# Chapter 3

# Working with Surfaces

## Learning Objectives

***After completing this chapter, you will be able to:***

- *Create various types of surfaces*
- *Create surfaces by adding data*
- *Add data from different sources*
- *Create and edit surface styles*
- *Apply surface properties*

# SURFACES

In Civil 3D, a surface is considered as a building block and is created from triangular networks or grid data. Each triangle on the surface is created by connecting three nearest points of elevation. Also, each triangle has a defined slope.

A surface is a basic requirement of any Civil 3D project. All volume calculations, profiles, corridors, sections, and grading objects that are generated in a project are based on the surface created. You can create a surface by importing data from Triangular Irregular Network (TIN) files, Digital Elevated Model (DEM) files, points, point files, existing AutoCAD objects, contours, breaklines, point groups, and also from Google Earth.

## Types of Surfaces

There are four types of surfaces in Civil 3D: TIN Surface, TIN Volume Surface, Grid Surface, and Grid Volume Surface. The TIN Volume Surface and Grid Volume Surface are used for calculating volumes. These surfaces are discussed next.

### TIN Surface

A TIN surface is generated by the triangulation of points. The TIN surface is a set of contiguous non-overlapping triangles. The edges that form the triangles of the surface connect points to form an irregular triangular network. Each edge of the triangle is bound by two vertices. Each vertex in the surface has a definite coordinate and elevation. The elevation of any point on the surface is calculated by interpolating the elevation of the vertices of the triangle in which the point lies. The TIN surfaces are mostly used in case of irregular and variable surfaces such as streams, roads, and so on.

### TIN Volume Surface

A TIN volume surface is created by computing the difference between two surfaces: the base surface and the comparison surface. The TIN volume surface is used for calculating the cut, fill, and net volumes between the two surfaces (base and comparison). The elevation of any point in the TIN volume surface is defined by the difference in the elevation of the base surface and the comparison surface at that point.

### Grid Surface

A grid surface is created from the points that lie on the grid. A grid comprises points with elevation information spaced at regular intervals. You can either create or import a grid surface. You can also use a grid surface for mapping the surfaces that have uniform topography.

### Grid Volume Surface

A grid volume surface is created by computing the difference between the comparison surface and the overlaying base surface. The elevation value of a grid volume surface is the difference between the elevation values of the comparison and base surfaces. The grid volume surface helps you to quickly generate a volume that can be used for iterative site design. Figure 3-1 shows a grid volume surface.

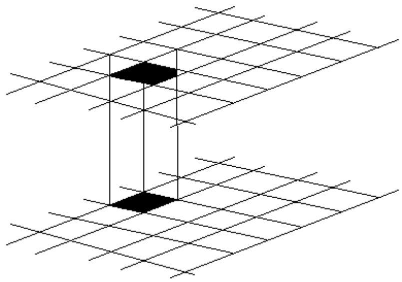


*Figure 3-1 The grid volume surface*

# CREATING AND EDITING SURFACES

A surface can be created by using points, point groups, AutoCAD objects, and boundaries. A surface can also be created directly by importing the DEM and TIN files that are generated from the survey. The methods of creating a surface and its different aspects are discussed next.

## Creating a Surface

| | |
|---|---|
| **Ribbon:** | Home > Create Ground Data > Surfaces drop-down > Create Surface |
| **Command:** | CREATESURFACE |

You can create a surface by using the **Create Surface** tool. Invoke the **Create Surface** tool from the **Create Ground Data** panel in the **Surfaces** drop-down; the **Create Surface** dialog box will be displayed, as shown in Figure 3-2.

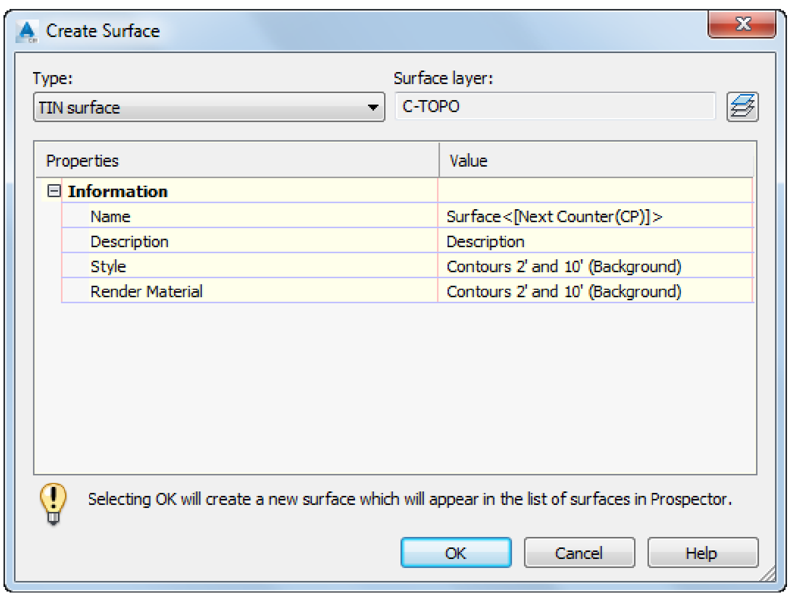



*Figure 3-2 The **Create Surface** dialog box*

Alternatively, you can invoke the **Create Surface** dialog box by using the **TOOLSPACE** palette. To do so, select the **Surfaces** node from the **Prospector** tab of the **TOOLSPACE** palette and right-click; a shortcut menu will be displayed. Choose the **Create Surface** tool from the shortcut menu to display the **Create Surface** dialog box. In this dialog box, you can specify the surface type, name, and description of the surface. The options in this dialog box are discussed next.

## Type

The **Type** drop-down list shows four different types of surfaces that you can create in Civil 3D. Select the required surface type from this drop-down list.

## Surface layer

The **Surface layer** text box displays the default layer on which the surface will be created. . By default, the **C-TOPO** layer is used as the surface layer. To assign a different layer, choose the button on the right of the text box to display the **Object Layer** dialog box. You can use this dialog box to assign a new layer to the surface.

The surface properties displayed under the **Information** node of the **Create Surface** dialog box are discussed next.

### Name

This property specifies the default name of the surface. Click on the **Value** field of the **Name** property; a button will be displayed on the right of the default name. Choose the button; the **Name Template** dialog box will be displayed. Specify a name in the **Name** edit box and choose the **OK** button. You can also specify a name directly in the **Value** field after clicking on the default surface name.

### Description

This property is used to briefly describe the surface. Click on the **Value** field of the **Description** property and enter a short description of the surface, if required.

### Style

This property specifies the default surface style assigned to the surface. To modify the surface style, click in the **Value** field of this property; the browse button will be displayed. Choose the browse button; the **Select Surface Style** dialog box will be displayed. You can use one of the options from the **Select a style** drop-down list in this dialog box to define the surface style based on the project requirement.

You can create a new surface style in Civil 3D. To do so, click in the **Value** field of the **Style** property; a drop-down list will be displayed. Select the **Create New** option from the drop-down list. You will learn how to create a new surface in detail later in this chapter.

**Note**

*The styles available in the* ***Select Surface Style*** *dialog box vary depending on the template used.*

### Render Material

This property specifies the default material value applied to the surface. Click on the default render material in the **Value** column and choose the button that will be displayed on the right; the **Select Render Material** dialog box will be displayed. You can select the required option from the **Select from list** drop-down list in this dialog box. Select the **By Block** option to use the render material of an associated block. You can also select the **By Layer** option from the drop-down list to use the render material assigned to the layer on which the object is created or select the **Global** option to use the global render material for the surface.

On specifying the values of the surface properties, choose the **OK** button; the **Create Surface** dialog box will be closed and a new TIN surface will be created and added to the **Surfaces** node in the **Prospector** tab of the **TOOLSPACE** palette. To view the newly created surface, expand the **Surfaces** node by clicking on the **+** button next to it. The new surface with the name **Surface1** is added to the **Surface** node.

**Note**
*As the added surface does not contain any data, so it will not be visible in the drawing area.*

## Adding DEM Files, Point Data from AutoCAD Objects, Point Files, Point Groups, and Contours to a Surface

You can add data to a surface from different sources such as points, point files, DEM and contours. The data obtained from these objects will be used to create a surface. The **Definition** subnode in the **Prospector** tab of the **TOOLSPACE** palette lists the different sources that can be used to add data in a surface. Expand the **Definition** subnode to view the sources, as shown in Figure 3-3. The methods of adding different types of data are discussed next.

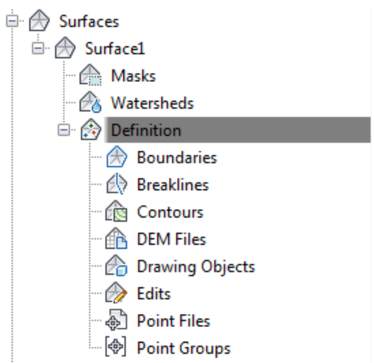
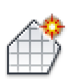



***Figure 3-3** Expanded **Definition** subnode*

### Adding DEM Files (Creating a Surface from DEM)

| | |
|---|---|
| **Ribbon:** | Surface > Modify > Add Data drop-down > DEM Files |
| **Command:** | ADDSURFACEDEMFILE |



Digital Elevation Model or DEM is defined as the digital representation of a continuously varying surface. DEM represents the elevation or height of any point in the data from a datum. The DEM file stores and transfers large amount of data related to the topographic relief information of the land. This information can be used in surveying, planning, and other engineering projects. The DEM files basically consists of XYZ coordinates of points at regular grid spacing intervals.

To add the DEM files, choose the **DEM Files** tool from the **Modify** panel; the **Add DEM File** dialog box will be displayed. Alternatively, expand the **Definition** subnode of the surface in the **Prospector** tab of the **TOOLSPACE** palette, select the **DEM Files** option, and right-click; a shortcut menu will be displayed. Choose the **Add** button from the shortcut menu; the **Add DEM File** dialog box will be displayed, as shown in Figure 3-4. Next, click on the button next to the **DEM file name** edit box; the **Grid Surface from DEM** dialog box will be displayed. Next, browse to the required location and select the required DEM file and then choose the **Open** button in the dialog box; the DEM file will be added to the current drawing file.

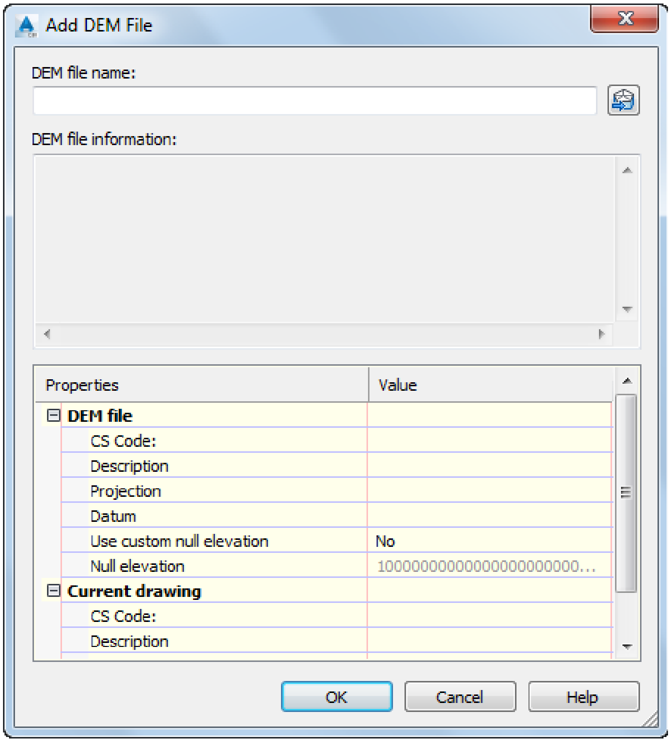



*Figure 3-4 The **Add DEM File** dialog box*

The **Properties** column of the **Add DEM File** dialog box displays various properties such as the coordinate system of the DEM file, description of the coordinate system, projection used, and so on. The properties and their respective values for both the DEM file and the current drawing are displayed in the **Add DEM File** dialog box under two categories, **DEM file** and **Current drawing**. By default, no coordinate system is assigned to the DEM file. You can assign a coordinate system to it to match with the coordinate system of the current drawing, especially if you are creating a grid surface from the DEM file.

To change the coordinate system of the DEM file, click in the **Value** field of the **CS Code** property and choose the button that is displayed on the right; the **Select Coordinate Zone** dialog box will be displayed. Select the required category from the **Categories** drop-down list. Next, select the coordinate zone from the **Available coordinate systems** drop-down list and choose the **OK** button; the **Select Coordinate Zone** dialog box will be closed. Again, choose the **OK** button from the **Add DEM file** dialog box; the DEM data will be added to the drawing and the surface will be created. To view the surface, enter **ZE** in the command line and press ENTER. The surface will be displayed based on the surface style set.

**Note**

*You can directly create a surface from the DEM file by using the options available in the* ***TOOLSPACE*** *palette. To do so, choose the* ***Surface*** *option in the* ***Prospector*** *tab of the* ***TOOLSPACE*** *palette and right-click; a shortcut menu will be displayed. Next, choose the* ***Create Surface from DEM*** *option from the shortcut menu; the* ***Grid Surface from DEM*** *dialog box will be displayed. In this dialog box, select the required DEM file and choose the* ***Open*** *button; the surface will be created from the selected DEM file.*

## Adding Data to Surface by Using Drawing Objects

| | |
|---|---|
| **Ribbon:** | Surface > Modify > Add Data drop-down > Drawing Objects |
| **Command:** | ADDSURFACEDRAWINGOBJECTS |

You can add surface data from different types of AutoCAD drawing objects. To do so, choose the **Surface** tool from the **Ground Data** panel, as shown in Figure 3-5. On doing so, the **Surface** tab will be displayed in the ribbon.

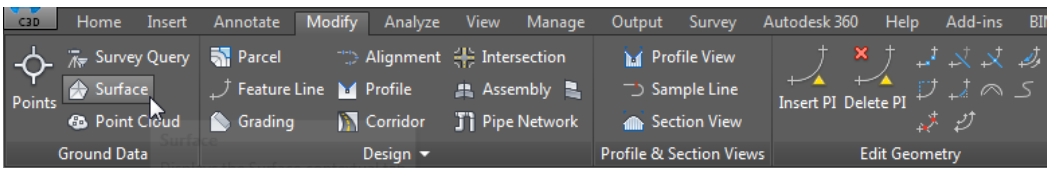



***Figure 3-5*** *The* ***Surface*** *tool chosen from the* ***Ground Data*** *panel*

Next, choose the **Drawing Objects** tool from the **Modify** panel; the **Add Points From Drawing Objects** dialog box will be displayed, as shown in Figure 3-6. To add the drawing object, select the object type from the **Object type** drop-down list. The drop-down list contains six different types of drawing objects such as lines, points, blocks and so on. The points from these objects are then added to the surface. For example, on selecting the **Points** option from the **Object type** drop-down list, the XYZ coordinates of the AutoCAD points will be used for creating the surface points.

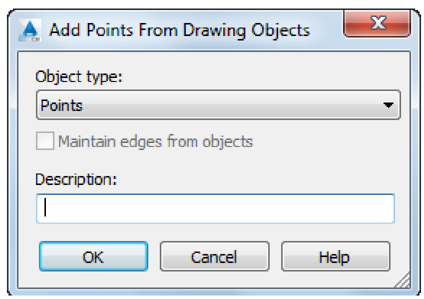



***Figure 3-6*** *The* ***Add Points From Drawing Objects*** *dialog box*

Similarly, on selecting the **Lines** option, the XYZ coordinates of the endpoints of the line project will be used to create the surface points. By default, the **Maintain edges from objects** check box in the **Add Points From Drawing Objects** dialog box is inactive. If you select the **Lines**, **Polyface**, or **3D faces** options from the **Object type** drop-down list, the **Maintain edges from objects** check box will be activated. This check box is used to maintain a distance between the surface edges and the edges of the drawing objects. On selecting this check box, the edges of the surface will be created at a distance from the edges of the drawing object selected. If this check box is not selected, the edges of the surface will coincide with the edges of the drawing object. Optionally, you can enter the description of the point data in the **Description** text box and choose the **OK** button; the dialog box will be closed and the surface will be created with the surface border visible (according to the surface style settings) in the drawing.

You can also add objects to the surface by using the **TOOLSPACE** palette. To do so, right-click on the **Drawing Objects** option in the **Definition** subnode of the surface in the **Prospector** tab; a shortcut menu will be displayed. Choose the **Add** option from the shortcut menu; the **Add Points From Drawing Objects** dialog box will be displayed. To add the drawing object, select the object type from the **Object type** drop-down list. The points from these objects are then added to the surface.

Note that whenever you add data to a surface, a symbol will be displayed on the left of the data option in the **Definition** subnode. This symbol indicates that the data has been added and it also symbolizes the type of data added to the surface. For example, on adding data to the surface by using the **Drawing Objects** option, a symbol will be displayed on the left of the **Drawing Objects** option in the **Definition** subnode of the surface, as shown in Figure 3-7.

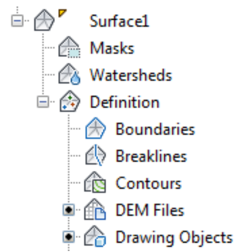



***Figure 3-7** The symbol displayed on the left of the **Drawing Objects** option*

**Note**

*AutoCAD objects can be added only in case of the **TIN surface** type.*

## Adding Point Files

| | |
|---|---|
| **Ribbon:** | Surface > Modify > Add Data drop-down > Point Files |
| **Command:** | ADDPOINTFILE |

You can create surface data by adding point files consisting of point data. Point files can be added to the surface only after selecting the right file format. AutoCAD Civil 3D supports many types of file formats. You can select a point file format for the surface depending upon the point data format in the file. The point files can be added only to the TIN surfaces.

To add a point file data to a surface, choose the **Point Files** tool from the **Modify** panel; the **Add Point File** dialog box will be displayed, as shown in Figure 3-8. From the **Specify point file format** list box in the dialog box, select the format to import the point file. For example, the **PENZ (space delimited)** format will include point numbers, easting, northing, and elevation values separated by space. Next, choose the button available on the right of the **Selected Files** list box in the **Add Point File** dialog box; the **Select Source File** dialog box will be displayed. Select the required point file from the dialog box and choose the **Open** button; the file name and its path will be displayed in the **Source File(s)** area. Choose **OK** in the **Add Point File** dialog box; the data in the point file will be added to the surface.

Alternatively, you can add point file data by using the options in the **TOOLSPACE** palette. To do so, expand the **Definition** subnode and right-click on **Point Files**; a shortcut menu will be displayed. Choose the **Add Files** option from the shortcut menu; the **Add Point File** dialog box will be displayed. Select the relevant format to import the point file data.

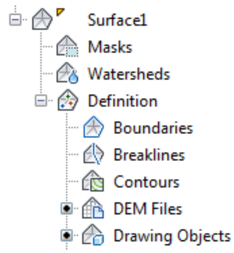



***Figure 3-8** The symbol displayed on the left of the **Drawing Objects** option*

## Adding Point Groups

**Ribbon:** Surface > Modify > Add Data drop-down > Point Groups
**Command:** ADDSURFACEPOINTGROUP

AutoCAD Civil 3D allows you to create a surface from a group of points or use the existing point groups to add the required point data to a surface.

To add a point group to a surface, choose the **Point Groups** tool from the **Modify** panel; the **Point Groups** dialog box will be displayed, as shown in Figure 3-9. Next, select the point group that you want to add to the surface. Note that by default, the point group **_All Points** is created when you import points. Choose the **OK** button; the points of the selected point group will be added to the surface. Point file and Point groups are the most common types of data sources that are added to the surfaces.

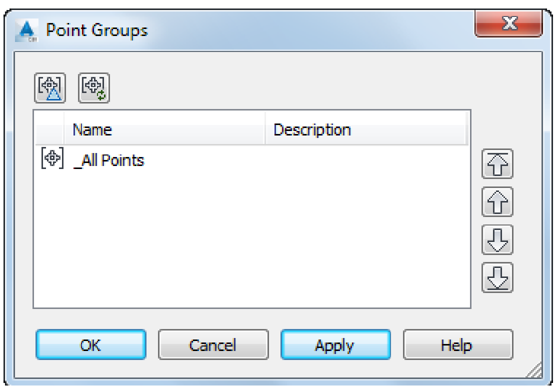



***Figure 3-9** The **Point Groups** dialog box*

## Adding Contours

**Ribbon:** Surface > Modify > Add Data drop-down > Contours
**Command:** ADDSURFACECONTOURS

Apart from the point files and point groups, contours also provide the data that can be used to create a surface. To add contours to a surface, choose the **Contours** tool from the **Modify** panel; the **Add Contour Data** dialog box will be displayed, as shown in Figure 3-10.

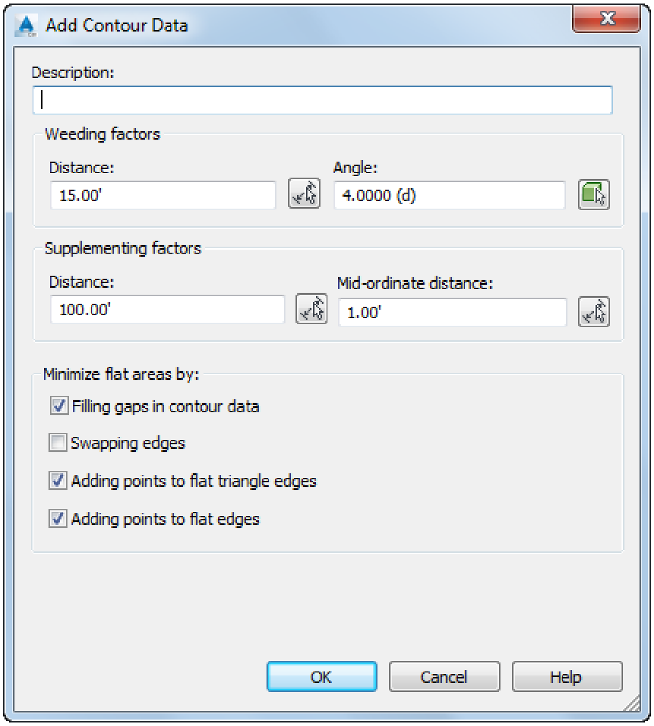



***Figure 3-10** The **Add Contour Data** dialog box*

Alternatively, you can add contours using the **TOOLSPACE** palette. To do so, right-click on **Contours** in the **Definition** subnode; a shortcut menu will be displayed. Choose the **Add** option from the shortcut menu; the **Add Contour Data** dialog box will be displayed.

You can use the options in this dialog box to add contour data to a surface. Enter a description about the contour data in the **Description** text box. The options in the **Weeding factors** and **Supplementing factors** areas are used to remove or add vertices along the contours. Specify the weeding distance in the **Distance** edit box or choose the button on the right of this edit box to specify the weeding distance in the drawing. Similarly, you can specify the weeding angle in the **Angle** edit box. In the **Supplementing factors** area, specify the supplementing distance and the mid-ordinate distance in the **Distance** and **Mid-ordinate** distance edit boxes, respectively. In the **Minimize flat areas by** area, the **Filling gaps in contour data**, **Adding points to flat triangle edges**, and **Adding points to flat edges** check boxes are selected by default to enable the filling of small gaps in contours and adding new vertices at the required locations.

After you have specified the required data, choose the **OK** button; the dialog box will be closed and you will be prompted to select the contours. Select the required contours and press ENTER; the contour data will be added to the surface and the surface will be displayed according to the surface style used.

## Adding Surface Boundaries

| | |
|---|---|
| **Ribbon:** | Surface > Modify > Add Data drop-down > Boundaries |
| **Command:** | ADDSURFACEBOUNDARIES |

A boundary can be defined as a closed polygon entity used to represent the edge or limit of a surface. You can control the visibility of the surface triangulation within or outside the boundary. Any closed polygon or polyline object can be added to the surface as a surface boundary. To add a boundary to a surface, choose the **Boundaries** tool from the **Modify** panel; the **Add Boundaries** dialog box will be displayed, as shown in Figure 3-11.

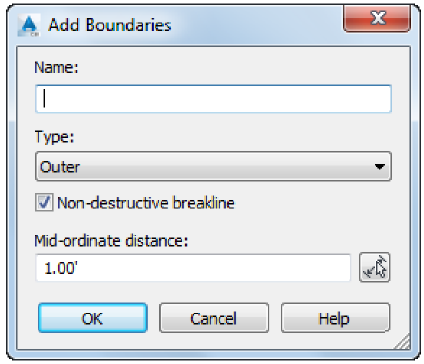



***Figure 3-11*** *The* ***Add Boundaries*** *dialog box*

Alternatively, you can add a boundary to a surface by using the options in the **TOOLSPACE** palette. To do so, expand the **Definition** subnode of the surface from the **Prospectors** tab and right-click on **Boundaries**; a shortcut menu will be displayed. Choose the **Add** option from the shortcut menu; the **Add Boundaries** dialog box will be displayed. In the dialog box, you can specify a name for the boundary in the **Name** edit box. Next, select the type of boundary from the **Type** drop-down list. A surface can have four different types of boundaries that are discussed next.

### Outer

By default, the **Outer** option is selected in the **Type** drop-down list. As a result, the boundary is created such that all the TIN lines outside the boundary are deleted and the TIN lines inside the boundary are retained. Figures 3-12 and 3-13 show the surface before and after adding the outer boundary, respectively.

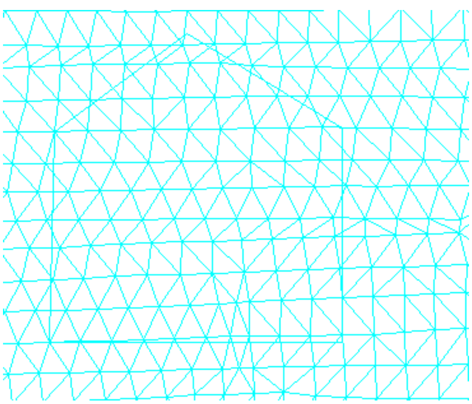


***Figure 3-12*** *Surface before adding the outer boundary*

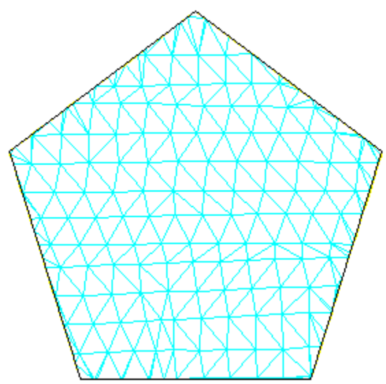


***Figure 3-13*** *Surface after adding the outer boundary*

Note that the **Non-destructive breakline** check box in the **Add Boundaries** dialog box is selected by default. As a result, a new surface is created by cutting the triangles lying across the boundary. If this check box is cleared, an internal border will be created and the triangles that are lying completely inside the boundary will be retained. In other words, the internal border will include only those triangles whose all three edges are inside the boundary. Figure 3-14 shows the internal border of the surface created after adding the outer boundary by clearing the **Non-destructive breakline** check box.

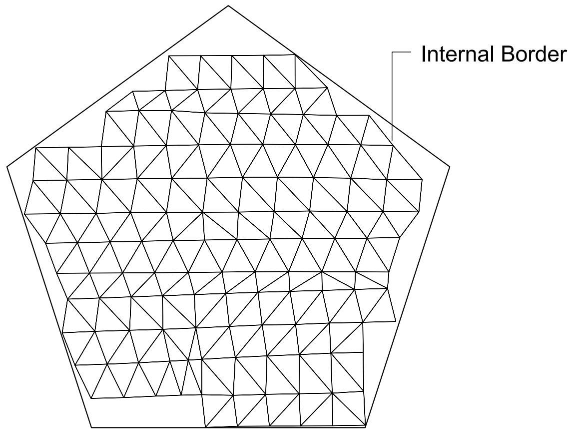



***Figure 3-14*** *Adding the outer boundary after clearing the* ***Non-destructive breakline*** *check box*

**Note**

*You can create multiple outer boundaries. However, in such a case, Civil 3D surface will display only the last boundary created. To switch between multiple outer boundaries, select the check box corresponding to the required boundary in the* ***Definition*** *tab of the* ***Surface Property - <Surface Name>*** *dialog box.*

## Hide

The **Hide** option from the **Type** drop-down list enables you to hide the surface triangulation in a particular area. The triangles lying inside the related boundary will not be displayed. You can use this boundary to create voids or holes for some areas such as wetlands and buildings. The surface inside the boundary will not be included in the area and volume calculations. Figure 3-15 shows the surface after adding the boundary with the **Hide** option and the **Non-destructive breakline** check box selected. Figure 3-16 shows the surface after selecting the **Hide** option from the **Type** drop-down list and clearing the **Non-destructive breakline** check box in the **Add Boundaries** dialog box. In Figure 3-15, the triangles are deleted exactly at the boundary as the **Non-destructive breaklines** check box was selected by default. In Figure 3-16, an internal border is created and all the triangles that were lying completely inside the boundary are affected, thus making them hidden.

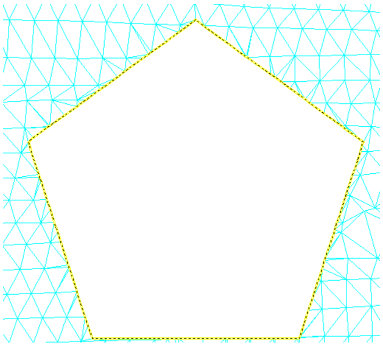


***Figure 3-15*** *Surface created after adding the boundary using the **Hide** option*

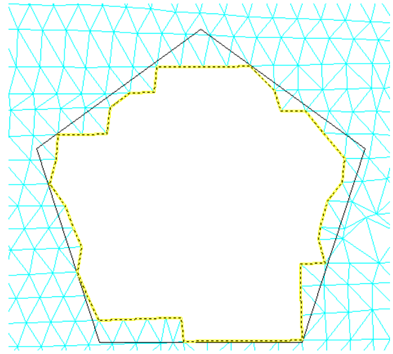


***Figure 3-16*** *Surface created after clearing the **Non-destructive breakline** check box*

## Show

Selecting the **Show** option from the **Type** drop-down list enables you to display the triangles inside the hide boundary. You can use this option to display the hidden triangles when the **Hide** option is selected.

### Data Clip

Selecting the **Data Clip** boundary option from the **Type** drop-down list helps you to create a surface boundary using the polygon object from the drawing itself such as feature lines, parcels, circles, and so on. This type of boundary is used to define a region on a surface where you want to import a set of surface data exclusively. For example, if you want to import a high resolution Light Detection and Ranging (LIDAR) Data to a corridor but not to the surrounding surface, you can use the **Data Clip** boundary option to import the data only to the corridor region. The data imported will be clipped at the **Data Clip** boundary.

After you have selected the boundary type from the **Type** drop-down list, choose the **OK** button from the **Add Boundaries** dialog box; the dialog box will be closed and you will be prompted to select the objects. Select the required polygon or polyline object and then press ENTER; the selected boundary will be added to the surface. Note that the boundaries that you add to the surface will be listed along with their names and types in the **TOOLSPACE** Item View, as shown in Figure 3-17.

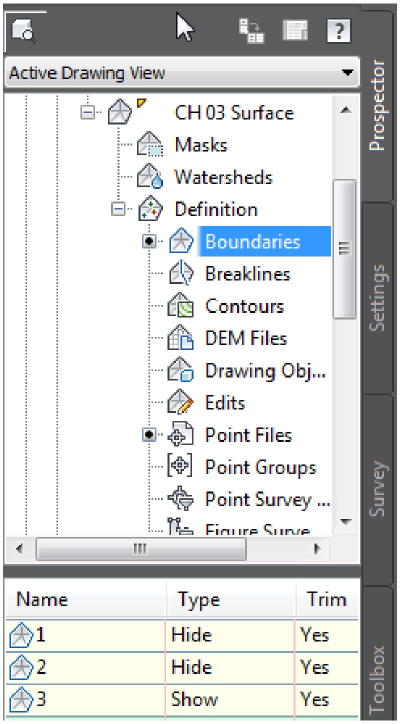



***Figure 3-17*** *The **TOOLSPACE** Item View of the **Toolspace** palette displaying the added boundaries*

To view the properties of the boundary, select the required boundary in the **TOOLSPACE** Item View and right-click on it; a shortcut menu will be displayed. Choose the **Properties** option from the shortcut menu; the **Boundary Properties** dialog box will be displayed. You can also use this dialog box to edit the name of the boundary view. To do so, enter the desired name in the **Name** edit box in this dialog box.

## Deleting a Boundary

To delete a boundary, select it from the **TOOLSPACE** Item View and right-click to display a shortcut menu. Choose the **Delete** option from the shortcut menu; a warning message will be displayed, as shown in Figure 3-18. Choose the **OK** button; the boundary will be deleted from the drawing as well as from the **TOOLSPACE** Item View. Remember that the **Delete** option will not be displayed in the shortcut menu if you are trying to delete the current boundary. A small triangular symbol on the left of the boundary name in the **TOOLSPACE** item view indicates that a boundary is currently in use.

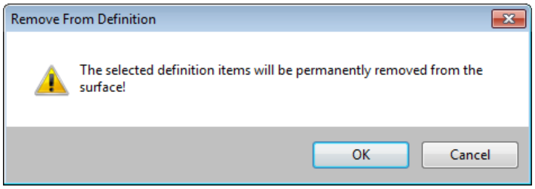



***Figure 3-18*** *The **Remove From Definition** warning message window*

If you remove a boundary from the surface or make any changes in the surface, a yellow symbol will be displayed on the left of the surface name, as shown in Figure 3-19. This symbol indicates that a change has been made in the surface and the surface is out of date. To update the surface, select the surface in the **Prospector** tab and right-click to display a shortcut menu. Choose the **Rebuild** option from the shortcut menu; the surface will be updated automatically and the symbol will disappear.

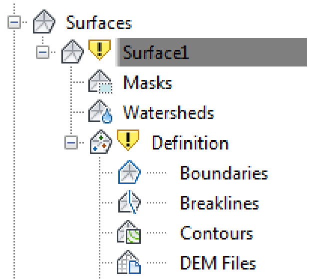



***Figure 3-19*** *The symbol displayed on the left of* ***Surface1***

## Adding Breaklines to the Surface

| | |
|---|---|
| **Ribbon:** | Surface > Modify > Add Data drop-down > Breaklines |
| **Command:** | ADDSURFACEBREAKLINES |

Breaklines are polylines, feature lines, or 3D lines used to restrict the triangulation of the surface along the breakline. Breaklines basically represent the features such as streams, retaining walls, ditches, and so on. They can be added only to TIN surfaces. To add breaklines, choose the **Breaklines** tool from the **Modify** panel; the **Add Breaklines** dialog box will be displayed.

Alternatively, right-click on the **Breaklines** option in the **Definition** node of the surface in the **Prospector** tab of the **TOOLSPACE** palette; a shortcut menu will be displayed. Choose the **Add** option from the shortcut menu; the **Add Breaklines** dialog box will be displayed, as shown in Figure 3-20. Enter a short description or name for the breakline in the **Description** text box. This description will be displayed in the **Description** column of the **TOOLSPACE** Item View. The description will help you identify the breaklines. If you do not enter any description, Civil 3D will automatically name it as **Breakline set1:1**. Next, select the type of breakline that you want to add to the surface from the **Type** drop-down list. There are five types of breaklines, which are discussed next.

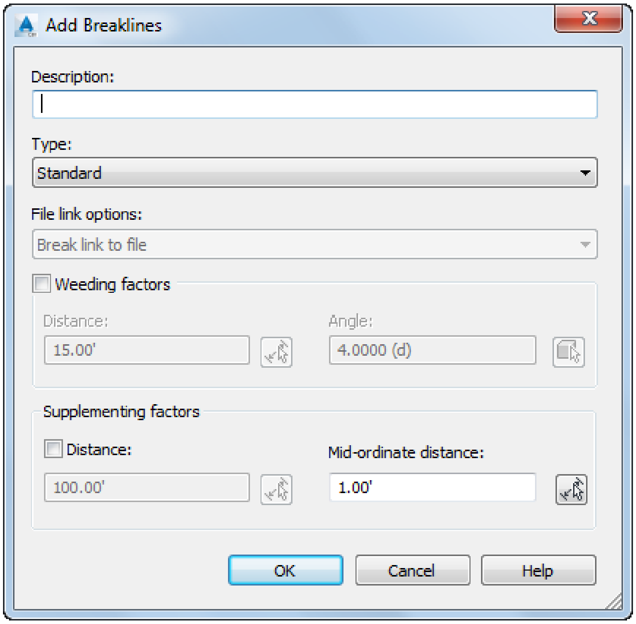



***Figure 3-20** The **Add Breaklines** dialog box*

## Standard

This type of breakline is a three-dimensional object created from 3D lines, polylines, and feature lines. The standard breaklines are created by connecting the existing elevation points of polylines or feature lines. Therefore, you do not need to associate point objects with their vertices. The breaklines used for defining a marshy area or swales are good examples of **Standard** breaklines. The **Standard** breaklines can also have arc segments. For a breakline of the **Standard** type, you can specify the weeding and supplementing factors by selecting the **Weeding** and **Supplementing factors** check box respectively. You specify the weeding distance and angle in the **Distance** and **Angle** edit boxes, respectively. Alternatively, choose the buttons on the right of the **Distance** and **Angle** edit boxes to specify the distance and angle. Similarly, specify the distance and mid-ordinate distance for the curved segments in the respective edit boxes in the **Supplementing factors** area. The mid-ordinate distance is used to tessellate the polyline arcs from which the breakline will be created. The supplementing factors are used to add more triangles along the straight segments of the breakline to make the surface smoother. Figures 3-21 and 3-22 illustrate the effect of adding breaklines with and without using the supplementing factors.

## Proximity

The **Proximity** breakline is a two-dimensional object created from polylines or feature lines. The XYZ coordinates or Northing, Easting, and Elevations of the proximity breakline are calculated for each vertex of the parent polyline in reference to the TIN surface point that is nearest to the vertex. These types of breaklines cannot have curved segments.

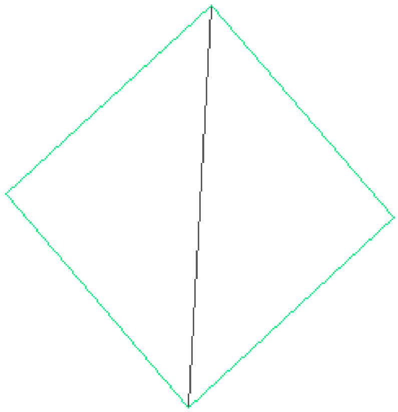


*Figure 3-21 Breaklines created without selecting the **Distance** check box*

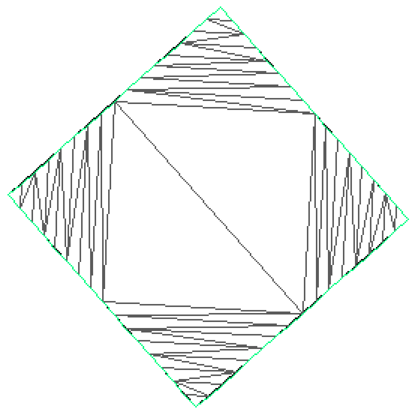


*Figure 3-22 Breaklines created after selecting the **Distance** check box*

### Wall

Like the **Standard** breakline, the **Wall** breakline is also a three-dimensional object created from an existing polyline and representing the top or bottom of the wall. Wall breaklines can be used in areas where there is vertical face in the surface such as retaining walls, curbs, and so on. The **Wall** breaklines help you to represent the surface in a more accurate way. For example, for a retaining wall, you can define the differences in the elevations between the materials on both sides of the wall.

After selecting the **Wall** breakline from the **Type** drop-down list, choose the **OK** button; the **Add Breaklines** dialog box will be closed and you will be prompted to select the objects. Select the required polyline objects and press ENTER; you will be prompted to pick the offset side. Click on any side of the first polyline vertex; you will be prompted to select an option for specifying the wall height. You can enter **ALL** or **Individual**. Enter **A** for **ALL** and **I** for **Individual**. If you press ENTER, the **ALL** option will be selected by default. If you select the **All** option, you will be prompted to enter elevation difference for offset points or elevation. Next, enter the required elevation value and press ENTER; all vertices of the polyline object will have the same elevation and the **Wall** breaklines will be created.

The **Individual** option is used to specify the offset elevations of each vertex of the polyline. To specify the elevations using this option, enter **I** in the command line and press ENTER; you will be prompted to specify the elevation at the offset points individually. Specify the elevation in the command line and press ENTER until the elevation for all points is specified. Note that adding the wall breaklines can change the surface slope values.

### From file

The **From file** breakline is created directly from the FLT(*.flt) file format without drawing the breaklines. The FLT file consists of the name, type, coordinates, and elevations of the breakline. On selecting the **From file** option from the **Type** drop-down list, the **File link options** drop-down list in the **Add Breaklines** dialog box will be enabled. There are two types of file linking options in this drop-down list. The **Break link to file** option in this drop-down list is used to add the breakline to the **Definition** subnode of the surface. If you select this option, then after importing the breaklines from the file, the link to the file will be broken. However, on selecting the **Maintain link to file** option, the link to the file will be maintained.

After you have selected the required file link option, choose the **OK** button from the **Add Breaklines** dialog box; the **Import Breakline File** dialog box will be displayed. You can use this dialog box to select the required FLT file and import it to the drawing.

### Non-destructive

The **Non-destructive** breaklines are used to restrict the surface triangulations along its length, without affecting the surface elevation at the intersections and surface triangulation.

After selecting the type of breakline and other required options in the **Add Breaklines** dialog box, choose the **OK** button; you will be prompted to select the object in the drawing. Select the polyline or 3D lines in the drawing and press ENTER; the breaklines will be added to the surface and **Breakline set1:1** will be displayed in the **TOOLSPACE** Item View.

## Editing a Surface

You can perform various surface edits manually. To view various edit operations, choose the **Edit Surface** drop-down from the **Modify** panel; all the edit tools will be displayed. Choose any of the tools and perform the desired edit operation on the surface. Alternatively, you can edit surface by selecting the **Edits** option from the **Definition** subnode of the surface in the **Prospector** tab and right-click; a shortcut menu will display all surface edit options. The entire history of the edit operations performed on the surface will be added and can be viewed in the **TOOLSPACE** Item View. The **Edit Surface** tools are discussed next.

**Note**

*The options in the **Edits** shortcut menu of the **Prospector** tab are displayed based on the type of the surface selected.*

### Add Line

| | |
|---|---|
| **Ribbon:** | Surface > Modify > Edit Surface drop-down > Add Line |
| **Command:** | ADDSURFACELINE |

The **Add Line** tool is used to add a line to the TIN surface by connecting two endpoints of the existing surface lines. It helps you to modify the surface triangulation. Before using the **Add Line** edit option, make sure that the surface triangles are visible in the drawing. To add a new line to the surface, choose the **Add Line** tool from the **Modify** panel; you will be prompted to select the endpoints. Select the endpoints of the TIN lines of the triangles that cross the other TIN lines; the TIN lines will be added. Note that adding the TIN line affects the surface triangulation. Figure 3-23 shows the surface before adding the TIN lines and Figure 3-24 shows the surface after adding the TIN lines.

Continue selecting the endpoints of TIN lines to add more lines or press ENTER to end the command. The **Add Line** operation will be listed in the **TOOLSPACE** Item View. The coordinates of the selected points will also be displayed in the **Description** column of the **TOOLSPACE** Item View, as shown in Figure 3-25.

**Note**

*The **Add Line** edit option can be used in all four types of surfaces.*

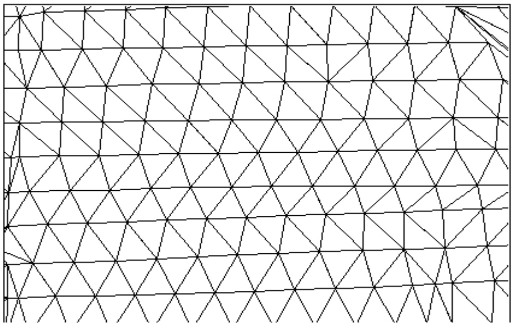


***Figure 3-23*** *Surface before adding TIN lines*

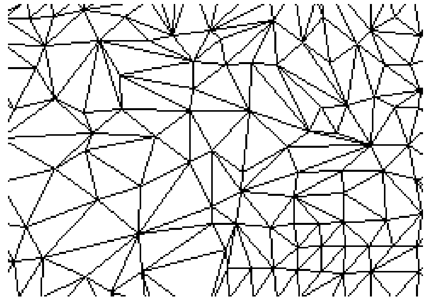


***Figure 3-24*** *Retriangulation of the surface after adding TIN lines*

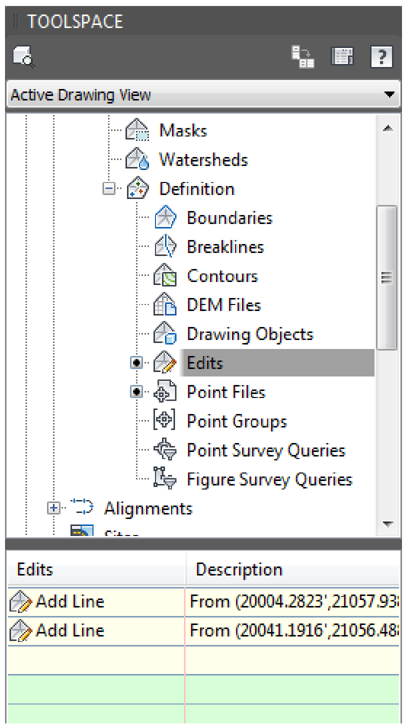



***Figure 3-25*** *The **Toolspace** Item View displaying the **Add Line** operation in the **Edits** column and the coordinates in the **Description** column*

**Tip**. *Alternatively, to use the **Add Line** edit option, expand the **Definition** subnode of the surface and right-click on **Edits**; a shortcut menu will be displayed. Choose **Add Line** from the shortcut menu to add a line. Similarly, you can use other edit options.*

## Delete Line

**Ribbon:** Surface > Modify > Edit Surface drop-down > Delete Line
**Command:** DELETESURFACELINE

The **Delete Line** tool is used to delete a TIN line or Grid line from an existing surface. To delete the required entity, choose the **Delete Line** tool from the **Modify** panel; you will be prompted to select the edges. Select the edges or TIN lines that you want to remove and press ENTER; the edges will be removed from the surface and a boundary will be created in the area from where the lines or edges of the triangles are deleted. This tool is useful to reduce the unwanted TIN or Grid lines so that an accurate surface is created. Moreover, it can be used to delete the lines where the surface is not required at all such as a pond or any other water resource. For example, using the **Delete Line** tool, you can create a surface with only the center line points of the road survey. On deleting lines from a surface, an interior border will be created around the area from where the lines are deleted, as shown in Figures 3-26 and 3-27.

**Note**
*The **Delete Line** tool can be used for all types of surfaces. However, you need to ensure that the surface triangles visibility is turned on before using this tool.*

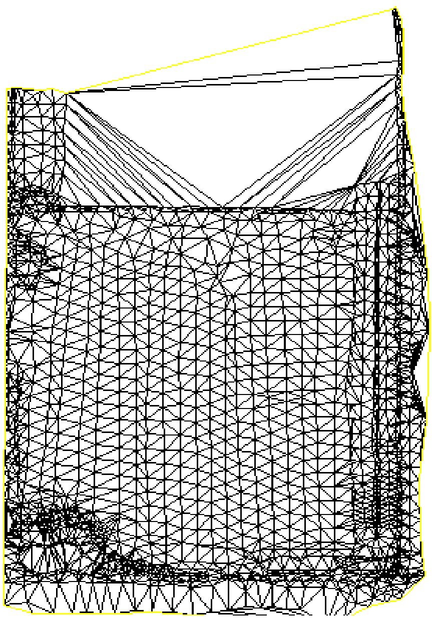


***Figure 3-26** The TIN surface before deleting TIN lines*

***Figure 3-27** The TIN surface with an interior border after deleting TIN lines*

## Swap Edge

**Ribbon:** Surface > Modify > Edit Surface drop-down > Swap Edge
**Command:** EDITSURFACESWAPEDGE

The **Swap Edge** tool is used to swap or change the direction of the two triangular faces by changing the direction of the common edge. To swap the edge of the triangles, choose the **Swap Edge** tool from the **Modify** panel; you will be prompted to select an edge. Select the required edge(s); the orientation of the edge and the facing of the triangles will be changed. Press ENTER to end the command. Note that this tool cannot be used in grid surfaces and grid volume surfaces. Figures 3-28 and 3-29 show a surface before and after swapping the edges.

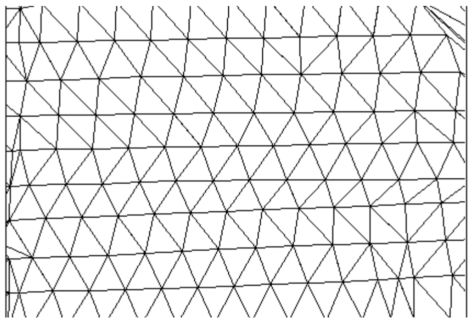


***Figure 3-28*** *Surface before swapping the edges*

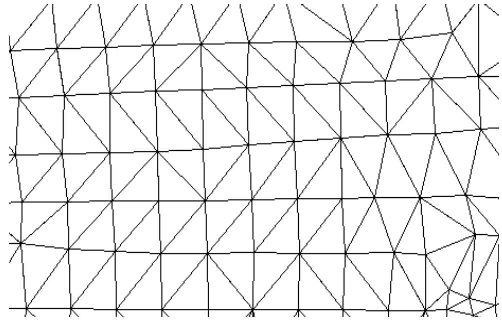


***Figure 3-29*** *Surface after swapping the edges*

## Add Point

| | |
|---|---|
| **Ribbon:** | Surface > Modify > Edit Surface drop-down > Add Point |
| **Command:** | ADDSURFACEPOINT |

The **Add Point** tool is used to add a point data manually to the surface by specifying the location and elevation of a point. To add a point, choose the **Add Point** tool from the **Modify** panel; you will be prompted to select a point. Click inside or outside the existing surface to add a point; you will be prompted to specify the elevation of the point. Enter the required elevation in the command line and press ENTER; the point will be added to the surface and the surface will re-triangulate accordingly. Figure 3-30 shows a surface after adding a point on its exterior border. In this figure, the surface has re-triangulated according to the elevation of the added point.

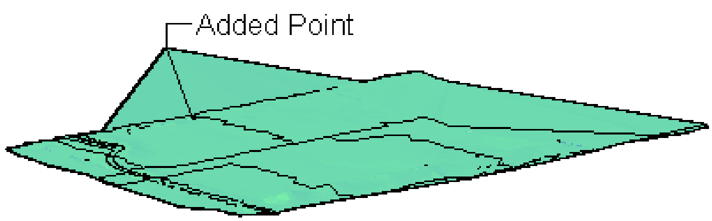



***Figure 3-30*** *Surface after adding a point on its exterior border*

The added point and its coordinates will be displayed in the **TOOLSPACE** Item View. The **Add Point** tool can be used in all types of surfaces. You can also use this tool to add elevation to the surfaces with flat slopes.

**Note**

*In case of grid surfaces, you can perform this operation only if you add points outside the existing surface, holes, or areas that have no existing points.*

## Delete Point

| | |
|---|---|
| **Ribbon:** | Surface > Modify > Edit Surface drop-down > Delete Point |
| **Command:** | DELETESURFACEPOINT |

This tool is used to remove unnecessary or unwanted surface points. To delete surface points, the visibility of the **Points** component should be turned on in the **Display** tab of the **Surface Style** dialog box. You will learn more about the surface styles later in this chapter. If you try to delete the points in the surface without turning the **Points** visibility on, Civil 3D will display a message box, as shown in Figure 3-31.

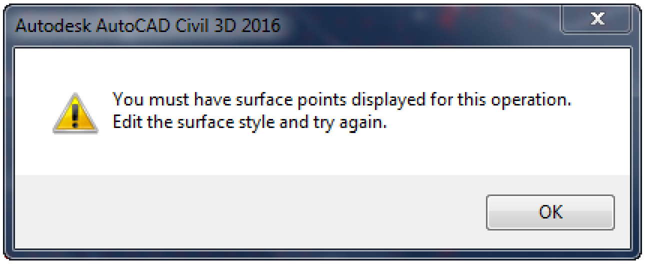



***Figure 3-31** The **AutoCAD Civil 3D 2016** message box*

To delete points from a surface, choose the **Delete Point** tool from the **Modify** panel; you will be prompted to select the points. Select the points that you want to delete from the surface and press ENTER; the selected points will be deleted from the surface and the surface will be updated automatically.

## Modify Point

| | |
|---|---|
| **Ribbon:** | Surface > Modify > Edit Surface drop-down > Modify Point |
| **Command:** | EDITSURFACEPOINT |

The **Modify Point** tool is used to modify the elevation of surface points. To change the elevation of surface points, choose the **Modify Point** tool from the **Modify** panel; you will be prompted to select the points to modify. Select the points that you want to modify and press ENTER; you will be prompted to specify the new elevation for the selected points. Specify the new elevation in the command line and press ENTER. You can continue selecting more points on the surface or press ENTER to end the command.

Alternatively, you can modify the elevation of surface points by selecting **Edits** in the surface **Definition** subnode in the **Prospector** tab of the **TOOLSPACE** palette and right-click; a shortcut menu will be displayed. Choose the **Modify Point** option from the shortcut menu; you will be prompted to select the point and specify the new elevation.

**Note**

*Before using the **Modify Point** tool, make sure that the points are visible in the drawing.*

## Move Point

**Ribbon:** Surface > Modify > Edit Surface drop-down > Move Point
**Command:** MOVESURFACEPOINT

The **Move Point** tool is used to change the location of the existing surface points. To move a point to a new location, choose the **Move Point** tool from the **Modify** panel; you will be prompted to select the point that you want to move. Select the required point from the surface by clicking on the point. Click again to specify the new location for the point. You can also specify the location in the command line by entering its coordinates. The surface retriangulates and updates automatically according to the new location of the point. Press ENTER to end the command. Note that this tool can be used only in case of TIN surfaces and TIN volume surfaces.

## Minimize Flat Areas

**Ribbon:** Surface > Modify > Edit Surface drop-down > Minimize Flat Areas
**Command:** MINIMIZESURFACEFLATAREAS

This tool is used to minimize the flat areas in a surface as these areas can make the surface inaccurate. These flat areas are the surface triangles whose points are obtained from a contour at same elevation. These points of the triangles will have the same elevation and therefore, the triangles will have no slope. This type of problem occurs when a contour data is added to a TIN surface.

The **Minimize Flat Areas** tool helps you to remove the triangles whose all three points are at the same elevation, thus creating a flat area. Also, you can find and remove the triangle edges connecting the points on the contours that have the same elevation. To do so, choose the **Minimize Flat Areas** tool from the **Modify** panel; the **Minimize Flat Areas** dialog box will be displayed, as shown in Figure 3-32. This dialog box displays different options under the **Minimize flat areas by** area. These options are discussed next.

### Filling gaps in contour data

This option is used to fill small gaps that can occur between two consecutive contours. If the contours lie close to each other, Civil 3D will join their ends by adding a triangular edge in the gap. This will make the contour a single continuous contour.

**Note**

*This option is applicable only on the contours that are displayed in the drawing. It does not affect the original contour data added to the surface.*

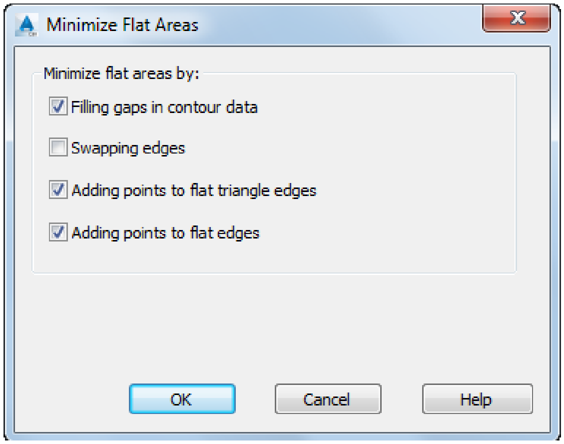



***Figure 3-32*** *The* ***Minimize Flat Areas*** *dialog box*

**Swapping edges**

On selecting this option, Civil 3D will find all flat triangles that share a non-contour edge with the non-flat triangles in the surface. If the two triangles of the surface that share a common edge form a convex quadrilateral, the common edge will be swapped creating two non-flat triangles. This option removes most of the flat triangles from the surface without affecting the number of surface triangles and points. In this way, the original size of the surface will be retained.

**Adding points to flat triangle edges**

On selecting the **Adding points to flat triangle edges** option, a new point will be created. The point will be created on the flat edge that is between the two triangles. In this case, instead of swapping the common edge of the two triangles, a point will be added at the midpoint of the common edge. The elevation of the point will be automatically calculated by Civil 3D. The surface will retriangulate automatically. This option helps you remove the flat areas more effectively than the **Swapping edges** option. This results in the increase of the number of triangles and points, thus affecting the size of the surface.

## Raise/Lower Surface

**Ribbon:** Surface > Modify > Edit Surface drop-down > Raise/Lower Surface
**Command:** RAISELOWERSURFACE

This tool is used to raise or lower the elevation of a surface by increasing or decreasing the value of the elevation of surface points. You can raise the surface by adding a positive value and lower the surface by adding a negative value for the elevation in the command line. This tool can be used for testing the gradings and adjusting the surfaces at the required elevation to calculate the cut and fill volumes.

To change the elevation of a surface, choose the **Raise/Lower Surface** tool from the **Modify** panel; you will be prompted to add the amount to all elevations. Enter a value (positive or negative) in the command line and press ENTER; the surface will be raised or lowered as per the specified elevation value. This tool is mainly used in TIN and Grid surfaces.

## Smooth Surface

| | |
|---|---|
| **Ribbon:** | Surface > Modify > Edit Surface drop-down > Smooth Surface |
| **Command:** | SMOOTHSURFACE |

This tool is used to smoothen the surface contours by adding additional points to the surface. You can calculate the elevation of the added points using two methods, NNI (Natural Neighbor Interpolation) and Krigging.

The NNI method is used to calculate the elevation of the arbitrary points by interpolating the elevations of the neighboring points. This method can also interpolate the elevation lying within the surface. The Krigging method is a complex method as compared to the NNI method. This method would require information about the spatial continuity and a sample of surface data to interpolate the elevations making it more complex than the NNI method.

To smoothen a surface using NNI method, choose the **Smooth Surface** tool from the **Modify** panel; the **Smooth Surface - <surface name>** dialog box will be displayed, as shown in Figure 3-33. Click and select the **Natural neighbor interpolation** option from the drop-down list in the **Value** column of the **Select method** property. On doing so, the options in the **Krigging Method** category will be disabled.

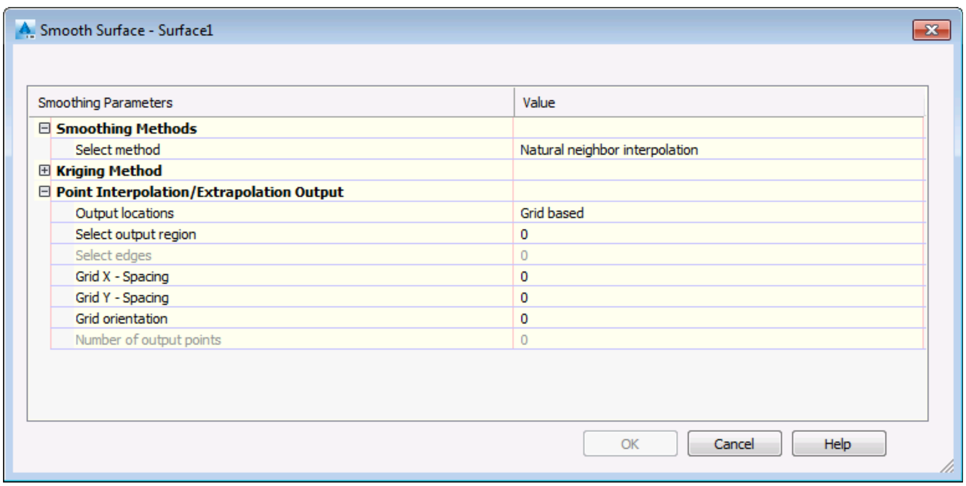



*Figure 3-33 The **Smooth Surface** dialog box*

Click in the **Value** field of the **Output locations** property and select the required option from this drop-down list. If you select the **Grid based** option, the generated points will lie on a grid defined within the specified polygon areas selected in the drawing. The **Centroids** option can be selected from the drop-down list to interpolate the points at the centroids of the triangles of the surface. The **Random points** option can be selected to interpolate number of points within the boundary of the polygon. The **Edge midpoints** option can be used to interpolate the points at the midpoints of the triangle edges. To use this option, the visibility of the surface triangles should be turned on.

Next, click in the **Value** field of the **Select output region** property and choose the browse button displayed on the right; the dialog box will disappear and you will be prompted to select an output region. Select the required surface or region bounded by a rectangle or polygon and choose the **OK** button from the **Smooth Surface - <surface name>** dialog box; Civil 3D will start smoothening the surface. Similarly, you can use the **Krigging** method to smoothen the surface. Figures 3-34 and 3-35 show the surface before and after smoothening using the **NNI** method.

## Paste Surface

| | |
|---|---|
| **Ribbon:** | Surface > Modify > Edit Surface drop-down > Paste Surface |
| **Command:** | EDITSURFACEPASTE |

The **Paste Surface** tool can be used to paste one surface over another surface. This tool is useful for creating composite surfaces such as creating corridor surfaces and then pasting the corridor surface on the existing surface.

This tool is also useful in surface gradings and comparing the design surface with the existing surface. To paste a surface on another surface, choose the **Paste Surface** tool from the **Modify** panel; the **Select Surface to Paste** dialog box will be displayed. In the dialog box, select the surface to be pasted on the existing surface from the list box and then choose **OK** to exit the dialog box.

***Figure 3-34*** *Surface contours before using the* ***NNI*** *method*

***Figure 3-35*** *Surface contours after using the* ***NNI*** *method*

## Simplify Surface

| | |
|---|---|
| **Ribbon:** | Surface > Modify > Edit Surface drop-down > Simplify Surface |
| **Command:** | SIMPLIFYSURFACE |

This tool is used to simplify a surface by reducing the number of points on the surface or by contracting the triangle edges on a TIN surface. To simplify a surface, choose the **Simplify Surface** tool from the **Modify** panel; the **Simplify Surface - <surface name>** wizard will be displayed. By default, the **Simplify Methods** page will be displayed in the wizard, as shown in Figure 3-36.

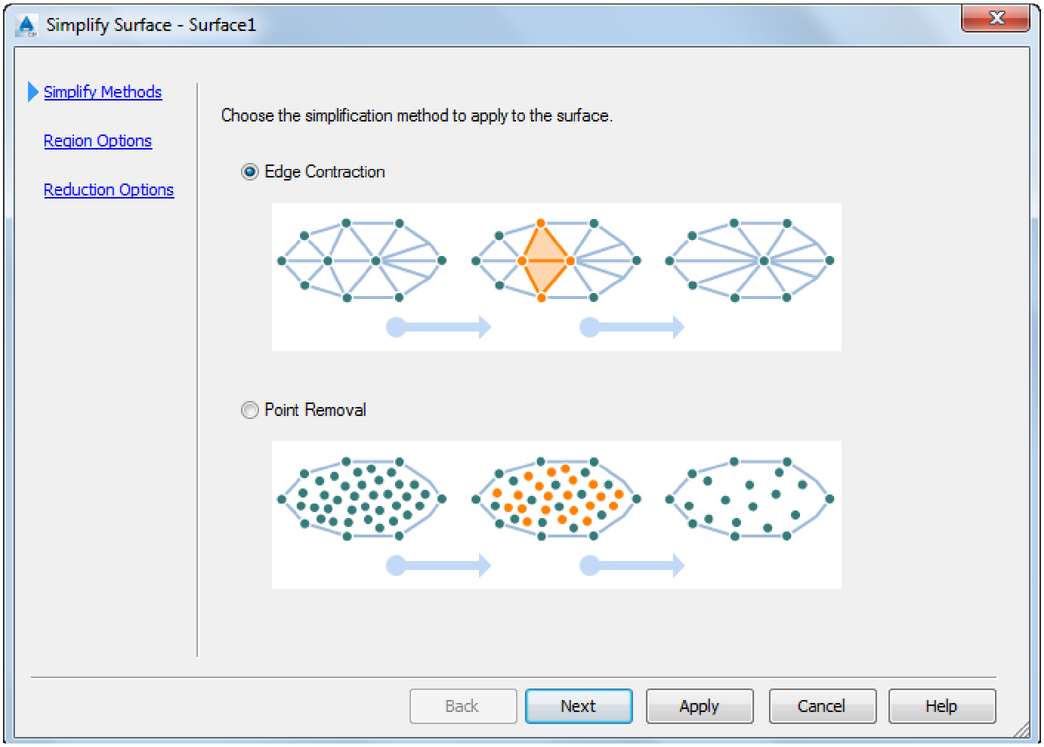



*Figure 3-36 The **Simplify Surface-Surface1** wizard with the **Simplify Methods** page displayed*

Select the required method for simplifying the surface and choose the **Next** button; the **Region Options** page will be displayed. Specify the region that you want to simplify in this page. You can select the surface border to simplify the entire surface, or select polygons or objects from a specific region to simplify it. Choose the **Next** button to display the **Reduction Options** page. Specify the percentage of the points to be removed from the surface or the selected region using the slider bar in the **Reduction Options** area of this page.

Next, choose the **Finish** button; the wizard will be closed and the surface will be simplified. You will notice a change in the surface triangulation due to the removal of surface points. Note that this option does not affect the accuracy of the surface. Figures 3-37 and 3-38 show a surface before and after using the **Simplify Surface** option.

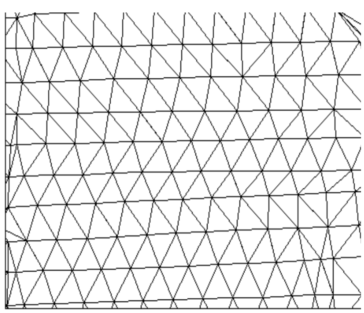


*Figure 3-37 Surface before using the **Simplify Surface** option*

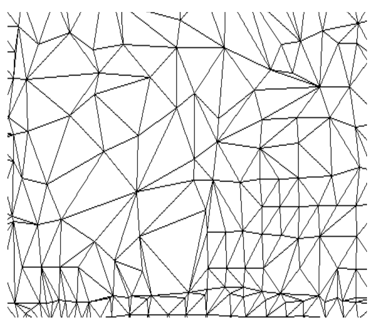


*Figure 3-38 Surface after using the **Simplify Surface** option*

## Creating Surface Masks

Surface masks are used to block the display of a surface area or a specific area on the surface, and to assign it a different render material. The masking region can be defined by using closed polygons, polylines, 3D polylines, rectangles, feature lines, surfaces, and so on. To create a mask, expand the **Surfaces** node in the **Prospecto**r tab of the **TOOLSPACE** palette and then expand the required surface node. Next, right-click on **Masks** to display a shortcut menu and choose the **Create Mask** option from it; you will be prompted to select objects. Select the required polyline or other objects to define the boundary of the masking region and press ENTER; the **Create Mask** dialog box will be displayed. Specify a name, masking type, render material, and other values in the dialog box and choose the **OK** button; the masked region will be created.

The surface outside or inside the masking region will be hidden depending upon the type of mask used. You can create two types of masks, inside and outside. On creating the inside mask type;the area inside the polygon boundary will be hidden. On selecting the outside mask, the area outside the polygon will be hidden. The surface masks are listed under the **Masks** head in the respective surface node.

## Creating Volume Surfaces

To create a volume surface, you need minimum two surfaces; a base surface and a comparison surfaces. Volume surfaces are classified into two types, TIN volume surfaces and Grid volume surfaces.

To create a volume surface, expand the **Surfaces** node in the **Prospector** tab of the **TOOLSPACE** palette and right-click; a shortcut menu will be displayed. Choose the **Create Surface** option from the shortcut menu; the **Create Surface** dialog box will be displayed. In this dialog box, select the **TIN volume surface** option from the **Type** drop-down list; the properties of the TIN volume surface will be displayed in the dialog box. Click in the **Value** field of the **Base Surface** property; a browse button will be displayed. Choose the browse button; the **Select Base Surface** dialog box will be displayed. Select the surface that is to be assigned as the base surface from the list of surfaces in the dialog box and choose the **OK** button. Similarly, click in the **Value** field of the **Comparison Surface** property and then select a surface to be assigned as the comparison surface from the **Create Surface** dialog box. Specify values of the other properties in this dialog box and choose the **OK** button; the dialog box will be closed and the volume surface will be created and listed in the **Surfaces** node. Similarly, you can also create grid volume surface using the **Create Surface** dialog box.

# SURFACE STYLES

The surface styles control the display of the surfaces. You can create different types of surface styles as per your project requirements. The surface styles are created and managed by using the options in the **Surface Style** subnode in the **Settings** tab of the **TOOLSPACE** palette. Civil 3D also has some in-built surface styles that are listed in the **Surface Styles** node of the **Settings** tab. The surface styles that are listed in the **Surface Styles** node depend upon the type of template selected.

## Creating a Surface Style

You can create a new surface style based on the project requirement. To create a new surface style**,** expand the **Surface** node in the **Settings** tab of the **TOOLSPACE** palette; the **Surface Styles** subnode will be displayed. Now, right-click on the subnode; a shortcut menu will be displayed. Choose the **New** option from the shortcut menu; the **Surface Style-New Surface Style** dialog box will be displayed, as shown in Figure 3-39. The options in different tabs in this dialog box are discussed next.

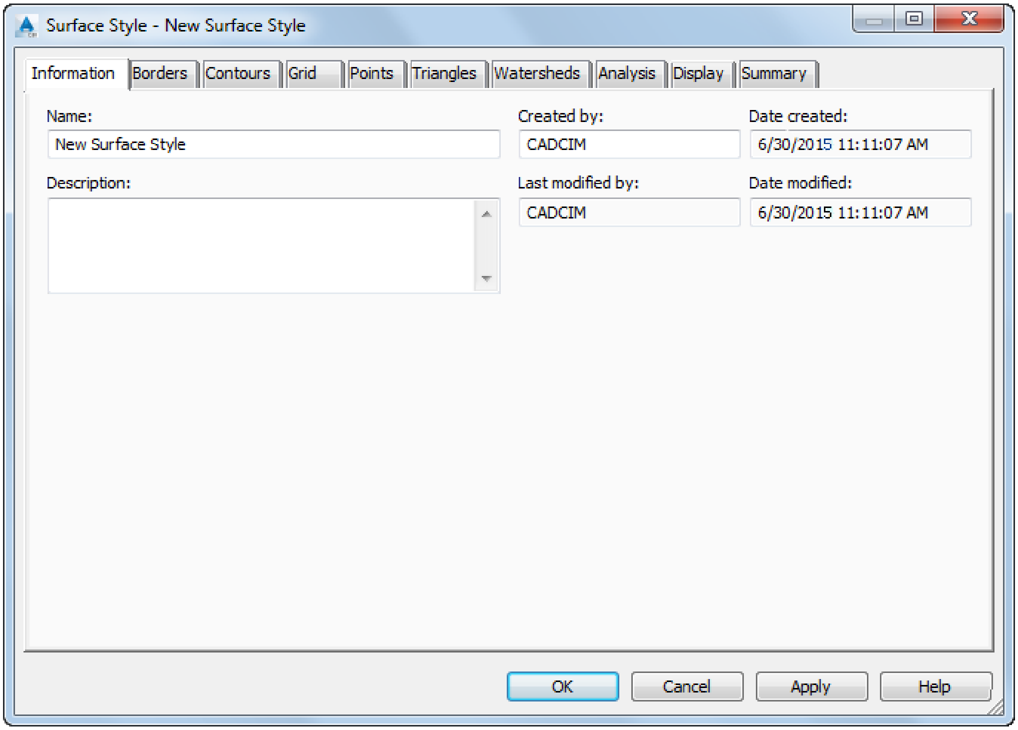



*Figure 3-39 The **Surface Style - New Surface Style** dialog box*

## Information Tab

The options in this tab are used to specify the name of the surface style, date of creation, description, and other basic information about the surface style. Specify the name for the surface style in the **Name** edit box. Optionally, you can specify the description in the **Description** text box.

## Borders Tab

This tab is used to set the display settings of the surface border by specifying the values of the border properties. These properties are listed under three categories: **3D Geometry**, **Border Types**, and **Datum**. These properties are discussed next.

### 3D Geometry

The properties in this category control the display of surface of the border. To control the display of the border, click in the **Value** field of the **Border Display Mode** property, and select the required mode from the drop-down list that will be displayed. The **Use Surface Elevation** option in this drop-down list is used to display the surface border at its original elevation. The **Flatten Elevations** option is used to display the border at the specified elevation. If you select this option from the drop-down list, the **Flatten Border to Elevation** property will be enabled in the dialog box. Specify the required elevation in the **Value** field of this property. If you select the **Exaggerate Elevation** option from the drop-down list, you will be able to exaggerate and scale the border according to the scale factor. You can specify the scale factor in the **Value** field of the **Exaggerate Borders by Scale Factor** property that will be enabled on selecting the **Exaggerate Evaluation** option.

### Border Types

The properties in this category help you control the visibility of the borders. The visibility of the outer border of the surface can be controlled by using the **Display Exterior Borders** property. The outer border is the border that represents the extents of the surface. By default, the value of this property is set to **True**. To hide the surface border, click in the **Value** field of this property and select **False** from the drop-down list.

The **Display Interior Borders** property in this category controls the visibility of the interior borders such as borders of ponds, wells, holes, or areas, where no data is available. By default, the value of this property is set to **True**. To make the interior border invisible, click on the **Value** field and select **False** from the drop-down list; the interior border of the surface will be hidden. Figure 3-40 shows the exterior and interior borders of a surface.

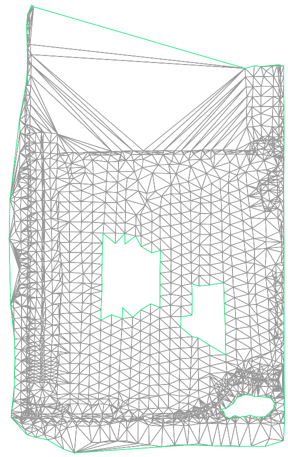


***Figure 3-40*** *Surface displaying the exterior and interior borders*

### Datum

The **Datum** category is used to specify whether to use a datum or not. By default, the value of this property is set to **False**. To show the datum, set the value to **True** and specify the datum elevation in the **Value** field of the **Datum Elevation** property. To project this boundary to the datum, set the value of the **Project Grid to Datum** property to **True**. The datum projection is visible only in the 3D view, as shown in Figure 3-41.

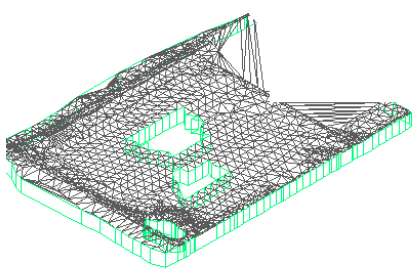


***Figure 3-41*** *A surface border projected to the datum in 3D view*

## Contours Tab

This tab is used to define the settings for the display of the surface contours. The properties in the **Contours** tab are listed under six different categories that are discussed next.

### Contour Ranges

Expand the **Contour Ranges** category to display various properties that control the display of the contours in the contour analysis. Contour analysis will be explained in detail in the next chapter. In the **Contour Ranges** category, the **Group Values by** property specify the grouping of contours. In the **Value** field of the **Group Values by** property, the **Quantile** option is selected by default. This option is used to divide the data equally in the number of contour ranges specified. The **Number of Ranges** property specifies the number of ranges in which the contours will be divided. To set the number of ranges, click in the **Value** field of this property and set the value using the spinner displayed on the right of this property. The **Range Precision** property specifies the rounding value for the contour ranges. To specify the range precision, click in the **Value** field of this property and select the required option from the drop-down list. The **Use Color Scheme** property specifies whether or not to use a color scheme for the contour range. To use the color schemes, click in the **Value** field and select **True** from the drop-down list. On doing so, the **Major Color Scheme** and **Minor Color Scheme** properties will be enabled. Click on the **Value** field of these properties and select the required color scheme from the drop-down lists displayed.

### 3D Geometry

Expand the **3D Geometry** category. The properties in this node are similar to those explained in the **3D Geometry** category of the **Borders** tab.

### Legend

Expand the **Legend** category. The property in this category specifies the default contour legend style. The contour legend provides information of the surface contours in a consolidated form. To modify the contour style, click on the **Value** field of this property and choose the browse button displayed on the right. On doing so, the **Contour Legend Style** dialog box will be displayed. Select the required style from the dialog box and choose the **OK** button; the specified style will be applied to the contour legends.

### Contour Intervals

Expand the **Contour Intervals** category to view properties in this category. There are three properties under this category. The **Base Elevation** property specifies the relative base elevation for the major or minor contour intervals. The major and minor contour intervals are defined with base elevation as the reference. Click on the **Value** field of the **Base Elevation** property and specify the required elevation or choose the button displayed on the right to specify the base elevation from the drawing area. Similarly, you can specify the minor contour interval in the **Value** field of the **Minor Interval** property. Note that when you change the value of the **Minor Interval** property, the value of the **Major Interval** property will change automatically. The value of the major contour interval depends upon the minor contour interval and is always a multiple of the minor contour interval value. For example, if the **Minor Interval** value is **5**, the value for the **Major Interval** should be **25**.

### Contour Depressions

Expand the **Contour Depressions** category. The properties in this category are used to control the display of the depression contours. The **Display Depression Contours** property in this category controls the display of depression in the contours. By default, the value of this property is set to **False**. Click on the **Value** field of this property and select **True** from the drop-down list to make the depression contours visible on the surface. The depression contours are indicated by tick marks around the contour boundary. You can set the interval between the tick marks in the **Tick Mark Interval** property. This property controls the distance between the tick marks displayed along the contour periphery. The number of tick marks generated along the contour boundary will vary according to the distance. To specify the interval between tick marks, click on the **Value** field and choose the button displayed on the right; you will be prompted to specify the distance. Specify the distance in the command line and press ENTER or pick points on the drawing to specify the distance. Similarly, specify the length of the tick marks in the **Value** field of the **Tick Mark Length** property to view them clearly. Figure 3-42 shows the depression contours displayed after setting the value of the **Display Depression Contours** property to **true** and with the tick marks facing inward.

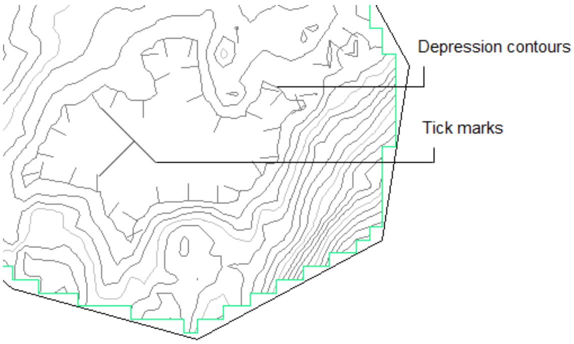



***Figure 3-42*** *Depression contours on the surface with the tick marks facing inside*

**Note**

*Contours will be displayed on the surface only if the visibility of the **Major Contour** and **Minor Contour** layers is turned on in the **Display** tab of the **Surface Style - New Surface Style** or **Surface Style - <surface>** dialog box.*

### Contour Smoothing

Expand the **Contour Smoothing** category to view the properties available in this category. The **Smooth Contours** property enables smoothing of contour lines by adding new vertices or drawing contours as spline curve. By default, the contour smoothening value is set to **False**. To enable contour smoothening, click in the **Value** field of this property and select **True** from the drop-down list that is displayed. This action will activate the **Smoothing Type** property in the **Contour Smoothing** category. Next, click in the **Value** field of the **Smoothing Type** property to select the type of contour smoothening. The **Add vertices** type of smoothening adds vertices on the contours to give them a smooth appearance. The **Spline curve** type of smoothening creates a spline curve through the contour points.

You can also control smoothening by using the slider bar in the **Contour smoothing** area at the bottom of the **Surface Style** dialog box. Note that this slider is available only for the **Add vertices** type of smoothening.

The **Display table** at the bottom of the tab shows the properties of the contour range in different columns. The **Number** column in this table displays the contour ranges specified. The number displayed in the **Number** column corresponds to the value specified in the **Number of Ranges** property. The **Major Display** and the **Minor Display** columns specify the display of the major and minor contours in that particular range.

You can specify the linetype, lineweight, and color for the contours by using the three buttons in the **Minor Display** and **Major Display** columns. To specify the linetype for the major contour, choose the first button out of the three buttons in the **Major Display**column; the **Select Linetype** dialog box will be displayed. In this dialog box select the required **Linetype** and then choose the **OK** button; the dialog box will be closed and the selected linetype will be assigned to the major contour. Similarly, you can specify the line weight and color for the contour by using the options available in the **Lineweight** and **Select Color** dialog boxes, respectively. These dialog boxes can be invoked by choosing the second and third button, respectively.

## Grid Tab

This tab controls the grid display of the grid surface. The grid properties are divided under three categories, **3D Geometry**, **Primary Grid**, and **Secondary Grid**. These categories are discussed next.

### 3D Geometry

Expand the **3D Geometry** category. The properties in this node are similar to those explained in the **3D Geometry** category of the **Borders** tab.

### Primary Grid

The properties in this category control the display of the primary grid on the surface. The primary grid represents the grid lines along the Y direction. The **Use Primary Grid** property in this category specifies whether to display the primary grid in the drawing or not. By default, the display of the primary grid is set to **true**, which means the primary grid lines will be displayed automatically.

The **Interval** property of the primary grid specifies the distance between the primary grid lines. Click in the **Value** field of this property and choose the button displayed on the right; you will be prompted to specify the distance. Specify the distance in the command line and press ENTER or pick points in the drawing.

The **Orientation** property specifies the direction or orientation of the primary grid lines. Specify the orientation for the grid lines in the **Value** column of this property. You can also use the button displayed on the right of this field to specify the orientation.

### Secondary Grid

The secondary grid represents the grid lines along the X direction. The properties in this category are same as those of the primary grid.

## Points Tab

This tab is used to control the display of the surface points. The categories and their properties in this tab are discussed next.

### 3D Geometry

The properties in this category are same as those discussed in the **Borders** tab except that here they are used for the surface points.

### Point Size

The properties in this category specify the units and size of the surface points. The **Point Scaling Method** property in this category specifies the scaling method to determine the point size. To select the scaling method, click on the **Value** field of this property and select the required method from the drop-down list. The **Point Units** property specifies the value used for the point size. To specify the point size, click on the **Value** field of this property and specify the value or choose the button displayed on the right to specify the point units by picking points in the drawing.

### Point Display

The properties in this category are used to control the display of surface points. The **Data Point Symbol** property in this category specifies the symbols used to display the surface point. Click in the **Value** field of this property; a button will be displayed on the right. On choosing this button, different symbols for the point display will be displayed, as shown in Figure 3-43. You can select the required symbol to display the surface points.

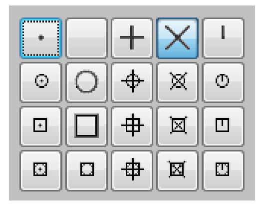


***Figure 3-43*** *Different symbols available for the point display*

The **Data Point Color** property specifies the color of the surface points. Click in the Value field of this property; a button will be displayed on the right. Choose the button; the **Select Color** dialog box will be displayed. Select the desired color for the data points from this dialog box and choose the **OK** button. The **Derived Point Symbol** property is used to specify the symbol for the display of the derived points. The derived points are the points that are added during surface smoothening and other calculations. You can select the required symbol and color for the derived points in the same way as discussed for the surface data points. Similarly, select a symbol and color for the non-destructive points. The non-destructive points are the points that are created when the non-destructive breaklines are added to the surface.

**Note**

*The surface points will be visible in the drawing only if the visibility of points is turned on in the **Display** tab of the **Surface Style** dialog box.*

## Triangles Tab

This tab is used to specify the properties of the triangle components of the TIN surface. The properties in this tab are discussed next.

### 3D Geometry

The **Triangle Display Mode** property specifies the mode by which the triangles will be displayed. You can select the options to assign a specific display mode for the triangles on the surface from the drop-down list available in the **Value** column of this property. There are three options available in this drop-down list, **Use Surface Elevation**, **Flatten Elevations**, and **Exaggerate Elevations**.

On selecting the **Use Surface Elevation** mode from the drop-down list, the triangles will be displayed at the surface elevation which is the actual surface elevation, as shown in Figure 3-44. On selecting the **Flatten Elevations** mode, the triangles will be flattened upto a specified elevation on the surface, as shown in Figure 3-45. Note that the elevated areas in the surface have been flattened and the surface has shifted from its original elevation.

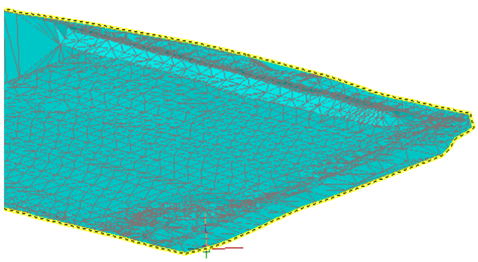


***Figure 3-44*** *Surface border and triangles created after using the* ***Use Surface Elevation*** *option*

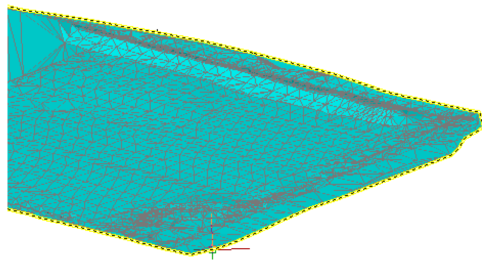


***Figure 3-45*** *Surface border and triangles created after using the* ***Use Surface Elevation*** *and* ***Flatten Elevations*** *options*

On selecting the **Exaggerate Elevations** mode, the triangles on the surface will be exaggerated by a specified scale factor, as shown in Figure 3-46. In this surface, the elevated areas have been exaggerated and elevated by a specified scale factor. Also, the surface has been elevated from its original elevation that is displayed by the surface border.

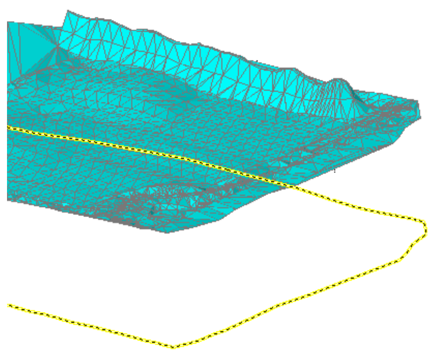


***Figure 3-46*** *Surface triangles created using the* ***Exaggerate Elevations*** *option and the border below the exaggerated surface*

## Watersheds Tab

Watershed is an area drained by a river or any other watercourse. The watershed areas help you to determine the drain points. The **Watersheds** tab is used to control the display of watersheds in a surface. The watershed properties are listed in different categories, as shown in Figure 3-47. The properties in this tab are discussed in detail in the next chapter.

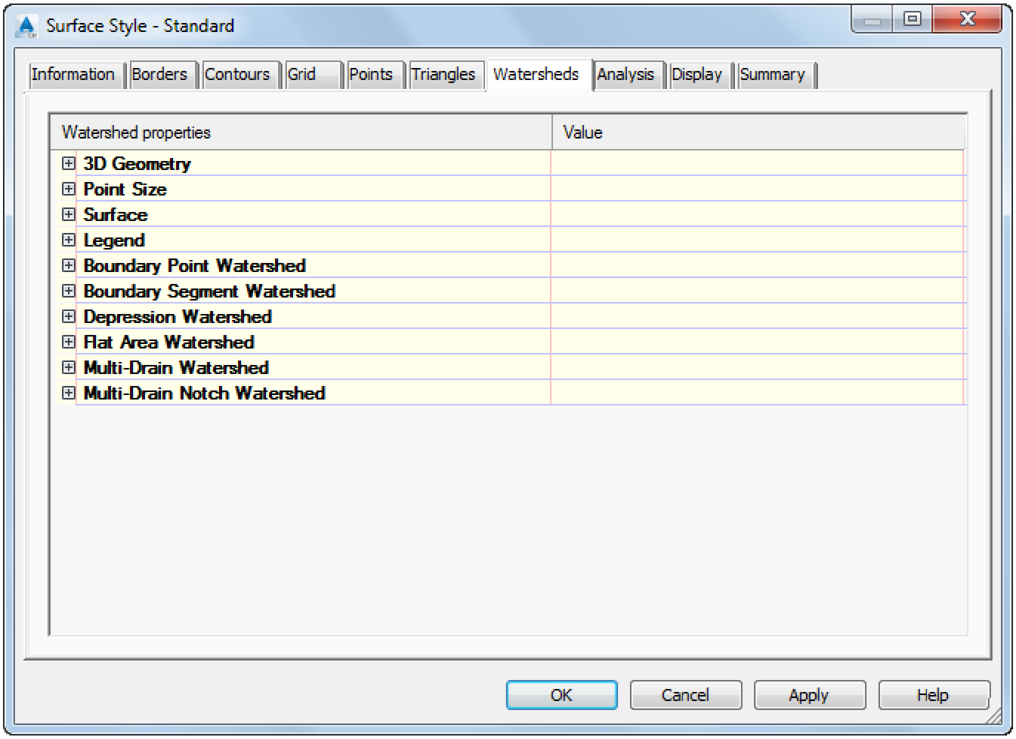



*Figure 3-47 The **Watersheds** tab of the **Surface Style - Standard** dialog box*



## Analysis Tab

This tab is used to specify the parameters that control the display of the surface analysis. There are four main types of analyses that can be used to study a surface. These are directions, elevations, slope, and slope arrows. The properties in this tab are discussed next.

### Directions/Elevations/Slopes/Slope Arrows Categories

Expand these categories. The properties in these categories specify the display settings of the directional type of surface analysis (discussed in the next chapter). The **Scheme** property specifies the color scheme of the ranges. Click in the **Value** field of the respective category and select the required color scheme from the drop-down list. The **Group by** property specifies the method to create and group the direction ranges. Click in the **Value** field of this property and select the grouping method from the drop-down list. To specify the number of ranges in which the directions will be grouped, click on the **Value** field of the **Number of Ranges** property and use the spinner on the right of this field to set the number. Note that the default value for this property is **8**.

To set the range precision, click in the **Value** field of the **Range Precision** property and select the value for the precision from the drop-down list. The **Display Type** property specifies the entity created during surface analysis. Click in the **Value** field and select the required entity from the drop-down list. You can create 2D faces, 3D faces, 2D solids, or meshes using the options in this drop-down list. However, only the meshes and 3D faces can be viewed in the 3D mode.

To set the legend style for these categories, click in the **Value** field of the **Legend Style** property of the respective category and choose the button that is displayed on the right; the **Legend Style** dialog box will be displayed. Select the required legend style using the options in this dialog box and choose the **OK** button to close this dialog box. The different types of surface analyses and the process to execute them are discussed in detail in next chapter.

## Display Tab

This tab controls the visibility and display settings of the surface components. You can use the properties under this tab to change the visibility, color, or layer for the surface components at various phases of the project and also in a specific view direction. The options in this tab are discussed next.

### View Direction

This option specifies the direction in which you can specify the display settings and view the surface. The **Plan** view is the default view direction that helps you to view the drawing in the 2D mode. The **Model** view helps you specify the settings in 3D view. The **Section** option helps you to view the surface in a section. You can specify the settings of the surface components in all the three view directions.

### Component Display Area

The **Component Display** area displays various components and view properties of the surface components in different columns. The **Component Type** column lists the names of all components of the surface or any Civil 3D object. The **Visible** column specifies the visibility of the components. To turn the visibility of a component on or off, click on the bulb icon in the **Visible** column. The yellow color of the bulb icon indicates that the component's visibility is turned on. As a result, the component will be displayed on the surface.

The **Layer** column displays the layers of the component. By default, the **0** layer is assigned to all components. Click on the default layer value for a specific component in the **Layer** column; the **Layer Selection** dialog box will be displayed. Select or create a new layer for the component by using this dialog box. You can also set the color, linetype, and lineweight for all the components displayed in the **Layer** column.

## Summary Tab

This tab displays the summary of surface style properties and the assigned parameters in the **Value** columns under different categories. You can expand the categories to view and edit the properties.

On specifying the values of the surface style properties, choose the **OK** button from the **Surface Style - New Surface Style** dialog box; the dialog box will be closed and the selected surface style will be added to the **Surface > Surface Styles** subnode of the **Settings** tab.

# Editing Surface Styles

| | |
|---|---|
| **Ribbon:** | Surface > Modify > Surface Properties drop-down > Edit Surface Style |
| **Command:** | EDITSURFACESTYLE |

To edit a surface style, choose the **Edit Surface Style** tool from the **Modify** panel; you will be prompted to select the surface. Select the required surface and press ENTER; the **Surface Style - <style name>** dialog box will be displayed. You can edit the properties of the surface style in this dialog box as per your requirement and choose the **OK** button to close it.

**Tip.** *Alternatively, choose the **Settings** tab in the **Toolspace** palette and expand **Surface > Surface Styles** subnode. Select the required surface style and right-click on it; a shortcut menu will be displayed. Choose the **Edit** option from the shortcut menu; the **Surface Style - <style name>** dialog box will be displayed.*



## Editing Surface Properties

**Ribbon:** Surface > Modify > Surface Properties drop-down > Surface Properties
**Command:** EDITSURFACEPROPERTIES

Every Civil 3D surface has some properties that can be viewed and edited at any stage of the project. The surface properties can be viewed in the **Surface Properties** dialog box. To view and edit the surface properties, choose the **Surface Properties** tool from the **Modify** panel; the **Surface Properties - <surface name>** dialog box will be displayed.

Alternatively, to display this dialog box, choose the **Prospector** tab of the **TOOLSPACE** palette and expand the **Surfaces** node. Next, select the required surface and right-click; a shortcut menu will be displayed. Choose the **Surface Properties** option from the shortcut menu; the **Surface Properties - <surface name>** dialog box will be displayed. The dialog box has four tabs that are discussed next.

### Information Tab

This tab displays the general information about the surface such as surface name, description, surface style, and render material. The options in this tab are as follows:

#### Name

This option specifies the default name of the surface. Specify the name of the surface in the **Name** edit box.

#### Description

You can optionally enter the description of the surface in the **Description** text box.

#### Default styles

The **Default styles** area is used to specify the default surface style and render material style. Select the desired option from the **Surface style** and **Render Material** drop-down lists in this area to specify the surface style and render material. Instead of selecting an in-built surface style in Civil 3D, you can also create a new style, or copy and edit the current surface style in the **Default styles** area. To do so, choose the down arrow on the right from the respective drop-down list; a flyout will be displayed. Choose the required option from the flyout to perform the corresponding operations.

#### Object locked

If you select this check box, you will not be able to edit a surface. Clear the check box to edit the surface.

#### Show tooltips

This check box is selected by default. It displays the tooltips for the objects in the drawing. The tooltips display the information about the objects such as elevation, coordinates and description.

## Definition Tab

This tab is used to edit the surface definition items. The options in this tab are discussed next.

**Build:** Expand the **Build** options. The properties in this category are discussed next.

**Copy deleted dependent objects:** This property specifies whether the object information is to be copied to the surface definition in case the object is deleted. For example, if the polyline used for creating a boundary is deleted, the boundary is also deleted from the surface and its definition. If you set the value of this property to **Yes**, the objects such as boundary, breaklines or a point group will be retained even if you remove the parent polyline or any other object. The information of the deleted component will be copied and saved in the surface's definition.

**Exclude elevations less than:** This property is used to set the minimum limit of the surface elevation such that the value less than the specified value will be excluded while creating the surface. Click in the **Value** field of this property and select **Yes** from the drop-down list to exclude the points that have elevations less than the specified value.

**Elevation:** This property will be activated only if you have set the value for the **Exclude elevation less than** property to **Yes**. In the **Value** column of this property, you can set the value for the minimum limit of the surface elevation, such that the value less than the specified value will be excluded while creating the surface.

**Exclude elevations greater than:** This property sets the maximum limit of the surface elevation such that the values greater than the specified value will be excluded from the surface. Click in the **Value** field of this property and select **Yes** from the drop-down list to exclude the elevations greater than the specified value. Specify the maximum value in the **Value** field of the **Elevation** property. The elevation greater than this value will be excluded from the surface.

**Use maximum angle:** This property sets the maximum limit of the angle between the adjacent TIN lines such that values greater than the specified values will be excluded from the surface. Click in the **Value** field of this property and select **Yes** to exclude the angles greater than the specified value. Specify the maximum value in the **Value** field of **Maximum angle between adjacent TIN lines** property. The angle greater than this value will be excluded from the surface.

**Maximum angle between adjacent TIN lines:** This property will be activated only if you have set the value for the **Use maximum angle** property to **Yes**. In this property, you can specify the limit value in the **value** column for the maximum angle between the adjacent TIN lines when the surface is built.

**Use maximum triangle length:** This property specifies whether the triangles that have length more than the specified length should be removed from the surface or not. Click in the **Value** field of this property and select **Yes** from the drop-down list to remove the triangles with a length greater than the specified length of the triangle.

**Maximum triangle length:** This option is activated after you select **Yes** for the **Use maximum triangle length** option in the **Value** field. Click in the **Value** field and specify the value for the length. The triangles with length greater than the length specified in the column will be removed while creating the surface.

**Convert proximity breaklines to standard:** This option specifies whether to convert the proximity breaklines into standard breaklines or not. Click in the **Value** field of this property; a down arrow will be displayed. Choose the down arrow and select **Yes** from the drop-down list to convert the proximity breaklines into standard breaklines.

**Allow crossing breaklines**. This property is used to specify whether to allow the crossing of breaklines or not. If you set the value of this property to **Yes**, then you need to specify the elevation at the point of intersection.

**Elevation to use:** This property is available only when the value for **Allow crossing breaklines** property is set to **Yes**. Click in the **Value** field of this property and select any one of the following options from the drop-down list. Select the **Use first breakline elevation at intersection** option to use the elevation of the first breakline to determine the elevation at the intersection. Select the **Use last breakline elevation at intersection** option to use the elevation of the last breakline to determine the elevation at the intersection. Select the **Use average elevation at intersection** option to use the average of the first and last breakline elevation values to determine the elevation at the intersection.

#### Data operations

Expand the **Data operations** options to view the data operations performed on a surface. You can specify whether or not to include the data operations in the surface build. To include the required data operation, select the check box next to it from the **Operation Type** list box at the bottom of the dialog box. You can change the visibility, color, or layer for the surface components at various phases of a project. By default, the value for all the data operations is set to **Yes**. Click in the **Value** field of a data operation; a down arrow will be displayed. Choose the down arrow and select **No** to exclude the data operation from the surface build.

#### Edit operations

Expand the **Edit operations** options to view different surface edit operations. By default, the value for all the edit operations is set to **Yes**. Click in the **Value** field of a data operation; a down arrow will be displayed. Next, choose the down arrow and select **No** to exclude the edit operation from the surface build. You can also specify whether or not to include the edit operations in the surface build by using the **Operation Type** list box. To do so, select or clear the edit operations check boxes in this list box.

#### Operation Type

The **Operation Type** area of the **Surface Properties** dialog box displays all surface operations and their parameters in the order in which these operations were performed. These parameters are listed in the **Parameters** column. You can clear the check box for any surface operation listed in the **Operation Type** column to exclude it from the surface build.

### Analysis Tab

This tab is used to specify the properties of the surface analysis. You can also modify the properties of existing surface analysis using the options in this dialog box. Various options in the **Analysis** tab are discussed next.

#### Analysis type

This option specifies the type of analysis selected. You can select the type of surface analysis from the **Analysis type** drop-down list. By default, **Elevations** is selected from the drop-down list. The properties of the selected analysis type will be displayed in the tab.

### Legend

This option specifies the legend style for the selected analysis type. Select the required legend style from the **Select a Style** drop-down list in the **Legend** area.

### Preview

Select the **Preview** check box to display the legend style in the **Preview** area, as shown in Figure 3-48. You can use the **ViewCube** in the **Preview** area to rotate and adjust the legend table.

### Ranges

The **Ranges** area displays the number of ranges required for the surface analysis. These ranges will be displayed in the **Range Details** area below the **Create ranges by** drop-down list. Use the **Number** spinner in this area to set the number of ranges. The default value displayed in the spinner is 8. You can use the **Run analysis** button to run the surface analysis after specifying the required properties. The **Range Details** area of this tab displays the result of analysis in tabular form.

## Statistics Tab

The **Statistics** tab displays the statistical information about the surface grouped in categories. This information includes maximum and minimum values for elevation, slope/grade, coordinate in X and Y direction, number of triangles, and number of points.

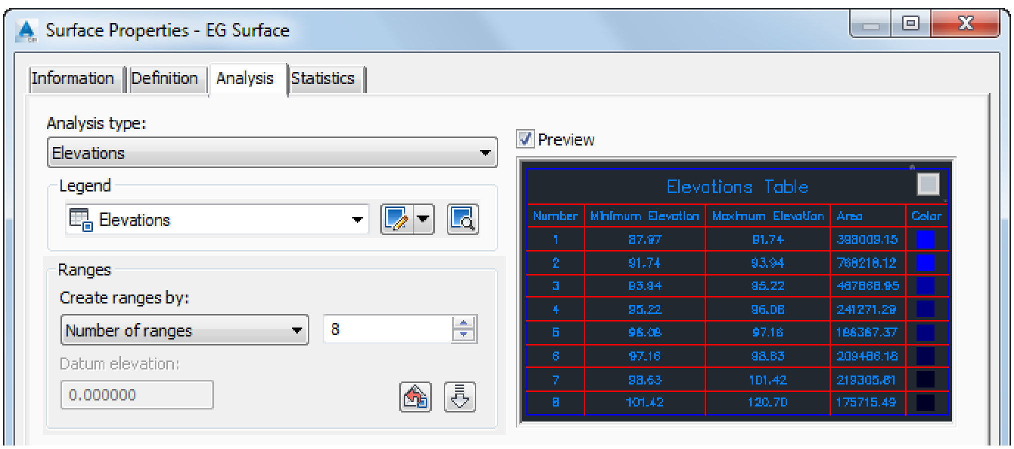



*Figure 3-48 Partial view of the* ***Surface Properties - EG Surface*** *dialog box with the* ***Preview*** *check box selected and legend style displayed*

**Note**

*The information displayed in the third category in the* ***Statistics*** *tab varies according to the type of surface created. For example, in case of TIN surface, the number of triangles, maximum triangle area, and so on will be displayed in the TIN category. If the surface is a grid surface or volume surface, then properties displayed will vary accordingly.*

# SURFACE TOOLS

AutoCAD Civil 3D provides various tools which make the surface object more interactive. These tools can be accessed from the **Surface Tools** panel in the **Surface** contextual tab. To display the **Surface** tab, choose the **Surface** tool from the **Ground Data** panel of the **Modify** tab. The Surface tools are discussed next.

## Drape Image

| | |
|---|---|
| **Ribbon:** | Surface > Surface Tools > Drape Image |
| **Command:** | DRAPEIMAGE |

This tool is used to drape an image imported from Google Earth over an existing surface. Before using this tool, you need to import and insert an image from the Google Earth into Civil 3D and then overlay the image on the surface using the **Drape Image** tool.

To drape an image, choose the **Drape Image** tool from the **Surface Tools** panel; the **Drape Image** dialog box will be displayed, as shown in Figure 3-49.

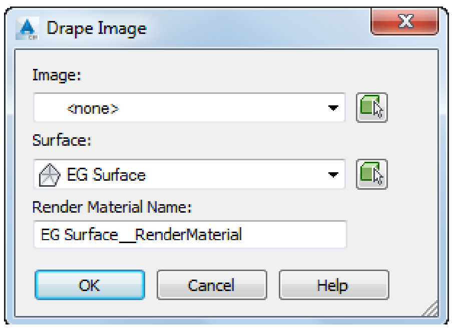



***Figure 3-49*** *The **Drape Image** dialog box*

This dialog box is used to select the required image and overlay it over the surface. Select the required image from the drop-down list in the **Image** area. You can also choose the **Select from the drawing** button on the right of this drop-down list and select the image from the drawing. Note that the images imported from Google Earth are named automatically when imported to Civil 3D. Next, select a surface from the **Surface** drop-down list or select it from the drawing by choosing the **Select from the drawing** button on the right of the drop-down list. The render material is created automatically as Civil 3D renders the image draped over the surface. Now, choose the **OK** button; the dialog box will be closed. The image will be draped but you will not be able to view it. Before you view the image, select the imported image and right-click to display a shortcut menu. Choose **Isolate Objects > Hide Selected Objects** from the shortcut menu; the image will disappear and you will see the surface only. At this stage, you can view the surface in 3D mode only. To view the rendered image, choose the **View** tab and then select the **Realistic** visual style from **Visual Style** drop-down list in the **Views** panel, as shown in Figure 3-50. The gray scale Google Image will be draped over the surface and the image will be clipped according to the surface.

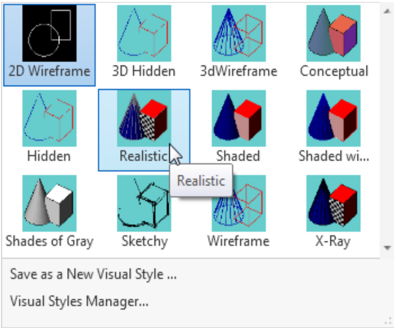



***Figure 3-50** The **Realistic** visual style selected from the **Visual Style** drop-down list*

## Extract Solids from Surface

| | |
|---|---|
| **Ribbon:** | Surface > Surface Tools > Extract from Surface drop down>Extract Solids from Surface |
| **Command:** | EXPORTSURFACETOSOLID |

This tool is used to extract solids from the surface based on the surface triangles and surface borders. To extract a solid from a surface, choose the **Extract Solids from Surface** tool from **Surface Tools** panel; the **Extract Solid from Surface** dialog box will be displayed, as shown in Figure 3-51.

Select the surface from the **Surface** drop-down list or choose the **Select from the drawing** button to select the surface from drawing.

In the **Vertical definition** area, the **Depth** radio button is selected by default. Enter the negative or positive integer value for the depth.. If you enter a positive value, the solid is extruded upward from the surface. If you enter a negative value, the solid is extruded downward from the surface. Select the **At fixed elevation** radio button to specify that the bottom of solid is at a fixed elevation. On doing so, the edit box below will be activated. Enter the required elevation value in the edit box. Select the **At a Surface** radio button to create a solid from a surface. On selecting this radio button, the edit box below will be activated. Enter the required value int he edit box. Alternatively choose, the **Select from the drawing** button; next to the edit box. The solid defined is at a relative distance from another surface.

In the **Drawing Output** area, the **Insert Into Current Drawing** radio button is selected by default and is used to create the solid in the current drawing. To create solid in another drawing, select the **Add To a New Drawing** radio button and browse to open the **Specify new drawing** dialog box, where you can specify the drawing name and location.

In the **Layer** area, specify the layer on which the solid is created and in the **Color** area, specify the color for the solid. Now, choose the **Create Solid** button. The solid surface will be saved with *.dwg* file extension.

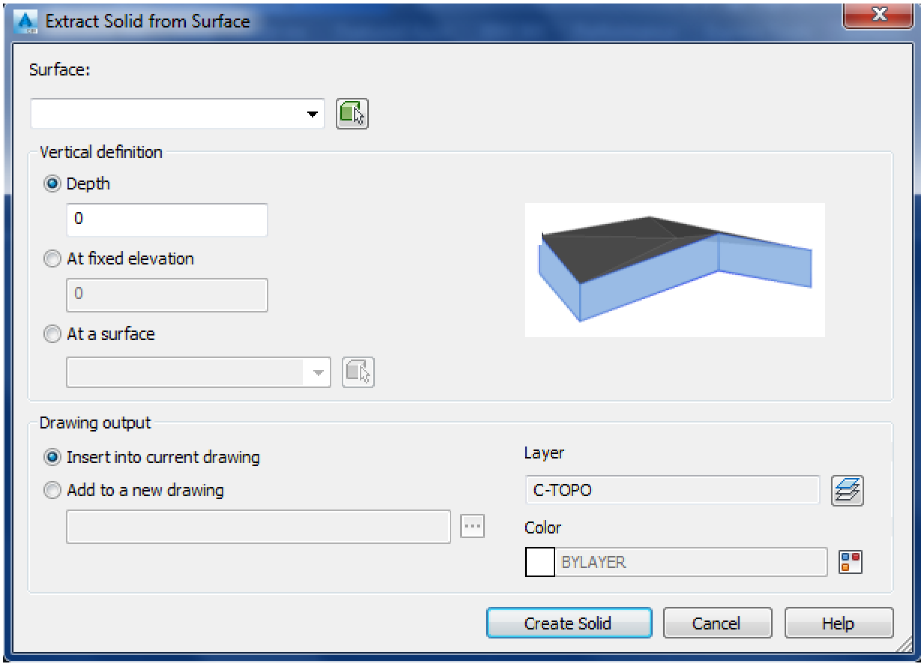



*Figure 3-51 The **Extract Solid from Surface** dialog box*

## Export to DEM

| | |
|---|---|
| **Ribbon:** | Surface > Surface Tools >Extract from Surface drop down>Export to DEM |
| **Command:** | SURFACEEXPORTTODEM |

AutoCAD Civil 3D allows you to export the surface data to DEM files that can be used in different programs such as AutoCAD Map 3D, ArcGIS, and Raster Design. To export the surface data to a DEM file, choose the **Export to DEM** tool in the **Surface Tools** panel; the **Export Surface to DEM** dialog box will be displayed, as shown in Figure 3-52. This dialog box is used to specify the parameters to export a file. The **Selected Surface** category of this dialog box displays information of the surface to be exported and the **Export** category displays the properties of the DEM file to be created.

To specify the DEM file name, click on the folder symbol in the **Value** field of the **Dem file name** property; the **Export Surface to DEM** dialog box will be displayed. In this dialog box, browse to the required location and specify a file name in the **File name** edit box. Next, select the file type from **Files of type** drop-down list. You can either select the **.dem* or **.tif* file format. Choose the **Save** button to close the dialog box. Similarly, click on the browse button in the **Value** field of the **Export coordinate zone** property; the **Select Coordinate System** dialog box will be displayed. Specify a coordinate zone for the DEM file in the dialog box and choose the **OK** button to return to the **Export Surface to DEM** dialog box. Choose the **OK** button; the **Export Surface to DEM** dialog box will be displayed until the surface is exported to DEM file. The file will be saved with **.tif* or **.dem* file extension. You can now import and use this DEM file in other programs supporting the USGS (.dem) or GEOTIFF (.tif) file types.

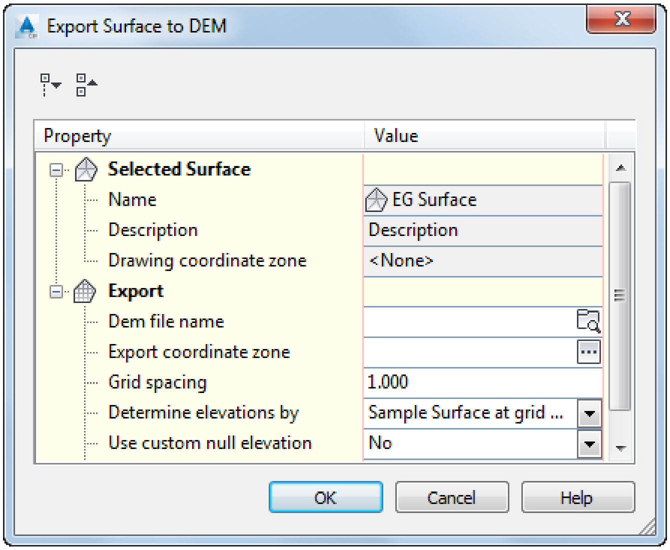



*Figure 3-52 The **Export Surface to DEM** dialog box*

## Extract Objects

| | |
|---|---|
| **Ribbon:** | Surface > Surface Tools >Extract from Surface drop down> Extract Objects |
| **Command:** | SURFACEEXTRACTOBJECTS |

The **Extract Objects** tool is used to extract the surface components and modify them without exploding them. In such cases, the extracted components retain their original properties. These components can be modified and used for developing further designs. For example, catchment areas, water drop paths, watersheds, and so on can be extracted and the data obtained from these objects can be used to design ponds, sewers, and so on. These components can be used by the Hydrological department for further investigations.

To extract objects from a surface, choose the **Extract Objects** tool from the **Surface Tools** panel; the **Extract Objects from Surface - <surface name>** dialog box will be displayed, as shown in Figure 3-53.

The dialog box displays the existing components of the surface in the **Property** column. By default, Civil 3D will extract all components. To extract the desired components, choose the down-arrow in the **Value** field of the corresponding property and then select the **Select from Drawing** option from the drop-down list; the **Select from Drawing** button will be displayed on the right. Choose the button; you will be prompted to select the required components. Select the required components and then press ENTER; the **Extract Objects from Surface - <Surface 1>** dialog box will be displayed with the required objects selected. Next, choose the **OK** button; the selected objects will be extracted. Now, enter **List** in the command line and press ENTER; you will be prompted to select the objects. Select the required extracted object and press ENTER; the **AutoCAD Text Window** will be displayed showing the information of the component that has been extracted.

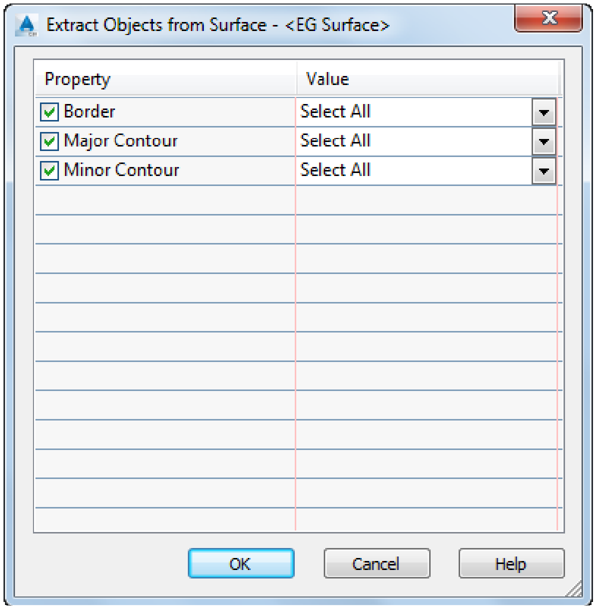



***Figure 3-53** The **Extract Objects from Surface - <EG Surface >** dialog box*

**Note**

*If there are multiple surfaces, you will be required to select the relevant surface. To do so, press ENTER; the **Select a Surface** dialog box will be displayed. Now, select the surface and then choose **OK**.*

# TUTORIALS

1. Download the *c02_c3d_2016_tut.zip* file from *http://www.cadcim.com*. The path of the file is as follows: *Textbooks > Civil/GIS > AutoCAD Civil 3D > Exploring AutoCAD Civil 3D 2016*.

2. Now, save and extract the downloaded file at the following location:

   *C:\c3d_2016\c03_c3d_2016_tut*

**Note**

*While opening the tutorial file, the **PANORAMA** window with an error message may appear. Close this window to proceed further.*

## Tutorial 1 — Creating Surface

In this tutorial, you will create a surface, as shown in Figure 3-54, and add point data to it. Then, you will apply a surface style and perform edit operations on it. **(Expected time: 30 min)**

The following steps are required to complete this tutorial:

a. Open a template file.

b. Create a TIN surface.
c. Create and assign a new surface style to the surface.
d. Save the file.

## Opening the Template File

1. Choose **New** from the **Application Menu**; the **Select template** dialog box is displayed.

2. In the **Select template** dialog box, select *_AutoCAD Civil3D (Imperial) NCS.dwt* template and choose the **Open** button.

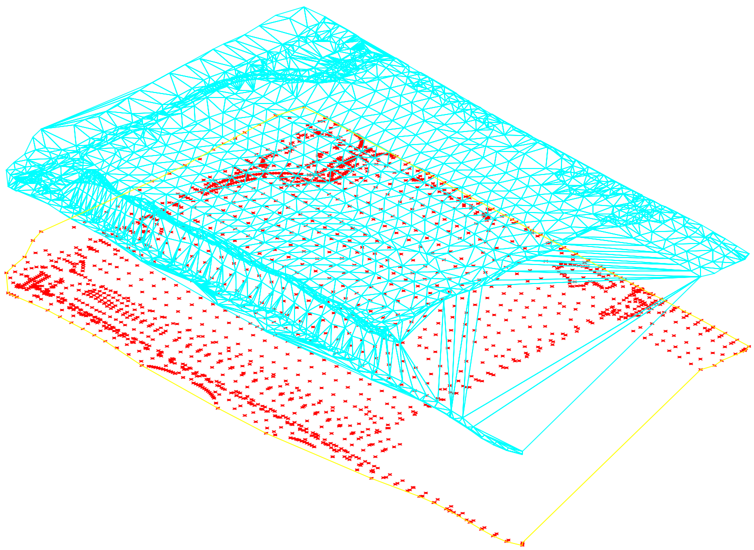


***Figure 3-**54 The surface displayed in 3D view*

## Creating the Surface

1. Choose the **Create Surface** tool from **Home > Create Ground Data > Surfaces** drop-down; the **Create Surface** dialog box is displayed.

2. Click in the **Value** field of the **Name** property and choose the browse button displayed on the right; the **Name Template** dialog box is displayed, as shown in Figure 3-55.

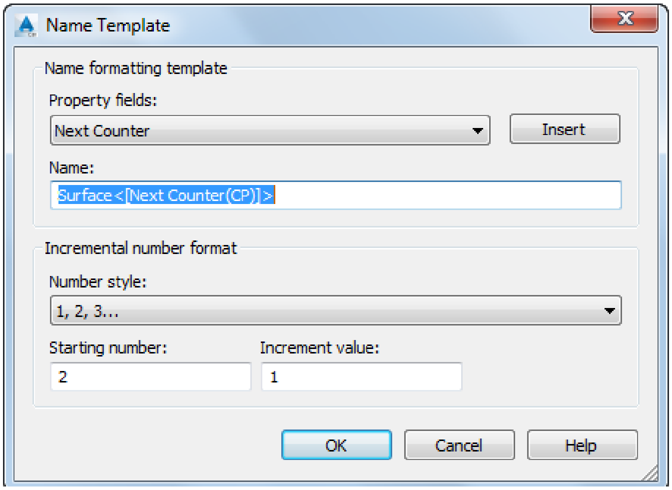



***Figure 3-55** The **Name Template** dialog box*

3. Enter the name **EG Surface** in the **Name** edit box of the dialog box and choose the **OK** button to close the dialog box.

4. Now, ensure that the **TIN surface** option is selected in the **Type** drop-down list of the **Create Surface** dialog box and choose the **OK** button; the dialog box is closed. Also the surface is created and added to the **Surfaces** node in the **Prospector** tab of the **TOOLSPACE** palette, as shown in Figure 3-56.

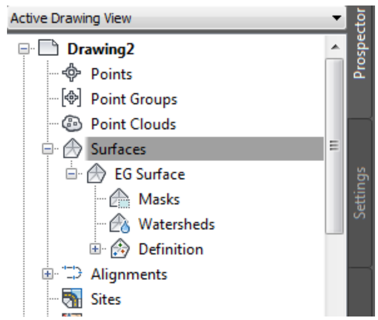



***Figure 3-56** The **EG Surface** added in the **Surfaces** node of the **Prospector** tab*

**Note**

*You cannot see the surface at this stage because it is empty and you need to add data to make it visible.*

## Adding Point File to the Surface

1. Expand **Surfaces > EG Surface > Definition** in the **Prospector** tab of the **TOOLSPACE** palette.

2. Select **Point Files** from the **Definition** node and right-click; a shortcut menu is displayed.

3. Choose **Add** from the shortcut menu; the **Add Point File** dialog box is displayed, as shown in Figure 3-57.

4. From the **Specify point file format** list box, select the **PENZD (space delimited)** option. This file format is used for the point files that are saved with the extension *.txt*.

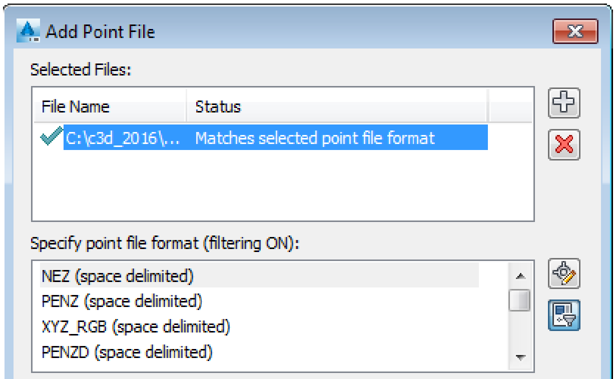



***Figure 3-57*** *Partial view of the **Add Point File** dialog box*

**Note`**
*If you select the wrong format for the point file from the **Specify point file format** list box, Civil 3D will display an error message indicating that the wrong file format has been selected.*

5. Next, choose the button on the right of the **Selected Files** list box; the **Select Source File** dialog box is displayed.

6. Browse to the folder *c03_c3d_2016_tut* and select the *c03 _c3d_2016_tut01- PENZD* text file. Choose the **Open** button; the file name and path is displayed in the **Selected Files** list box, refer to Figure 3-57.

7. Choose the **OK** button from the **Add Point File** dialog box; the data is added to the surface.

## Viewing the Surface

1. Enter **ZE** in the command line and then press ENTER; the surface is displayed with a surface border, and the major and minor contours, as shown in Figure 3-58. This is due to the default surface style settings of the template file.

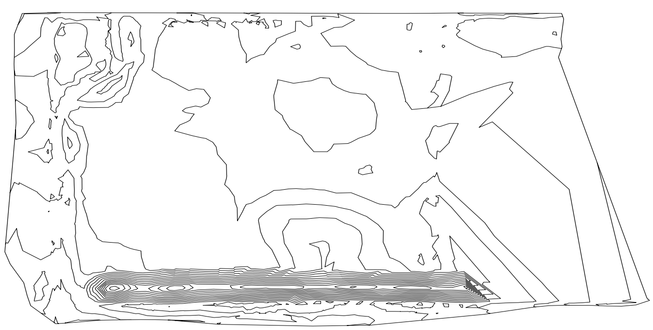


***Figure 3-58*** *The **EG Surface** displayed in the drawing area*

## Creating a New Surface Style

1. Choose the **Settings** tab of the **TOOLSPACE** palette; the items in the tab are displayed.

2. In the **Settings** tab, expand **Surface > Surface Styles**; you will notice that an orange triangular symbol is displayed on the left of the three surface styles namely: **Contours 1' and 5' (Design)**, **Contours 2' and 10' (Background)**, and **Contours 2' and 10' (Design)**, indicating that these surface styles are currently in use.

3. Select **Surface Styles** and right-click; a shortcut menu is displayed.

4. Choose **New** from the shortcut menu; the **Surface Styles - New Surface Style** dialog box is displayed. The **Information** tab is chosen by default in this dialog box.

5. Enter **EG Style** in the **Name** edit box of the **Information** tab.

6. Choose the **Apply** button; the name of the dialog box is changed to **Surface Style - EG Style**.

7. Next, choose the **Points** tab from the **Surface Style - EG Style** dialog box.

8. Click on the **Value** field of the **Point Units** property and enter **7.00'**.

9. Next, expand the **Point Display** category and click in the **Value** field of the **Data Point Symbol** property; a browse button is displayed on the right of this field.

10. Choose the button displayed; a small window with different point symbols is displayed. Retain the default settings.

11. Next, choose the **Display** tab. Make sure that the **Plan** option is selected in the **View Direction** drop-down list of the **Display** tab. The settings of the surface style can only be viewed in the plan.

12. Click on the bulb icon in the **Visible** column and turn on the visibility of **Points** and **Triangles**. Also, ensure that the **Major Contour** and **Minor Contour** layers are turned on so that the objects in these layers are visible when viewed in the plan view.

13. Choose the **Apply** button to configure the settings for the plan view of the surface.

14. Next, select the **Model** option from the **View Direction** drop-down list.

15. In the **Display** tab, turn on the visibility of **Points** and **Border**. Also, ensure that the visibility of **Triangles** is turned on.

16. Choose the **Triangles** tab and click in the **Value** field of the **Triangle Display Mode** property and select the **Exaggerate Elevation** option from the drop-down list. Next, enter **5.000** in the **Value** field of the **Exaggerate Triangles by Scale Factor** property.

17. Choose the **Apply** button; the settings are configured for the 3D view/model of the surface.

18. Next, choose the **OK** button in the **Surface Styles - EG Style** dialog box; the **EG Style** is created and added in the **Surface Styles** node of the **Settings** tab.

## Assigning the EG Style to the Surface

1. To assign the **EG Style** to the surface, choose the **Prospector** tab of the **TOOLSPACE** palette and expand the **Surfaces** node.

2. Select **EG Surface** and right-click; a shortcut menu is displayed.

3. Choose **Surface Properties** from the shortcut menu; the **Surface Properties - EG Surface** dialog box is displayed. The **Information** tab is chosen by default in this dialog box.

4. In the **Default styles** area of the tab, select **EG Style** from the **Surface style** drop-down list, as shown in Figure 3-59.

5. Choose the **Apply** button; a TIN surface is displayed with the surface points.

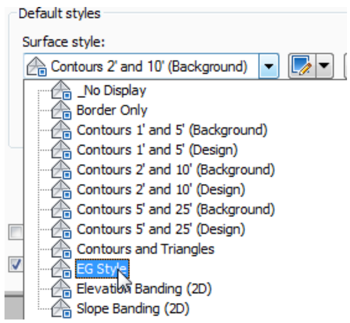



***Figure 3-**59 Selecting the **EG Style** option from the **Surface style** drop-down list*

6. Choose the **OK** button to close the **Surface Properties - EG Surface** dialog box. Figures 3-60 and 3-61 show the TIN surface with the surface points and the point style selected, respectively. Note that the surface points are represented by the symbol that was selected for the point display in the surface style.

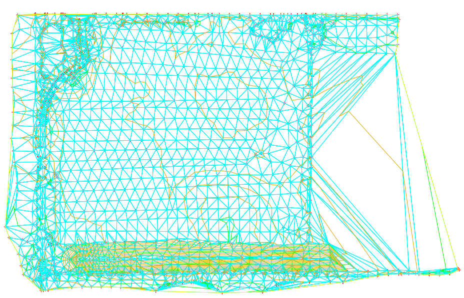


***Figure** 3-60 The TIN surface with the surface points*

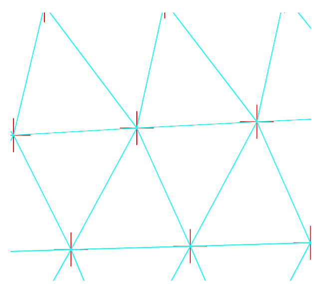


***Figure 3-**61 Points represented by the selected point symbol*

7. Choose the **SE Isometric** button from the **Views** panel of the **View** tab, as shown in Figure 3-62 to view the surface in 3D view; the surface is displayed with triangles at an exaggerated elevation, and the surface border and points are displayed at the original elevation, as shown in Figure 3-63.

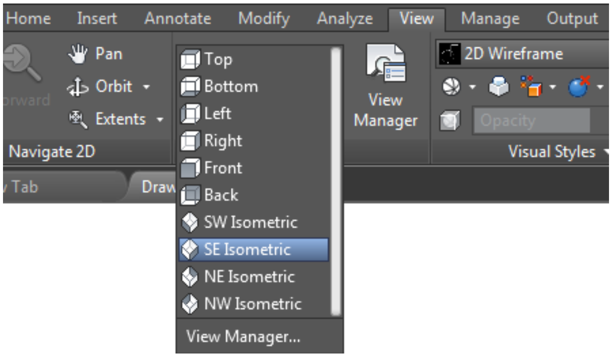



***Figure 3-62** The **SE Isometric** button chosen from the **View** tab*

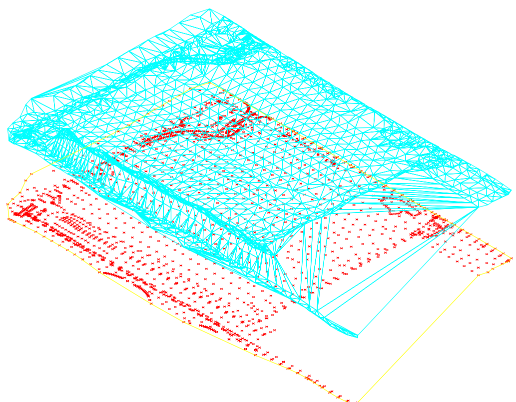


***Figure 3-63** The **SE Isometric** view of the **EG Surface***

**Note**

*Figure 3-63 displays the model view of the surface using the model view settings previously defined for surface style.*

## Saving the Drawing

1. Choose the **Save As** from the **Application Menu**; the **Save Drawing As** dialog box is displayed.

2. In this dialog box, browse to the following location:

   *C:\c3d_2016\c03_c3d_2016_tut*

3. In the **File name** edit box, enter **c03_c3d_2016_tut01a**.

4. Next, choose the **Save** button; the file is saved with the name *c03_c3d_2016_tut01a* at the specified location.

## Tutorial 2 Surface Pasting

In this tutorial, you will create two surfaces by using the point group and breaklines. Also, you will use some of the surface editing commands and paste the surface, as shown in Figure 3-64.
**(Expected time: 30 min)**

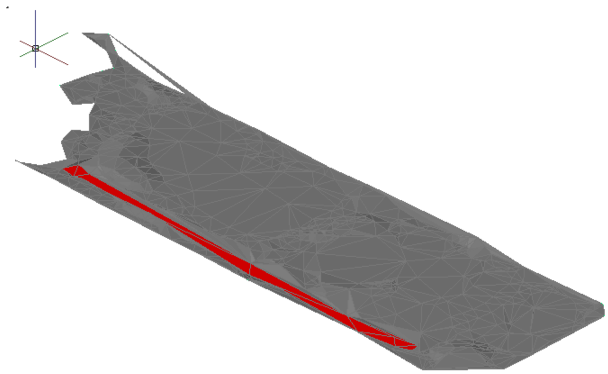


***Figure 3-64*** *The pasted surface*

The following steps are required to complete this tutorial:

a. Open the file.
b. Create a surface by using breaklines.
c. Create a surface by using point groups.
d. Use the **Delete Line** tool to edit the surface.
e. Paste the surface.
f. Save the file.

### Opening the File

1. Choose **Open** from the **Application Menu**; the **Select File** dialog box is displayed.

2. In the dialog box, browse to the location *C:\c3d_2016\c03_c3d_2016* where you have saved the file.

3. Select the file *c3d_2016_c03_tut02* and then choose the **Open** button to open the file.

### Creating E Street Surface

1. Choose the **Create Surface** tool from **Home > Create Ground Data > Surfaces** drop-down; the **Create Surface** dialog box is displayed.

2. Select the **TIN surface** option in the **Type** drop-down list of the displayed dialog box, if it is not selected by default.

3 Enter **E Street** in the **Value** field of the **Name** property.

4. Set the style to **E Style** in the **Value** field of the **Style** property. To do so, click on the

browse button in the **Value** field of the **Style** property; the **Select Surface Style** dialog box is displayed. In this dialog box, select the **E Style** option from the drop-down list and then choose the **OK** button; the selected style is displayed in the **Value** field of the **Style** property.

5. Next, choose the **OK** button from the **Create Surface** dialog box; the dialog box is closed and a surface is created. Now, you will add data to this surface.

6. Choose the **Prospector** tab and expand **Surfaces > E Street > Definition**.

7. In the **Definition** node, right-click on **Breaklines** and choose **Add** from the shortcut menu displayed; the **Add Breaklines** dialog box is displayed.

8. Enter **E Lines** in the **Description** text box and ensure that the **Standard** option is selected in the **Type** drop-down list.

9. Next, choose the **OK** button; the dialog box is closed and you are prompted to select the objects.

10. Select the two polylines from the drawing and then press ENTER; the breaklines are added to the **E Street** surface, as shown in Figure 3-65.

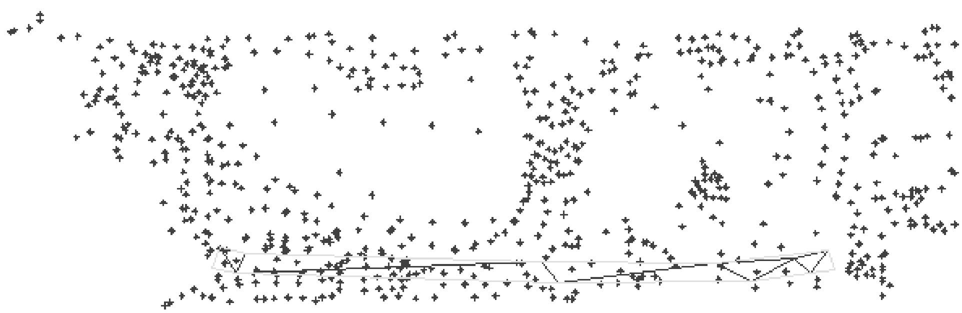


***Figure 3-65*** *Points and the* ***E Street*** *surface after adding breaklines*

**Note**

*The breaklines are added to the* ***Breaklines*** *subnode of the* ***E Street*** *surface's* ***Definition*** *node and they are also displayed in the List View.*

## Creating the Existing Ground Surface

1. Repeat steps 1 to 4 given in the previous section and then set the following values in the **Create Surface** dialog box:

   Surface type: **TIN surface** Name: **EG** Style: **Basic**

2. Now, choose the **OK** button; the **EG** surface is created.

3. Expand **Surfaces > EG > Definition** in the **Prospector** tab of the **TOOLSPACE** palette.

4. Right-click on **Point Groups** and choose **Add** from the shortcut menu displayed; the **Point Groups** dialog box is displayed.

5. Select the **_All Points** group and choose the **OK** button; tthe point group is added as the definition to the surface, as shown in Figure 3-66.

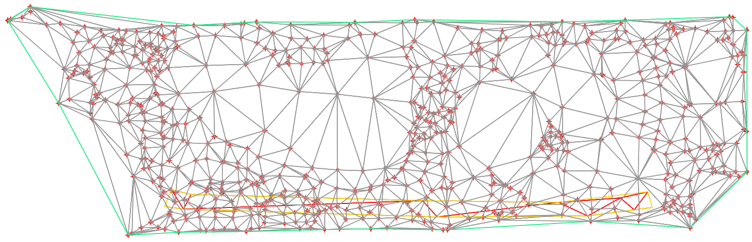


***Figure 3-66*** *The **EG** surface created after adding the point group*

## Editing the EG surface

1. Choose the **Surface** tool from the **Ground Data** panel of the **Modify** tab; the **Surface** contextual tab is displayed.

2. Next, choose the **Delete Line** tool from **Surface > Modify > Edit Surface** drop-down; you are prompted to select the surface from which you want to delete the lines.

3. Press ENTER; the **Select a Surface** dialog box is displayed.

4. Select the **EG** surface from the dialog box and choose the **OK** button; the dialog box is closed and you are prompted to select the required edges.

5. Select all the longer edges on the left side of the surface and shorter edges on the top left corner, as shown in Figure 3-67.

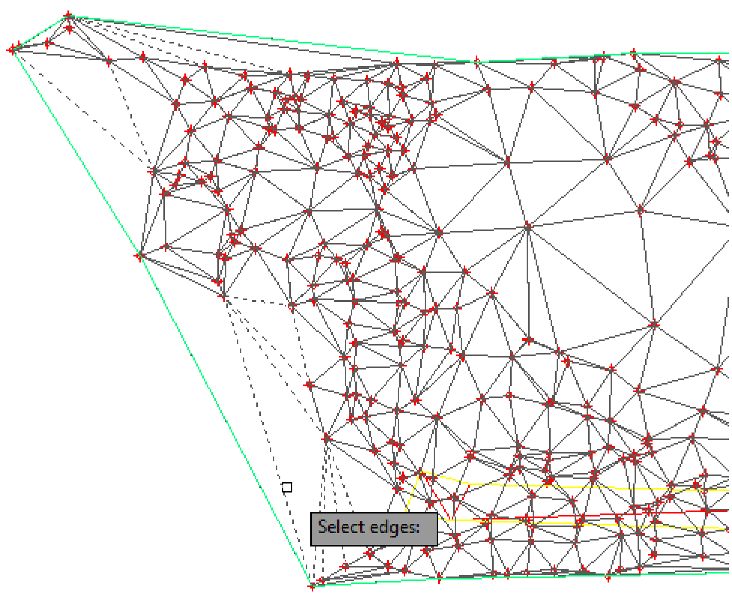



***Figure 3-67*** *Selecting the longer edges of the triangles*

6. Press ENTER; the selected lines are deleted and the surface is displayed, as shown in Figure 3-68. Press ENTER to exit the command.

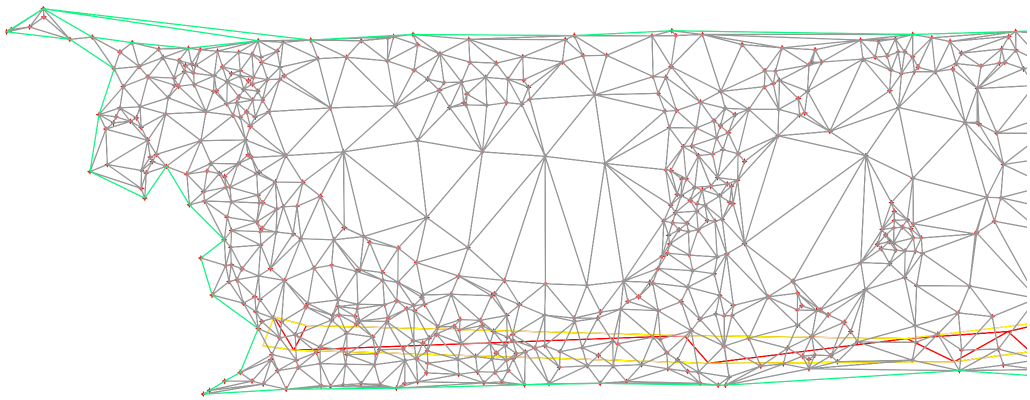


***Figure 3-68*** *The **EG** surface after deleting the TIN lines*

7. Before you paste the **E Street** surface on the **EG** surface, it is better to view the surfaces in a 3D view. To do so, select the **SE Isometric** option from the **Views** panel of the **View** tab; the **EG** surface is displayed in an isometric view, as shown in Figure 3-69.

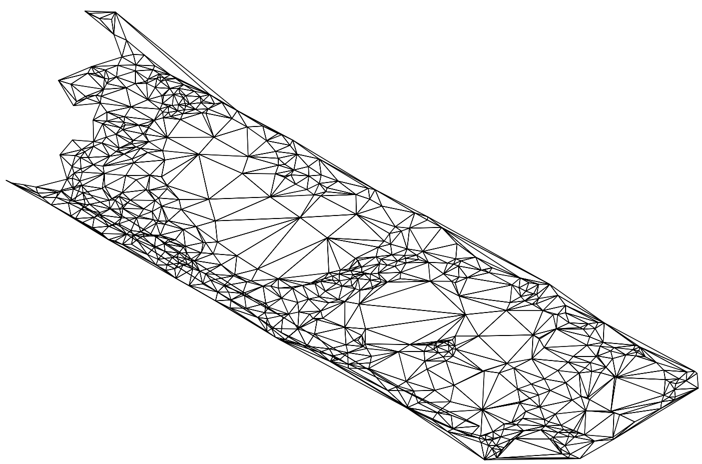


***Figure 3-69*** *The **EG** surface before pasting the **E Street** surface*

## Pasting the Surface

1. Choose the **Paste Surface** tool from **Surface > Modify > Edit Surface** drop-down; you are prompted to select a surface.

2. Press ENTER; the **Select a Surface** dialog box is displayed.

3. In the dialog box, select the **EG** surface and choose the **OK** button; the **Select Surface to Paste** dialog box is displayed.

4. Select **E Street** from the dialog box and choose the **OK** button; the dialog box is closed and the **E Street** surface is pasted on the **EG** surface. Civil 3D has now recalculated triangulation and has built the **EG** surface taking breaklines into account. Notice the change of the **EG** surface in the region of added breaklines, refer to Figure 3-70.

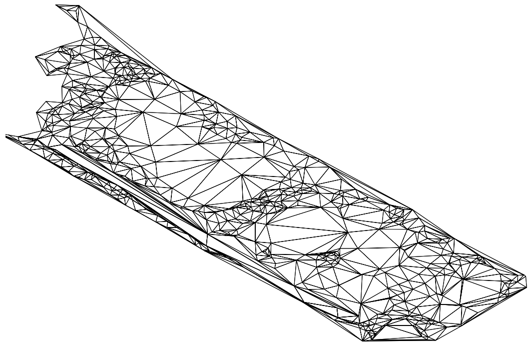


***Figure 3-70*** *The **EG** surface after pasting the **E Street** surface*

5. Choose the **View** tab and then select the **Realistic** visual style from the **Visual Style** drop-down list in the **Views** panel; the **E Street** surface is displayed, as shown in Figure 3-71.

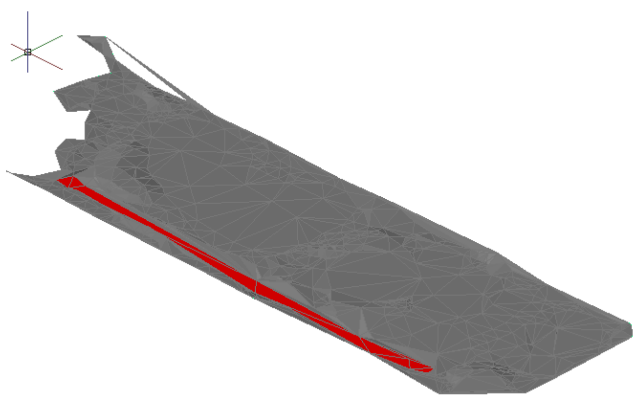


***Figure 3-71*** *The realistic view of the surface after pasting the **E Street** surface over the **EG** surface*

**Note**

*1. The **E Street** surface will disappear as soon as you choose another command or zoom in or out the surface.*

*2. Sometimes you may not be able to paste one surface over another. This happens due to the difference in the elevation of the two surfaces.*

## Saving the File

1. Choose **Save As** from the **Application Menu**; the **Save Drawing As** dialog box is displayed.

2. In this dialog box, browse to the following location:
   *C:\c3d_2016\c03_c3d_2016*

3. In the **File name** edit box, enter **c03_c3d_2016_tut02a** .

4. Next, choose the **Save** button; the file is saved with the name *c03_c3d_2016_tut02a* at the specified location.

## Tutorial 3 Creating Surface From GIS Data

In this tutorial, you will create a surface using a shape file, import a raster image, and drape it over the surface, refer to Figure 3-72. **(Expected time: 45 min)**

The following steps are required to complete this tutorial:

a. Open a new drawing file.
b. Set the parameters of the drawing.
c. Create a surface from GIS data.
d. Insert the georeferenced image.
e. Drape the raster image.
f. View the result of draping.
g. Save the file.

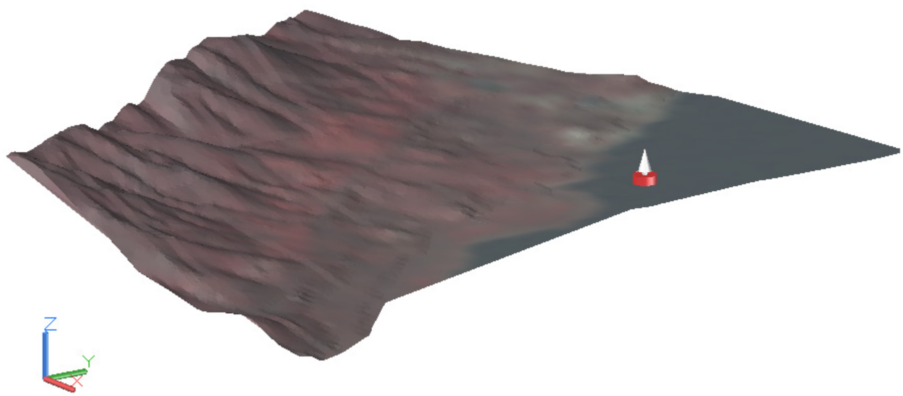


***Figure 3-72*** *The isometric view of the surface with the draped raster image*

### Opening a New Drawing File

1. Choose **New** from the **Application Menu**; the **Select Template** dialog box is displayed**.**

2. In this dialog box, select the **_AutoCAD Civil 3D (Imperial) NCS** template; a new file is opened.

### Setting the Drawing Parameters

In this section, you will specify the drawing settings such as the coordinate system, drawing units, and the ambient settings for the drawing.

1. Right-click on the drawing name in the **Settings** tab of the **TOOLSPACE** palette; a flyout is displayed. Choose the **Edit Drawing Settings** option from the displayed flyout; the **Drawing Settings - <Drawing Name>** dialog box is displayed.

2. Choose the **Units and Zone** tab in the dialog box. Next, select the **Meters** option from the **Drawing Units** drop-down list and the **UTM, NAD 83 Datum** option from the **Categories** drop-down list.

3. In the **Available coordinate systems** drop-down list, select the option **UTM with NAD83 datum, Zone 11, Meter; Central Meridian 117d W** and then choose the **Apply** button from the **Drawing Settings** dialog box. The drawing is set to UTM Zone 11 projection and the datum is set to NAD 83, refer to Figure 3-73.

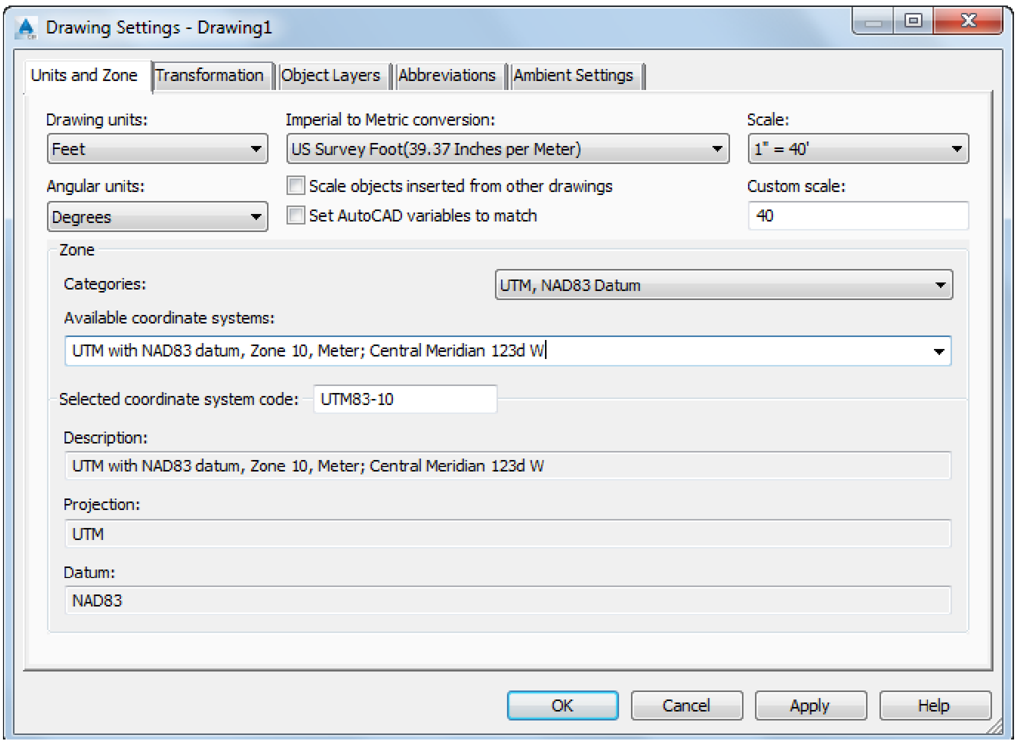



***Figure 3-73*** *The settings in the* ***Units and Zone*** *tab of the* ***Drawing Settings - Drawing 1*** *dialog box*

4. Next, choose the **Ambient Settings** tab. Expand the **Coordinate** node and click on the cell in the **Value** column corresponding to the **Unit** property; a drop-down list is displayed. Select the **meter** option from the drop-down list, as shown in Figure 3-74.

5. Expand the **Elevation** node and click on the cell in the **Value** column corresponding to the **Unit** property; a drop-down list is displayed. Select the **meter** option from the drop-down list.

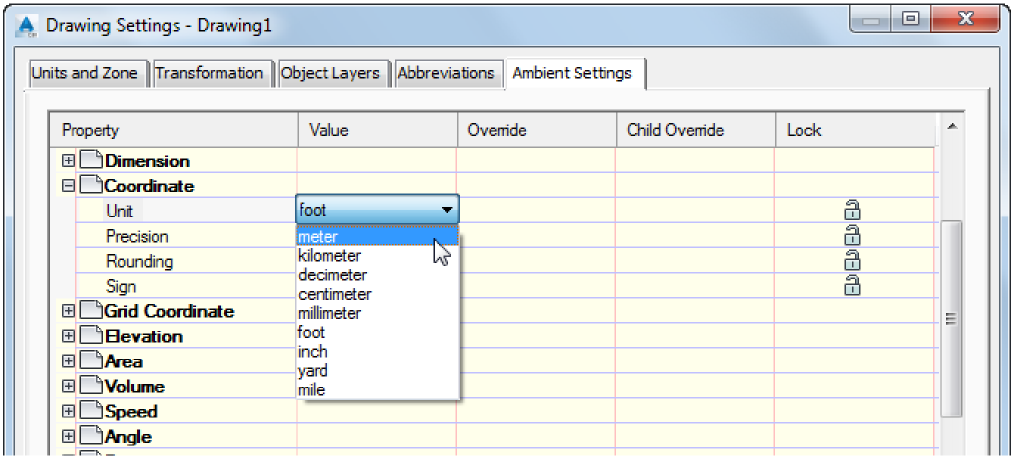



*Figure 3-74 Selecting the **meter** option for the **Unit** cell from the drop-down list*

6. Expand the **Lat Long** node and click on the cell in the **Value** column corresponding to the **Format** property; a drop-down list is displayed. Select the **decimal** option from the drop-down list.

7. Choose the **Apply** button to set the parameters and then choose the **OK** button; the **Drawing Settings** dialog box is closed.

## Creating a Surface from GIS Data

1. Choose the **Create Surface from GIS Data** tool from **Home > Create Ground Data > Surfaces** drop-down; the **Object Options** page of the **Create Surface from GIS Data** wizard is displayed, as shown in Figure 3-75.

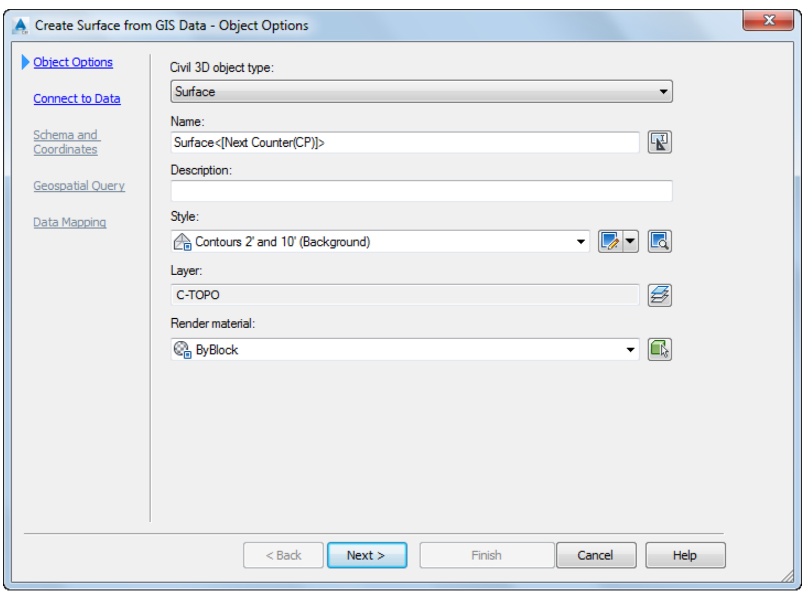



*Figure 3-75 The **Object Options** page of the **Create Surface from GIS Data** wizard*

2. In the **Name** text box, delete the existing text and type **LakeHenshawRegion** as the surface name. Choose the **Next** button; the **Connect to Data** page of the wizard is displayed.

3. In the **Data source type** area of this dialog box, select the **SHP** radio button.

4. In the **Connection parameter** area, choose the browse button corresponding to the **SHP path** edit box; the **Select a SHP file** dialog box is displayed.

5. Browse to the location *C:\c3d_2016\c03_c3d_2016* and select the *LakeHenshaw-ElevationPoints* file. Choose the **Open** button; the **Select a SHP File** dialog box is closed and the path of the selected file is displayed in the **SHP path** text box.

6. Choose the **Login** button in the **Connect to Data** page; the **Schema and Coordinates** page of the wizard is displayed.In the **Schema and Coordinates** page, notice that the **LakeHenshaw-ElevationPoints** file is displayed under the **Feature class** column and its CRS is displayed under the **Coordinate System** column.

7. Now, select the check box for the **LakeHenshaw-ElevationPoints** file and choose the **Next** button; the **Geospatial Query** page of the wizard is displayed.

8. Accept the default settings in the **Geospatial Query** page and choose the **Next** button; the **Data Mapping** page is displayed.

9. In the **Map GIS data to Civil3D properties** list box of the **Data Mapping** page, click in the **Civil3D Property** cell corresponding to the **Contour** property; a drop-down list is displayed. Select **Elevation** from this drop-down list and choose **Finish** to exit the wizard.

10. Enter **ZE** at the Command prompt and press ENTER to zoom to the extent of the drawing. The surface created from GIS data is displayed, as shown in Figure 3-76.

***Figure 3-76*** *Surface created from GIS data*

## Inserting the Georeferenced Image

For the purpose of overlaying the raster on the surface, you shall use the **MAPIINSERT** command.

1. Enter **MAPIINSERT** command in the command line and press ENTER; the **Insert Image** dialog box is displayed.

2. In this dialog box, browse to the location *C:\c3d_2016\c03_c3d_2016_tut* and select the *LakeHenshaw.tif* file.

3. Next, choose the **Open** button; the **Insert Image** dialog box is closed and the **Image Correlation** dialog box is displayed.

4. In the **Source** tab of the **Image Correlation** dialog box, select the **World File** option from the **Correlation Source** drop-down list, if it is not selected. Make sure the values of the parameters in the **Source** tab of the dialog box are set as shown in Figure 3-77.

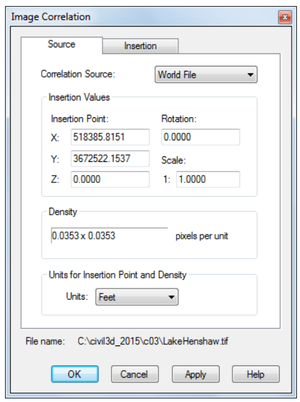



***Figure 3-77*** *The* ***Image Correlation*** *dialog box*

6. To make the surface visible in the drawing, select the image and right-click; a shortcut menu is displayed. From the shortcut menu, choose the **Display Order** option; a flyout is displayed, as shown in Figure 3-78.

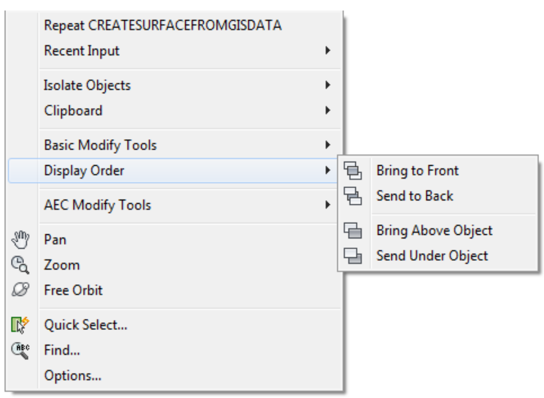



***Figure 3-78*** *The flyout showing the* ***Display Order*** *options*

7. Choose the **Send to Back** option from this flyout; the surface is now displayed over the image, as shown in Figure 3-79.

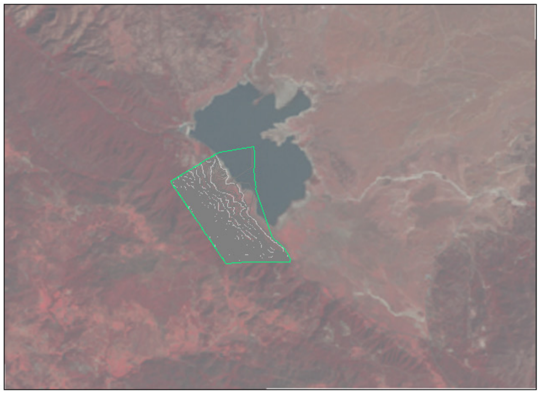


***Figure 3-79*** *Surface displayed over the inserted raster image*

## Draping the Image

1. Choose the **Surface** tool from the **Ground Data** panel of the **Modify** tab; the **Surface** tab is displayed in the ribbon.

2. In this tab, choose the **Drape Image** tool from the **Surface Tools** panel; the **Drape Image** dialog box is displayed, as shown in Figure 3-80. Accept the default values in the dialog box and choose the **OK** button; the dialog box is closed and the image is draped over the surface.

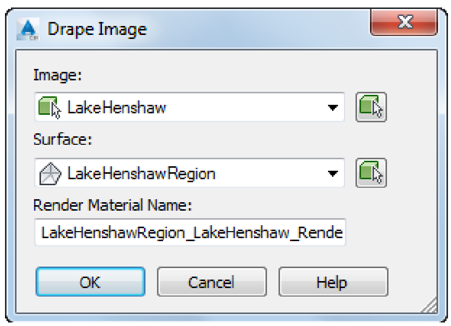



***Figure 3-80*** *The* ***Drape Image*** *dialog box*

## Viewing the Image

1. Select the raster image and right-click; a shortcut menu is displayed.

2. Choose **Isolate Objects > Hide Selected Objects** from the shortcut menu.

3. Select the **Realistic** visual style from the **Visual Style** drop-down list in the **View** tab in the Ribbon.

4. Choose the **Orbit** tool from the **Navigation 2D** panel of the **View** tab and orient the drawing to a suitable viewing angle. Exit the **Orbit** tool by pressing the ESC key. The drawing area now shows the surface with draped raster image, as shown in Figure 3-81.

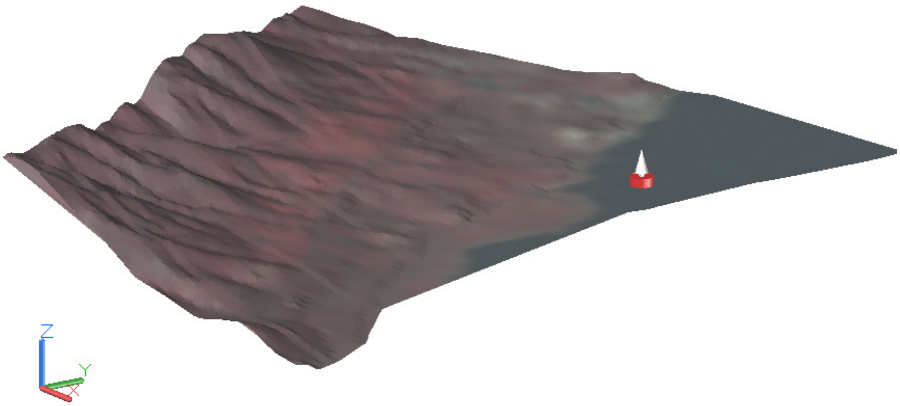


***Figure 3-81*** *The surface with the draped raster image*

## Saving the File

1. Choose **Save As** from the **Application Menu**; the **Save Drawing As** dialog box is displayed.

2. In this dialog box, browse to the following folder:

   *C:\c3d_2016\c03_c3d_2016_tut*

3. In the **File name** edit box, enter **c03_tut03a**.

4. Next, choose the **Save** button; the file is saved with the given name at the specified location.

## Self-Evaluation Test

**Answer the following questions and then compare them to those given at the end of this chapter:**

1. Which of the following tools is used to create a surface from a digital elevation model?

   a) **Create Surface** b) **Create Surface from DEM**
   c) **Create Surface from TIN** d) **Create Surface from GIS Data**

2. Which of the following properties in the **Add DEM Files** dialog box is used to change the coordinate system of the DEM file?

   a) **CS Code** b) **Datum**
   c) **Projection** d) None of these

3. Which of the following features can be represented by breaklines?

    a) Streams　　b) Retaining walls
    c) Ditches　　d) All of the above

4. Which of the following type of analysis places a slope directional arrow at the centroid of each triangle?

    a) **Slopes**　　b) **Slope Arrows**
    c) **Directions**　　d) **Elevations**

5. Which of the following option is used to control the visibility of the surface triangulation within or outside a defined region?

    a) **Boundaries**　　b) Drawing Area
    c) **Breaklines**　　d) **Surface Styles**

6. The DEM is used to represent the ____________ of a continuously varying surface.

7. A surface boundary is a _________ entity and is also used to control the visibility of the surface triangulation.

8. In a Civil 3D drawing file, the _________ tab of the **TOOLSPACE** palette lists the available surface styles.

9. The _________ node in the **Prospector** tab of the **TOOLSPACE** palette lists the surfaces in the current drawing file.

10. A surface can be defined as a network of grids or TIN lines. (T/F)

11. You can use only point data to create a Civil 3D surface. (T/F)

12. The surface is not visible unless the data is added to it. (T/F)

13. You can create a volume surface by comparing two surfaces. (T/F)

14. You can add breaklines only to a TIN surface. (T/F)

15. The **Extract Objects** tool is used only to extract the triangles from the TIN surface. (T/F)

## Review Questions

**Answer the following questions:**

1. The _________ tool is used to identify the flat areas and gaps in the surface after adding the contour data.

2. The _________ tab of the **Surface Properties** dialog box displays the overall view of the surface.

3. You need to use the __________ tool to create a surface from an SHP file.

4. You can create the ___________ and __________ volume surfaces using the **Create Surface** tool.
5. The surface name is displayed in both the **Prospector** and **Settings** tabs of the **TOOLSPACE** palette. (T/F)

6. The volume surface is also known as differential surface. (T/F)

7. You can change the location of a surface point. (T/F)

8. You can change the display of the surface at any stage of the project by modifying the surface styles and surface properties. (T/F)

9. The objects which are extracted from the surface do not retain their original properties. (T/F)

10. The **Definition** subnode provides you with the surface edit options. (T/F)

## Exercises

### Exercise 1 DEM Surface

Download the *c03_c3d_2016_ex01.dem* file from the CADCIM website. The path of the file is *Textbooks > Civil/GIS > AutoCAD Civil 3D > Exploring AutoCAD Civil 3D 2016*. Next, create a surface from DEM. Also, create a new surface style in the **Contours** tab of the **Surface Style - New Surface Style** dialog box using the following parameters:

**(Expected time: 30 min)**

1. Contour Display Mode: **Exaggerate Elevations**
2. Scale Factor: **3**
3. Minor Interval: **10**
4. Contour Smoothening: **True**
5. Smoothening type: **Spline curvee**

**Hint:**

Settings in the **Display** tab

1. In **Plan** view direction, turn on the visibility of **Borders**, **Minor Contours**, and **Major Contours**.

2. Set the color of **Border** to **blue**, **Minor Contour** to **red**, and **Major Contour** to **green**.

3. In the **Model View** direction, turn off the visibility of **Border** and turn on the visibility of **Major Contour** and **Minor Contour** and set the color of **Minor Contour** to **magenta** and **Major Contour** to **cyan**.

Assign the newly created surface style to the surface and then choose the **SE Isometric** option from the list box in the **Views** panel of the **View** tab to view the surface. The SE Isometric view of the DEM Surface after assigning the new surface style is displayed, as shown in Figure 3-80.

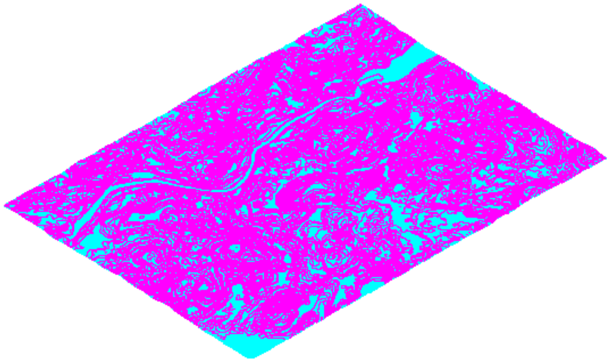


***Figure 3-80*** *The SE Isometric view of the DEM Surface after assigning the new surface style*

## Exercise 2 Drape Image

Download the *c03_c3d_2016_ex02.dem* file from the CADCIM website. The path of the file is *Textbooks > Civil/GIS > AutoCAD Civil 3D > Exploring AutoCAD Civil 3D 2016*. Next, create a surface from DEM, and insert and drape *c03_c3d_2016_ex02_image* over the surface using the following parameters: **(Expected time: 45 min)**

1. Coordinate System: **UTM with NAD27 datum, Zone 10, Meter**
2. Drawing Units: **Meter**

**Hint:**

Set the units for insertion point and density in feet during the image insertion.

## Answers to Self-Evaluation Test

**1.** b, **2.** a, **3.** d, **4.** b, **5.** a, **6.** Surface elevation, **7.** Polygon **8. Settings**, **9. Surfaces**, **10.** T, **11** F, **12.** T, **13.** T, **14.** T, **15.** F