Chapter 4

Surface Volumes and Analysis

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the concept of surface volumes*
- *Compute surface volumes*
- *Understand the concept of surface analysis*
- *Perform surface analysis*
- *Create and add legend tables*

SURFACE VOLUMES

Earthwork estimation is an important part of any Civil Engineering project. Civil 3D provides you with a streamlined method to calculate total cut, fill, and net volumes of earthwork between two surfaces. The earthwork volumes between two surfaces can be calculated either by creating a volume surface (third surface) or by directly calculating the total cut and fill volumes between them.

Civil 3D makes earthwork calculations simple and fast. For earthwork calculations, you need two surfaces. Civil 3D analyzes both surfaces and calculates the total volume of cut and fill by comparing the vertical distance between points in these surfaces. The distance between the elevations of two surfaces can then be used to create a third surface, known as volume surface. The various methods to calculate the surface volumes are discussed next.

Volumes Dashboard

Ribbon: Analyze > Volumes and Materials > Volumes Dashboard
Command: VOLUMESDASHBOARD



Using the Volumes dashboard, you can break the surfaces in phases or parcels for volume analysis. The tools in the dashboard can be used to calculate and analyze the volume in multiple volume surfaces and bounded areas, which fall within the boundary of the volume surfaces. In the dashboard, you can save and use the information of the analysis across multiple drawing sessions. The result of volume analysis can be exported as a summary report or can be inserted into the drawing using the dashboard.

To calculate surface volumes, choose the **Volumes Dashboard** tool from the **Volumes and Materials** panel of the **Analyze** tab; the **PANORAMA** window with the **Volumes Dashboard** tab will be displayed, as shown in Figure 4-1.

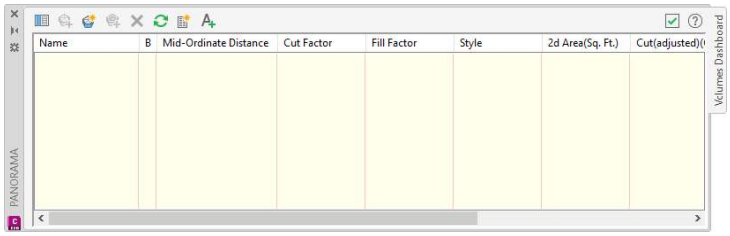

Figure 4-1 *The* ***PANORAMA*** *window with the* ***Volumes Dashboard*** *tab*

Components of Volumes Dashboard

The various tools displayed in the **Volumes Dashboard** tab of the **PANORAMA** window that are used to calculate volume surfaces and generate report are discussed next.

Toggle Net Graph Panel



The **Toggle Net Graph Panel** button is used to toggle the display of the auxiliary panel. This panel displays the total volume (cut and fill), net volume, and cut and fill

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

graph for the selected volume surfaces or bounded areas. To display the surface statistics in the panel, select one or more surfaces using the corresponding check box.

Add Volume Surface



You can choose the **Add Volume Surface** button to add an existing volume surface to the Volumes dashboard.



Note

*The **Add Volume Surface** button remains inactive if no volume surface exists in the current drawing.*

Create new volume surface



You can choose the **Create new volume surface** button to create and add a new volume surface to the Volumes dashboard. On choosing this button, the **Create Surface** dialog box will be displayed. Select the appropriate volume surface type from the options in the **Type** drop-down list. Specify base surface and comparison surface in the **Volume Surface** node. To specify the base surface, click in the corresponding **Value** cell; a browse button appears in the cell. Choose the browse button; the **Select Base Surface** dialog box is displayed. In the dialog box, select the required surface and choose the **OK** button. The dialog box closes and the name of the selected surface now appears in the **Value** cell of the base surface in the **Create Surface** dialog box.



Note

*You are required to specify the values for the **Grid Parameters** when you choose the **Grid volume surface** option from the **Type** drop-down list.*

Add Bounded Volume



You can choose the **Add Bounded Volume** button to limit the area for volume calculation of a surface to a specified region. You can use AutoCAD entities such as Polyline, Polyline (3D), Polyline (2D), Parcel Segment, Feature Line, Circle, Ellipse, Survey Figure, and Parcel to define the boundary of the area.



Note

*You must create and select a volume surface in the **Volumes Dashboard** tab to activate the **Add Bounded Volume** button.*

To add a bounded volume to the surface, choose the **Add Bounded Volume** button; you will be prompted to select the bounding object. Select the bounding object in the drawing area; the bounded volume is added to the surface and the volume calculations are displayed in the surface node.

Remove Selected Entry



You can choose the **Remove Selected Entry** button to delete the selected row from the **Volumes Dashboard** tab of the **PANORAMA** window.

Re-compute volumes



You can choose the **Re-compute volumes** button to recalculate the volume. This tool is useful where an updated surface requires recalculations of volume.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Generate Cut/Fill Report

You can choose this button to generate the cut or fill reports from the selected volume surfaces and bounded areas.

Note

*The **Cut/Fill Report** is generated and displayed in a new browser. You can save the report as a file to include in your project report.*

Insert Cut/Fill Summary

You can choose this button to insert the result of surface volume calculations into the drawing in tabular form. To insert the table, choose the **Insert Cut/Fill Summary** button; you will be prompted to specify the point of insertion. Click in the drawing area to specify insertion point; the **Cut/Fill Summary** will be added at the specified location.

Composite Volumes

In the composite volume method, the cut, fill, and net volumes are calculated by using a pair of surfaces consisting of a top (Comparison) and a bottom (Base) surface. The Base surface is the existing ground surface and the Comparison surface will be the proposed surface. Civil 3D analyzes and compares the two surfaces to calculate the difference of elevation between each vertex point of the TIN or a cell in the Grid surfaces or at the intersection of the triangle edges of two surfaces. This elevation difference is then used to create a TIN volume or a TIN grid surface.

To use the composite volume method, type **reportsurfacevolume** in the command line; the **PANORAMA** window with the **Composite Volumes** tab will be displayed, as shown in Figure 4-2. The components of the tab and the method used for computing composite volume are discussed next.

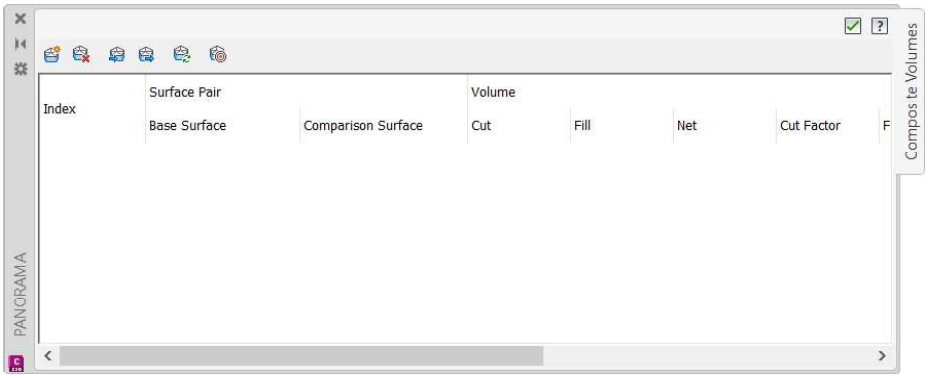

***Figure 4-2** The **PANORAMA** window with the **Composite Volumes** tab*

Components of Composite Volumes Tab

The components in this tab help you to calculate the volume based on the existing and proposed surface. The components are used to calculate the net volume by comparing the two surfaces. Volumes are also calculated for the purpose of cut and fill. The options available in this tab are discussed next.

Create new volume entry



You can choose the **Create new volume entry** button to create a new composite volume entry.

Delete volume entry



You can choose the **Delete volume entry** button to delete the existing composite volume entry from the **PANORAMA** window.

Import volume entries from file



You can choose the **Import volume entries from file** button to import the existing composite volume entries from a file. The file should be in XML format. On choosing this button, the **Import Volume Entries** dialog box will be displayed. Select the required XML file of the volume entry from this dialog box and choose the **Open** button; the volume entries will be imported and added to the **PANORAMA** window.

Export volume entries to file



You can choose the **Export volume entries to file** button to export volume entries. On choosing this button, the **Export Volume Entries** dialog box will be displayed. Specify a name for the file in the **File name** edit box and choose the **Save** button; the file will be saved in the XML format at the specified location.

Recompute volumes



You can choose the **Recompute volumes** button to recalculate volumes. This button is used when you have updated or edited any surface of the surface pair. Civil 3D recomputes the volumes based on the changes made in the surface.

Create new volume entry from surfaces



You can choose the **Create new volume entries from surfaces** button to create a volume entry by selecting surfaces from the drawing. Civil 3D prompts to select a base and a comparison surface when the **Create new volume entries from surfaces** button is chosen. Select the surfaces from the drawing; the cut and fill volumes will be displayed in the **PANORAMA** window.

Computing Composite Volumes

To calculate composite volumes, first you need to make a composite volume entry in the **PANORAMA** window. To add a new volume entry, choose the **Create new volume entry** button; the new volume entry will be displayed in the list box. Next, in the **Surface Pair** column, click on **<select surface>** in the **Base Surface** sub-column and select the required base surface from the drop-down list. Similarly, select a comparison surface from the **Comparison Surface** column and press ENTER or click on the free space in the window. The total cut and fill volumes will be calculated and then displayed in the **Volume** column of the following sub-columns:

Cut

The **Cut** sub-column displays the quantity of material to be removed from the Base surface to make it equal to the Comparison surface.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Fill

The **Fill** sub-column displays the quantity of earth material to be added to the Base surface, thus making it equal to the Comparison surface.

Net

The **Net** sub-column displays the net difference between the cut and fill volume of the two surfaces.

Net Graph

The **Net Graph** column shows the percentage representation of the net volume, net cut volume, and net fill volume. The red bar in this graph indicates the net cut volume, which signifies the extra material that needs to be removed from the existing site. The green bar indicates that the material needs to be added to the existing site.

The **Cut Factor** and **Fill Factor** columns display the cut and fill factor values for the specified surface. These values are used to calculate the net volume of material required to cut or fill the base surface so as to make it conform to the planned (Comparison) surface. The net material required to cut or fill the base surface on applying the cut or fill factor is displayed in the **Cut (adjusted)** and **Fill (adjusted)** columns, respectively.

Bounded Volumes

This method helps you to calculate the volume of a surface or specified part of a surface. The area, whose volume has to be calculated, can be defined by a bounding polygon or a parcel. The bounded volume can be calculated using a volume surface or any other surface. In case of volume surfaces, the volume is calculated using the difference in elevation between two surfaces. For terrain surfaces, the volume is calculated from zero elevation to the elevation in the area bounded by a polygon or parcel.

To calculate the bounded volume between two specified areas, enter **reportsurfboundedvolume** in the command line; you will be prompted to select the bounding polygon. Select the polygon that bounds the required area; the cut, fill, and net volumes will be displayed in the command line. Alternatively, press the F2 key to display the bounded volume results in the **AutoCAD Text Window**.

CREATING VOLUME SURFACES

The volume surfaces help you review earthwork calculations of a site. The volume surface is a surface that is created by using the elevation difference between two surfaces. You can then analyze the surface and review the total amount of the cut or fill volume so that a balance between the total cut and fill volumes can be produced on the site. Civil 3D allows you to create two types of volume surfaces. The methods to create these volume surfaces are discussed next.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Creating the TIN Volume Surface

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



To create a TIN volume surface, you need a pair of TIN surfaces. In Civil 3D, you can create a TIN volume surface by choosing the **Create Surface** tool from the **Create Ground Data** panel, as discussed in the previous chapter. On doing so, the **Create Surface** dialog box will be displayed. Next, select the **TIN volume surface** option from the **Type** drop-down list in this dialog box. Specify the name, style, and description of the surface in the respective **Value** fields. Next, click in the **Value** field of the **Base Surface** property; a browse button will be displayed. Choose this button to display the **Select Base Surface** dialog box. In this dialog box, select the required option from the list box and then choose the **OK** button; the **Select Base Surface** dialog box will be closed. Similarly, select the Comparison surface from the **Select Comparison Surface** dialog box. Specify the cut and fill factors for the material. Now, choose the **OK** button from the **Create Surface** dialog box; the dialog box will be closed and a TIN volume surface will be created and added to the **Surfaces** node. You can edit and analyze this surface the same way as you analyze to any other surface.



Note

*Due to increase in volume, the **Cut Factor** for the excavated material will be greater than 1 while the **Fill Factor** will be less than 1 as the volume of the filling material decreases due to compaction.*

Creating the Grid Volume Surface

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



You can create a grid volume surface by using a pair of grid surfaces. To create a Grid volume surface, first you need to create a grid surface, as discussed in previous chapter. Next, choose the **Create Surface** tool from the **Create Ground Data** panel; the **Create Surface** dialog box will be displayed. Select **Grid volume surface** from the **Type** drop-down list; the properties of the grid volume surface will be displayed in the **Properties** column of the dialog box. Specify the information such as name, description, and style for the grid volume surface in the respective **Value** fields. Similarly, you can specify the grid parameters such as grid spacing in both X and Y directions and also in the grid's orientation. Next, select the Base surface as well as the Comparison surface as explained in the previous section. Enter the cut and fill factors and then choose the **OK** button. The **Create Surface** dialog box will be closed and the grid volume surface will be added in the **Surfaces** node.



SURFACE ANALYSIS

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



After creating a surface, you need to analyze it for collecting information from it. This information can be used in the development of the project design. Surface analyses helps you understand and visualize the surface topography in a better way. For example, if you are

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

generating the slope analyses of a surface, it would help you to find slopes of various points in the surface. Civil 3D supports seven ways to analyze a surface. These are Contours, Directions, Elevations, Slopes, Slope Arrows, Watersheds, and User-defined contours.

To carry out a surface analysis, choose the **Surface Properties** tool from the **Modify** panel; the **Surface Properties - <surface name>** dialog box will be displayed. In this dialog box, choose the **Analysis** tab; the options in this tab will be displayed, as shown in Figure 4-3.

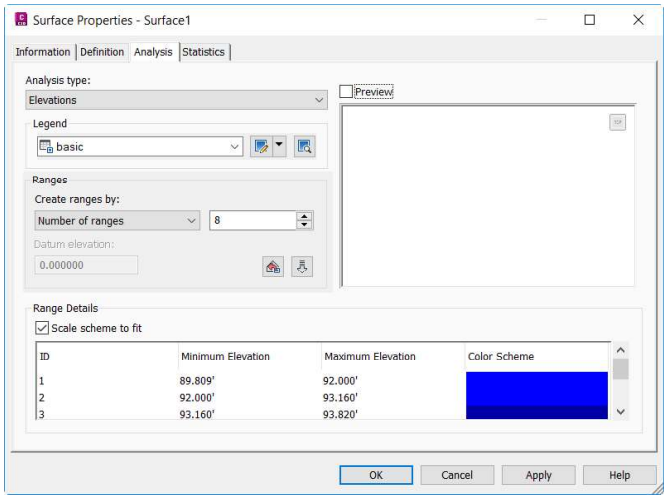

Figure 4-3 *The options displayed in the* ***Analysis*** *tab of the* ***Surface Properties*** *dialog box*

Note

If there are multiple surfaces in the drawing, you will be prompted to select a surface. To select a surface from the list, press ENTER; the ***Select a Surface*** *dialog box will be displayed. Select the required surface using this dialog box.*

Alternatively, choose the **Prospector** tab in the **TOOLSPACE** palette and expand the **Surfaces** node. Right-click on the required surface in the **Surfaces** node; 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.

In the **Analysis** tab, select the required option from the **Analysis type** drop-down list. The options in this drop-down list are used to analyze the elevation, contour, slope, directions, Slope Arrows, and Watershed. After selecting the analysis type, select the legend style from the drop-down list in the **Legend** area. Legend is a tabular representation of the Surface analysis data. You can create legend style depending upon the information that you want to display in the legend table. You can select the **Preview** check box to preview the created legend style. Note that the preview of the legend is displayed only after running the analysis. Next, in the **Ranges** area, specify the number of ranges in the **Number** spinner; the surface data will be divided into the specified number of ranges and analyzed accordingly. After specifying the number of ranges, choose the **Run analysis** button; the result of the analysis will be displayed in tabular

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

form in the **Range Details** area. This result table may have different columns depending on the analysis type selected. In the **Range Details** area, the **Scale scheme to fit** check box is selected by default. This check box scales the number of entries in the **Scheme** column of the **Range Details** area to cover the ranges uniformly. This implies that a uniform sampling is taken from the scheme between the first and last entries in the **Scheme** column. After performing the analysis, you can preview the information of the legend in the **Preview** area. Choose the **OK** button to close the dialog box.

To visualize the analysis results in the drawing, select the surface that you have analyzed and right-click to display a shortcut menu and then choose the **Edit Surface Style** option from it; the **Surface Style - <style name>** dialog box will be displayed. By default, the **Display** tab is chosen in the dialog box. In this tab, turn on the layers of those components whose results you want to view from the analysis. For example, if you run the Elevation analysis of the surface, you need to turn on the visibility of the **Elevations** component in the **Surface Style** dialog box. The methods for performing various types of analyses by using the options in the **Analysis type** drop-down list are discussed next.

Contour Analysis

The **Contours** option in the **Analysis type** drop-down is used to analyze and display surface contours by using different colors and linetypes. To analyze contours, select the **Contours** option from the **Analysis type** drop-down list of the **Surface Properties - <surface name>** dialog box. The **Legend** area displays the default legend style for the selected analysis type. Civil 3D has an in-built legend style for each of its analysis types. However, you can create your own legend style and display the required information in the legend. This is discussed later in this chapter. In the **Analysis** tab, you can also preview the legend style. To preview the legend tables with analysis information, select the **Preview** check box. You can also add legend tables to the drawing, which is also discussed later in this chapter.

In the Contour analysis, you can divide and group surface contours in different number of ranges based on their elevations. To do so, specify the desired value in the **Number** spinner in the **Ranges** area of the **Analysis** tab. The contours will be grouped depending on the value specified in the **Number** spinner.

In the **Range Details** area of the **Analysis** tab, each contour range is assigned a maximum and a minimum elevation. The major and minor ranges of this contour range are assigned a specific color, linetype, and lineweight. The value of the results can be modified in this area.

On specifying the legend properties and the number of ranges, perform the Contour analysis by choosing the **Run analysis** button. Next, choose the **Apply** button and then the **OK** button to close the **Surface Properties - <surface name>** dialog box. To view contours in the surface, you need to turn on the visibility of the major and minor contours in the **Display** tab of the **Surface Styles** dialog box.



Note

*The **Range** area of the **Analysis** tab is used to display the default value of the number of ranges for each analysis type. This default value is derived from the range settings in the **Surface Style** dialog box.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Direction Analysis

The **Directions** option in the **Analysis type** drop-down list of the **Analysis** tab is used to classify surface triangles on the basis of the direction they face. Direction analysis generates aspect map that graphically displays the surface slopes with reference to the cardinal direction. For analyzing the surface, select the **Directions** option from the **Analysis type** drop-down list of the **Surface Properties - <surface name>** dialog box. Specify the options in the **Legend** area, as discussed earlier. In the **Ranges** area, specify the value in the **Number** spinner and choose the **Run analysis** button; the result of this analysis will be displayed in terms of maximum and minimum directions of each range with the assigned color scheme in the **Range Details** area of the **Analysis** tab.

You can modify the maximum and minimum direction values and the color scheme for a particular range by double-clicking on default values in the respective columns of the **Range Details** area. After modifying these values, choose **Apply** and then the **OK** button to close the dialog box. To view the results of the Direction analysis, turn on the visibility of the **Directions** component in the **Surface Style** dialog box.

Elevations Analysis

The Elevations analysis is the most important type of Surface analysis. It is used to group the surface triangles based on their elevation. It helps the site planners review and judge the total amount of earthwork required to create a leveled finished surface. To run this type of analysis, select the **Elevations** option from the **Analysis type** drop-down list in the **Analysis** tab of the **Surface Properties - <surface name>** dialog box. Specify the options in the **Legend** area as discussed earlier. In the **Ranges** area, select the required option from the **Create ranges by** drop-down list. You will need to set the value in the spinner, if you select the **Number of ranges** option from the drop-down list. Next, run the analysis; the details such as maximum elevation, minimum elevation, and color scheme for each range of elevation will be displayed in the **Range Details** area of the **Analysis** tab. Also, elevation bands will be displayed in different shades of color specified in the **Color Scheme** property of the **Analysis** tab in the **Surface Style** dialog box. You can also modify the color scheme in the **Surface Style** dialog box. Note that the default color scheme selected is **Blue**.

Alternatively, in the **Analysis** tab of the **Surface Properties - <surface name>** dialog box, you can change the color of each range by double-clicking on the default color in the column and selecting the color from the **Select Color** dialog box. After modifying the color or elevation values of a range, choose the **Apply** button in the **Select Color** dialog box. To view the Elevation analysis results, turn on the visibility of **Elevations** in the **Display** tab of the **Surface Style <name>** dialog box. To invoke this dialog box, select the required surface from the **Surfaces** node in the **Prospector** tab and right-click; a shortcut menu will be displayed. Choose the **Edit Surface Style** option from the shortcut menu; the **Surface Style <name>** dialog box will be displayed. In this dialog box, choose the **Display** tab and turn on the visibility of the **Elevations** layer. Choose **Apply** and then the **OK** button. On doing so, you will see different elevation bands represented by different colors, as shown in Figure 4-4.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

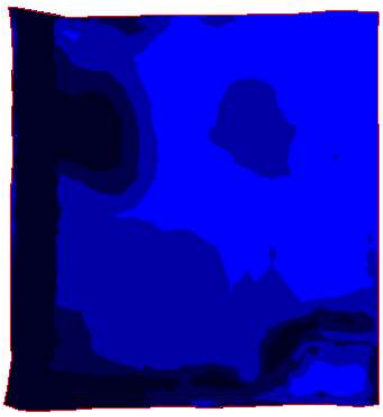

Figure 4-4 *Elevation bands represented by different colors*

Slopes Analysis

The Slopes analysis is another important analysis carried out to analyze varying slopes in the surface. The slope of a surface is defined as a change in its elevation over a certain distance. It can be expressed as a ratio or a percentage (grade). For example, slope of 4:1 means an elevation raise of one unit for a distance of four units. In percentage, it can be expressed as 25% of grade. Similarly, a slope of 3: -1 means an elevation fall of one unit for a distance of three units. In percentage, it can be expressed as a grade of -33%. The **Slopes** option in the **Analysis type** drop-down is used to display the slope of the ground. In Slope analysis, the surface triangles are grouped according to their slope. Each range is assigned a color and the triangles in a particular slope based on range will be displayed using the same color scheme.

To analyze slopes, select the **Slopes** option from the **Analysis type** drop-down list of the **Surface Properties - <surface name>** dialog box. Specify the options in the **Legend** area as discussed earlier. In the **Ranges** area, specify the value in the **Number** spinner as discussed earlier and choose the **Run analysis** button; the result of this analysis will be displayed in terms of the minimum and maximum slope values in the **Range Details** area.

Turn on the visibility of **Slopes** in the **Display** tab of the **Surface Style** dialog box; triangles will be displayed according to the color scheme of the slope range they belong to. Civil 3D calculates the slopes of triangles of the TIN surface at their centroids. The result of the Slope analysis helps the designers and engineers in designing the drainage and sewer systems.

Slope Arrows Analysis

The **Slope Arrows** option in the **Analysis type** drop-down list is used to perform Surface analysis on the basis of the directions of slopes of the triangles. The result of this analysis is displayed using the direction arrows. These arrows show the slope direction and are placed at the centroid of each triangle in a TIN surface. The slope arrows pointing each other represent a valley whereas the arrows pointing away from each other signify a hill. In the Watershed analysis, the slope arrows represent the direction of the water flow across the surface.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

To analyze slope directions, select the **Slope Arrows** option from the **Analysis type** drop-down list of the **Surface Properties-<surface name>** dialog box. Specify the options in the **Legend** area as discussed earlier. In the **Ranges** area, specify the value in the **Number** spinner, as discussed earlier and choose the **Run analysis** button; the result of this analysis will be displayed in terms of the minimum and maximum slope values in the **Range Details** area.

After performing the analysis, turn on the visibility of **slopes arrows** in the **Display** tab of the **Surface Style** dialog box. You will notice that the slope arrows are displayed according to their color range. Figure 4-5 shows the slope arrows after running the Slope Arrows analysis.

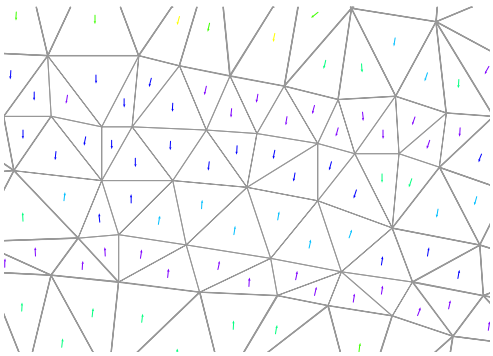

Figure 4-5 *Slope arrows displayed on performing the Slopes Arrows analysis*



Note

The slope analysis and the slope arrows analysis display the same information. But the slope analysis displays results through colored triangles, whereas the slope arrows analysis displays results using the coloured slope arrows.

Drainage Analysis

Autodesk Civil 3D 2026 introduces the Drainage Analysis extension, which brings the tools of **InfoDrainage** directly into the Civil 3D environment. With this integration, you can design, analyze, and review stormwater drainage systems including catchments, pipes, manholes, culverts, and ponds without switching to another application. Working within one unified platform ensures data consistency, minimizes errors, and streamlines the overall design process.

This extension also provides powerful hydraulic and hydrologic analysis features, allowing users to simulate rainfall, calculate runoff, and assess how the drainage network performs under different conditions. Analysis results such as flow rates, velocities, and water levels are displayed directly within Civil 3D for easy access and review. The Drainage Analysis extension enhances design accuracy, efficiency, and sustainability while simplifying stormwater management workflows.

User-defined Contours Analysis

The **User-defined contours** option in the **Analysis type** drop-down is used to carry out the analysis for the user-defined contour lines of the surface. These contour lines are defined according to their elevation ranges. For example, consider that the site on which you are working has contour lines upto 100 feet of elevation and your project requires analyzing contours upto

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

70 feet. In such a case, you can use the User-defined contours analysis. This analysis divides the surface and displays contour lines at specified contour intervals upto the elevation range defined by the user. As a result, in the example discussed above, after performing this analysis, the surface will display contour lines upto a range of 70 feet at specific contour intervals. Moreover, the contour lines are displayed in different colors, linetype, and lineweight.

To analyze User- defined contours, select the **User-defined contours** option from the **Analysis type** drop-down list of the **Surface Properties - <surface name>** dialog box. Specify the options in the **Legend** area as discussed earlier. In the **Ranges** area, specify the value in the **Number** spinner as discussed earlier and choose the **Run analysis** button; the result of this analysis is displayed in terms of elevation in the **Range Details** area. In this analysis, the contour lines are segregated according to a specific elevation assigned with a unique combination of colors, linetype, and lineweight upto a range that can be modified. To view the contours, turn on the visibility of **Use Contours** in the **Display** tab of the **Surface Style** dialog box.
Figures 4-6 and 4-7 show the **Range Details** area of the Contour analysis and the User-defined Contours analysis, respectively. A comparison between these two analysis types shows that the Contour analysis displays the maximum and minimum elevations for a range whereas the User-defined analysis shows contour lines at a specified interval and elevation and within a defined range.

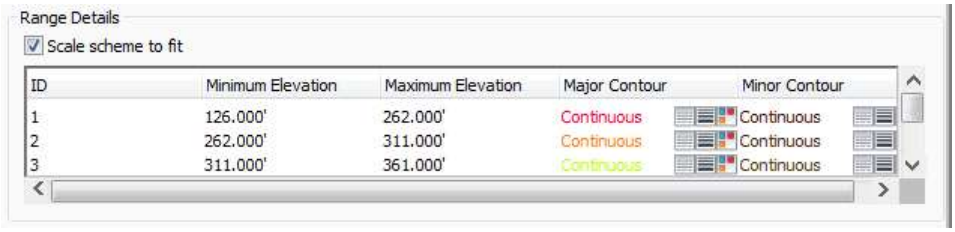

Figure 4-6 *The* ***Range Details*** *area of the Contour analysis*

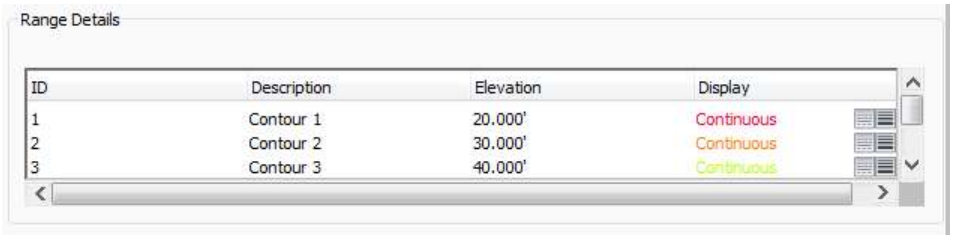

Figure 4-7 *The* ***Range Details*** *area of the user-defined Contour analysis*

Watersheds Analysis

Watershed is an area that is drained by watercourse. It is the area of land that catches rain, snow, and other forms of precipitation, and then drains it to another watercourse such as river, lake, and so on. The TIN lines of the surface are used to calculate the areas through which the water would flow along the surface. These areas help you determine the drain targets. Drain target is an area from where the water flow stops or passes over the surface. The region of the surface from where water drains to these targets is called a watershed.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

In Civil 3D, the Watersheds analysis helps you analyze the flow of water along and off the surface by determining the regions or watersheds. Various types of watersheds and Watershed Analyses are discussed next.

Type of Watersheds

Depending on the type of drain targets, watershed is classified into six types. These types are discussed next.

Boundary Point Watershed

In this type of watershed, the drain target point is the boundary point. The boundary point is the lowest point at the end of the channel.

Boundary Segment Watershed

If an edge on the surface boundary belongs to a triangle that slopes down toward it, the water flow will find its path along this edge and flows off the surface. A boundary segment watershed is a sequence of connection of such edges, which are the drain target of the water flow.

Depression Watershed

A depression is a set of connected points that are at the same elevation and whose neighbors are at higher elevation. This connected set of points is a single drain target where the water flow has no downhill place to go.

In a Depression watershed, the drain target is determined by a set of points at lower elevation than the surrounding points. This type of watershed is classified into two types, namely Ambiguous Depression and Shallow Depression.

The Ambiguous Depression watershed does not include any drain targets but water flowing on it can reach more than one drain target. This type of watershed is created when the depth of the Depression watershed is less than the threshold depth of the watershed and there are multiple neighboring watersheds at the points of minimum elevation of the boundary.

The Shallow Depression watershed is determined by the average depth of Depression watershed when it is filled to overflow. The average depth of depression is determined by dividing the volume of water in the depression by the area of the top surface of water. When this depth is less than the user-specified threshold depth, the Depression watershed is known as Shallow Depression watershed.

Flat Area Watershed

The Flat Area watershed is defined as the flat region from where the water can drain to other drain targets besides the flat area. A flat area is defined as a connected set of triangles in which vertices of all the triangles have the same elevation.

Multi-drain Watershed

The Multi-drain watershed is a split channel and water flowing through it can drain into multiple channels. Civil 3D keeps a record of the watersheds into which the water from the Multi-drain watersheds can drain.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Multi-drain Notch Watershed

In this type of watershed, a notch is formed by the two points of a flat edge and the water can flow to multi drains from the notch.

Running the Watershed Analysis

To run the Watershed analysis, select the **Watersheds** option from the **Analysis type** drop-down list of the **Surface Properties** dialog box. The options displayed in the **Analysis** tab of the **Surface Properties** dialog box are discussed next.

In the **Watershed Parameters** area of this tab, the **Merge depressions into single drain targets when Minimum Average Depth is less than** edit box is used to specify the minimum average depth of a depression in the surface on which a watershed can be created. This will avoid the minor depression depths from being defined as watersheds. Enter a value in this edit box to specify the minimum average depth of the depression in the surface at which a watershed is to be created. All the depressions with a depth less than the specified value will be ignored automatically, and therefore will not be considered as watersheds.

Alternatively, select the **Merge adjacent boundary watersheds** check box to enable all the boundary segments or boundary point watersheds along the outer boundary surface to be merged. Now, choose the **Run analysis** button to run the **Watershed** analysis; the details of the analysis will be displayed in various columns of the **Range Details** area, as shown in Figure 4-8.

In the **Range Details** area, the **ID** column displays the index number of every range. The **Type** column displays the type of watershed in that range. The bulb icon in this column indicates the visibility of the watershed. You can turn the visibility of the watershed on or off at any range. The **Drains Into** column specifies the sub areas into which the watershed drains.

For example, 101,103 indicate that the watershed drains into the sub-areas 101,103. The **Segment Display** column shows the segments of the watershed sub-areas. You can change the display of these segments by changing the lineweight, linetype, and color of the sub-area segments. To do so, choose the required button from the **Segment Display** column. The **Area Display** column specifies the hatch pattern that is used to represent the watershed sub-area. Choose the required button from this column to display the **Hatch Properties** dialog box. You can use this dialog box to specify a hatch pattern. To change the color of the hatch pattern, choose the button on the right of the hatch pattern and select the color from the **Select Color** dialog box that will be displayed after choosing the button.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

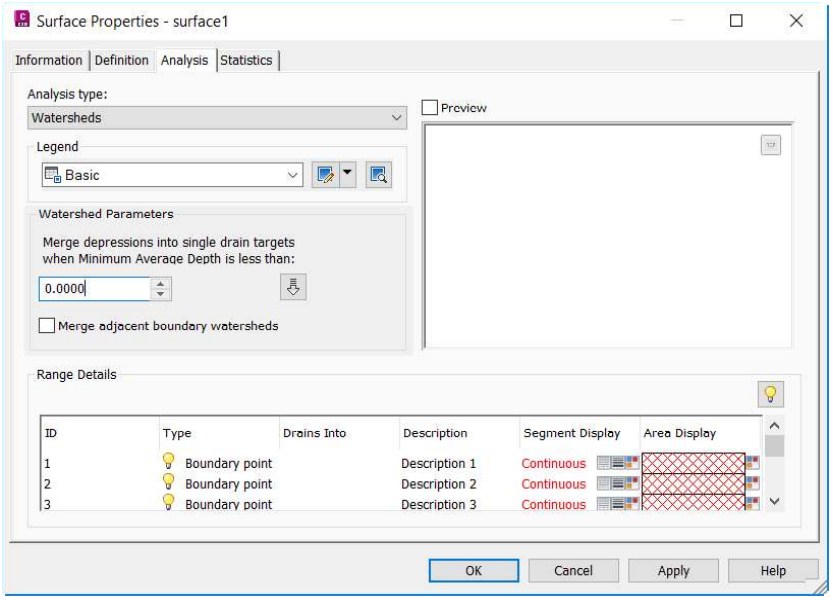

Figure 4-8 *The analysis details shown in the* ***Range Details*** *area for the* ***Watersheds*** *analysis*

Once the analysis is done, choose the **Apply** button. Next, choose the **OK** button to close the dialog box. The **Watersheds** analysis is completed. To view the results of the **Watersheds** analysis, turn on the visibility of **Watersheds** in the **Display** tab of the **Surface Style** dialog box. On doing so, watersheds are displayed in different hatch patterns and colors. Each watershed displays a tag inside it, which shows the range ID, type of watershed, and watershed area. Figure 4-9 shows the watershed with different hatch patterns and Figure 4-10 shows the watershed tags with watershed information.

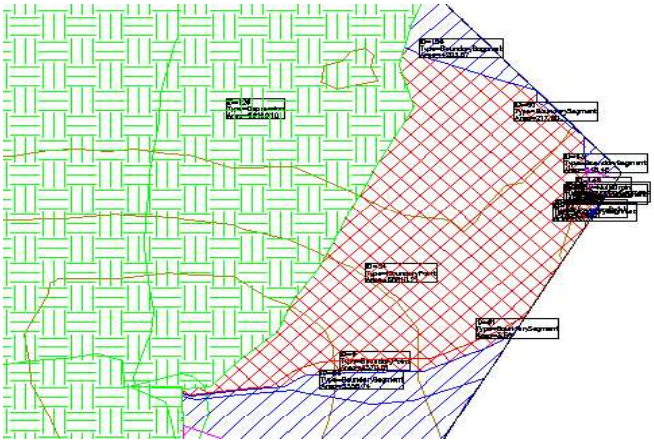

Figure 4-9 *Hatch patterns representing the watersheds*

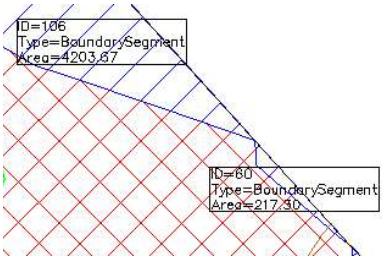

Figure 4-10 *Watershed tag displaying the watershed information*

SURFACE ANALYSIS TABLES

In Civil 3D, data collected after performing a Surface analysis is divided into ranges that have different colors assigned to them. The surface legend table is used to organize and consolidate the data in a tabular form. You can create surface legend tables for all types of surface analyses, such as **Directions**, **Elevations**, **Slopes**, **Slope Arrows**, **Contours**, and **Watersheds**.

You can add a surface legend table to your drawing only after you have performed the Surface analysis, modified the surface style to display the analysis type, and set up a surface legend table style to display the surface data. The method of creating a legend table style and adding a legend table to the drawing is discussed next.

Creating Legend Table Styles

Civil 3D offers a default legend table style for each type of Surface analysis. However, you can create your own legend table style and view the required information in the table.

The table styles are created, edited, and managed in the **Settings** tab of the **TOOLSPACE** like any other Civil 3D object. To create a new legend table style, expand the **Surface > Table Style** node in the **Settings** tab and then expand the subnode for which you want to create a style. For example, expand the **Elevation** node to create the legend style for the Elevation analysis of the surface. Next, you will notice that a default **Elevations** legend style name appears in the **Elevation** node. Now, right-click on **Elevation** and then choose **New** from the shortcut menu; the **Table Style - New Elevation Table Style** dialog box will be displayed, as shown in Figure 4-11. The tabs in this dialog box are discussed next.

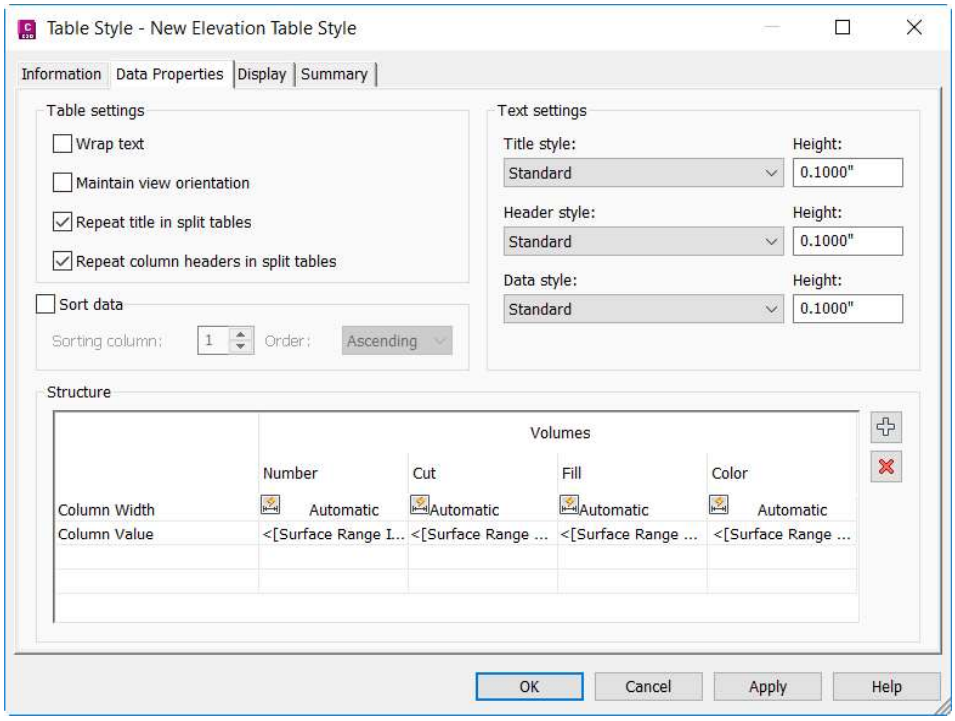

***Figure 4-11** The **Table Style - New Elevation Table Style** dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Information Tab

This tab is used to edit the name of the table style and to give description about the style, if required. Enter a name of the legend style in the **Name** edit box.

Data Properties Tab

This tab is used to define the table settings and the text style of the text in the table. The areas in this tab are discussed next.

Table settings Area

The options in this area are used to set the table format. Select the **Wrap text** check box to wrap text in the **Elevation Table** when the column width is set to **Manual** in the **Structure** area. Select the **Maintain view orientation** check box to align the legend table according to the orientation of the drawing view. The legend will be aligned according to the drawing, if rotated. Select the **Repeat titles in split tables** check box to enable the title of the legend to be repeated, if the legend is split into different sections. Select the **Repeat column headers in split tables** check box to ensure that the column headers of the legend repeat when they are split into different sections.

Text settings Area

The options in the **Text settings** area are used to set the display option of the text in the table. Select the text style from the **Title style** drop-down list for the title and then specify the height of the text title in the **Height** edit box. Similarly specify the text style and height for the column header and data using the respective drop-down list and edit boxes.

Sort data Area

The options in this area are used to sort data based on the specified column of the table. To do so, select the **Sort data** check box in this area; the **Sorting column** spinner and the **Order** drop-down list will be available. Use the spinner to specify the number of columns to sort out the data. Next, select the order from the **Order** drop-down list to sort the data.

Structure Area

The **Structure** area is used to specify the composition of the legend data in columns of the table. It allows you to add or delete columns from the table. The table title and the column headers vary according to the type of analysis being used. To modify the header or title text, double-click on the column header or title to open the respective text component editor window. For example, double-click on the **Number** header for the **Number** column indicating the range IDs, refer to Figure 4-11. On clicking a header column, the **Text Component Editor - Column Contents** window will be displayed. You can use this window to edit the header text, color, and style. Choose the **OK** button after editing the header text. Similarly, you can change the header text for other columns also.

To add a new column in the table structure, choose the **Add column** button on the right of the **Structure** area. Note that the added column does not have any header text. Next, double-click in the header area and specify the header for the column. For example, if you want to display the area of the elevation band, you can add **2D Area** as the column header.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Next, to specify the width for the column, click on the default **Automatic** symbol displayed in the **Column Width** row and then choose **Manual** from the flyout. Specify the column width in the edit box. By default, the column width is set to **Automatic**, which means the column width will be adjusted automatically. After specifying the column width, you need to specify the column value as the composition of that column. To do so, double-click on the **Column Value** cell in the required column; the **Text Component Editor - Column Contents** window will be displayed. Next, select the required option from the **Properties** drop-down list; the properties and values of the selected option will be displayed in the **Modifier** and **Value** columns, respectively. Modify the required values and then choose the arrow button to add the selected property in the **Text Editor**. Finally, choose the **OK** button; the selected value column will be displayed in the column.

Display Tab

This tab is used to specify the visibility, layer, color, linetype, and lineweight for the table components.

Summary Tab

The **Summary** tab is used to get the review of the legend style settings. After you have specified the settings for the table style, choose the **OK** button from the **Table Style - New Elevation Table Style** dialog box; the dialog box will be closed and the table style will be added in the respective subnode in the **Table Styles**.

Adding Surface Legend Tables

Ribbon:	Annotate > Labels & Tables > Add Tables drop-down > Add Surface Legend Table
Command:	ADDSURFACELEGENDTABLE

To add the result table of Surface analysis in the current drawing, choose the **Add Surface Legend Table** tool from the **Labels & Tables** panel; you will be prompted to enter the table type. Specify the type of table by entering a value in the command line or by selecting it dynamically from the dynamic input prompt; you will be prompted to specify the behavior of the table. Select the **Dynamic** option so that the legend table is updated automatically when you modify the surface. If you select the **Static** option, the legend table will not be updated automatically when you modify the surface. On selecting the behavioral option for the legend table, you will be prompted to specify the upper left corner of the table. Click in the drawing to specify the upper left corner; the legend table will be added in the drawing.



Note

*While adding the legend table if there are multiple surfaces, you will be prompted to select the surface or press ENTER; the **Select a Surface** dialog box will be displayed with the name of the surfaces available. Select the required surface from the list and choose the **OK** button to close the dialog box.*

To assign a new table style to the added legend table, select the table from the drawing and right-click; a shortcut menu will be displayed. Choose **Table Properties** from the shortcut menu; the **Table Properties** dialog box will be displayed, as shown in Figure 4-12.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

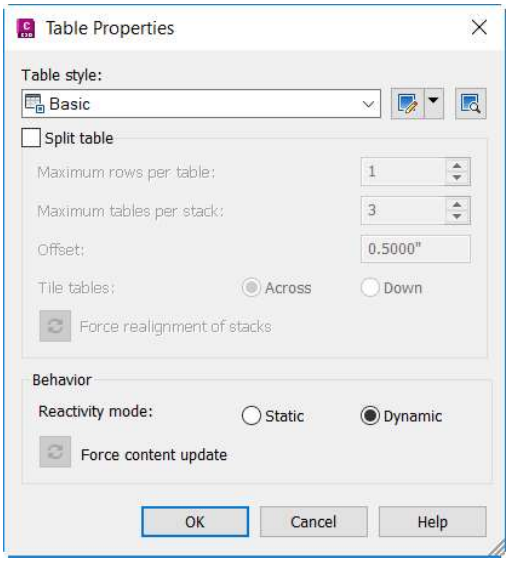

Figure 4-12 *The* ***Table Properties*** *dialog box*

Select the required table style from the **Table style** drop-down list. Next, you can select the **Split table** checkbox to split the table, if required. On doing so, the options in this dialog box will be activated. Specify the number of rows in each section of the split by using the **Maximum rows per table** spinner. Also, specify the number of splits using the **Maximum tables per stack** spinner. Next, specify the distance between the split sections of the table in the **Offset** edit box. In the **Tile tables** area, select the **Across** radio button to tile the split sections horizontally or select the **Down** radio button to tile the split sections vertically. Choose the **OK** button after selecting the table style and specifying the settings. The legend table will be displayed according to the assigned legend style.

Editing Legend Table Styles

You can edit the style settings for any legend table. To edit the legend table, expand the **Table Styles** subnode from the **Surface** node in the **Settings** tab of the **TOOLSPACE** palette. Next, expand the required legend table node for which you want to edit the legend table. Right-click on the current legend table style name and choose **Edit** from the shortcut menu; the **Table Style <style name>** dialog box will be displayed. Edit the table style settings such as data properties and the table component display in the **Data Properties** and **Display** tabs. Also, edit the table style name if required. Choose the **Apply** button. Next, choose the **OK** button to close the dialog box.



Tip

Select the required legend table in the drawing and right-click to display a shortcut menu. Choose ***Edit Table Style*** *from the shortcut menu to edit the legend table style.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

SURFACE LABELS

Ribbon: Annotate > Labels & Tables > Add Labels drop-down > Surface > Add Surface Labels
Command: ADDSURFACELABELS

After you have created and analyzed the surface, you can display the surface information by using surface labels. Surface labels have two components: labels and tag. The label component contains text and the tag component is used to mark surfaces with numbers. Surface labels are created and managed in the **Settings** tab of the **TOOLSPACE** palette. You can add labels for surface contours, slopes, and spot elevations by using the **Add Surface Labels** tool. The various label types are discussed next.

Adding Contour Labels

To add contour labels to a surface, choose the **Add Surface Labels** tool from the **Labels & Tables** panel; the **Add Labels** dialog box will be displayed. Select **Surfaces** from the **Feature** drop-down list and then select **Contour - Multiple at Interval** from the **Label type** drop-down list. Next, select the major contour label, minor contour label, and user contour label styles from the respective drop-down lists. Now, choose the **Add** button from the dialog box; you will be prompted to pick the first point. Click in the drawing to specify the first point of the line along which you want to label contours. Click again to specify the second point of the line. Next in the command line, specify the contour interval at which you want to label contours. Press ENTER; Civil 3D will label contours along the line at specified contour. Similarly, you can label multiple contours or a single contour by selecting the **Contour - Multiple** or **Contour - Single** option from the **Label type** drop-down list. Alternatively, you can label contours by choosing the **Contour - Multiple**, **Contour - Single**, or **Contour - Multiple at Interval** tool from the **Add Labels** drop-down, refer to Figure 4-13.

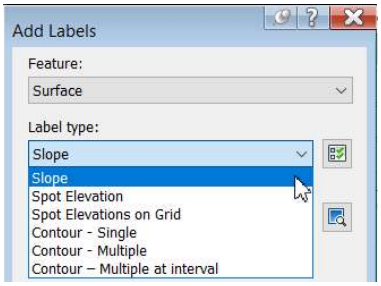

Figure 4-13 *Various options for labeling a surface*

Adding Slope Labels

You can add slope labels to a surface. To do so, choose the **Add Surface Labels** tool from the **Labels & Tables** panel; the **Add Labels** dialog box will be displayed. Select the **Surface** option from the **Feature** drop-down list in the dialog box. Next, select **Slope** from the **Label type** drop-down list.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Select the required slope label style from the **Slope label style** drop-down list. Now, choose the **Add** button; you will be prompted to choose between one point or two point method for specifying point.

The **One-point** slope label allows you to add a slope label by selecting a single point. The slope and direction are determined normal to the selected point. Press ENTER to accept the default **One-point** slope label and then pick a point in the drawing to add a slope label. The **Two-point** slope label allows you to add a slope label by selecting two points, the first point is the origin and the second point indicates the slope and the direction. To add a **Two-point** slope label, press the T key and then press ENTER. Pick the first point to add slope label and then pick the second point in the required direction; the slope label will be added to the specified direction.

Adding Spot Labels

Civil 3D allows you to add spot elevation labels and view the elevation of the surface at the spot. To add spot elevation labels, select the **Spot Elevation** option from the **Label type** drop-down list in the **Add Labels** dialog box. Select the required spot elevation label style option from the **Spot elevation label style** drop-down list. Also, select a marker style from the **Marker style** drop-down list to add a marker at the spot elevation label. Now, choose the **Add** button; you will be prompted to select the surface. Select the surface; you will be prompted to select a point in the drawing. Select the point in the drawing; the spot elevation along with the marker will be placed in the drawing for the selected point. Now, choose the **Close** button, the **Add Labels** dialog box will be closed. You can also use the **Spot Elevation on Grid** option from the **Label type** drop-down list in the **Add Labels** dialog box to add the spot elevation in a grid surface.

SURFACE LABEL STYLES

Surface styles control the display of surface, while label styles control the display of surface label styles. Civil 3D has various in-built label styles but you can also create your own label styles to label surface contours, slopes, watersheds, and spot elevations. These label styles control the visibility and display of surface labels. To create a surface label style, choose the **Settings** tab of the **TOOLSPACE** palette. Expand the **Label Styles** subnode from the **Surface** node in the **Settings** tab. Next, expand the required surface component node for which you want to create a label style. For example, in the **Contour** subnode, you will notice some in-built contour label styles. Now, if you want to create a new label style for contours, right-click on the **Contour** subnode to display a shortcut menu. Choose **New** from the shortcut menu; the **Label Style Composer - New Surface Contour Label Style** dialog box will be displayed, as shown in Figure 4-14. Various tabs in this dialog box are discussed next.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

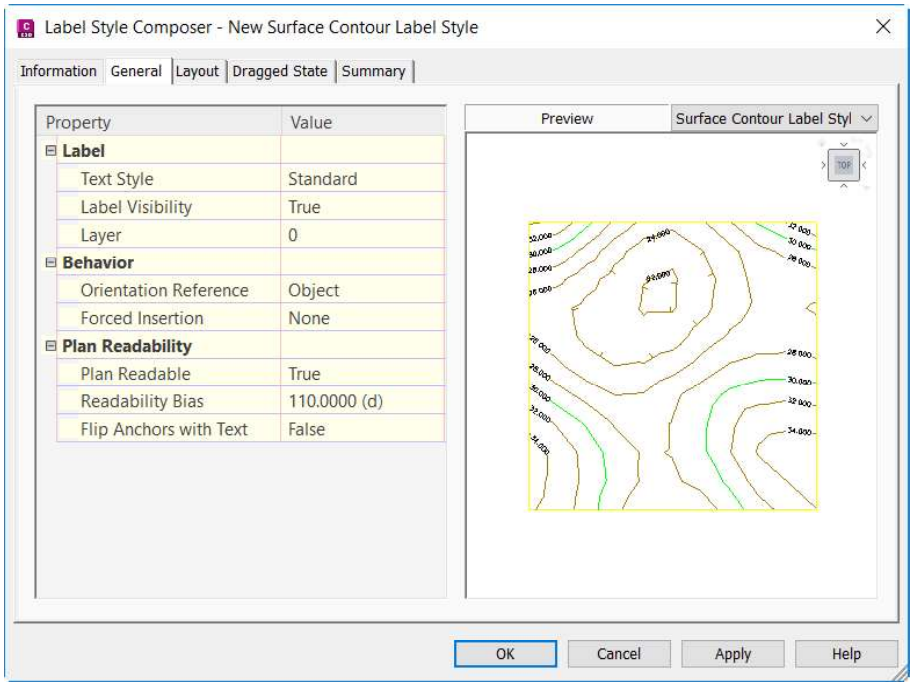

*Figure 4-14 The **Label Style Composer - New Surface Contour Label Style** dialog box*

Information Tab

Using this tab, you can specify the contour label style name in the **Name** edit box. Optionally, enter a description for the label style in the **Description** box. This tab also provides details about the label style.

General Tab

This tab is used to specify the general properties of label style such as label text style, label visibility, and label orientation. Properties of the label style are displayed in the **Property** column and values are displayed in the **Value** column in three different categories. These categories are discussed next.

Label

Expand the **Label** category. The **Text Style** property in this category specifies the text style of all the text components of a label. **Standard** is the default value of the text style. Click in the **Value** field of this property and choose the browse button; the **Select Text Style** dialog box will be displayed. Select the required type of text style from this dialog box. The **Label Visibility** property controls the visibility of the labels in the drawing. By default, its value is set to **True**. You can click in the **Value** field of this property and select **False** from the drop-down list to hide the label in the drawing. The **Layer** property specifies the layer of all components of the label. You can modify the layer on which the label will be placed.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Behavior

The **Orientation Reference** property specifies the orientation of the label. Click in the **Value** field of this property and select the required option from the drop-down list displayed. The **Object** option is selected by default in this drop-down list. The **View** option is used to realign a label to the current view orientation in both the model and plan views. The **World Coordinate System** option is used to adjust labels with respect to the angle between the current view and world view. The **Forced Insertion** property is used to specify the position of a label relative to an object. This property is applicable only when the **Orientation Reference** property is set to **Object** and the objects are lines, arc segments, or spline segments. The **None** value maintains the label position as composed, relative to the object. The **Top** option is used to add label above the object and the **Bottom** option is used to add label below the object.

Plan Readability

The **Plan Readable** property of this category specifies the text rotation to ensure that all the text components in labels are read easily in a specified view. Click in the **Value** field of the property and select **True** or **False** from the drop-down list. If you select **True**, the text in the label will be rotated so that it can be read easily in a specified view. Select the **False** option to keep the label in its original position, irrespective of the view.

Preview

The preview window displays the view of the label style. To change the view, select any option from the **Preview** drop-down list. Next, right-click on the preview window to access the view-related options in the shortcut menu.

Layout Tab

This tab is used to create, delete, or edit label components. You can also use this tab to add new components to a label, if required, and specify the text for the component. In this tab, you can specify text contents, height, color, and other related properties as well.

Dragged State Tab

This tab is used to specify label properties in the dragged state. You can control the display and appearance of labels when they are dragged from their insertion points.

Summary Tab

The **Summary** tab is used to display all the components of labels and the specified values of various label properties. After you have specified the label style properties, choose the **OK** button; the **Label Style Composer** dialog box will be closed and the new contour style will be added to the **Contour** node in the **Settings** tab of the **TOOLSPACE** palette. To view the label style, choose the **New Surface Contour Label Style** subnode from the **Label Styles** node in the **Settings** tab of the **TOOLSPACE** palette, as shown in the Figure 4-15.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com



Figure 4-15 *Choosing the* ***New Surface Contour Label Style*** *subnode from the* ***Label Styles*** *node*

Select the label style that you have created from the **Add Labels** dialog box before adding contour labels to the surface. Similarly, you can create your own styles for spot elevations, slopes, and watersheds and use them to label the surface.

TUTORIALS

1. Download the *c04_c3d_2026_tut.zip* file from *https://www.cadcim.com*. The path of the file is as follows: *Textbooks > Civil/BIM/GIS > AutoCAD Civil 3D > Exploring AutoCAD Civil 3D 2026> Tutorial Files*.

2. Save and extract the downloaded file at the following location:

   C:\ c3d_2026

The first and second tutorial of this chapter is available in video format.



Note

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

Tutorial 1 Surface Volume

In this tutorial, you will calculate the composite volume and create a TIN volume surface. Also, you will calculate the volume of an area bounded by a polygon. Figure 4-16 shows the surface whose volume has been determined. **(Expected time: 45 min)**

The following steps are required to complete this tutorial:

a. Open the file.
b. Compute composite volume.
c. Create a TIN volume surface.
d. Create the new legend style.
e. Add the legend table to the drawing.
f. Calculate the volume of the area bounded by the polygon.
g. Save the file.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

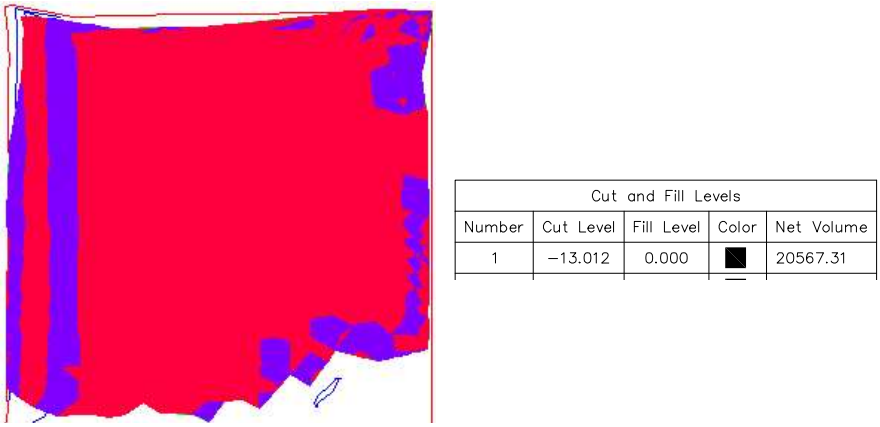

Figure 4-16 *Surface with the determined net volume*

Video

You can view the video of this tutorial by using the following link: https://www.cadcim.com/civil-3d-2026-ch-4-tutorial1.

Opening the File

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

2. In this dialog box, browse to the location *C:\c3d_2026\c04_c3d_2026_tut,* where you have saved the file.

3. Select the file *c04_c3d_2026_tut01.dwg* and choose the **Open** button to open it.

   The drawing file consists of two TIN surfaces: the **Existing Ground** surface with borders and contours, and the **Proposed Ground** surface with triangles. You will notice a magenta colored polygon in the **Proposed Ground** surface.

Computing Composite Volumes

1. Enter **REPORTSURFACEVOLUME** at the command prompt and press ENTER; the **Composite Volumes** tab in the **PANORAMA** window is displayed.

2. Choose the **Create new volume entry** button from the **Composite Volumes** tab of the **PANORAMA** window; a new volume entry is added to the list box.

3. Double-click on the **<select surface>** cell in the **Base Surface** column; a drop-down list is displayed. Select the **Existing Ground** option from this drop-down list, as shown in Figure 4-17.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

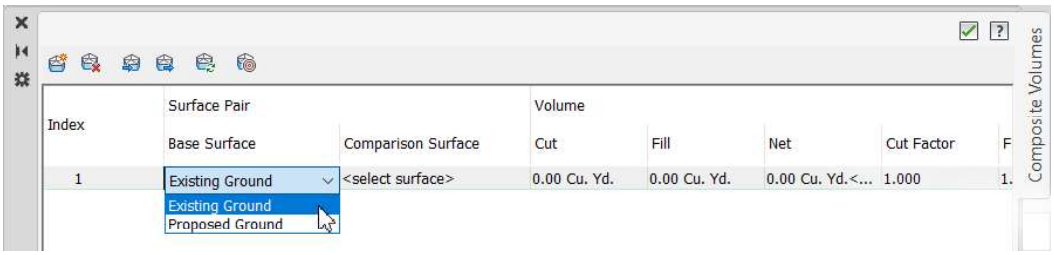

Figure 4-17 *Selecting the* ***Existing Ground*** *option from the drop-down list*

4. Similarly, select the **Proposed Ground** option from the drop-down list in the **Comparison Surface** column. On specifying the base and comparison surfaces, the net cut and fill volumes are displayed in the **Cut** and **Fill** columns, respectively, as shown in Figure 4-18.

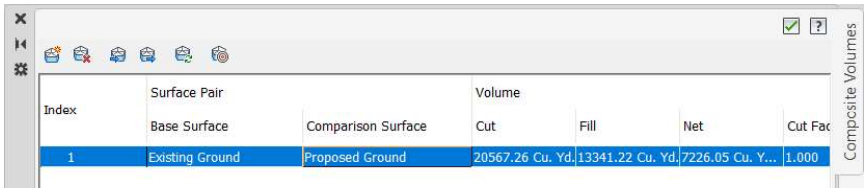

Figure 4-18 *The net cut and fill volumes displayed in the* ***Cut*** *and* ***Fill*** *columns*

The total volume of material to be removed from the existing ground is **20567.26 Cu. Yd.** and the total material required to be filled is **13341.22 Cu. Yd**.

5. Accept the default cut and fill factor values. Move the slider of the **PANORAMA** window to pan across the window to view the adjusted values for the cut, fill and net material required. The net volume of material is **7226.05 Cu Yd<Cut>**.

   This means that to create a balanced proposed ground, the **7226.05 Cu Yd** of material needs to be removed from the existing ground. Note that the red graph in the **Net Graph** column indicates that the material needs to be removed from the base surface.

6. Close the **PANORAMA** window.

Creating the TIN Volume Surface

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

2. In this dialog box, select the **TIN volume surface** option from the **Type** drop-down list.

3. Set the values of the TIN volume surface properties as follows:

   Name: **Volume Surface** Base Surface: **Existing Ground**
   Comparison Surface: **Proposed Ground**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

4. Choose the **OK** button; the **Create Surface** dialog box is closed. The surface is created and displayed in different colored bands.

5. Now, expand the **Surfaces** node in the **Prospector** tab of the **TOOLSPACE** palette.

6. Right-click on the **Volume Surface** node; a shortcut menu is displayed. From the shortcut menu, choose the **Surface Properties** option; the **Surface Properties - Volume Surface** dialog box is displayed.

7. Choose the **Statistics** tab from this dialog box if not selected by default.

8. Expand the **Volume** category in this tab; the cut, fill, and net volumes are displayed, as shown in Figure 4-19.

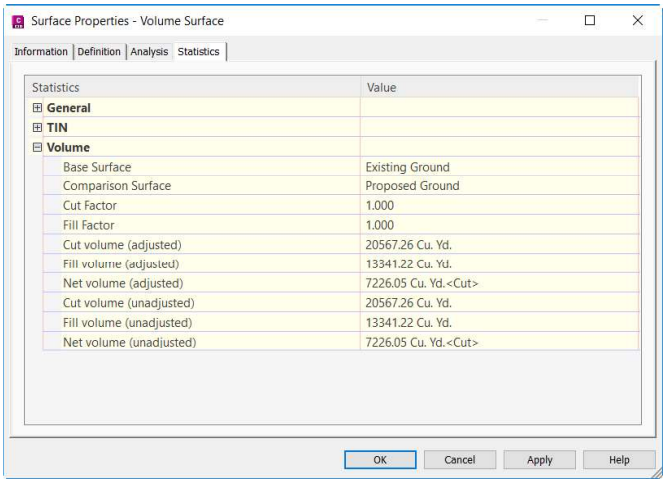

Figure 4-19 *The cut, fill and net volumes displayed in the* ***Volume*** *category*

Next, you will perform the Elevations analysis to visually differentiate various ranges of altitude in the surface.

9. Next, choose the **Analysis** tab in the **Surface Properties - Volume Surface** dialog box.

10. Ensure that the **Elevations** option is selected from the **Analysis type** drop-down list.

11. In the **Ranges** area, retain the default settings and choose the **Run analysis** button; the results are displayed in the **Range Details** area. Choose the **Apply** button; the volume surface shows elevation bands in different colors, as shown in Figure 4-20. Different color bands represent various elevation ranges in the surface.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

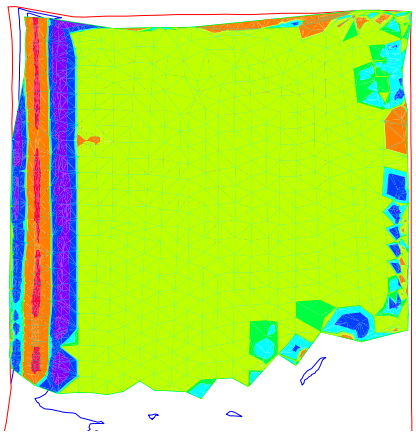

Figure 4-20 *The volume surface with elevation band*

Now perform analysis using two bands of elevation ranges.

12. Set the value in the spinner located next to the **Create ranges by** drop-down list to **2** and choose the **Run analysis** button; the analysis is performed. The range details are displayed in the **Range Details** area, as shown in Figure 4-21. Choose the **Apply** button to apply the changes to the drawing.

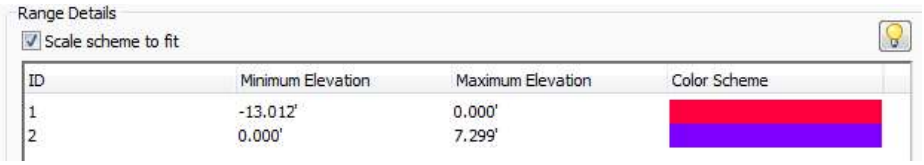

Figure 4-21 *Partial view of the* ***Range Details*** *area for Elevation analysis with two bands*

Creating the New Legend Style

In this section, you will create a new legend style.

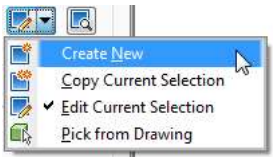

Figure 4-22 *Choosing the* ***Create New*** *option from the flyout*

1. In the **Analysis** tab, choose the down arrow button from the **Legend** area and then choose the **Create New** option from the flyout displayed, as shown in Figure 4-22. On doing so, the **Table Style - New Elevation Table Style** dialog box is displayed.

2. In the **Information** tab, enter **Cut and Fill Levels** in the **Name** edit box.

3. Choose the **Data Properties** tab from the dialog box. In the **Structure** area of this tab, double-click on the table name **Elevations Table**; the **Text Component Editor - Column Contents** dialog box is displayed.

4. Delete the existing text in the **Text Editor** text box and then enter **Cut and Fill Levels** in it, as shown in Figure 4-23.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

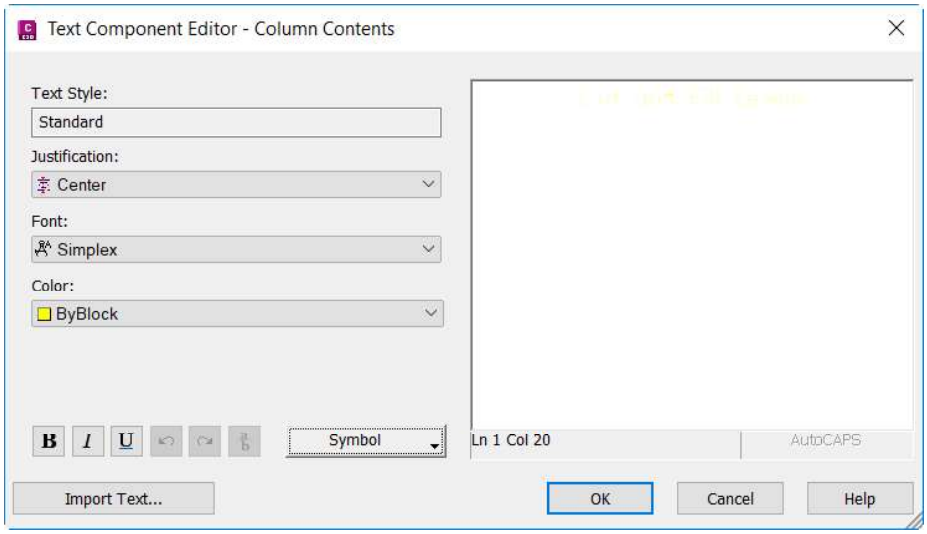

Figure 4-23 *The text entered in the* ***Text Editor*** *text box*

5. Choose the **OK** button to close the dialog box; the **Table Style - Cut and Fill Levels** dialog box is displayed. Notice that the name of the table in the **Structure** area is changed to **Cut and Fill Levels**.

6. In this dialog box, double-click on the **Minimum Elevation** header in the **Structure** area; the **Text Component Editor - Column Contents** dialog box is displayed.

7. Delete the existing text in the **Text Editor** text box and then enter **Cut Level** in it. Next, choose the **OK** button; the dialog box is closed. Notice that the **Minimum Elevation** column in the **Structure** area of the **Table Style - Cut and Fill Levels** dialog box is renamed as **Cut Level**.

8. Double-click on the **Maximum Elevation** header; the **Text Component Editor - Column Contents** dialog box is displayed.

9. Select and then delete the existing text in the **Text Editor** text box and enter **Fill Level**.

10. Next, choose the **OK** button; the **Text Component Editor - Column Contents** dialog box is closed.

11. Choose the **Add Column** button; a new column is added to the **Structure** area. 

12. Double-click on the header of the newly added column; the **Text Component Editor-Column Contents** dialog box is displayed.

13. In the **Text Editor** text box of the dialog box, enter **Net Volume** and then choose the **OK** button; the **Text Component Editor - Column Contents** dialog box is closed.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

14. Now, double-click in the **Column Value** cell of the **Net Volume** column; the **Text Component Editor - Column Contents** dialog box is displayed.

15. Now, select the **Surface Range Volume** option from the **Properties** drop-down list, as shown in Figure 4-24.

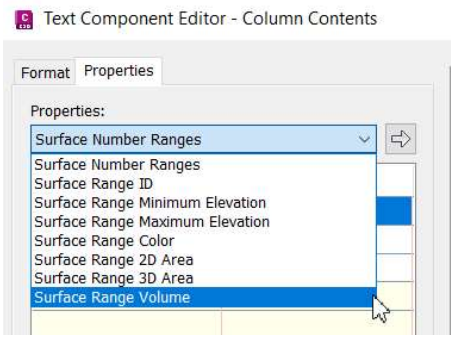

Figure 4-24 *Selecting the* ***Surface Range Volume*** *option from the* ***Properties*** *drop-down list*

16. Choose the arrow button to add this property to the **Text Editor** window. 

17. Next, choose the **OK** button; the **Text Component Editor - Column Contents** dialog box is closed.

18. Now, choose the **Display** tab from the **Table Style - Cut and Fill Levels** dialog box.

19. Set the **Overall Border** color display to **By Layer**. Also set color for the **Title Separator**, **Header Separator**, and **Data Separator** components to **blue**. Next, set the color of the **Title Text**, **Header Text**, and **Data Text** components to **red**.

20. Next, click on the bulb icon in the **Visible** column of the **Title Area Fill**, **Header Area Fill**, and **Data Area Fill** to turn off the display. Next, choose the **OK** button to close the **Table Style - Cut and Fill Levels** dialog box.

21. Select the **Preview** check box in the **Analysis** tab of the **Surface Properties - Volume Surface** dialog box to view the new **Cut and Fill Levels** legend table in the preview, as shown in Figure 4-25.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

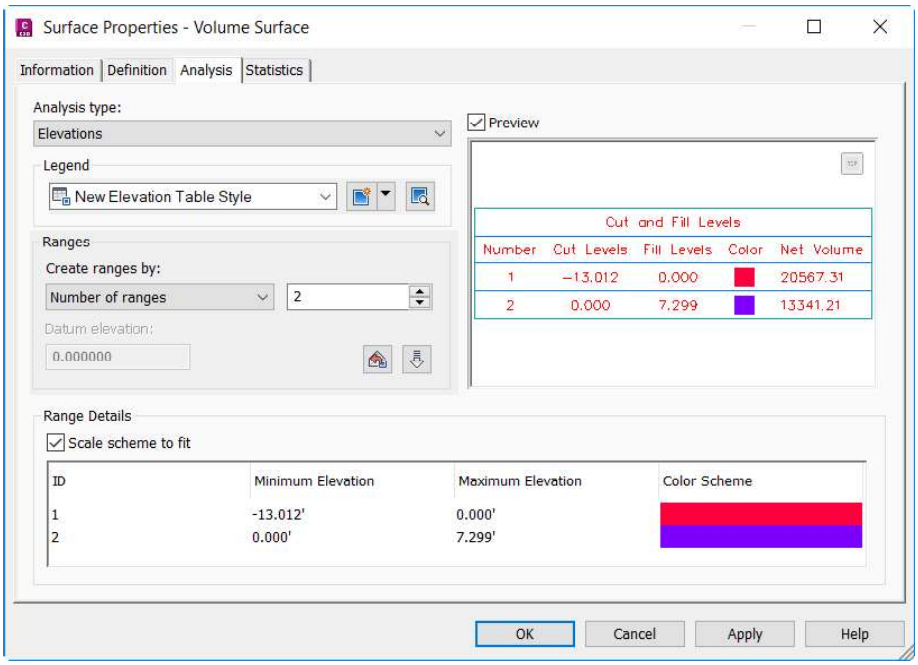

Figure 4-25 *The* ***Analysis*** *tab displaying the* ***Cut and Fill Levels*** *legend table in the* ***Preview*** *window*

22. Choose the **OK** button to close the **Surface Properties - Volume Surface** dialog box.

Adding the Legend Table to the Drawing

1. Choose the **Add Surface Legend Table** tool from **Annotate > Labels & Tables > Add Tables** drop-down; you are prompted to select the surface.

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

3. Select **Volume Surface** and choose **OK**; the dialog box is closed. Also, you are prompted to specify the type of the table.

4. Press ENTER to accept the default **Elevations** option; you are prompted to specify the behavior of the table.

5. Press ENTER to accept the default **Dynamic** option; you are prompted to select the upper left corner.

6. Click at the required location in the drawing; the legend table is added to the drawing and displays the cut and fill levels along with net volume, as shown in Figure 4-26.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Cut and Fill Levels				
Number	Cut Level	Fill Level	Color	Net Volume
1	−13.012	0.000		20567.31
2	0.000	7.299		13341.21

Figure 4-26 *The legend table displaying the cut and fill levels along with net volume*

Computing the Bounded Volume

1. In the **Prospector** tab of the **TOOLSPACE** palette, choose the **Volume Surface** subnode from the **Surfaces** node and right-click on it; a shortcut menu is displayed.

2. Choose the **Edit Surface Style** option from the shortcut menu; the **Surface Style - Elevation Banding (2D)** dialog box is displayed.

3. In this dialog box, choose the **Display** tab. Turn off the visibility of **Elevations** component in this tab. Choose the **OK** button to close the dialog box.

4. Enter **ReportSurfBoundedVolume** in the command line and press ENTER twice; the **Select a Surface** dialog box is displayed.

5. Choose the **Volume Surface** option from this dialog box.

6. Choose the **OK** button; the dialog box is closed and you are prompted to select the bounding polygon.

7. Select the **Finished Ground** displayed in magenta color from the drawing, as shown in Figure 4-27.

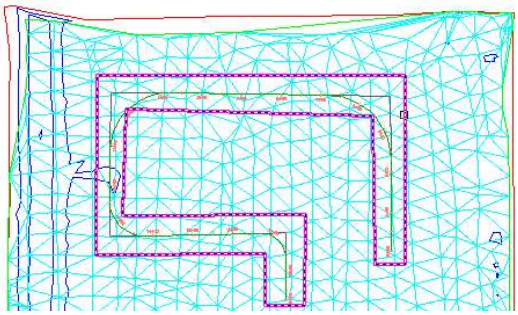

Figure 4-27 *Selecting the* ***Finished Ground*** *from the drawing*

8. Press the F2 key; the **AutoCAD Text Window** is displayed showing the cut, fill, and net volumes of the area bounded by the polygon, as shown in Figure 4-28.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

```
Select a surface <or press enter key to select from list>:

Surface:  Existing Ground
Current datum elevation: 0.000'
Select bounding polygon or [Datum]: *Cancel*

Command: *Cancel*

Command: REPORTSURFBOUNDEDVOLUME


Select a surface <or press enter key to select from list>:

Surface:  Volume Surface
Current datum elevation: 0.000'
Select bounding polygon or [Datum]:
          Net volume = 35.00 Cu. Yd.<Cut>
                 Cut = 35.00 Cu. Yd.
                Fill = 0.00 Cu. Yd.
Net volume (adjusted) = 35.00 Cu. Yd.<Cut>
      Cut (adjusted) = 35.00 Cu. Yd.
     Fill (adjusted) = 0.00 Cu. Yd.
Current datum elevation: 0.000'

Select bounding polygon or [Datum]:
```

Figure 4-28 *The* ***AutoCAD Text Window*** *displaying the cut, fill, and net volumes of the bounded area*

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_2026\c04_c3d_2026_tut

3. In the **File name** edit box, enter **c04_tut01a**.

4. Choose the **Save** button; the file is saved with the name *c04_tut01a.dwg* at the specified location.

5. Choose **Close** from the **Application Menu** to close the file.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Tutorial 2 Elevation Analysis

In this tutorial, you will carry out the Elevation analysis using two different methods of ranging elevations and viewing elevation bands in the surface, refer to Figure 4-29.

(Expected time: 30 min)

The following steps are required to complete this tutorial:

a. Open the file.
b. Create Surface Elevation analysis styles.
c. Add elevation legend.
d. Apply surface style and perform Surface analysis.
e. Save the file.

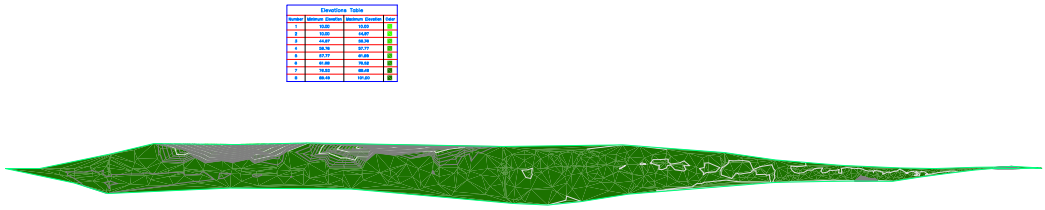

Figure 4-29 *Surface1 elevation bands*

Video

You can view the video of this tutorial by using the following link: https://www.cadcim.com/civil-3d-2026-ch-4-tutorial2.

Opening the File

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

2. In this dialog box, browse to the location *C:\c3d_2026\c04_c3d_2026_tut,* where you have saved the file.

3. Select the file *c04_c3d_2026_tut02.dwg* and choose the **Open** button from the dialog box.

The file contains a surface displaying its contours and border.

Creating a Surface Analysis Style

1. Choose the **Settings** tab from the **TOOLSPACE** palette.

2. Expand **Surface > Surface Styles** in the **Settings** tab to view the existing surface style (**Basic**).

3. Right-click on **Basic** and choose the **Edit** option from the shortcut menu displayed; the **Surface Style - Basic** dialog box is displayed.

4. Choose the **Information** tab and then enter **Elevation-Quantile** in the **Name** edit box.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

5. Choose the **Analysis** tab and expand the **Elevations** category node in it.

6. Click in the **Value** cell corresponding to the **Range color scheme** option; a drop-down list is displayed. Select the **Greens** option from this drop down list.

7. Ensure that the values of the properties in the **Elevations** category match the values shown in Figure 4-30.

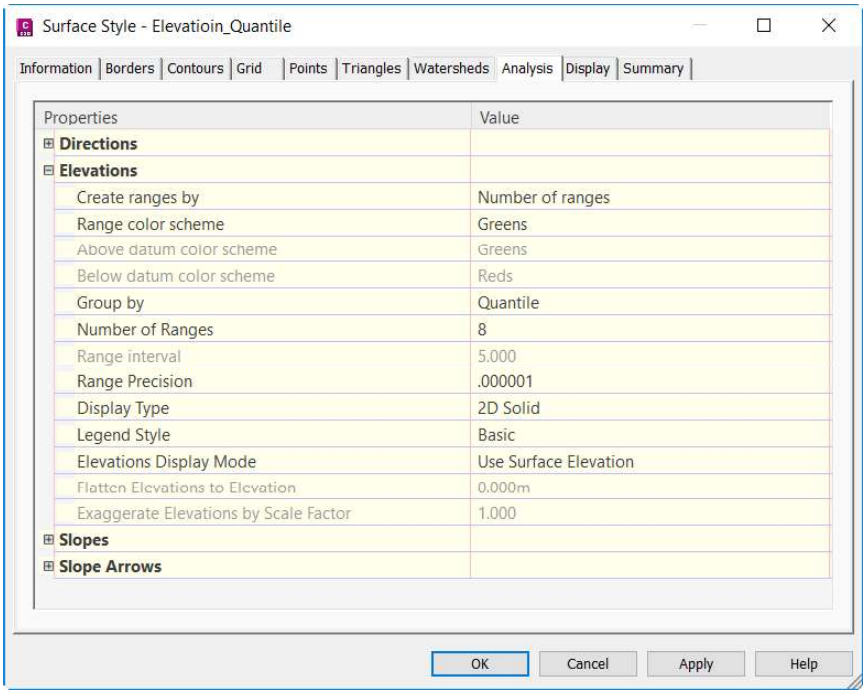

Figure 4-30 *Properties in the* ***Elevations*** *category of the* ***Analysis*** *tab*

8. Now, choose the **Display** tab from the **Surface Style - Elevation - Quantile** dialog box and turn on the visibility of the **Elevations** component.

9. Choose **Apply** and then the **OK** button; the dialog box is closed. The **Elevation-Quantile** style is added to the **Surface Styles** node, as shown in Figure 4-31. Also, elevation band are displayed in the surface, as shown in the Figure 4-32. Elevations in the surface are grouped by the Quantile method and displayed in different shades of green, representing elevations of different ranges.

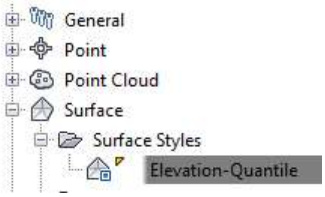

Figure 4-31 *The* ***Elevation-Quantile*** *style added to the* ***Surface Styles*** *node*



Figure 4-32 ***Surface1*** *surface displaying elevation bands*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Adding the Elevation Legend

1. Choose **Add Surface Legend Table** from **Annotate > Labels & Tables > Add Tables** drop-down.

2. Select **Elevations** from the dynamic input prompt if not selected by default; the table type is specified and you are prompted to specify the behavior type of the legend.

3. Press ENTER to select **Dynamic** from the dynamic input prompt; you are prompted to specify the upper left corner of the table.

4. Click at the required location in the drawing area to add the legend table, refer to Figure 4-26; the Elevations table showing eight ranges of elevations with a color assigned to each range is added to the drawing, refer to Figure 4-33.

Elevations Table			
Number	Minimum Elevation	Maximum Elevation	Color
1	10.00	98.37	
2	98.37	98.94	
3	98.94	99.32	
4	99.32	99.53	
5	99.53	99.74	
6	99.74	99.93	
7	99.93	100.21	
8	100.21	101.00	

Figure 4-33 *The Elevations table*

In this case, the dark shade represents the highest elevation range (101.00) whereas the light shade represents the ranges with lower elevations.

5. In the **Settings** tab, expand **Surface Styles** in the **Surface** node. Right-click on the **Elevation-Quantile** subnode; a shortcut menu is displayed.

6. Choose the **Copy** option from the shortcut menu; the **Surface Style - Elevation-Quantile [Copy]** dialog box is displayed.

7. Choose the **Information** tab. Enter **Elevations-Equal Interval** in the **Name** edit box.

8. Next, choose the **Analysis** tab. Expand the **Elevations** category.

9. Set the value of the **Range color scheme** property to **Greens** and the **Group by** property to **Equal interval**.

10. Choose the **Display** tab and turn on the visibility of **Elevations** if not turned on by default.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

11. Choose the **OK** button; the dialog box is closed and **Elevations - Equal Interval** is added to the **Surface Styles** node.

Applying Surface Style and Performing Surface Analysis

1. Select the surface in the drawing area and right-click; a shortcut menu is displayed.

2. Choose the **Surface Properties** option from the shortcut menu; the **Surface Properties - Surface1** dialog box is displayed.



Tip

*You can also invoke the **Surface Properties - Surface1** dialog box using the **Prospector** tab. To do so, expand the **Surfaces** node and then right-click; a shortcut menu is displayed. In this menu, choose the **Surface Properties** option.*

3. In the **Information** tab, select the **Elevations - Equal Interval** option from the **Surface style** drop-down list in the **Default styles** area.

4. Choose the **Apply** button.

5. Choose the **Analysis** tab and make sure the **Elevations** option is selected in the **Analysis type** drop-down list.

6. In the **Ranges** area of the tab, retain the default settings and then choose the **Run Analysis** button to run the analysis; the details are displayed in the **Range Details** area.

7. Choose the **OK** button to close the dialog box. Notice the difference in representation of the surface. Also, notice the updated values in the **Elevations** table.

   As the Dynamic option is selected, the values are updated while specifying the behavior of the legend.

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_2026\c04_c3d_2026_tut

3. In the **File name** edit box, enter **c04_tut02a**.

4. Next, choose the **Save** button; the file is saved with the name *c04_tut02a.dwg* at the specified location.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Tutorial 3 Slope Analysis

In this tutorial, you will create a style for the Slope analysis. Next, you will run the Slope analysis and the Slope Arrows analysis on the surface, as shown in Figure 4-34.

(Expected time: 30 min)

The following steps are required to complete this tutorial:

a. Open the file.
b. Perform the Slope analysis.
c. Add the slope legend table.
d. Perform the Slope Arrow analysis.
e. Save the file.

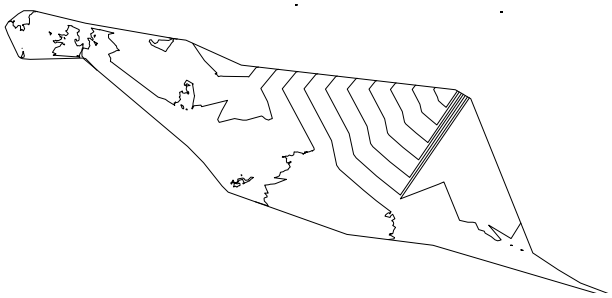

Figure 4-34 *The surface used for slope analysis and slope arrows analysis*

Opening the File

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

2. In this dialog box, browse to the location *C:\c3d_2026\c04_c3d_2026*, where you have saved the file.

3. Select the file *c04_c3d_2026_tut03.dwg* and choose the **Open** button to open it. The file consists of a surface that displays contours and border, refer to Figure 4-34.

Performing the Slope Analysis

1. Select **Existing Surface** from the drawing and right-click; a shortcut menu is displayed.

2. Choose the **Surface Properties** option from the shortcut menu; the **Surface Properties - Existing Surface** dialog box is displayed.

3. Choose the **Analysis** tab from the dialog box if not selected by default and then select the **Slopes** option from the **Analysis type** drop-down list.

4. Set the value to **5** in the **Number** spinner in the **Ranges** area of the dialog box if not set by default.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

5. Choose the **Run analysis** button; the information about minimum and maximum values of different ranges of slopes is displayed in the **Range Details** area. Five different ranges in the area are represented by different color shades.

6. Double-click on the color for the range ID **1** in the **Range Details** area; the **Select Color** dialog box is displayed.

7. Select the **red** color from the dialog box and then choose the **OK** button; the **Select Color** dialog box is closed and the color representing the range is changed to red.

8. Follow the procedure used in steps 6 and 7 to modify the color of ranges 2, 3, 4, and 5 to **yellow**, **green**, **cyan**, and **blue** respectively.

9. Choose **Apply** and then the **OK** button; the **Surface Properties - Existing Surface** dialog box is closed.

10. Now, expand the **Surface** node in the **Prospector** tab and right-click on **Existing Surface**; a shortcut menu is displayed.

11. Choose the **Edit Surface Style** option from the shortcut menu; the **Surface Style - Standard** dialog box is displayed.

12. Turn on the visibility of **Slopes** in the **Display** tab and choose **OK** to close the dialog box. Figure 4-35 shows different colors representing different slope bands created on the surface.

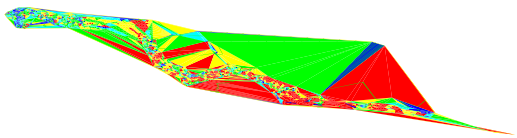

Figure 4-35 *Different colors representing different slope bands*

Adding the Slope Legend

1. Choose the **Add Surface Legend Table** tool from **Annotate > Labels & Tables > Add Tables** drop-down; you are prompted to select the table type.

2. Select **Slopes** from the dynamic input prompt to select the table type; you are prompted to specify the table behavior. Now, press ENTER; the **Dynamic** option is accepted as the default behavior and you are prompted to specify the top left corner of the table.

3. Click in the drawing area to add slope legend table; the slope legend table is added to the drawing showing information about the minimum/maximum slope of a range and the color representing the range.

4. Select the legend table and right-click; a shortcut menu is displayed.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com



Tip

The table displayed in the drawing area is very small due to the view scale. Use the ***Zoom Selected objects*** *option to zoom in the table for easy viewing. To do so, enter* ***Z*** *at the command line; you are prompted to select the mode of zooming. Enter* ***O*** *at the command line for specifying* ***Object****; you are prompted to select the object. Select the slope legend table from the drawing and press ENTER, the slope legend table in the drawing area is zoomed in.*

5. Choose the **Edit Table Style** option from the shortcut menu; the **Table Style - Standard** dialog box is displayed.

6. Choose the **Summary** tab from the dialog box and expand the **General** category.

7. Set the following values in the **General** category:

   Text Height: Title: **20.0000"** Text Height: Header: **18.0000"**

   Text Height: Data: **15.0000"**

8. Choose the **Apply** button and then the **OK** button to close the dialog box; the **Slopes Table** is displayed, as shown in Figure 4-36.

Slopes Table			
Number	Minimum Slope	Maximum Slope	Color
1	0.00%	1.32%	
2	1.32%	2.65%	
3	2.65%	4.46%	
4	4.46%	8.27%	
5	8.27%	75056.83%	

Figure 4-36 *The* ***Slopes Table***

Performing the Slope Arrows Analysis

1. Select **Existing Surface** from the drawing and right-click; a shortcut menu is displayed.

2. Choose the **Surface Properties** option from the shortcut menu;the **Surface Properties - Existing Surface** dialog box is displayed.

3. Choose the **Analysis** tab from the dialog box and select the **Slope Arrows** option from the **Analysis type** drop-down list.

4. Next, set the value to **5** in the **Number** spinner of the **Ranges** area of the dialog box if not set by default.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

5. Choose the **Run analysis** button; the information about the slope area is displayed in the **Range Details** area. It displays the minimum and maximum slopes for each range. Slope arrows are divided into five different ranges represented by different color shades.

6. Double-click on the currently selected color for the range ID **1**; the **Select Color** dialog box is displayed.

7. Select red from the dialog box if not selected by default and then choose the **OK** button; the **Select Color** dialog box is closed and the color representing the range is changed to red.

8. Using the procedure followed in steps 6 and 7 to modify the color of ranges **2**, **3**, **4**, and **5** to **yellow**, **green**, **cyan**, and **blue**, respectively.

9. Choose **Apply** and then the **OK** button; the **Surface Properties - Existing Surface** dialog box is closed.

10. Select the surface in the drawing area and right-click; a shortcut menu is displayed. Select the **Edit Surface Style** option from the shortcut menu; the **Surface Style - Standard** dialog box is displayed. In the dialog box, choose the **Display** tab and ensure that the visibility of **Slopes** is turned off and the visibility of **Slope Arrows** component is turned on. You can toggle the visibility state of the selected component by choosing the bulb icon in the **Visible** column.

11. Choose the **OK** button; the dialog box is closed and the drawing now displays the result of the Slope Arrows analysis**.** The result is displayed using different colored arrows that correspond to different slope ranges, as shown in Figure 4-37.

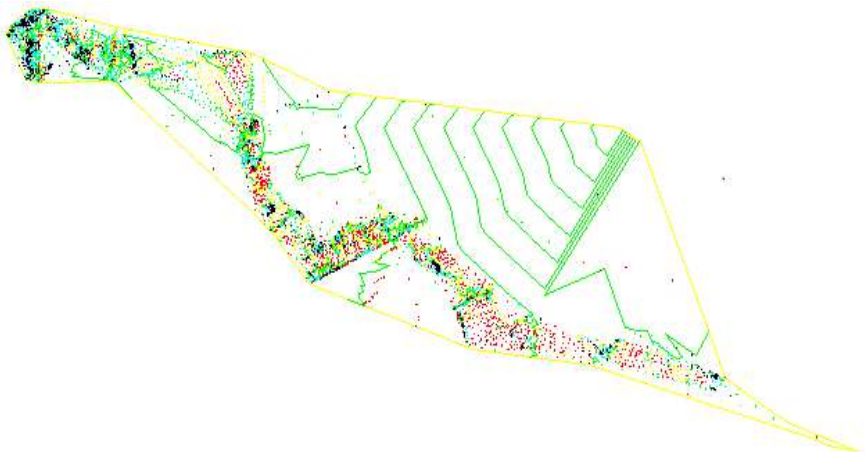

Figure 4-37 *The arrows displaying the Slope Arrows analysis*

12. Zoom in to view the **Slope Arrows** legend table. On doing so, arrows of different colors pointing in different directions are displayed.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

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_2026\c04_c3d_2026_tut

3. In the **File name** edit box, enter **c04_tut03a**.

4. Next, choose the **Save** button; the file is saved with the name *c04_tut03a.dwg* at the specified location.

Tutorial 4 Watershed and Water Drop Analyses

In this tutorial, you will perform the Watershed and Water Drop analyses for the given surface. You will also extract objects from the given surface, refer to Figure 4-38**.**

(Expected time: 30 min)

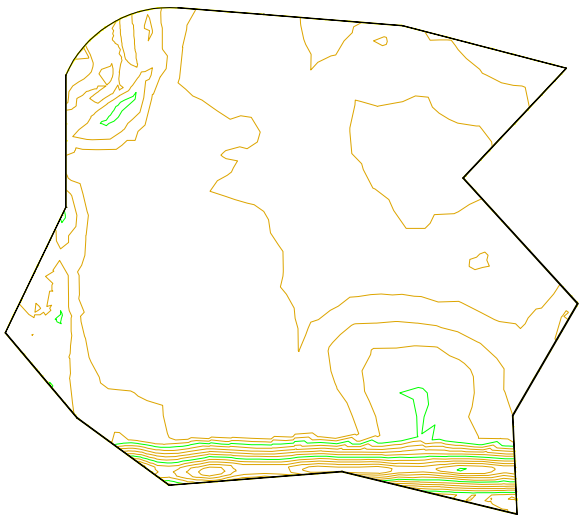

Figure 4-38 *Surface for Watershed and Water Drop analyses*

The following steps are required to complete this tutorial:

a. Open the file.
b. Create surface style.
c. Perform the Watersheds analysis.
d. Add the Watershed legend.
e. Extract objects from the surface.
f. Perform the Water Drop analysis.
g. Save the file.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Opening the File

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

2. In this dialog box, browse to the location *C:\c3d_2026\c04_c3d_2026_tut,* where you have saved the file.

3. Select the file *c04_c3d_2026_tut04.dwg* and choose the **Open** button to open the file. The file consists of a surface that displays its contours and surface border, refer to Figure 4-38.

Creating the Surface Style

1. Select the surface from the drawing area and right-click; a shortcut menu is displayed.

2. Choose the **Edit Surface Style** option from the shortcut menu; the **Surface Style - New Surface Style** dialog box is displayed.

3. In the dialog box, choose the **Watersheds** tab and expand the **Boundary Point Watershed** category.

4. Next, set the following values of properties as given below:

Color: **red** (default color) Use Hatching: **True**
Hatch Pattern: **ANS137**

5. Now, expand the **Boundary Segment Watershed** category and ensure that the following values of properties are set:

Color: **blue** Use Hatching: **True**
Hatch Pattern: **ANS131**

6. Next, expand the **Depression Watershed** category and set the following values:

Color: **green** (default color) Use Hatching: **True**
Hatch Pattern: **EARTH**

7. Expand the **Multi-Drain Watershed** category and set the following values:

Color: **magenta** Use Hatching: **True** Hatch Pattern: **ANS132**

8. Choose the **Apply** button.

9. Now, choose the **Display** tab and turn on the visibility for the **Watersheds** option.

10. Choose the **OK** button to close the **Surface Style - New Surface Style** dialog box.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Performing the Watersheds Analysis

1. Select **Surface 1** from the drawing and right-click; a shortcut menu is displayed.

2. Choose the **Surface Properties** option from the shortcut menu; the **Surface Properties - Surface1** dialog box is displayed.

3. Ensure that the **Analysis** tab is selected and then select the **Watersheds** option from the **Analysis type** drop-down list.

4. Choose the **Run analysis** button; the details of the **Watershed** analysis are displayed in the **Range Details** area.

5. Choose the button with a bulb icon at the top right corner of the **Range Details** area;  the **Watershed Display** dialog box is displayed.

6. Click on the bulb icons next to the **Flat area** and **Multi-drain notch** watershed types to turn off the display of these watersheds in the drawing, refer to Figure 4-39.

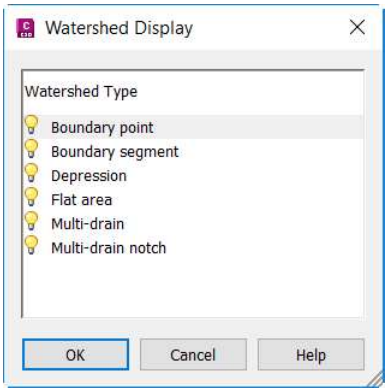

Figure 4-39 *The Flat area and Multi-drain notch watershed options*

7. Choose the **OK** button to close the dialog box.

8. Again, choose OK to close the Surface Properties - Surface1 dialog box; different hatch patterns representing different watershed regions in the surface are displayed, as shown in Figure 4-40. Each watershed area has a tag that displays information about that watershed.

9. Zoom in to view the watershed tag in each area, refer to Figure 4-41. The watershed tag displays the ID, type, and area of the watershed.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

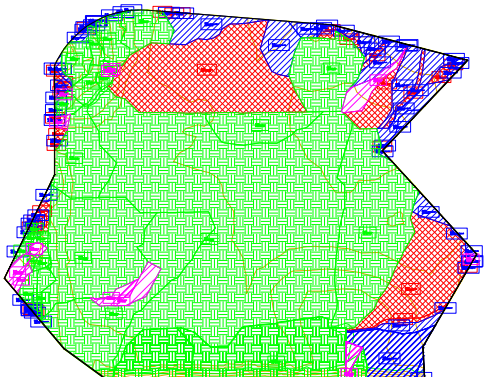

Figure 4-40 *Watersheds displayed after performing the* ***Watersheds*** *analysis*

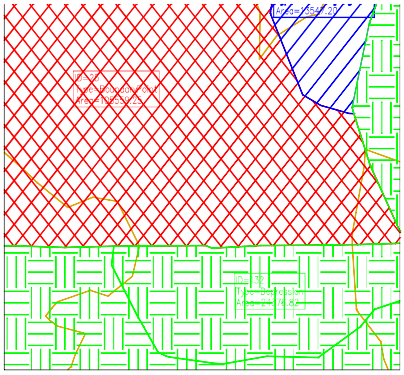

Figure 4-41 *The watershed tag for the given watershed*

Adding the Watershed Legend

1. Choose the **Add Surface Legend Table** tool from **Annotate > Labels & Tables > Add Tables** drop-down; you are prompted to select the table type.

2. Select **Watersheds** from the **Enter table type** dynamic input prompt displayed in the screen, as shown in Figure 4-42; you are prompted to specify the top left corner of the table.

Note

Dynamic input is displayed only if the ***Dynamic Input*** *option is turned on.*



Figure 4-42 *Selecting the* ***Watersheds*** *option from the* ***Enter table type*** *dynamic input prompt*

3. Select **Dynamic** as the behavior of the table from the input prompt.

4. Click at the required location in the drawing; the watershed legend table is added to the drawing and also the information about the watersheds is displayed in 155 ranges, refer to Figure 4-43.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Watersheds Table

ID	Type	Drains Into	Description	Segment Display	Area Display	Area
1	Boundary point		Description 1	———	179.80 Sq. Ft.	179.80
2	Boundary point		Description 2	———	11.02 Sq. Ft.	11.02
3	Boundary point		Description 3	———	78.29 Sq. Ft.	78.29
4	Boundary point		Description 4	———	5.91 Sq. Ft.	5.91
5	Boundary point		Description 5	———	146.03 Sq. Ft.	146.03
6	Boundary point		Description 6	———	413.02 Sq. Ft.	413.02
7	Boundary point		Description 7	———	722.31 Sq. Ft.	722.31
8	Boundary point		Description 8	———	3304.61 Sq. Ft.	3304.61
9	Boundary point		Description 9	———	2570.51 Sq. Ft.	2570.51
10	Boundary point		Description 10	———	338.51 Sq. Ft.	338.51

Figure 4-43 *Partial view of the Watersheds Table*

5. Now, select the **Watersheds Table** and right-click on it; a shortcut menu is displayed.

6. Choose the **Edit Table Style** option from the shortcut menu; the **Table Style - Basic** dialog box is displayed.

7. In the **Data Properties** tab, select the **Description** column from the **Structure** area and choose the **Delete column** button to delete the selected column. 

8. Choose **Apply** and then the **OK** button to close the dialog box. The **Description** column is removed from the watershed legend table.

9. Again, select the **Watersheds Table** and right-click; a shortcut menu is displayed. Choose the **Table Properties** option from the shortcut menu; the **Table Properties** dialog box is displayed, as shown in Figure 4-44.

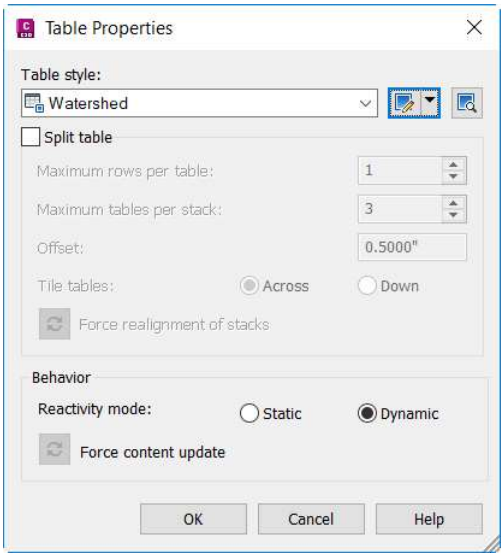

Figure 4-44 *The* ***Table Properties*** *dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

10. Select the **Split table** check box if not selected by default. Set the value to **10** in the **Maximum rows per table** spinner and **3** in the **Maximum tables per stack** spinner.

11. Choose the **OK** button; the dialog box is closed and the **Watersheds Table** is divided into three columns.

Extracting Objects from Surface

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

2. Choose the **Extract Objects** tool from the **Extract From Surface** drop down in the **Surface Tools** panel of the **Surface** tab; the **Extract Objects from Surface - <Surface1>** dialog box is displayed.

3. Clear all the check boxes except the **Watersheds** check box in the dialog box and then choose the **OK** button; the dialog box is closed and watersheds are extracted from the objects.

4. Move the cursor over the surface and click on any watershed area; you will notice that you can select the watershed areas individually. Also, on selecting an individual watershed area, a blue grip is displayed around the watershed area. This indicates that you have selected an individual object.

5. Keeping the watershed area selected, enter **LIST** in the command line and then press ENTER; the AutoCAD Text Window showing the parameters of the selected watershed area is displayed, as shown in Figure 4-45.

6. Press ENTER; you are prompted to select an object. Select another watershed and press ENTER to view the parameter of the selected watershed in the **AutoCAD Text Window**. Now, close the **AutoCAD Text Window**.

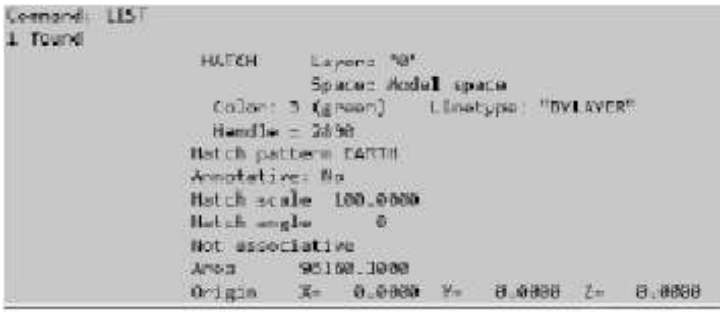

Figure 4-45 *The* ***AutoCAD Text Window*** *showing parameters of the selected watershed area*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Performing the Water-Drop Analysis

1. Choose the **Water Drop** tool from **Analyze > Ground Data > Flow Paths** drop-down; the **Water Drop** dialog box is displayed.

2. Click in the **Value** field of the **Path Layer** property; a **Browse** button is displayed. Choose the **Browse** button; the **Layer Selection** dialog box is displayed.

3. In this dialog box, choose the **New** button; the **Create Layer** dialog box is displayed.

4. In the **Create Layer** dialog box, click in the **Value** field of the **Layer Name** property and enter **C-TOPO-WDRP** in the text box displayed. Next, choose the **OK** button; a new layer is added to the **Layer Selection** dialog box.

5. Select the **C-TOPO-WDRP** layer in the **Layer Selection** dialog box and choose the **OK** button; the selected layer name is displayed in the **Value** field of the **Path Layer** property of the **Water Drop** dialog box.

6. In the **Water Drop** dialog box, click in the **Value** field corresponding to the **Place Marker at Start Point** property; a drop-down list is displayed. Select the **Yes** option from the drop-down list, refer to Figure 4-46.

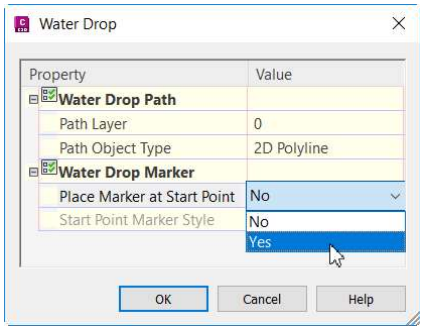

Figure 4-46 *The **Yes** option selected in the **Place Marker at Start Point** drop-down list*

7. Choose the **OK** button; the dialog box is closed and you are prompted to select the point.

8. Zoom in the bottom of the surface and click at different locations; the polylines and markers are displayed, as shown in Figure 4-47.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

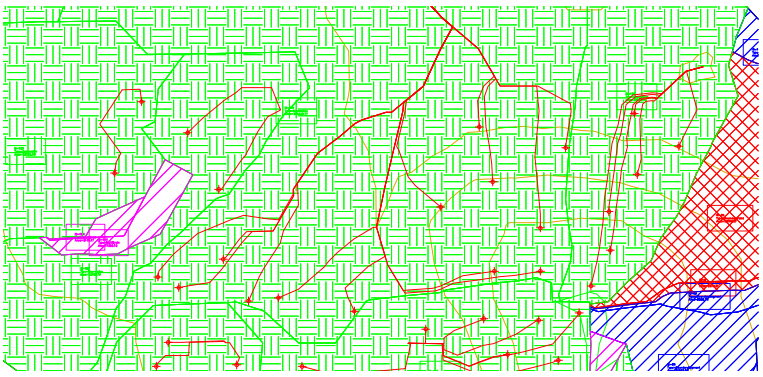

Figure 4-47 *Polyline and markers displaying the path of water flow*

9. The marker specifies the start point of the path and the polyline shows the path followed by water.

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_2026\c04_c3d_2026_tut

3. In the **File name** edit box, enter **c04_tut04a**.

4. Next, choose the **Save** button; the file is saved with the name *c04_tut04a.dwg* 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 analysis is used to determine the direction of slope?

   a) Slope
   b) Slope Arrow
   c) Contours
   d) Elevation

2. The _____ is used to investigate different properties of a surface.

3. The _____ tab in the **Surface Properties - <Surface Name>** dialog box is used to run the Surface analysis.

4. In the **Surface Properties** dialog box, you can view the legend table in the _____ area by selecting the **Preview** check box.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

5. The _____ provide you an organized way to view information about the Surface analysis.

6. The _____ area in the **Surface Properties** dialog box is used to specify the contents and the properties of the result table for the Surface analysis.

7. The _____ tab in the **Label Style Composer** dialog box is used to define the layout of a label by creating, deleting, or editing label components and their properties.

8. The _____ tab is used to define the properties for labels when they are dragged from their original position or insertion point.

9. The _____ tool is used to calculate the cut and fill quantities for a pair of surfaces.

10. The _____ and _____ factors are used for calculating the **Cut (adjusted)** and **Fill (adjusted)** volumes.

11. The **Slope Arrows** analysis is based on the slope directions. (T/F)

12. After carrying out the **Slope Arrows** analysis, an arrow will be displayed at the centroid of every triangle. (T/F)

13. In the **Watersheds** analysis, the depth less than the value specified in the **minimum average depth value** edit box is automatically ignored from being considered as watershed. (T/F)

14. The boundary point is the lowest drain target point at the end of the channel through which water continues to flow off the surface. (T/F)

15. The Multi-Drain Notch watershed is found on a surface where a flat edge exists between two points forming a notch and water flows into multiple drains from the notch. (T/F)

Review Questions

Answer the following questions:

1. Which of the following types of Surface analysis is carried out to analyze the flow of water across a surface?

a) **Slope** b) **Slope Arrows**
c) **Elevations** d) **Watersheds**

2. Which of the following are modes of a label?

a) Label mode and tag mode b) Dragged mode and invisible mode
c) Text mode and insertion mode d) None of these

3. Which of the following options allows you to create contour labels at specified interval?

a) **Contour- Single** b) **Add Labels**
c) **Contour -Multiple** d) **Contour -Multiple at Interval**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

4. The _____ method is used to compute the volumes of a specific area in the surface.

5. The _____ tab in the **Surface Properties** dialog box is used to display the computed volumes of a TIN volume surface.

6. The _____ key is used to display the result of the bounded volume in the **AutoCAD Text Window**.

7. The_____ analysis is used to divide elevation into bands of different colors representing various elevations.

8. The legend table styles are created, edited, and managed in the **Prospector** tab of the **TOOLSPACE** palette. (T/F)

9. The labels in the drawing can update automatically with a change in the surface. (T/F)

10. Watershed labels are added automatically when watersheds are displayed. (T/F)

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

EXERCISES

1. Log on to *www.cadcim.com* and browse to *Textbooks>Civil/BIM/GIS>AutoCAD Civil 3D>Exploring AutoCAD Civil 3D 2026>Exercise Files & Instructor Guide*. Next, *click on c04_c3d_2026 exer.zip* file to download the file.

2. Now, save and extract the downloaded folder to the following location:

   C:\c3d_2026

Note

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

Exercise 1

Open the *c04_c3d_2026_ex01.dwg* file and run the Contour analysis. Also, add contour labels. Figures 4-48 and 4-49 are given for your reference.

(Expected time: 30 min)

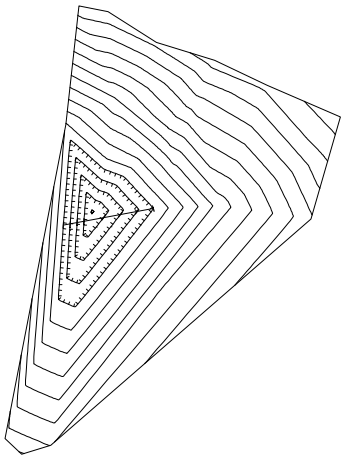

Figure 4-48 *Surface after contour analysis*

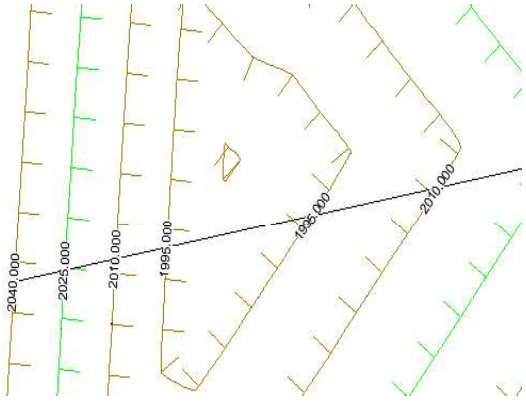

Figure 4-49 *Depression contours and contour labels*

Right-click on **Surface 1** in the **Prospector** tab and choose **Edit Surface Style** from the shortcut menu. Specify the following settings in the **Contours** tab of the **Surface Style** dialog box:

Number of Ranges: **5**
Minor Interval: **15.000'**
Display Depression Contours: **True**
Tick Mark Length: **3.000'**
Smooth Contours: **True**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Exercise 2

Open the *c04_c3d_2026_ex02.dwg* file and compute the volume between the surface1 and pool surface. Also, create a TIN volume surface. Assign the volume style to the TIN volume surface, refer to Figure 4-50. **(Expected time: 30 min)**

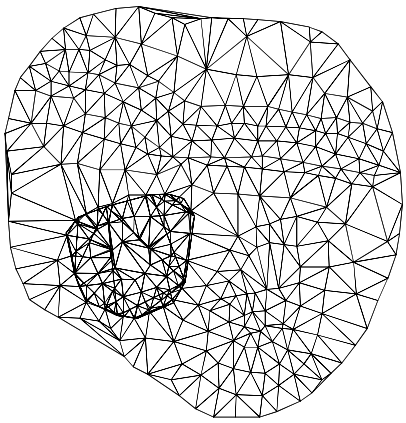

Figure 4-50 *The TIN volume surface*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Answers to Self-Evaluation Test

1. b, **2. Surface analysis**, **3. Analysis**, **4. Preview**, **5.** Range Details, **6.** Legends, **7. Layout**, **8. Dragged State**, **9. Volumes Dashboard**, **10.** Cut, Fill, **11.** T, **12.** T, **13.** T, **14**. T, **15**. T