the boundary polyline from the surface model, leaving a hole. An example of how these two boundaries can be used to modify a surface are shown below.



Original Surface

Hide Boundary
Non-destructive breakline checked



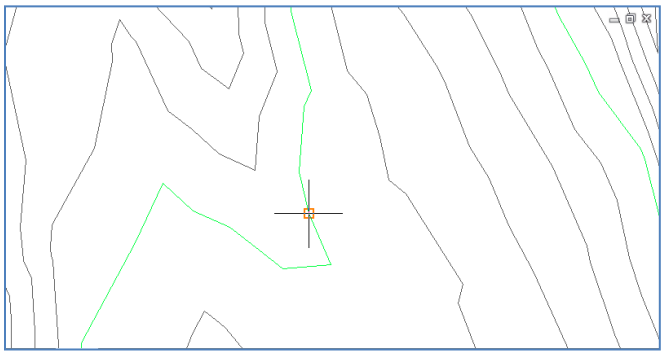
Show Boundary
Non-destructive breakline checked

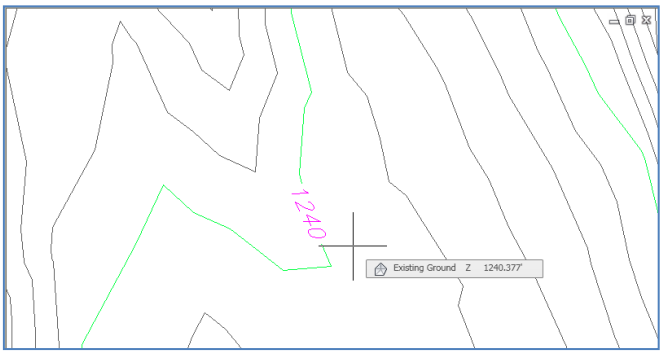
The elevations of surface contours can be labeled using contour labels. This guide covers the three types of labels that are available and how they are applied to contours in a surface.

To access the commands for adding contour labels to a surface, select the surface object in the drawing. A contextual ribbon for that surface will appear. The commands for contour labels are located on the *Add Labels* drop down menu on the *Labels & Tables* panel. Each of the contour label commands is covered below.

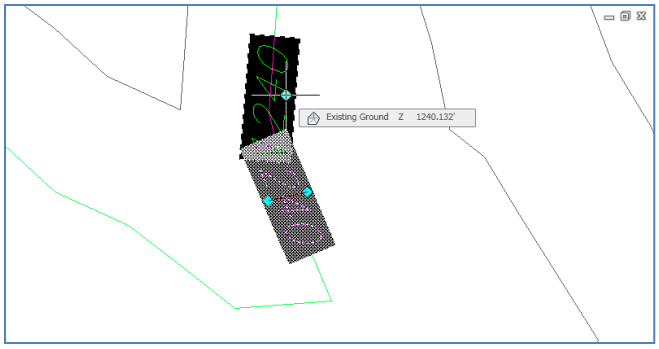
Contour – Single

The Contour – Single command places an elevation labels at a point that you select along the contour line. When you execute this command, you will be prompted to select the contour that you want to place the label on (bottom left). When you click on a location on a contour line, the label will be placed at the place on the contour (bottom right). Continue to add individual contour labels or right click with the mouse to exit the command.





Labels can be moved by selecting them and using their grips to slide them along the contour line.



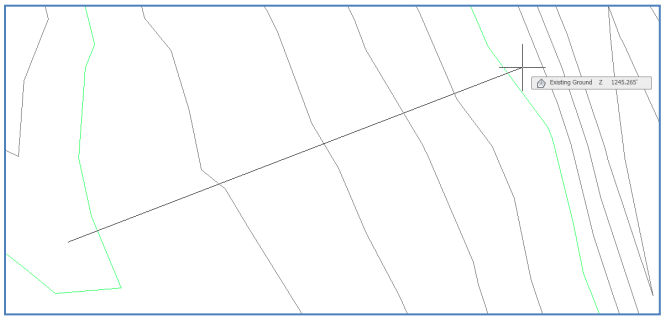
To delete a contour label, simply select the label and hit the delete button on the keyboard.

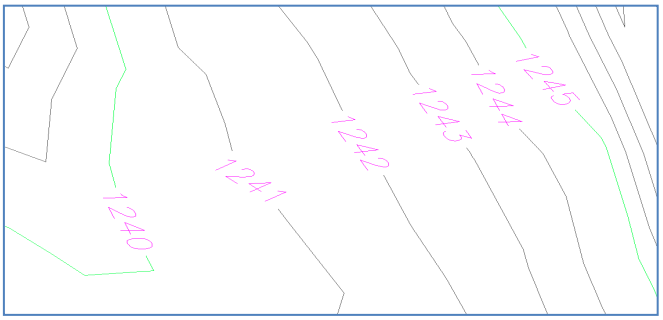
Contour – Multiple

The Contour – Multiple command places a series of elevation labels along a line that you draw across contours.

When you execute this command, you will be prompted to select the beginning and ending points of a line that crosses over the contours that you want to place labels on (bottom left). The labels will be placed where the line that you draw crosses over the contours (bottom right).

Continue to draw lines across contours or right click with the mouse to exit the command.

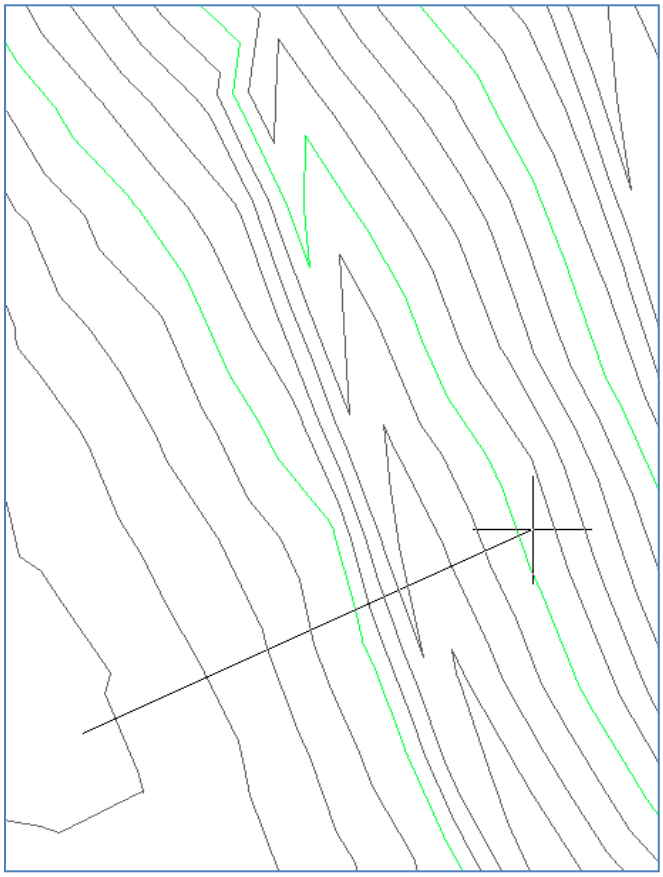


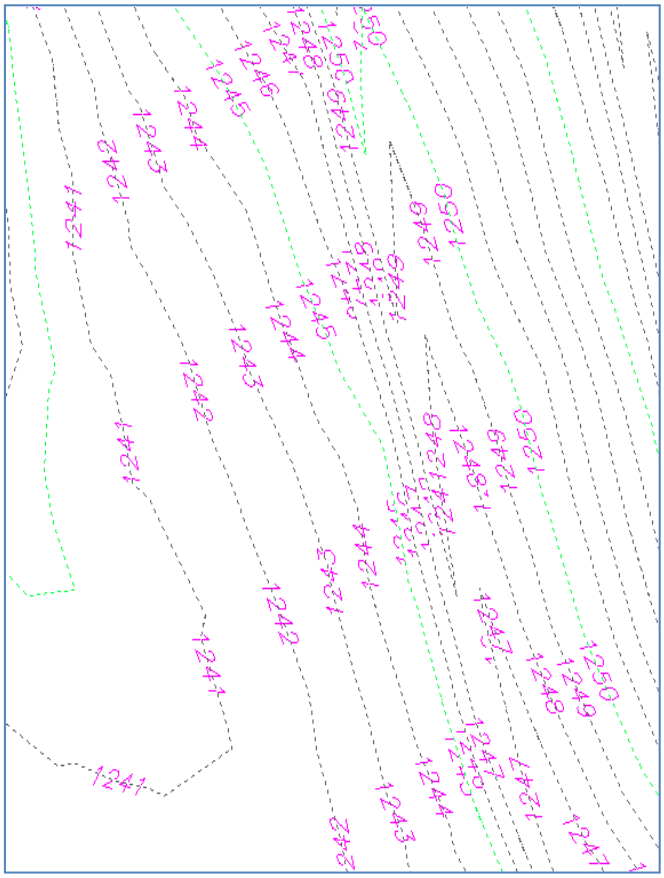


Contour – Multiple at Interval

The Contour – Multiple at Interval command places a series of elevation labels at a set distance interval along the contours that you select.

When you execute this command, you will be prompted to select the beginning and ending points of a line that crosses over the contours that you want to place labels on (bottom left). After you draw the line across contours, you will be prompted to specify an interval for the contour labels along the contour. The labels will be placed on the contours that you cross over with the line at the interval that you specify. (bottom right).





Spot elevations and slope labels can be placed on a surface. This guide covers the three types of spot labels that are available and how they are placed in a drawing.

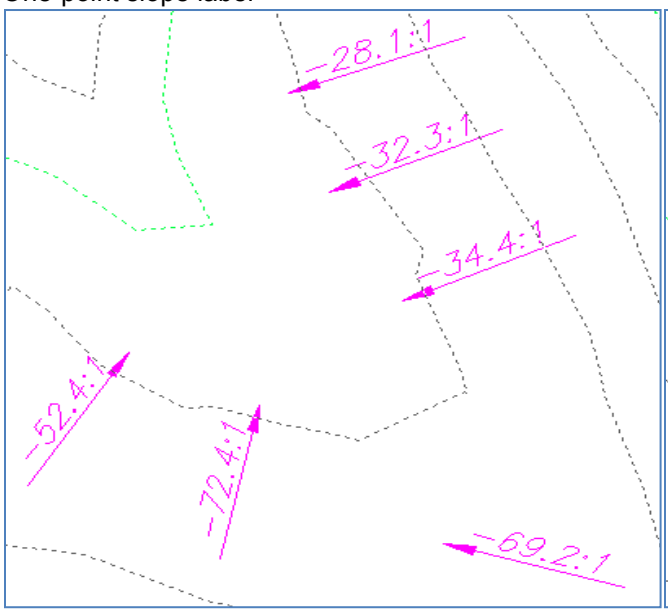
To access the commands for adding spot labels to a surface, select the surface object in the drawing. A contextual ribbon for that surface will appear. The commands for spot labels are located on the *Add Labels* drop down menu on the *Labels & Tables* panel. Each of the spot label commands are covered below.

Slope

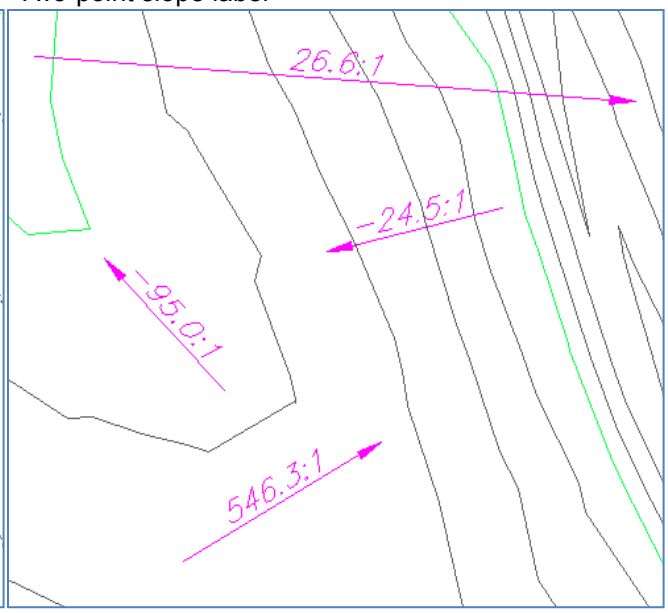
When you select the slope label command, you will have an option of creating a one-point or two-point slope label. The one-point slope label command will place a label listing the slope (in H:V format) at the point that you click on the surface. An arrow will automatically be placed under the slope label which indicates the direction of the slope.

If you select the two-point slope label command, you will be asked to specify the beginning and ending points on the surface that you want to calculate the slope between. The slope (in H:V format) will be placed in the drawing, along with an arrow drawn starting at the first point you selected and pointing toward the second point selected.

One-point slope label

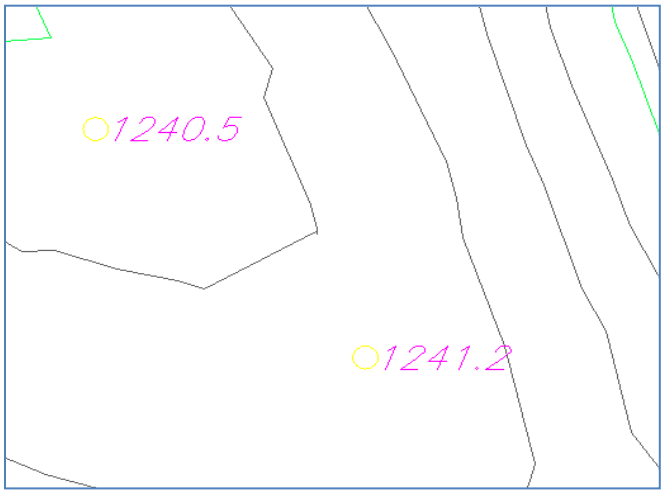


Two-point slope label



Spot Elevations

When you select the spot elevation command, you will select a point in the drawing where you want to display the spot elevation of a surface. The elevation label will be placed in the drawing along with a symbol indicating the point where the elevation was measured.



Spot Elevations on Grid

This command places spot elevation labels at a specific grid interval on a surface.

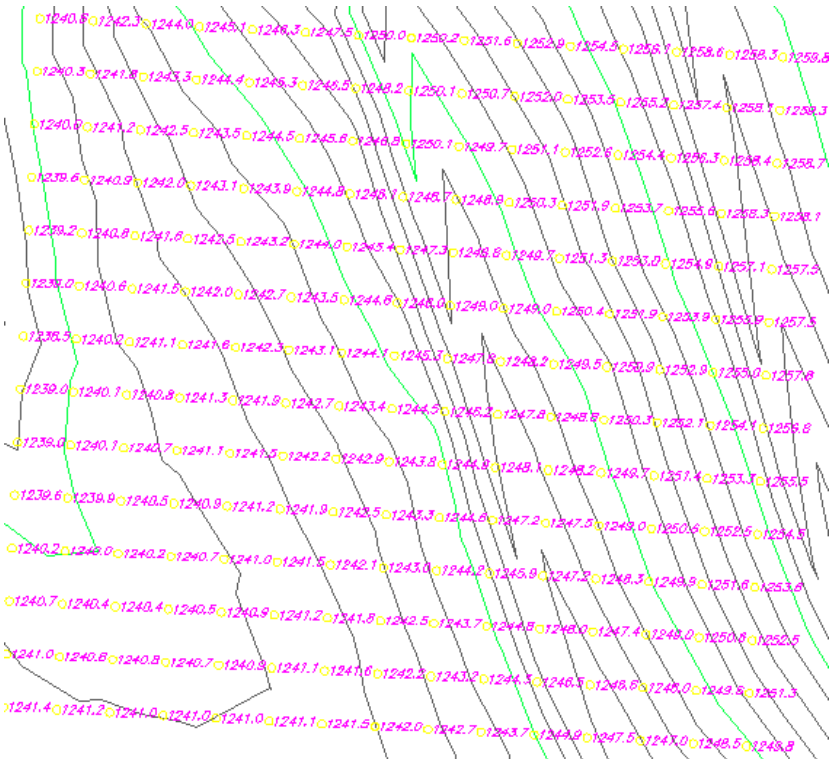
When you start the command, you will be asked to specify a base point for the grid of labels.

Next, you will be asked for a grid rotation angle. You can either click on a second point in the drawing to define the rotation angle, or you can enter in a rotation angle value at the command line.

After you specify the rotation angle you will be asked for the grid x and y spacing.

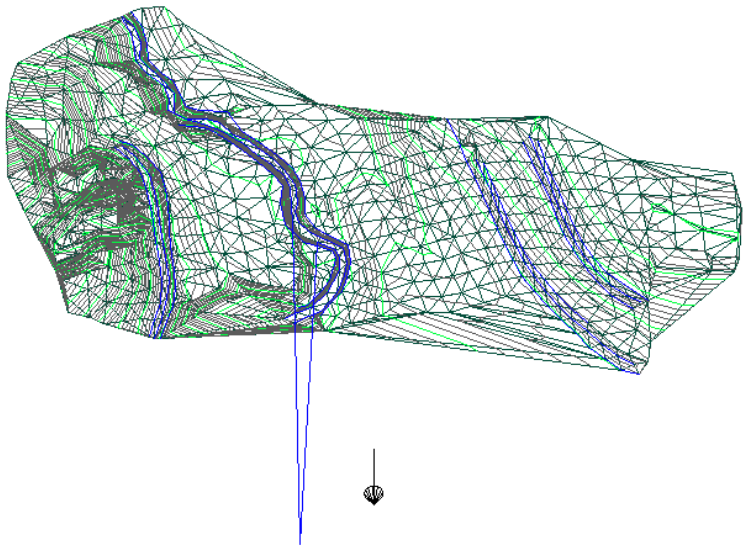
Next, you will be asked to specify the upper right location for the grid. The grid spacing you provided in the previous step will be displayed, and you can select on one of the grid points to specify the upper right hand corner of the window.

As a last step, the grid location window will be displayed in the drawing along with a smaller square that represents the x and y spacing that you specified. You will be given the option of changing size or rotation of the grid or grid squares at this point, or you can proceed to placing the spot elevations.

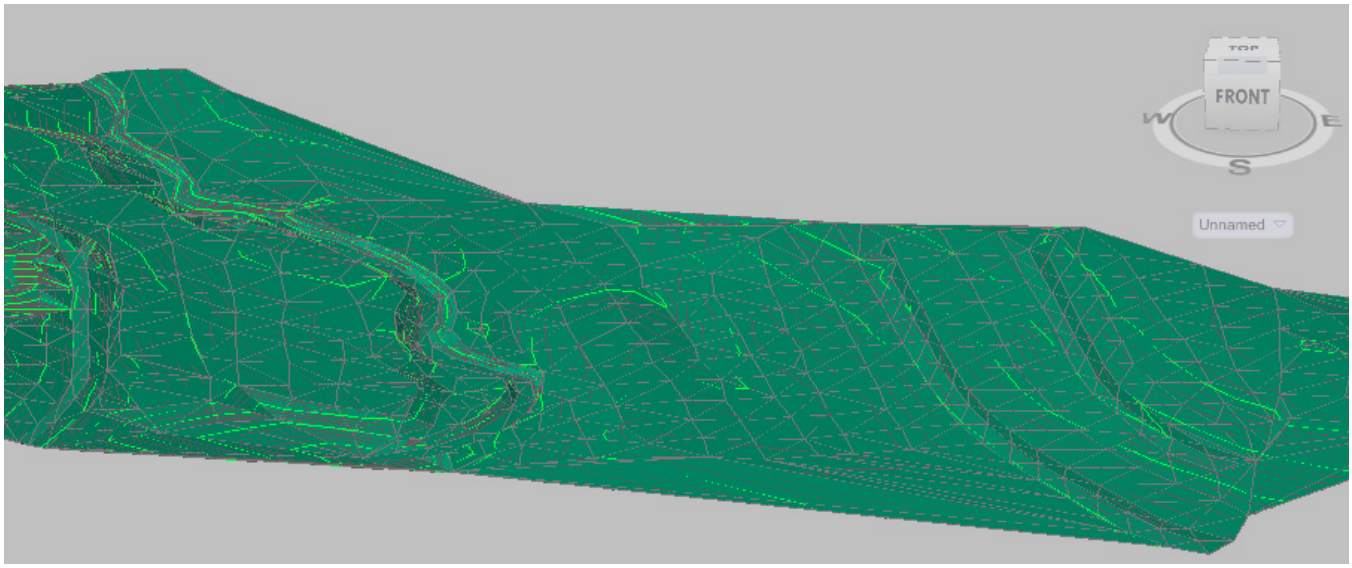


A useful way to visually inspect Civil 3D terrain surfaces is to rotate the view of the surfaces and view them from the side. This can make survey busts and other terrain irregularities readily apparent.

The figure below shows a terrain surface whose view has been rotated, which makes it easy to see that there is a problem with the elevation of one of the nodes on a 3D polyline that has been created to use as a breakline. A problem like this is very hard to detect when viewed from the top view of the drawing, but it is readily apparent when the objects are rotated.



A terrain surface that includes triangles can also be shaded to allow terrain relief to be seen, as shown in the example below. All of the surface styles included in the Minnesota drawing template are set to automatically display triangles when the surface is rotated, regardless of how the surface appears in plan view. This allows surface shading to be applied to any of the template's surface styles.



To shade a terrain surface, go to the *Views* panel on the *View* ribbon, and click on the down arrow to display the visual styles drop-down menu. The two shaded styles are the *Realistic* style and the *Conceptual* style. The primary difference between the two is that the edges between faces in the *Realistic* style are smoothed while the edges using the *Conceptual* style are more defined.

To return to a standard top-down plan view in the drawing, set the visual style to *2D Wireframe* on the drop-down menu on the *Views* panel of the *View* ribbon. Also on the *Views* panel, set the view to *Top*.

(This Page Intentionally Left Blank)