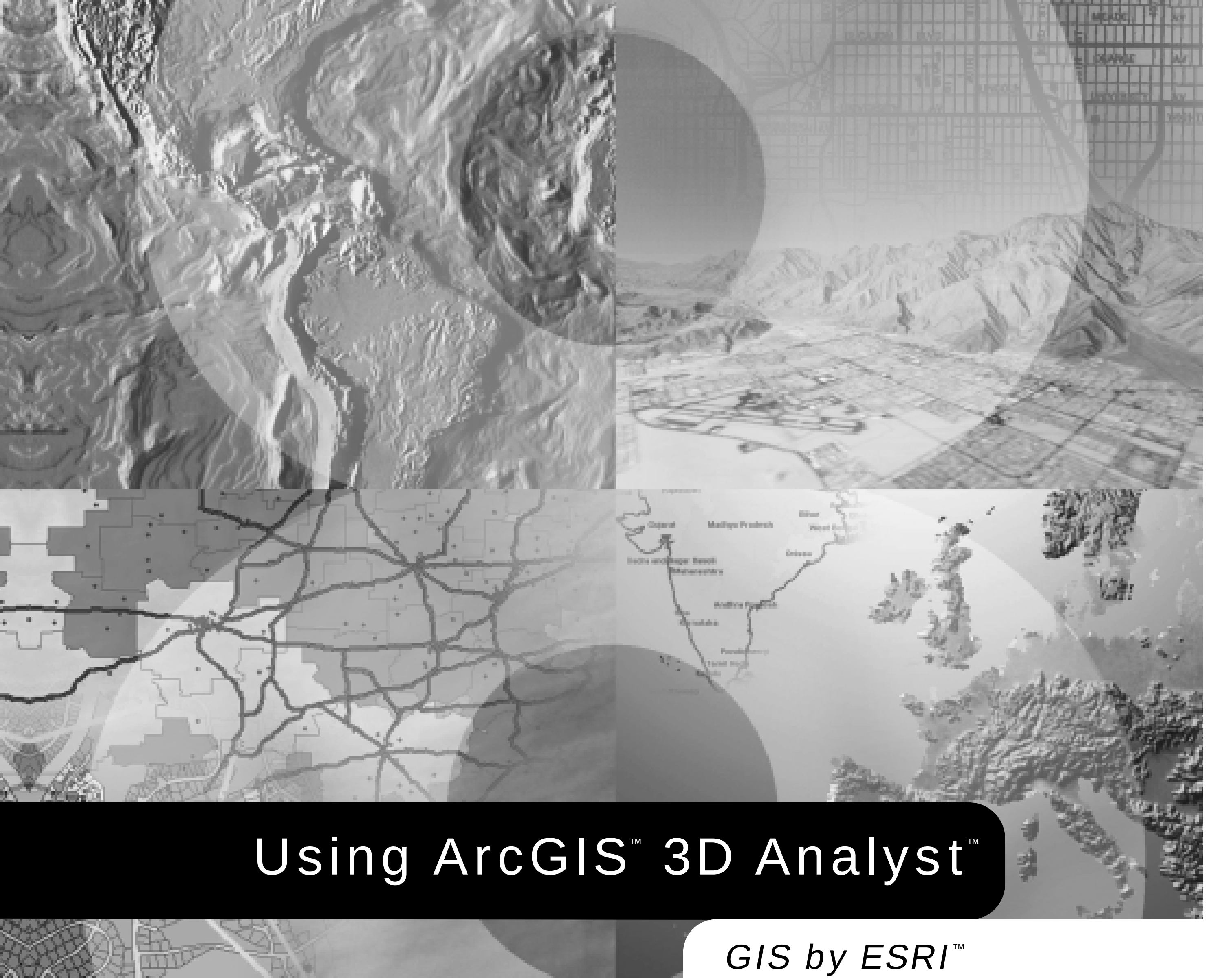


Using ArcGIS™ 3D Analyst™

GIS by ESRI™

DATA CREDITS

Exercise 1: Death Valley image data courtesy of National Aeronautics and Space Administration (NASA)/Jet Propulsion Laboratory (JPL)/Caltech.

Exercise 2: San Gabriel Basin data courtesy of the San Gabriel Basin Water Quality Authority.

Exercise 3: Belarus CS137 soil contamination and thyroid cancer data courtesy of the International Sakharov Environmental University.

Exercise 4: Hidden River Cave data courtesy of the American Cave Conservation Association.

Exercise 5: Elevation and image data courtesy of MassGIS, Commonwealth of Massachusetts Executive Office of Environmental Affairs.

CONTRIBUTING WRITERS

Steve Bratt, Bob Booth

Contents

Introducing ArcGIS 3D Analyst

1

IN THIS CHAPTER

- **What can you do with 3D Analyst?**
- **Tips on learning 3D Analyst**

Welcome to ESRI® ArcGIS™ 3D Analyst™, the three-dimensional (3D) visualization and analysis extension. 3D Analyst adds a specialized 3D viewing application, ArcScene™, to your desktop and adds new capabilities to ArcCatalog™ and ArcMap™.

ArcScene lets you make perspective view scenes in which you can navigate and interact with your geographic information system (GIS) data. You can drape raster and vector data over surfaces and extrude features from vector data sources to create lines, walls, and solids. You can also use 3D Analyst tools in ArcScene to create and analyze surfaces.

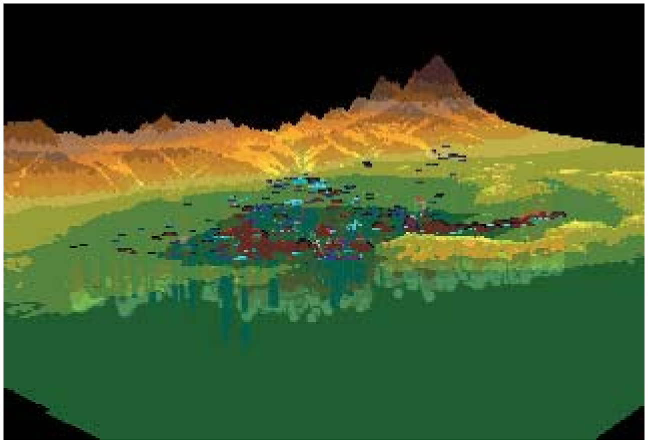
ArcCatalog is extended by 3D Analyst so you can manage 3D data and create layers with 3D viewing properties. You can preview scenes and data in 3D in ArcCatalog using the same 3D navigation tools you use in ArcScene.

ArcMap is extended by 3D Analyst so you can create new surfaces from your GIS data as well as analyze surfaces, query attribute values at a location on a surface, and analyze the visibility of parts of a surface from different locations. You can also determine the surface area and the volume above or below a surface and create profiles along a 3D line on a surface.

3D Analyst also lets you create 3D features from existing two-dimensional (2D) GIS data—or you can digitize new 3D vector features and graphics in ArcMap using a surface to provide the z-values.

What can you do with 3D Analyst?

3D Analyst provides you a set of tools in ArcScene for analysis and visualization of 3D data. It also extends ArcCatalog and ArcMap so you can more effectively manage your 3D GIS data and do 3D analysis and 3D feature editing in ArcMap.



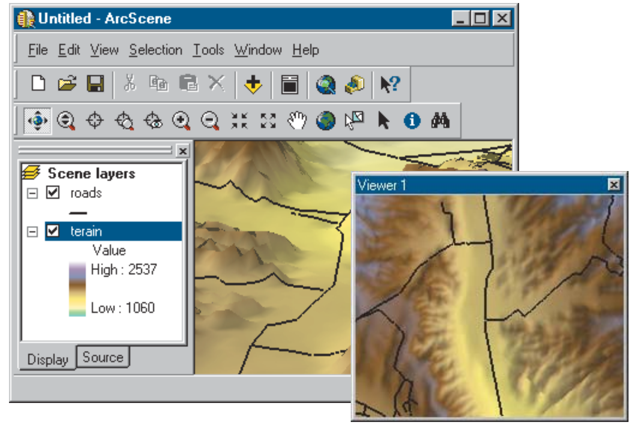
3D view of raster and vector data

What can you do with ArcScene?

The core of the 3D Analyst extension is the ArcScene application. ArcScene provides the interface for viewing multiple layers of 3D data for visualizing data, creating surfaces, and analyzing surfaces.

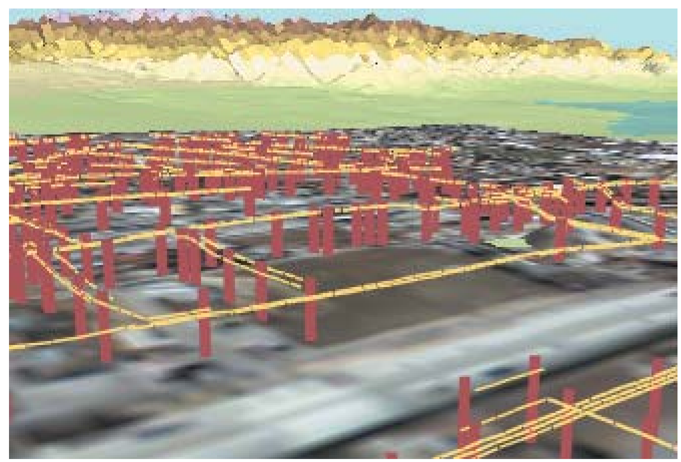
Visualizing data

3D Analyst lets you drape images or vector data over surfaces and extrude vector features from a surface. You can view a scene from multiple viewpoints using different viewers.

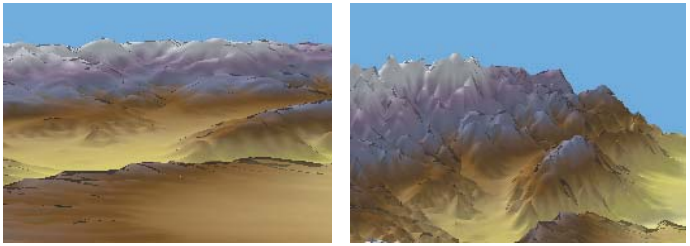


Viewing a scene from another perspective with another viewer

You can change the properties of 3D layers to use shading or transparency, and you can change the properties of a 3D scene to



3D view of utility poles and power lines

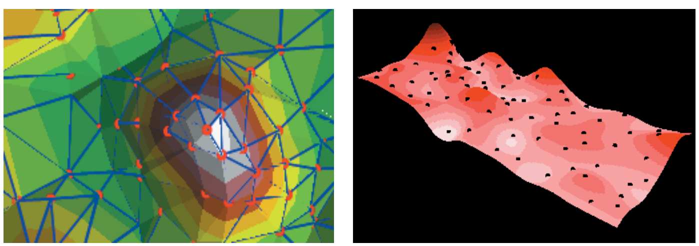


Exaggerating the vertical dimension of a scene

set the vertical exaggeration of terrain, the coordinate system and extent for the scene, and the illumination of the scene.

Creating surfaces

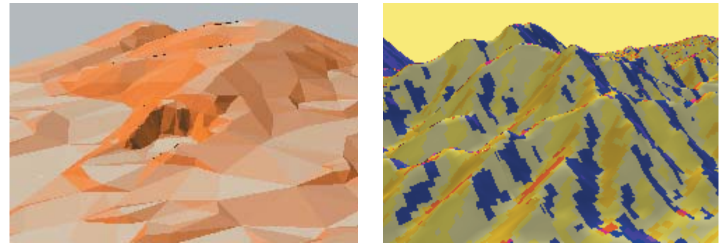
The 3D Analyst tools that are available in both ArcScene and ArcMap allow you to create surface models from your GIS data. You can interpolate raster surfaces and create or add features to triangulated irregular network (TIN) surfaces.



TIN and raster surfaces

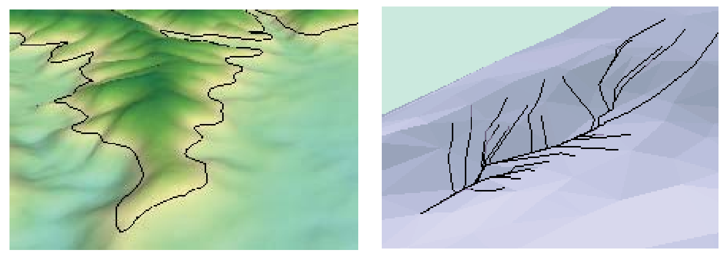
Analyzing surfaces

3D Analyst lets you interactively query the values in a raster surface and the elevation, slope, and aspect of TINs.



Slope and aspect rendering of surfaces

You can derive new rasters of slope and aspect from surface models, create contours, and find the steepest paths on a surface.

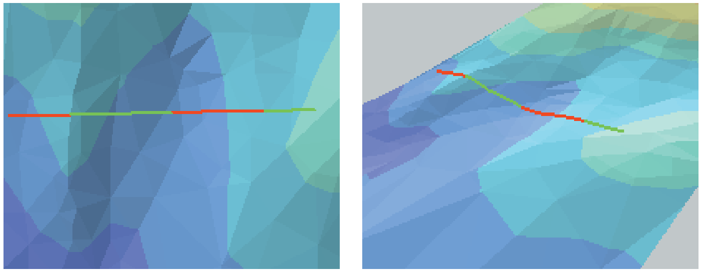


Contours and steepest paths

You can also analyze the visibility between different locations on a surface; create rasters that show the level of illumination of a surface (given a sun altitude and direction); and reclassify raster data for display, analysis, or feature extraction purposes.

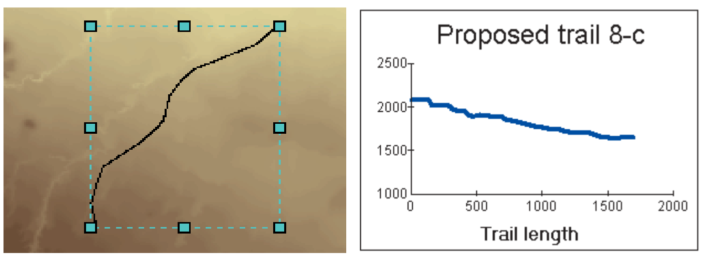
What can you do with ArcMap?

Installing 3D Analyst lets you add the 3D Analyst toolbar to ArcMap, so you can do all of the surface creation and analysis tasks in ArcMap that you can do in ArcScene. It also adds several tools that only operate in ArcMap: a tool that lets you find lines of sight on a surface, three tools for digitizing 3D features and



Line of sight created on a TIN in ArcMap, displayed in ArcScene

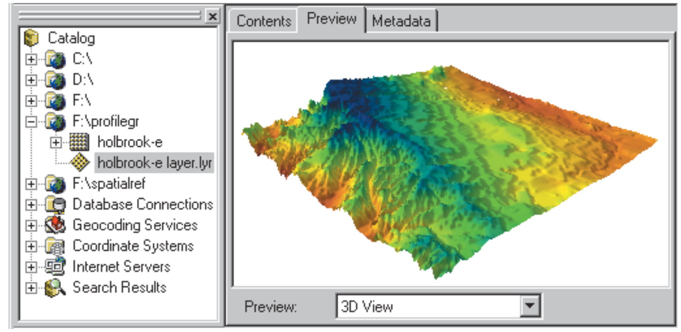
graphics using z-values from a surface, and a tool that lets you create graphs of the profile (change in elevation over distance) along a 3D line.



3D line graphic created on a raster in ArcMap and a profile graph of the line.

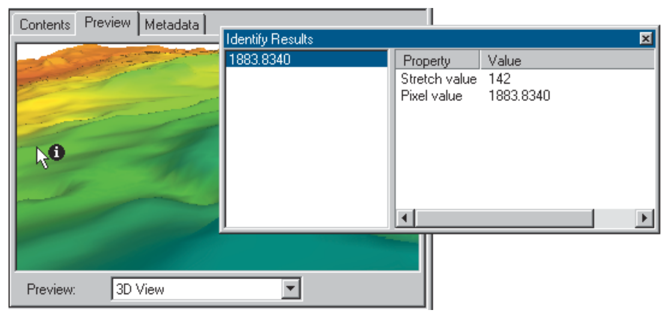
What can you do with ArcCatalog?

ArcCatalog is the ArcGIS application for managing your GIS data. 3D Analyst lets you preview and navigate around your 3D data. You can create GIS data layers and define their 3D viewing properties.



ArcCatalog lets you browse your data and create and preview layers in 3D.

You can also preview the scenes you've created in ArcScene. You can create metadata for your 3D GIS data including 3D



Navigate the 3D preview; identify features, raster cells, and TIN triangles.

thumbnails of scenes and data. You can create empty 3D feature classes or shapefiles in ArcCatalog that can then be populated by digitizing 3D features in ArcMap.

Tips on learning 3D Analyst

If you're new to GIS, take some time to familiarize yourself with ArcMap and ArcCatalog. The books *Using ArcMap* and *Using ArcCatalog* contain tutorials to show you how to make maps and manage GIS data.

Begin learning to use the 3D Analyst extension in Chapter 2, 'Quick-start tutorial', in this book. In Chapter 2 you'll learn how to open a 3D scene, add and query data in a 3D scene, create surfaces, and use ArcScene with ArcCatalog and ArcMap. The 3D Analyst extension comes with the data used in this tutorial, so you can follow along step by step at your computer. You can also read the tutorial without using your computer.

Finding answers to questions

Like most people, your goal is to complete your task while investing a minimum amount of time and effort in learning how to use the software. You want intuitive, easy-to-use software that gives you immediate results without having to read pages and pages of documentation. However, when you do have a question, you want the answer quickly so that you can complete your task. That's what this book is all about—getting you the answers you need, when you need them.

This book describes how to display your data in three dimensions, to create new surfaces from existing data, and to analyze three-dimensional surfaces. Although you can read this book from start to finish, you'll likely use it more as a reference. When you want to know how to do a particular task, such as adding shading to a surface, just look it up in the table of contents or the index. You'll find a concise, step-by-step description of how to complete the task. Some chapters also include detailed information that you can read if you want to learn more about the concepts behind the tasks. You can also refer to the glossary in this book if you come across any unfamiliar terms.

Getting help on your computer

In addition to this book, use the ArcGIS Desktop Help system to learn how to use 3D Analyst. To learn how to use Help, see *Using ArcMap*.

Contacting ESRI

If you need to contact ESRI for technical support, see the product registration and support card you received with ArcGIS 3D Analyst or refer to 'Contacting Technical Support' in the 'Getting more help' section of the ArcGIS Desktop Help system. You can also visit ESRI on the Web at *www.esri.com* and *support.esri.com* for more information on 3D Analyst and ArcGIS.

ESRI education solutions

ESRI provides educational opportunities related to geographic information science, GIS applications, and technology. You can choose among instructor-led courses, Web-based courses, and self-study workbooks to find educational solutions that fit your learning style. For more information go to *www.esri.com/education*.

Quick-start tutorial

2

IN THIS CHAPTER

The best way to learn 3D Analyst is to use it. In the exercises in this tutorial, you will:

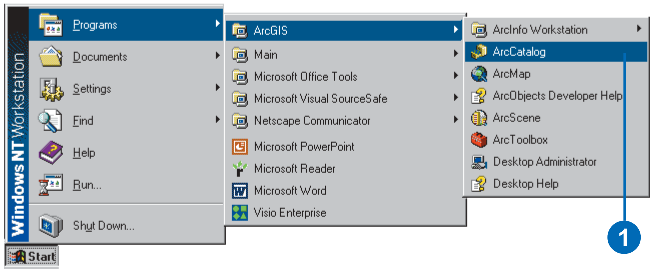
- Use ArcCatalog to find and preview 3D data.
- Add data to ArcScene.
- Set 3D properties for viewing data.
- Create new 3D feature data from 2D features and surfaces.
- Create new raster surface data from point data.
- Build a TIN surface from existing feature data.

In order to use this tutorial, you need to have the 3D Analyst extension and ArcGIS installed and have the tutorial data installed on a local or shared network drive on your system. Ask your system administrator for the correct path to the tutorial data if you do not find it at the default installation path specified in the tutorial.

Copying the tutorial data

First you will copy the tutorial data to a local drive. You will use ArcCatalog to browse to and copy the data.

1. Click Start, point to Programs, point to ArcGIS, and click ArcCatalog.

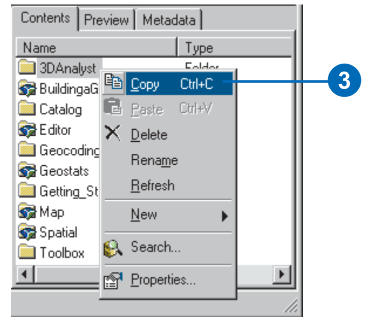


ArcCatalog lets you find and manage your data. The left side of the ArcCatalog window is called the *Catalog tree*; it gives you a bird's-eye view of how your data is organized and provides a hierarchical view of the geographic data in your folders. The right side of the Catalog window shows the contents of the selected branch of the Catalog tree.

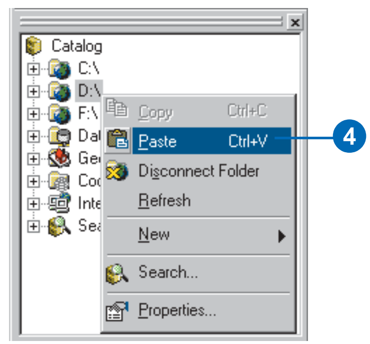
2. Click in the Location combo box and type the path to the \arcgis\ArcTutor folder on the drive where the tutorial data is installed. Press Enter.

The ArcTutor folder is now the selected branch of the Catalog tree. You can see its contents in the Contents tab.

3. Right-click the 3DAnalyst folder and click Copy.

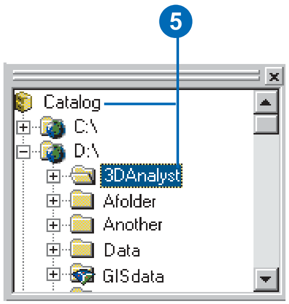


4. Right-click the local drive where you want to place the tutorial data and click Paste.



The folder is copied to your local drive. Now you will make a folder connection to the 3DAnalyst folder in the Catalog tree.

5. Click the 3DAnalyst folder on your local drive and drag it onto the top-level node, Catalog, of the Catalog tree.



There is now a folder connection in the Catalog for your local copy of the tutorial data.

In the graphics illustrating this tutorial, the ArcCatalog option to use a special folder icon for folders containing GIS data is turned on. That is why the folder GISdata, in the graphic above, looks different from the other folders. You can turn this option on in ArcCatalog, in the Options dialog box, on the General tab. ArcCatalog works faster when this option is turned off, so it is off by default.

Exercise 1: Draping an image over a terrain surface

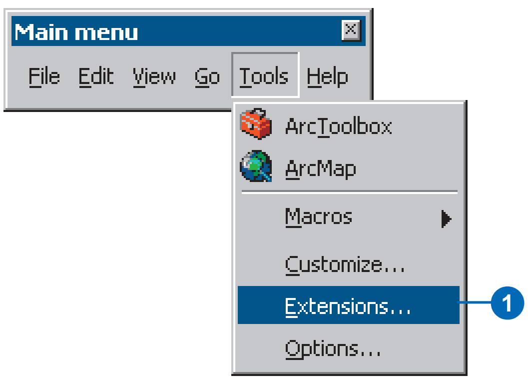
Viewing a remotely sensed image draped over a terrain surface can often lead to greater understanding of the patterns in the image and how they relate to the shape of the earth's surface.

Imagine that you're a geologist studying Death Valley, California. You have collected a TIN that shows the terrain and a satellite radar image that shows the roughness of the land surface. The image is highly informative, but you can add a dimension to your understanding by draping the image over the terrain surface. Death Valley image data was supplied courtesy of NASA/JPL/Caltech.

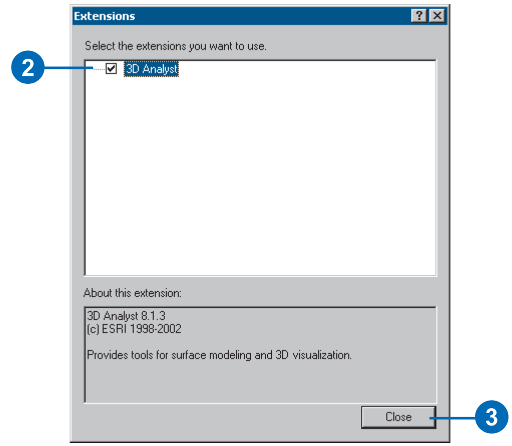
Turning on the 3D Analyst extension

You will need to enable the 3D Analyst extension.

1. Click Tools and click Extensions.



2. Check 3D Analyst.
3 Click Close.



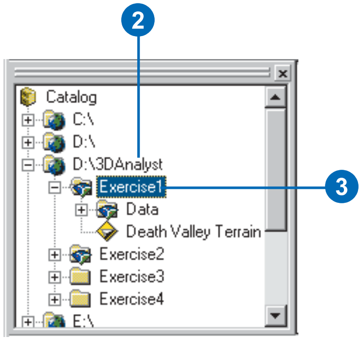
Previewing 3D data in ArcCatalog

Before you drape the image, you will browse to the terrain data and preview it in ArcCatalog.

1. Navigate to the 3DAnalyst folder connection in the Catalog tree.

2. Double-click 3DAnalyst.

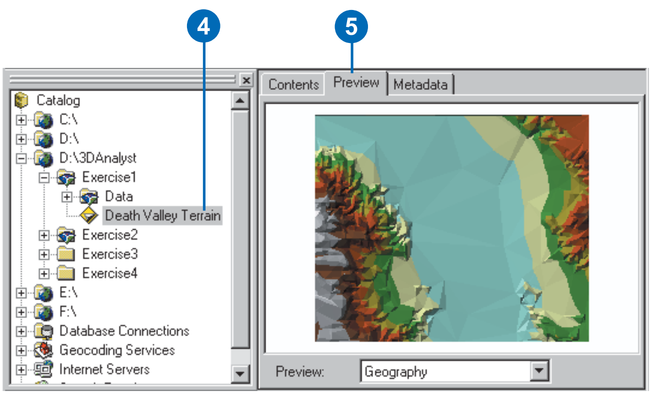
3. Double-click Exercise1.



You see a folder called Data and a TIN layer called Death Valley Terrain.

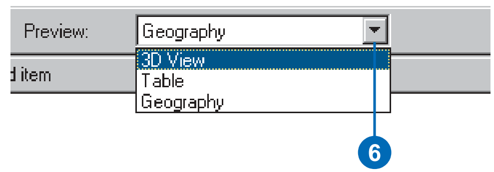
A layer is a shortcut to geographic data. It also stores information about how the geographic data should be drawn on a map or in a 3D scene.

4. Click Death Valley Terrain.

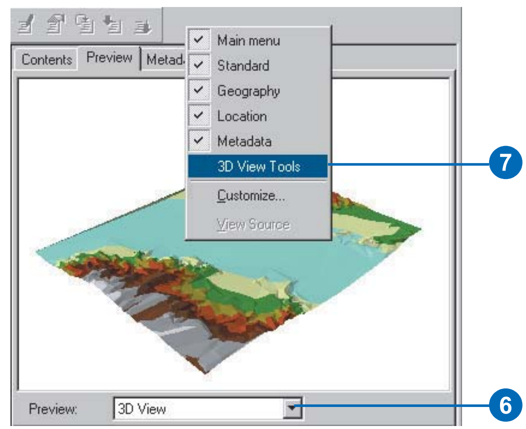


5. Click the Preview tab. You can preview your GIS data in ArcCatalog. With 3D Analyst installed, you can also preview some data in three dimensions.

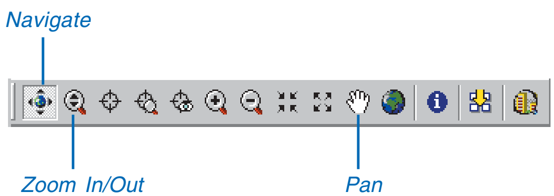
6. Click the Preview dropdown arrow and click 3D View.



7. Right-click above the preview window and click 3D View Tools.



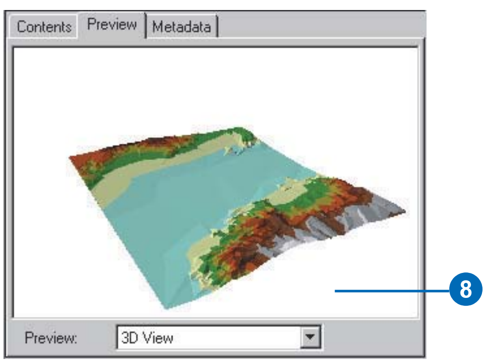
The preview becomes a 3D preview, and a new set of tools appears on the 3D View Tools toolbar.



The Navigate tool is active when you first preview data in 3D. You can see the names of tools by hovering the pointer over the tool.

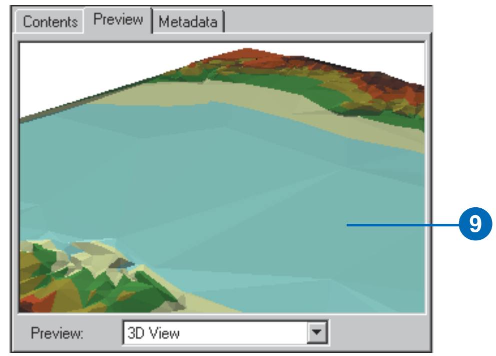
The Navigate tool allows you to rotate 3D data and change the apparent viewer height by clicking and dragging left and right and up and down, respectively, in the 3D preview.

8. Click on the 3D preview and drag to the right.



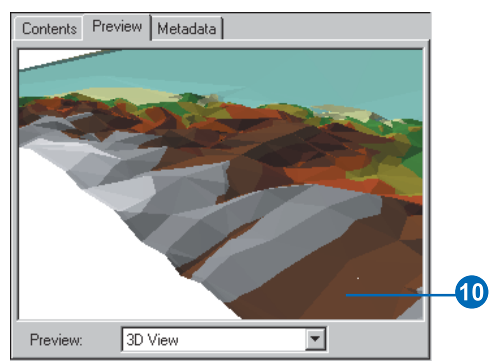
The data rotates around its center. The Navigate tool also allows you to zoom in and out and pan across the data, depending on the mouse button that you click while dragging in the 3D preview.

9. Right-click on the 3D preview and drag down.



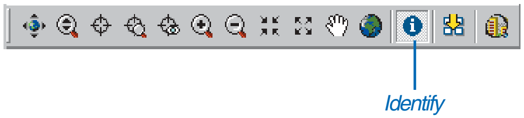
The pointer changes to the Zoom In/Out pointer, and the view zooms in to the data.

10. Click the middle button—or both the right and left buttons if you have a two-button mouse—and drag to the right.



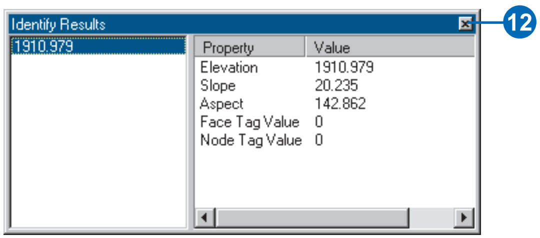
The pointer changes to the Pan pointer, and the view pans across the data.

11. Click the Identify button and click on the TIN.

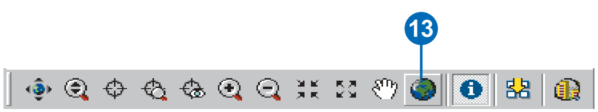


The Identify Results window shows you the elevation, slope, and aspect of the surface at the point you clicked.

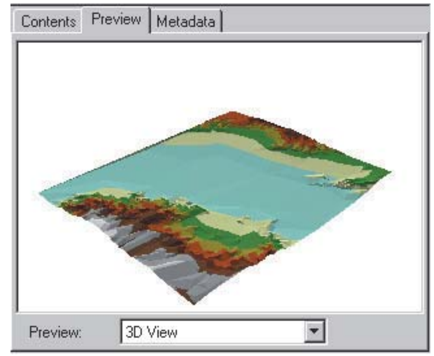
12. Close the Identify Results window.



13. Click the Full Extent button.



The view returns to the full extent of the data.

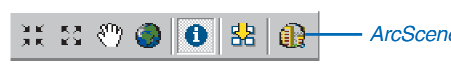


Now you've examined the surface data and begun to learn how to navigate in 3D. The next step is to start ArcScene and add your radar image to a new scene.

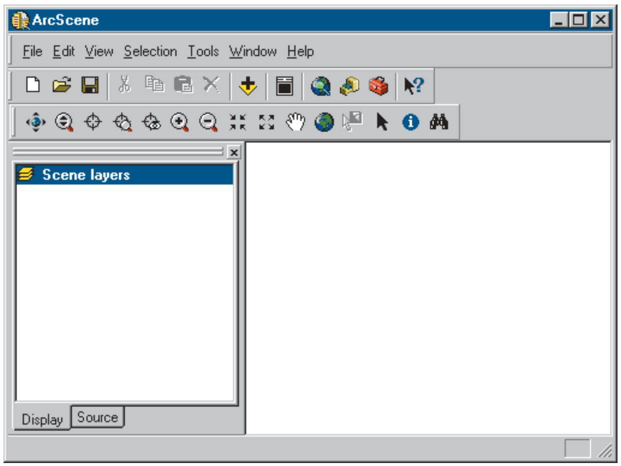
Starting ArcScene and adding data

ArcScene is the 3D viewer for 3D Analyst. Although you can preview 3D data in ArcCatalog, ArcScene allows you to build up complex scenes with multiple sources of data.

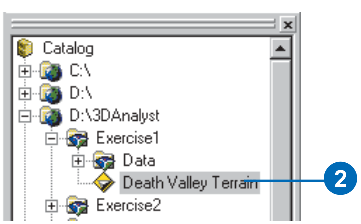
1. Click the ArcScene button on the 3D View Tools toolbar.



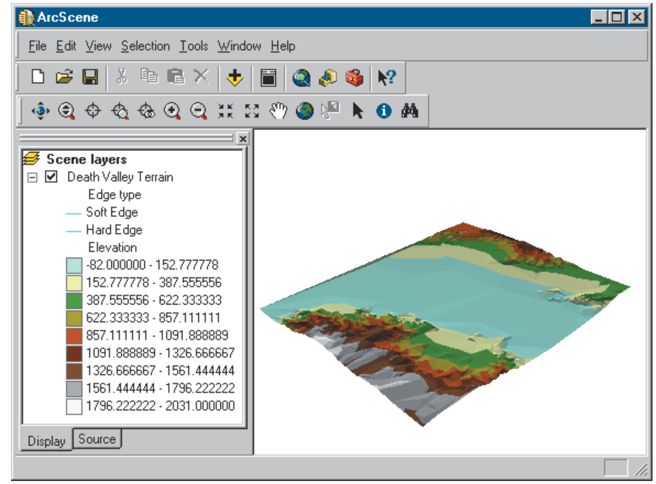
ArcScene starts. Note that many of the tools on the ArcScene Standard toolbar are the same as the 3D navigation tools that you see in ArcCatalog.



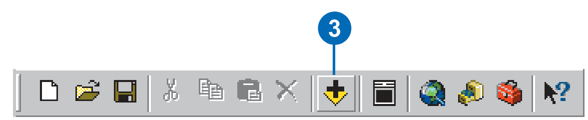
2. Click the Death Valley Terrain layer in the Catalog tree and drag it onto the right-hand side of the ArcScene window, then release the mouse button.



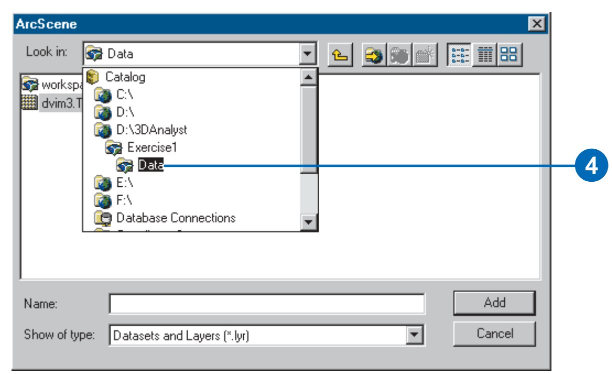
The TIN is drawn in the new scene.



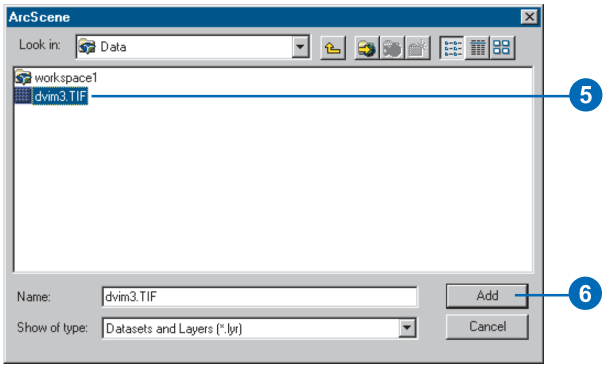
3. Click the Add Data button on the ArcScene Standard toolbar.



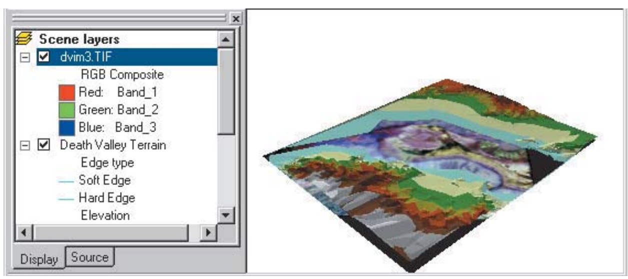
4. Navigate to the Data folder for Exercise1.



5. Click dvim3.TIF.
6. Click Add.

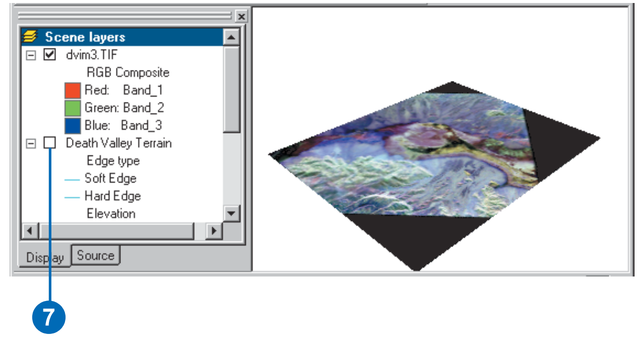


The image is added to the scene.



The image is drawn on a plane, with a base elevation value of zero. You can see it above the Death Valley terrain surface where the terrain is below 0 meters elevation (sea level); it is hidden by the terrain surface everywhere else.

7. Uncheck the Death Valley Terrain layer.



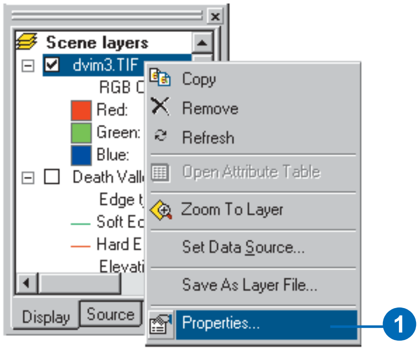
Now you can see the whole image. The black areas are parts of the image that contain no data and are a result of previous processing to fit the image to the terrain.

You have added the image to the scene. Now you will change the properties of the image layer so that the image will be draped over the terrain surface.

Draping the image

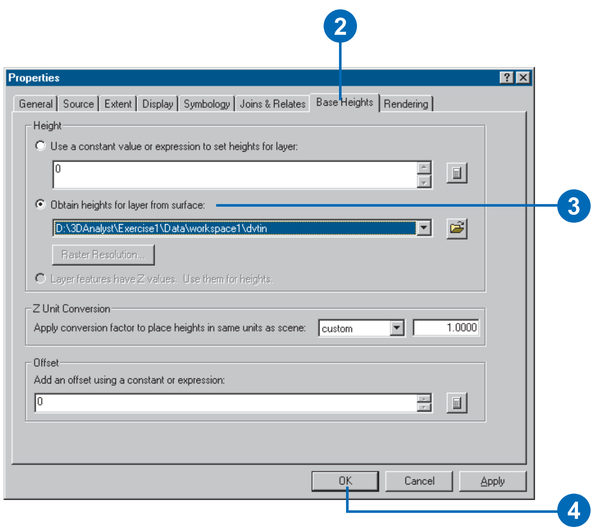
While the surface texture information shown in the image is a great source of information about the terrain, some relationships between the surface texture and the shape of the terrain will be apparent when you drape the image over the terrain surface. In ArcScene, you can drape a layer—containing a grid, image, or 2D features—over a surface (a grid or TIN) by assigning the base heights of the layer from the surface.

1. Right-click dvim3.TIF in the ArcScene table of contents and click Properties.



The layer Properties dialog box appears. You can change how a layer is drawn on a map or in a scene by setting its properties.

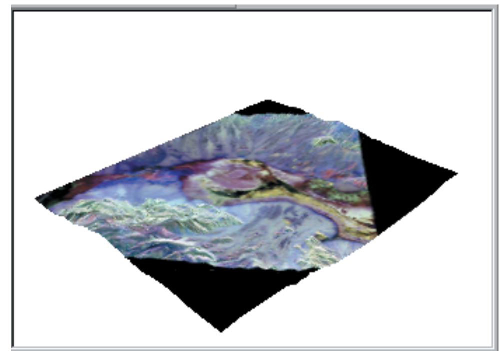
2. Click the Base Heights tab.



3. Click the option to Obtain heights for layer from surface.

   Because the TIN is the only surface model in the scene, it appears in the surface dropdown list.

4. Click OK.



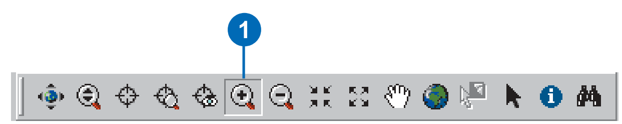
The image is draped over the terrain surface.

Now you will be able to navigate around the image and see the relationship between surface texture, as shown by the image colors, and the shape of the terrain.

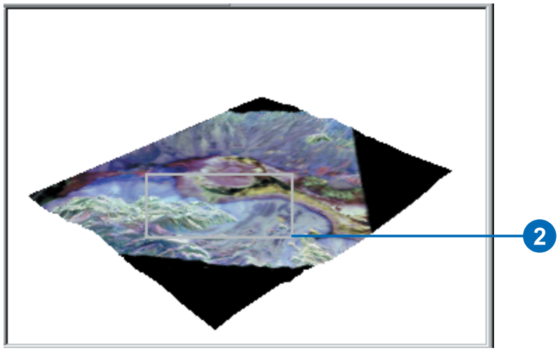
Exploring the image

You will use the navigation tools on the ArcScene Tools toolbar to explore the draped image.

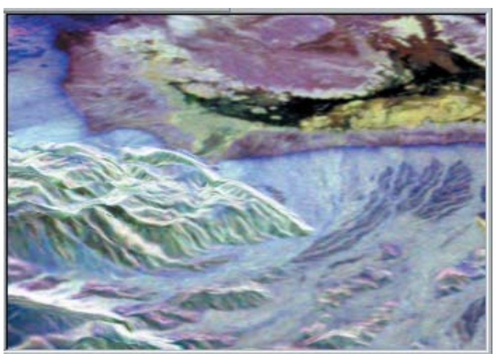
1. Click the Zoom in button.



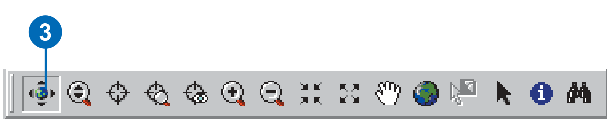
2. Click and drag a rectangle around the middle of the image.



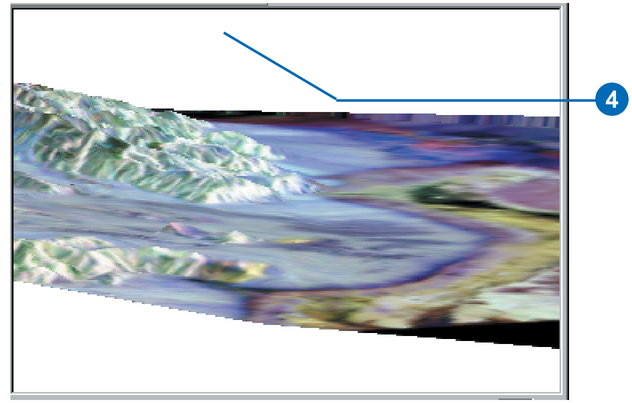
The scene zooms to the middle part of the image.



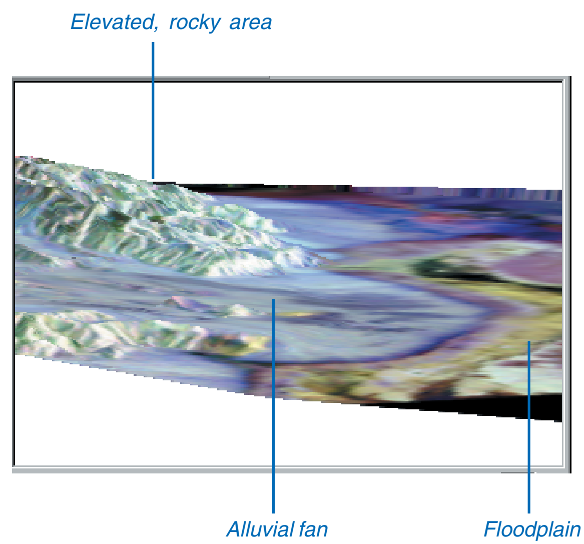
3. Click the Navigate button.



4. Click on the scene and slowly drag up and to the left.



The scene rotates, and the view angle lowers, so it looks as though you are looking down the valley, past the higher land on the left side of the scene.



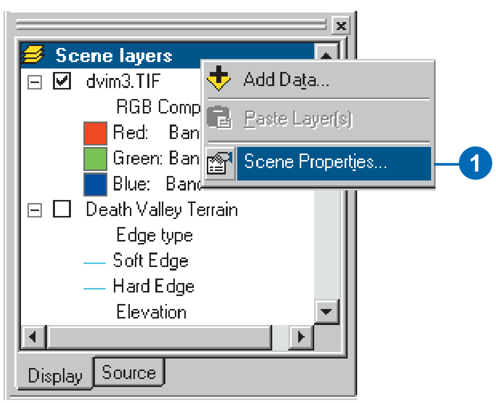
The elevated land is visibly rougher terrain than the flat valley bottom. The surface texture—and therefore the color, in the radar image—of this rocky area is different than the fine sediment of the floodplain—the yellow and black region in the valley bottom. The rocky area is also a different texture from the gently sloping alluvial fan that runs past it, down onto the valley floor.

Draping the radar image over the terrain surface allows you to see the relationship between the general shape of the land surface and the texture of the rocks and sediment that make up the surface.

Exaggerating the terrain

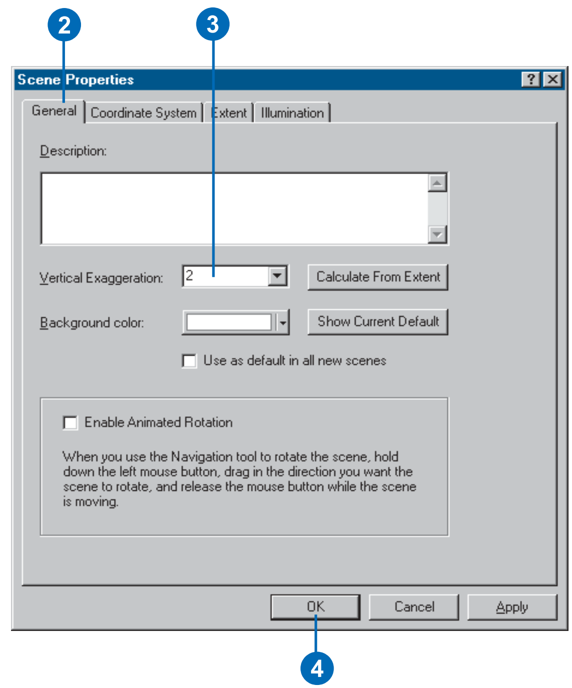
The valley is a broad area, relative to the height of the terrain, even though the mountains at the edge of the scene are more than 2,000 meters above the valley floor. In order to enhance the sense of depth in the scene, and to bring out subtle features in the terrain, you will exaggerate the height of the terrain.

1. Right-click Scene layers in the table of contents and click Scene Properties.



The Scene Properties dialog box lets you set properties that are shared by all of the layers in the scene. These include the vertical exaggeration, the background (sky) color, the coordinate system and extent of the data, and the way that the scene is illuminated (the position of the light source relative to the surface).

2. Click the General tab.

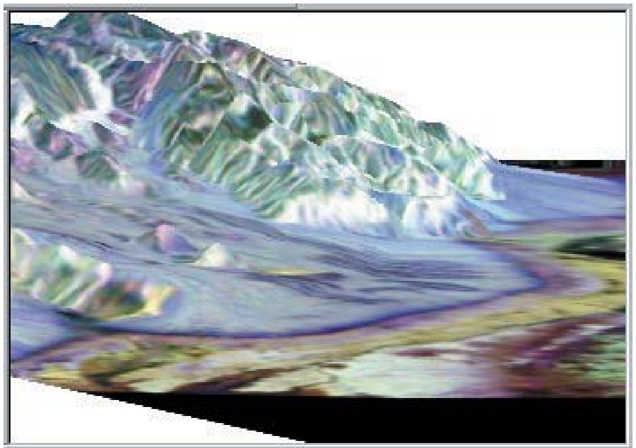


3. Type "2" in the Vertical Exaggeration combo box.

4. Click OK.

The apparent height of the terrain is now doubled.

You can now clearly see how the alluvial fan spreads out onto the valley floor, between the larger rocky area at the center of the scene and the smaller rocky area in the foreground at the left side of the scene.

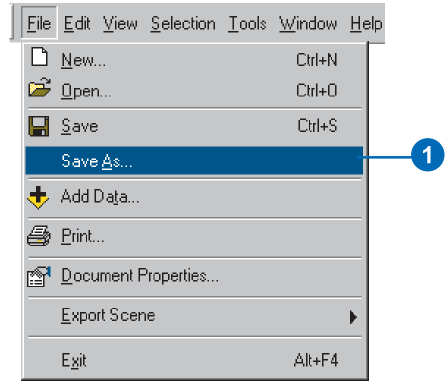


You have added depth to the radar image, explored the general relationship between the data in the image and the terrain data, and enhanced the scene so that you can perceive more subtle variations in the terrain.

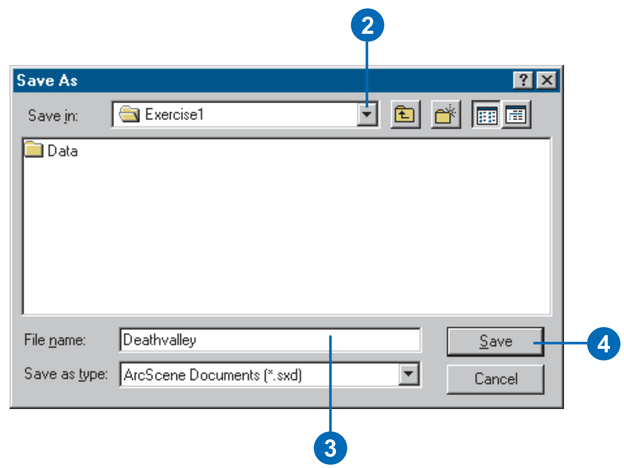
Now that you've built the scene, you will save it so that you can explore it later if you choose.

Saving the scene

Scenes, also called Scene Documents, are like maps. They contain information about how the layers that are in the scene should be rendered and where the data is located.



1. Click File and click Save As.



2. Navigate to the Exercise1 folder.
3. Type "Deathvalley".
4. Click Save.

The scene will now be available for you to open later.

Exercise 2: Visualizing contamination in an aquifer

Imagine that you work for a water district. The district is aware of some areas where volatile organic compounds (VOCs) have leaked over the years. Scientists from your department have mapped some plumes of VOCs in the aquifer, and you want to create a 3D scene to help officials and the public visualize the extent of the problem.

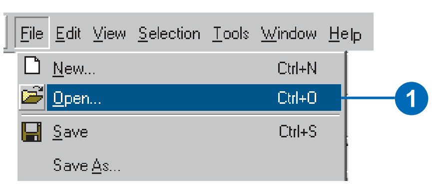
Some of the data for the scene has already been assembled in the Groundwater scene. You will modify the scene to better communicate the problem.

VOC data was supplied courtesy of the San Gabriel Basin Water Quality Authority.

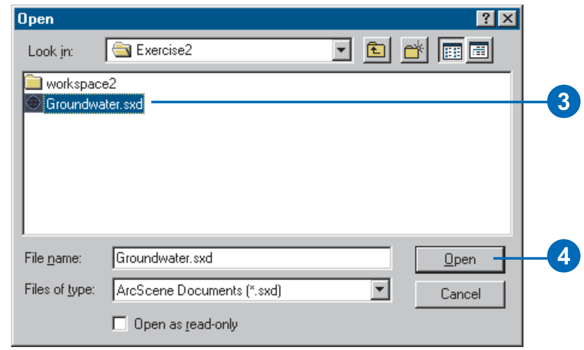
Opening the Groundwater scene document

This scene document contains a TIN that shows the shape of the contaminant plume, a raster that shows the concentration of the contaminant, and two shapefiles that show the locations of parcels and wells. You will drape the concentration raster over the plume TIN, extrude the building features and change their color, and extrude the well features so that the wells that are most endangered by the contamination may be more easily recognized.

1. In ArcScene, click File, then click Open.



2. Navigate to the Exercise2 folder.
3. Click Groundwater.sxd.



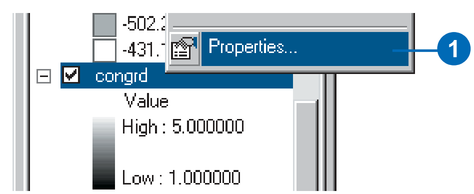
4. Click Open.

   The Groundwater scene opens. You can see the four layers in the table of contents.

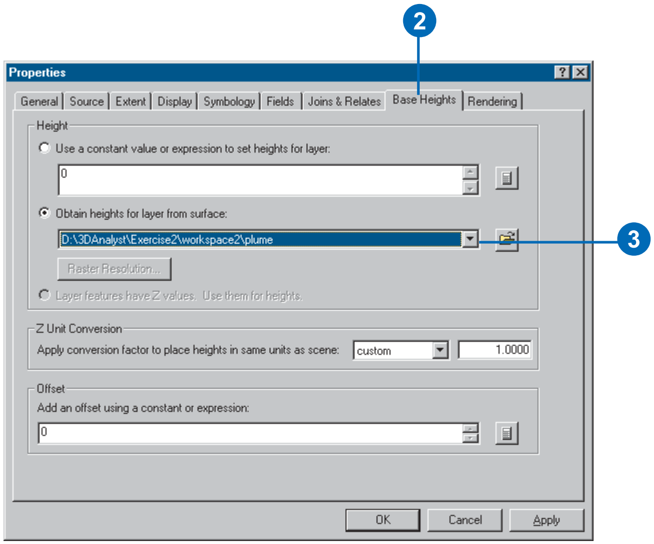
Showing the volume and intensity of contamination

You'll drape the raster of VOC concentration over the TIN of the contaminant plume surface to show the volume and intensity of contamination in the aquifer.

1. Right-click congrd and click Properties.



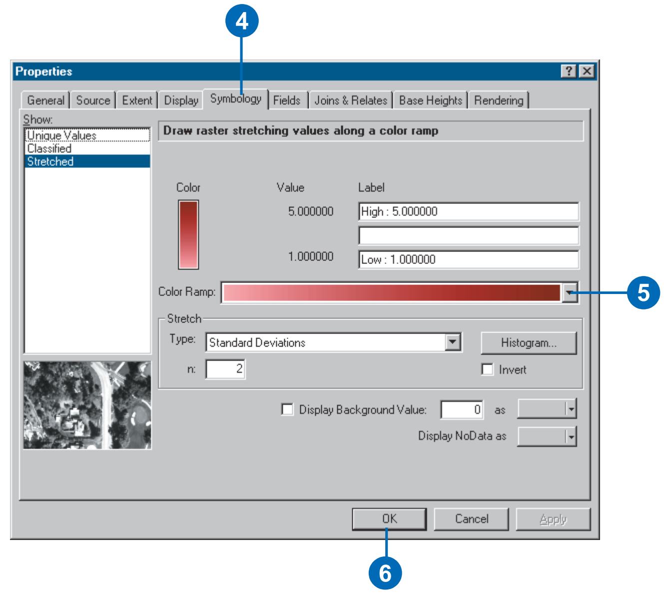
2. Click the Base Heights tab.



3. Click the dropdown arrow and click plume to get the heights from the plume TIN.

Now you will change the symbology of the raster to show the intensity of the contamination.

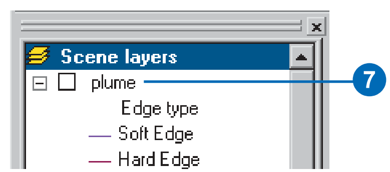
4. Click the Symbology tab.



5. Click the Color Ramp dropdown arrow and click a red color ramp for the raster.

6. Click OK.

7. In the table of contents, uncheck plume.



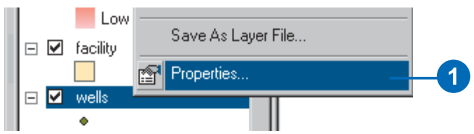
Now it is possible to see the shape of the plume and its intensity in 3D.

Showing the relationship of the plume to wells

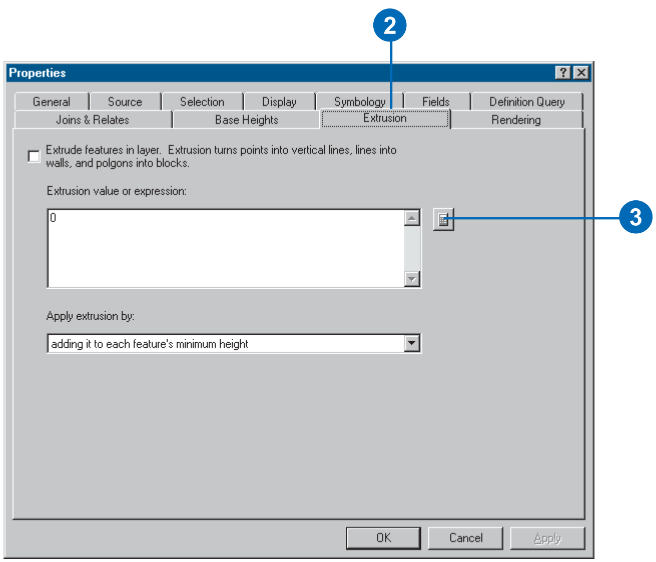
You can see that some of the wells are within the area of the plume. However, it is difficult to see which wells are most seriously affected because the contamination is more widespread but less concentrated at greater depths.

You will extrude the well features based on their depth attribute in order to see which wells intersect the plume.

1. Right-click wells and click Properties.



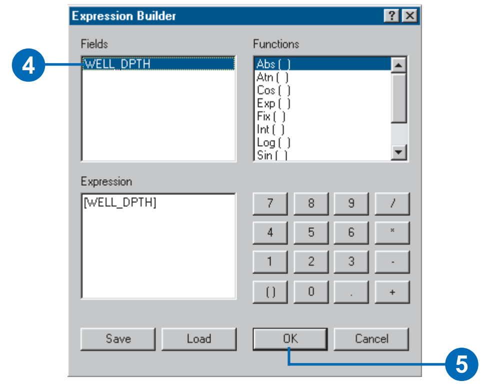
2. Click the Extrusion tab.



3. Click the Calculate Extrusion Expression button.

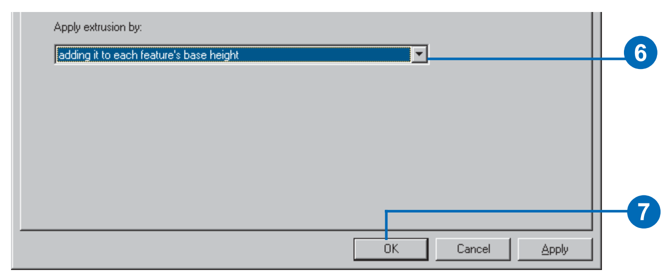
You will display the well points as vertical lines equal to the depth of the well. This information is stored in the WELL_DPTH field.

4. Click WELL_DPTH.



5. Click OK.

6. Click the dropdown arrow to apply the extrusion expression by adding it to each feature's base height. The well depths are expressed as negative values, so they'll be extruded downward.



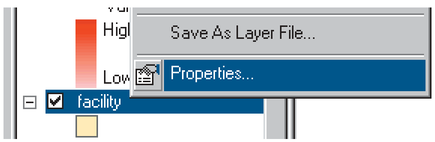
7. Click OK.

You can see the places where the wells intersect, or are close to, the plume. Now you will modify the scene to show the priority of various facilities that have been targeted for cleanup.

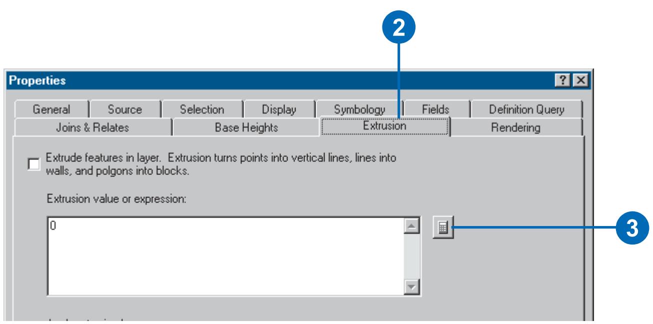
Showing the facilities with a high cleanup priority

Analysts in your department have ranked the facilities according to the urgency of a cleanup at each location. You'll extrude the facilities into 3D columns and color code them to emphasize those with a higher priority for cleanup.

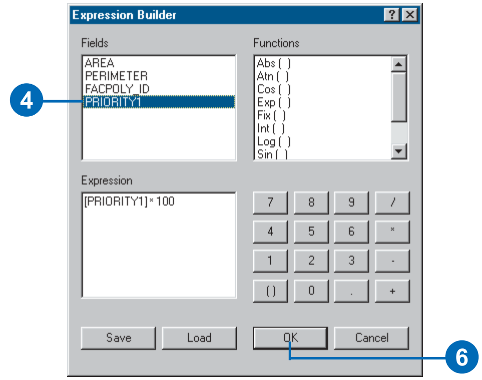
1. Right-click facility and click Properties.
2. Click the Extrusion tab.



3. Click the Calculate Extrusion Expression button.

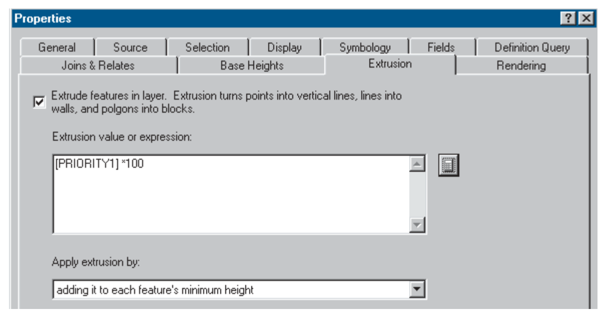


4. Click PRIORITY1.
5. Type "* 100".

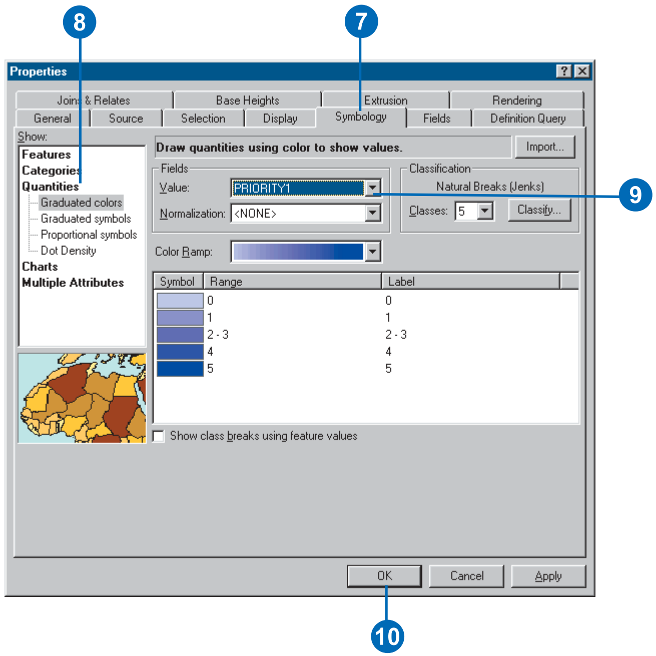


6. Click OK.

The expression you created appears in the Extrusion value or expression box.



7. Click the Symbology tab.
8. Click Quantities.
9. Click the Value dropdown list and click PRIORITY1.



10. Click OK.

The facilities are now extruded in proportion to their priority score. The scene now shows the shape and intensity of the contamination, the wells in relationship to the plume, and the facilities that need to be cleaned up in order to prevent further pollution of the groundwater.

Now you'll save your changes to the scene.

11. Click the Save button.

Exercise 3: Visualizing soil contamination and thyroid cancer rates

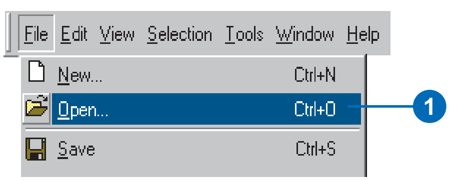
In 1986, after the catastrophic accident at the Chernobyl nuclear power plant in Ukraine, a large amount of radioactive dust fell on Belarus. Since then, scientists have studied the aftermath of the accident. One tool for exploring the data is 3D visualization. In this exercise, you will create two surfaces from point data collected in Belarus. One set of points contains measurements of soil CS137 concentrations. CS137 is one of several radioactive isotopes released by the accident. The other set of points shows the rates of thyroid cancer, aggregated by district, with the sample point placed near the district centers.

The CS137 contamination and thyroid cancer data was supplied courtesy of the International Sakharov Environmental University.

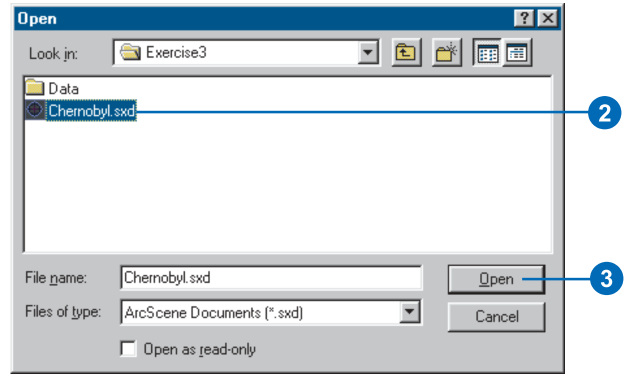
Viewing the point data

First, you will open the Chernobyl scene and view the point data.

1. Click File and click Open.



2. Navigate to Exercise3 and click Chernobyl.
3. Click Open.



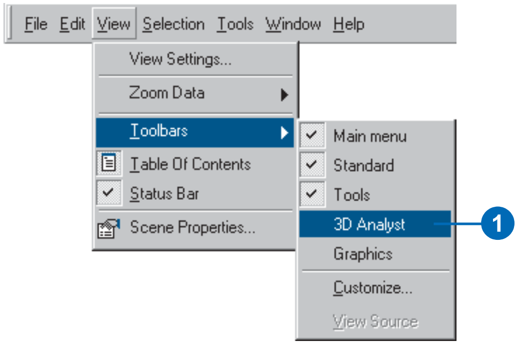
The CS137 soil measurements are shown with small point symbols, using a graduated color ramp to show the intensity of the contamination. The districts' thyroid cancer rates are shown with larger symbols, using a different color ramp.

Creating 3D point features

The soil CS137 samples are 2D points with some attributes. One way to view 2D points in 3D is by setting an extrusion expression, or a base height. You can also incorporate a z-value into a feature's geometry to allow it to be directly viewed in 3D without the need to set a base height from a surface or an attribute.

First, you'll add the 3D Analyst toolbar to ArcScene.

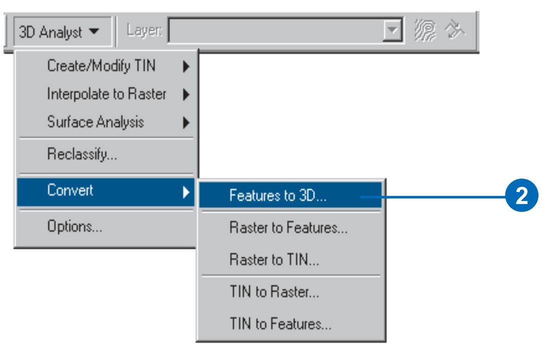
1. Click View, point to Toolbars, and click 3D Analyst.



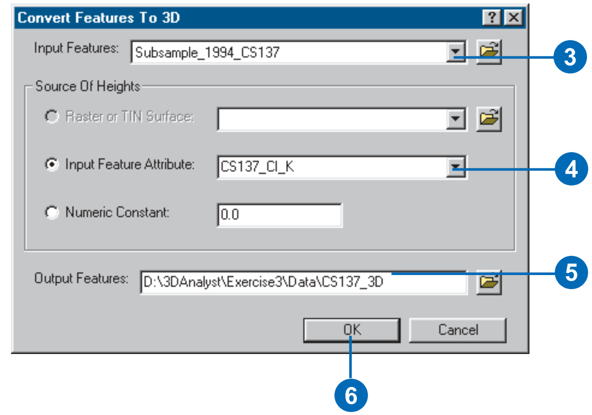
The 3D Analyst toolbar in ArcScene contains several 3D analysis and data conversion tools. The ArcMap 3D Analyst toolbar contains the same tools, plus several additional tools that you can use in ArcMap.

Now you will create 3D point features from the soil CS137 points.

2. Click 3D Analyst, point to Convert, and click Features to 3D.



3. Click the Input Features dropdown list and click Subsample_1994_CS137.



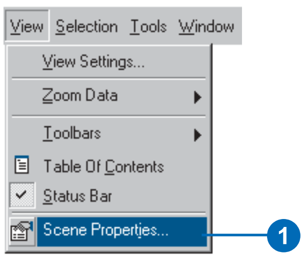
4. Click the Input Feature Attribute button, then click the Input Feature Attribute dropdown list and click CS137_CI_K.
5. Change the output feature name to CS137_3D.
6. Click OK.

The features are converted to 3D point features. However, they still seem to be resting on a flat plane because the CS137 concentration values range from 0 to 208.68, which is small relative to the horizontal extent of the data.

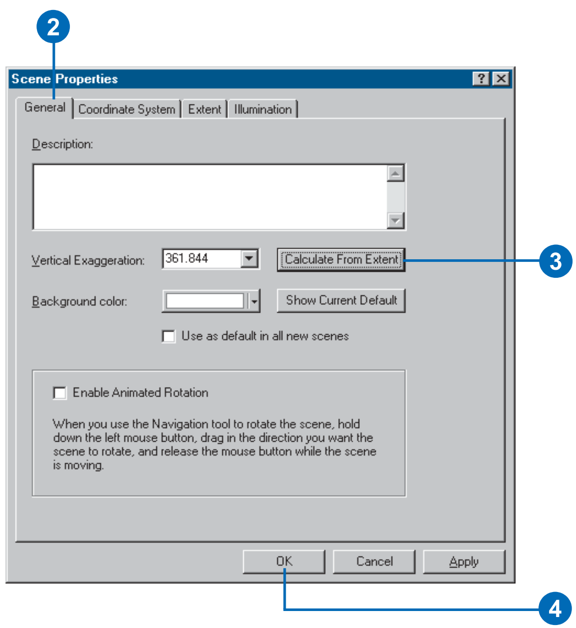
Increasing the vertical exaggeration

You will exaggerate the scene to show the new points with their height embedded in the feature geometry.

1. Click View and click Scene Properties.



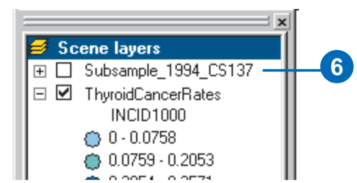
2. Click the General tab.



3. Click Calculate From Extent.
4. Click OK.
5. Click the Full Extent button.

Now that you can see the new 3D points in the scene, you can turn off the original CS137 sample point layer.

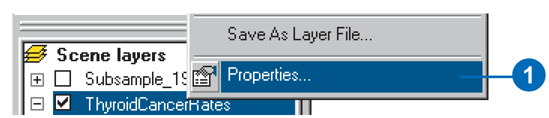
6. Uncheck the box in the table of contents beside Subsample_1994_CS137 and click the minus sign beside the box to hide the classification.



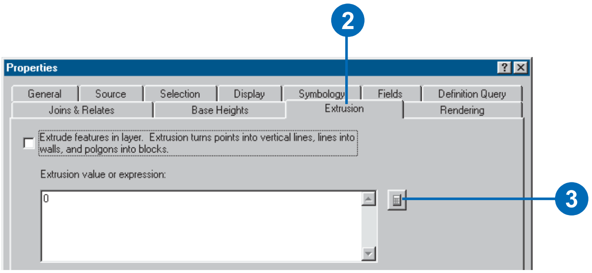
Extruding columns

Viewing points in 3D space is one way to investigate data. Another way is to extrude points into columns. You will extrude the thyroid cancer points into columns to compare them to the contamination data.

1. Right-click ThyroidCancerRates and click Properties.

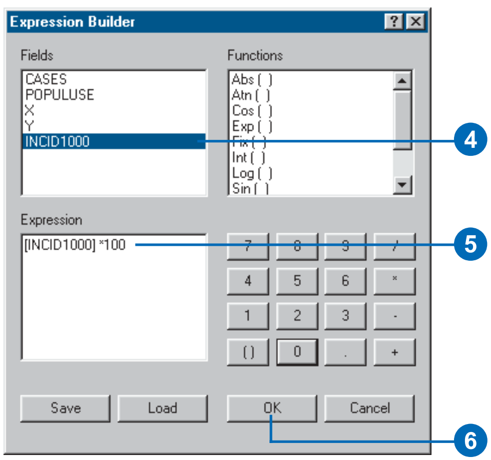


2. Click the Extrusion tab.



3. Click the Calculate Extrusion Expression button.

4. Click INCID1000 (the rate of cases per 1,000 persons).



Because the z-values of the phenomena that you are comparing have different ranges, you will multiply the cancer rate by 100 to bring the values into a range similar to that of the CS137 measurements.

5. Type "* 100".

6. Click OK on the Expression Builder dialog box.

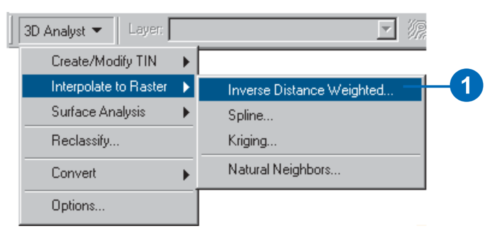
7. Click OK on the Layer Properties dialog box.

   Now the district centroid points are shown with columns proportionate to the thyroid cancer rates. If you navigate the scene you will see that the areas with the highest contamination levels also tend to have high thyroid cancer rates, although there are areas with lower CS137 contamination levels that also have high cancer rates.

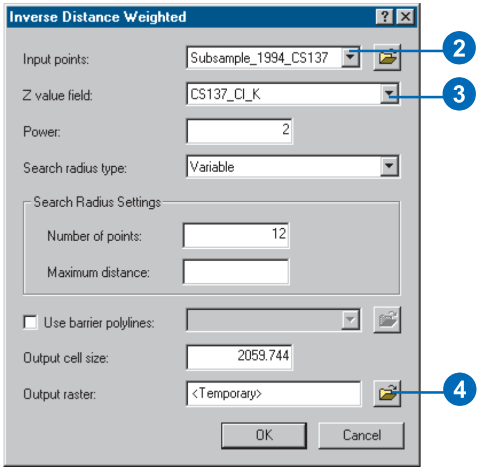
Creating a surface from point sample data

You know what the soil concentrations of CS137 are at the sample point locations, but you do not know what they are at the locations between sample points. One way to derive the information for locations between sample points is to interpolate a raster surface from the point data. There are many ways to interpolate such surfaces, which result in different models of varying accuracy. In this exercise you will interpolate a surface from the samples using the Inverse Distance Weighted (IDW) interpolation technique. IDW interpolation calculates a value for each cell in the output raster from the values of the data points, with closer points given more influence and distant points less influence.

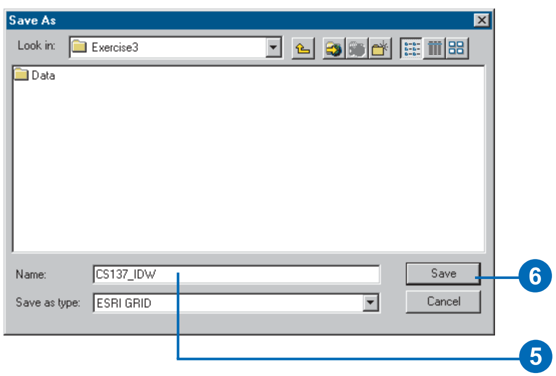
1. Click 3D Analyst, point to Interpolate to Raster, and click Inverse Distance Weighted.



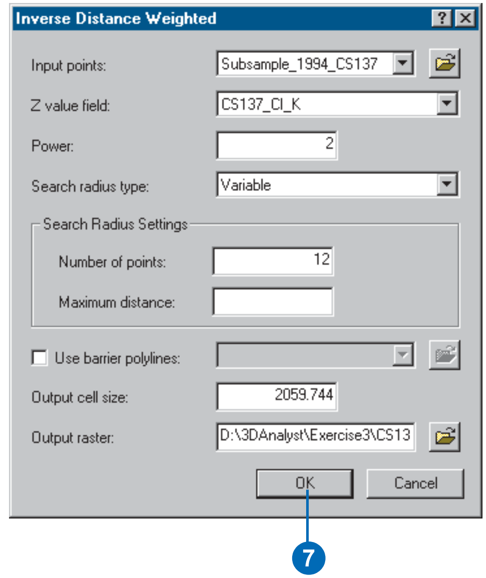
2. Click the Input points dropdown list and click Subsample_1994_CS137.



3. Click the Z value field dropdown list and click CS137_CI_K.
4. Click the Browse button.
5. Navigate to the Exercise3 folder and type “CS137_IDW” in the Name field.



6. Click Save.
7. Click OK.

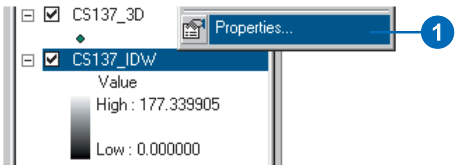


ArcScene interpolates the surface and adds it to the scene.

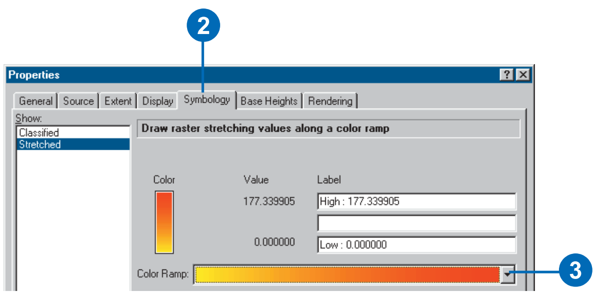
Viewing the interpolated surface

Now that the surface has been added to the scene, you can see that there are two areas with very high concentrations of CS137. You will view the surface in perspective, with a new color ramp, to better see its shape.

1. Right-click CS137_IDW and click Properties.

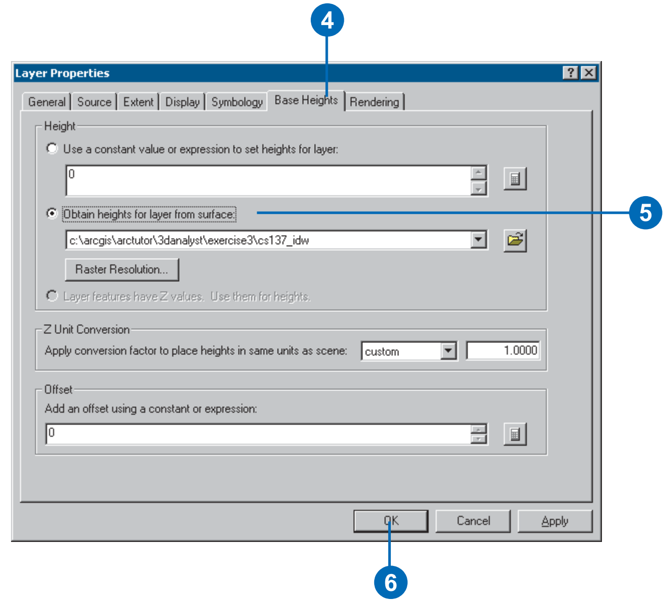


2. Click the Symbology tab.

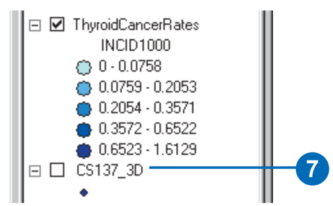


3. Click the Color Ramp dropdown arrow and click a new color ramp.

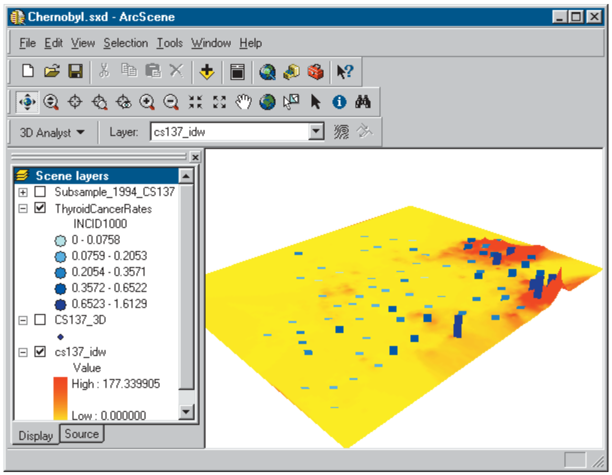
4. Click the Base Heights tab.



5. Click Obtain heights for layer from surface.
6. Click OK.
7. Uncheck CS137_3D in the table of contents.



Now you can see the interpolated surface of CS137 contamination, along with the thyroid cancer rate data.

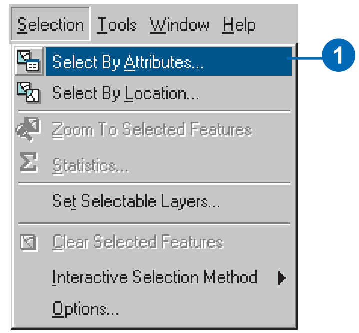


Next, you will select the province centers with the highest rates of thyroid cancer.

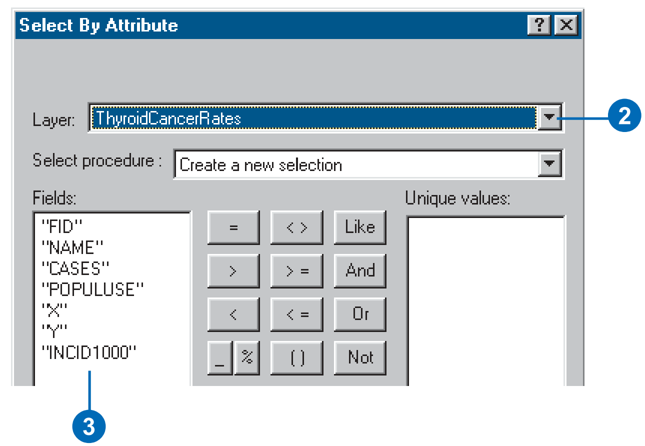
Selecting features by an attribute

Sometimes it is important to focus on a specific set of data or specific features. You can select features in a scene by their location, by their attributes, or by clicking them with the Select Features tool. You will select the province centers by attribute to find the locations with the highest rates.

1. Click Selection and click Select By Attributes.

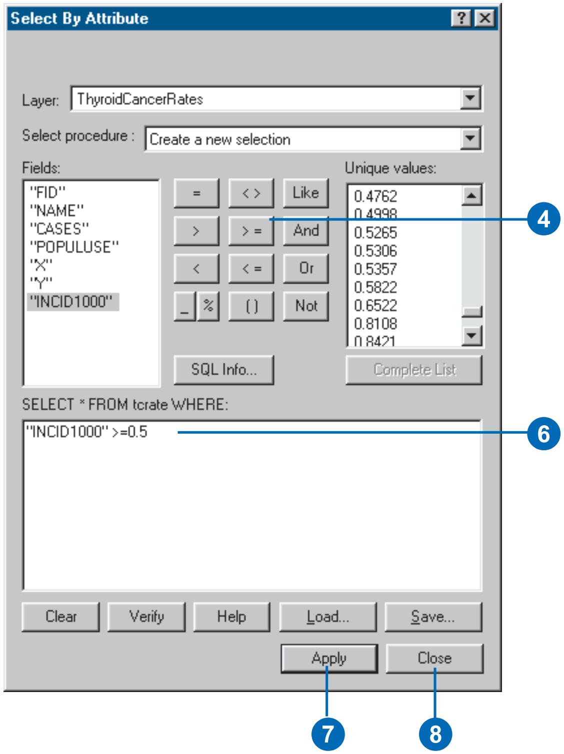


2. Click the Layer dropdown arrow and click ThyroidCancerRates.



3. Double-click INCID1000 in the Fields list.

4. Click the >= button.



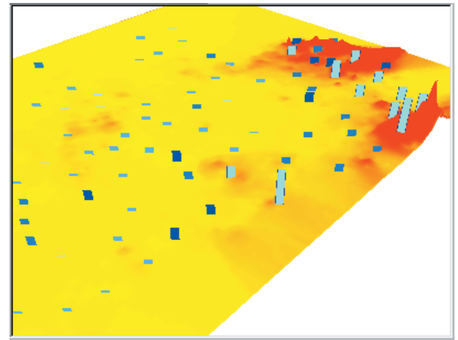
5. Type "0.5".

6. Check the selection expression you've built.

7. Click Apply.

8. Click Close.

The province centers with thyroid cancer rates greater than 0.5 cases per 1,000 are now selected in the scene.

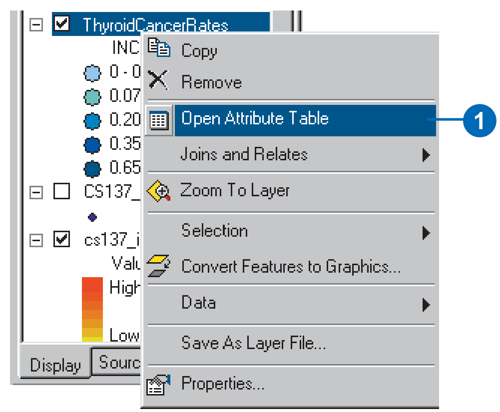


They are drawn in light blue to indicate that they are selected.

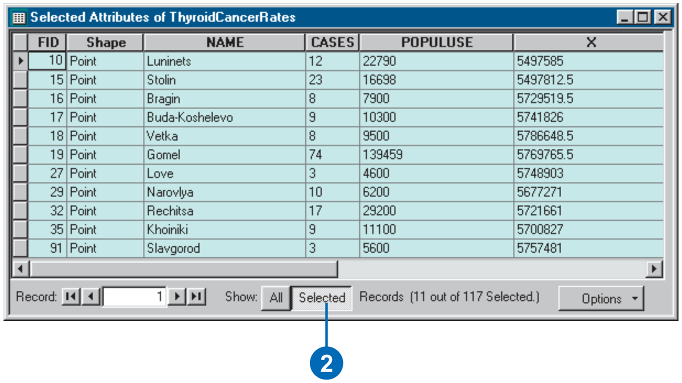
Viewing the attributes of features

You will investigate attributes of the selected locations and find out how many cases of thyroid cancer occurred in these districts.

1. Right-click ThyroidCancerRates and click Open Attribute Table.

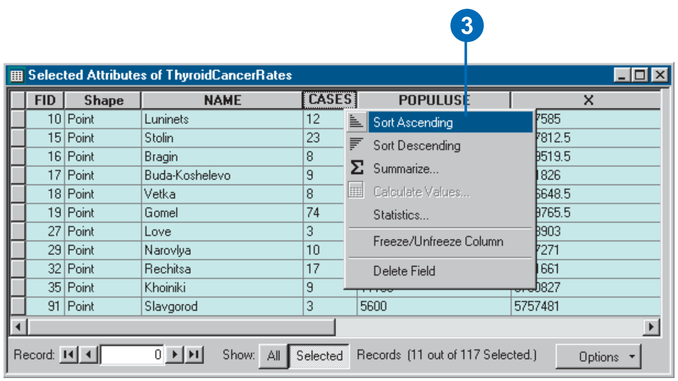


2. Click the Selected button.



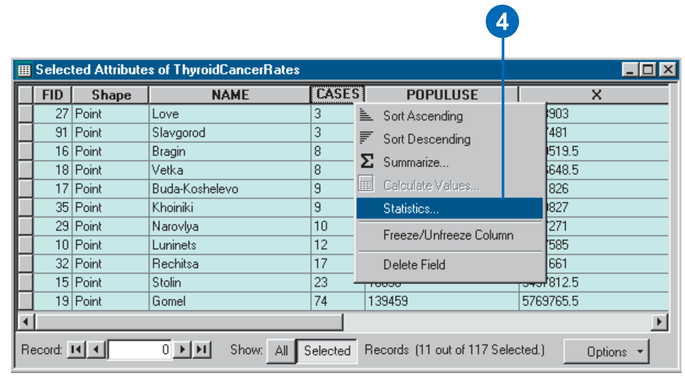
The table now shows only those features that you selected.

3. Right-click CASES and click Sort Ascending.



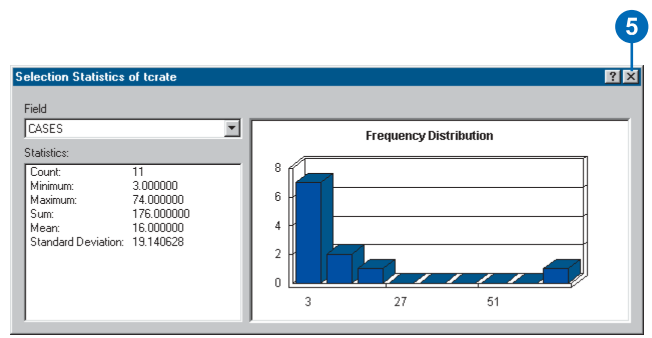
The selected province centers are sorted according to the number of cases.

4. Right-click CASES and click Statistics.

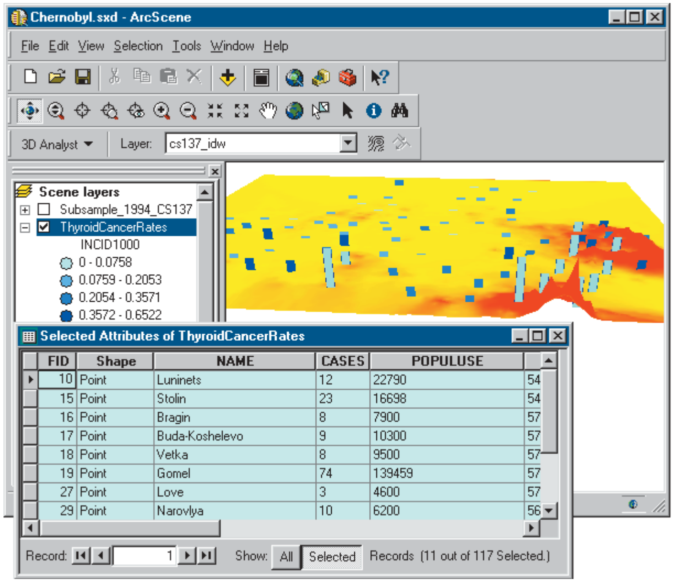


The total number of cases in the selected set of 11 province centers is 176.

5. Close the Selection Statistics dialog box.



6. Click the Navigate button and click on the scene.



You can work in ArcScene while the attribute table is open.

7. Click the Save button.

In this exercise you have created 3D features, extruded point features, and interpolated a raster surface from a set of data points. You've compared the extruded vector data to the surface data and explored the attributes of the vector data.

Exercise 4: Building a TIN to represent terrain

The town of Horse Cave, Kentucky, is situated above a cave that once served as the source of drinking water and hydroelectric power for the town. Unfortunately, the groundwater that flows in the cave was polluted by household and industrial waste dumped on the surface and washed into sinkholes. Dye tracing studies and a three-dimensional survey of the cave revealed the relationship between the cave passages and the town and demonstrated the connection between open surface dump sites and contamination of the groundwater in the cave below.

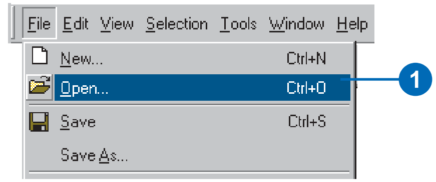
Thanks to the development in 1989 of a new regional sewage facility and the joint efforts of the Cave Research Foundation and the American Cave Conservation Association (ACCA), the groundwater is cleaner, and the cave has been restored. It is now operated as a tour cave and educational site by the ACCA.

Cave data was provided courtesy of the ACCA.

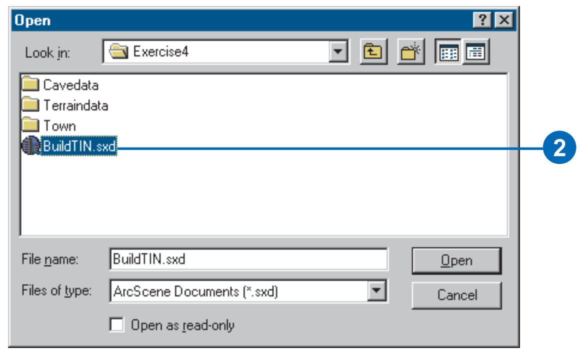
Viewing the cave and the landscape

First you will open the BuildTIN scene and view the cave survey and some terrain data layers. You'll use this terrain data to create a TIN and drape some other layers on it to visualize the relationship of the cave to the town.

1. Click File and click Open.

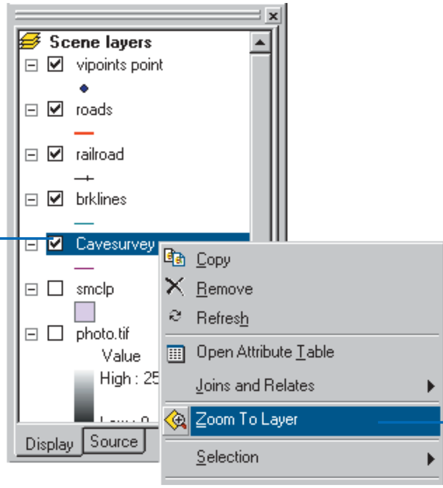


2. Navigate to the Exercise4 folder and double-click BuildTIN.sxd.



The scene opens, and you can see the location of roads and railroads, some sample elevation points, and a few significant contour lines. In the table of contents, you can see that some layers have been turned off.

3. Check the box to show the Cavesurvey layer.



4. Right-click Cavesurvey and click Zoom To Layer.

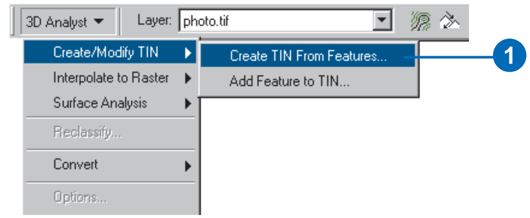
The cave survey data consists of PolylineZ features, which are automatically drawn in 3D because they have z-values embedded in their geometry. They appear above the rest of the data because all of the other layers are drawn with the default elevation of 0.

In the next steps you will build a TIN to provide the base heights for the streets and a photo of the town.

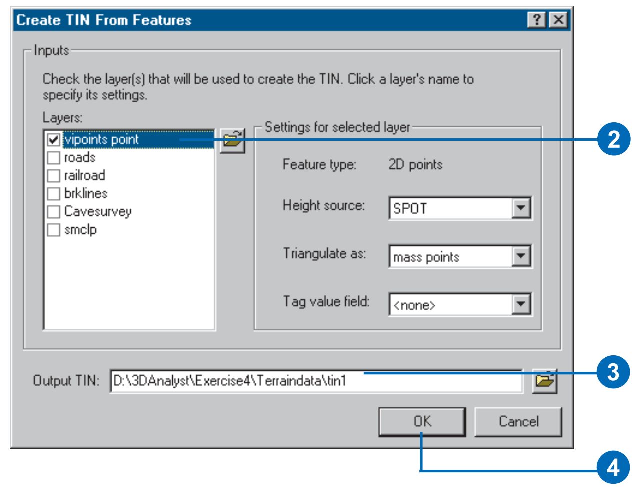
Creating a TIN from point data

You have a point layer called vipoints point. This coverage consists of points with an attribute called SPOT that contains elevation values taken at these points. You'll create the TIN surface model from these points.

1. Click 3D Analyst, point to Create/Modify TIN, and click Create TIN From Features.



2. Check vipoints point.

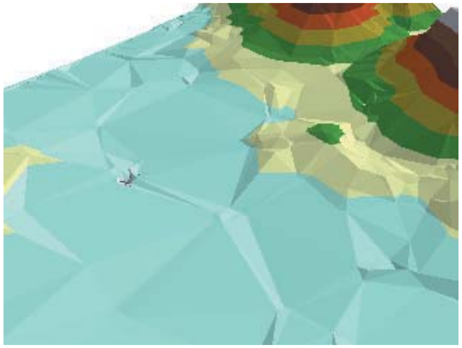


The SPOT field name appears in the Height source dropdown list, and the layer will be triangulated as mass points.

3. Change the default path so that the new TIN will be created in the Exercise 4\Terraindata\ folder.

4. Click OK.

The TIN is created and added to the scene. Note that it is drawn above the Cavesurvey layer; the elevation values in the TIN define its base height.

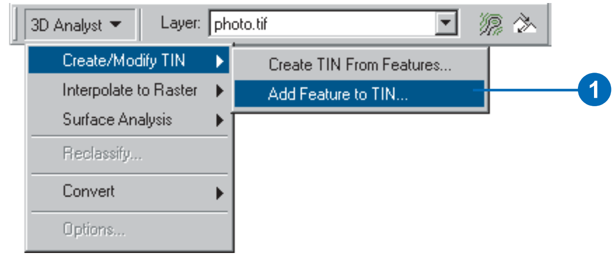


While this TIN is a fairly good model of the surface, you can make it more accurate by adding some more features.

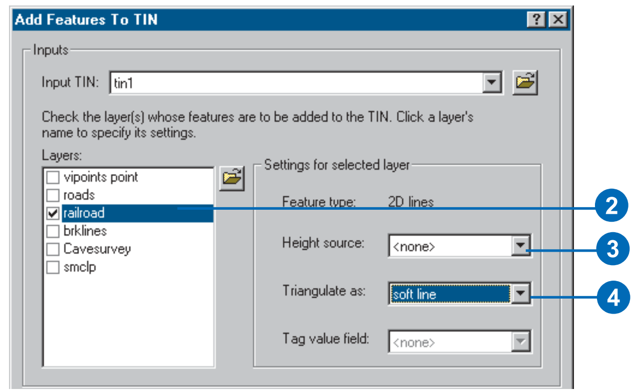
Adding features to a TIN

Now you will add hard and soft breaklines and a clip polygon to the TIN. You'll add the railroad features as soft breaklines, so they'll be represented on the surface but won't influence the shape of the surface. You'll add the brklines features as hard breaklines with elevation values to refine the shape of the surface in areas that you're most interested in. Finally, you will add the smclp polygon as a soft clip polygon to more smoothly define the edge of the TIN.

1. Click 3D Analyst, point to Create/Modify TIN, and click Add Feature to TIN.

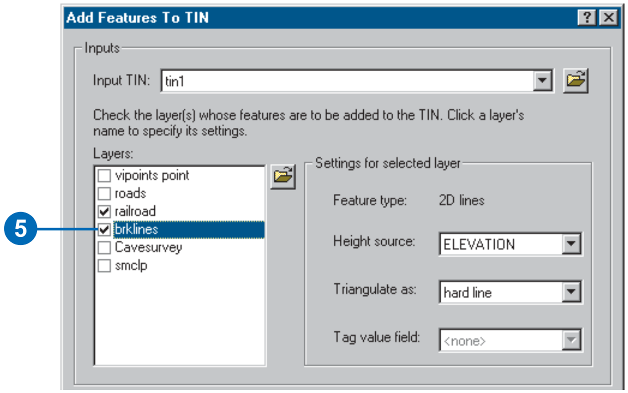


2. Check railroad.



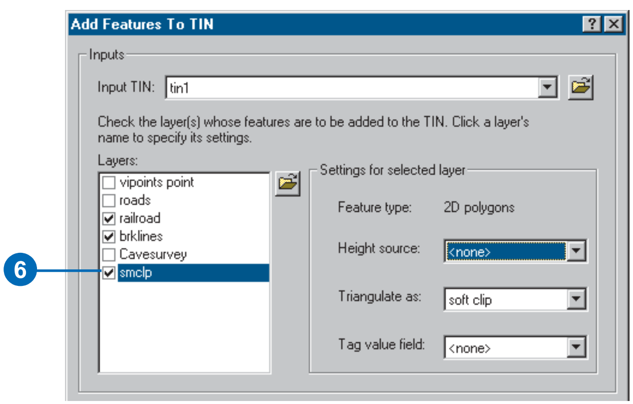
3. Click the Height source dropdown arrow and click <none>.
4. Click the Triangulate as dropdown arrow and click soft line.

5. Check brklines.



The Add Features To TIN tool detects that there is an ELEVATION field and uses it for the height source. You will accept the default and triangulate them as hard breaklines.

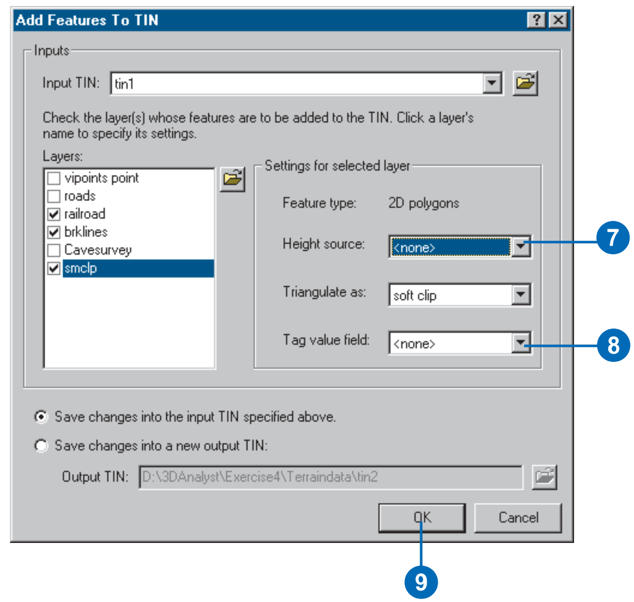
6. Check smclp.



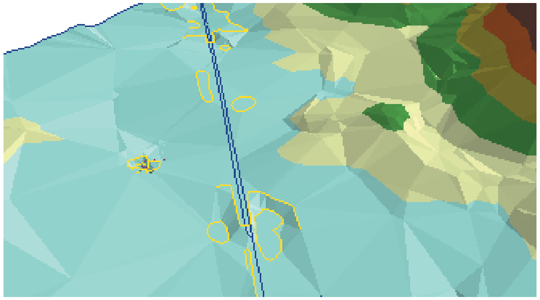
7. Click the Height source dropdown arrow and click <none>.

8. Click the Tag value field dropdown arrow and click <none>.

You have defined the feature layers that you want to add to your TIN and specified how they should be integrated into the triangulation.



9. Click OK.



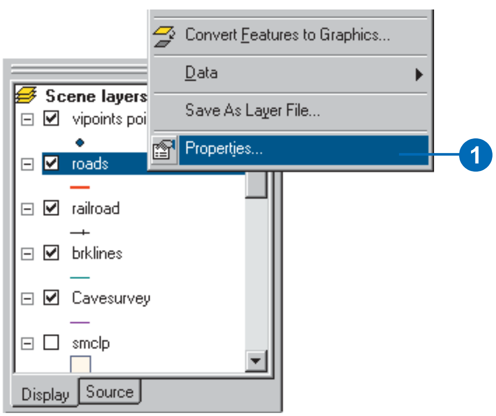
The new features are added to the TIN.

You can see that the railroad follows a bed that has been leveled somewhat relative to the surface.

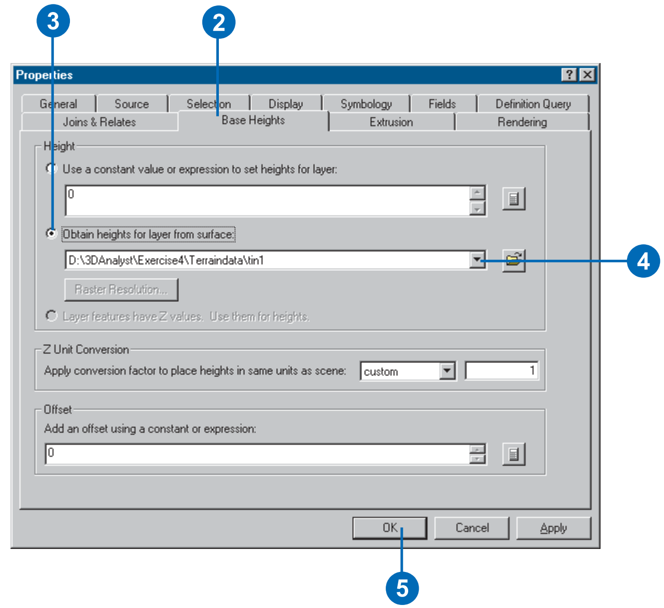
Setting features' base heights from the TIN

Now you will set the base heights for the road and railroad features from the new TIN.

1. Right-click roads and click Properties.



2. Click the Base Heights tab.

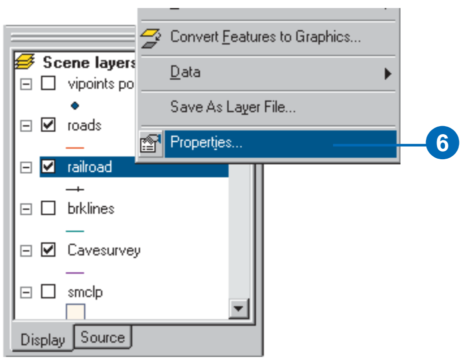


3. Click Obtain heights for layer from surface.
4. Click the dropdown arrow and click tin1.
5. Click OK.

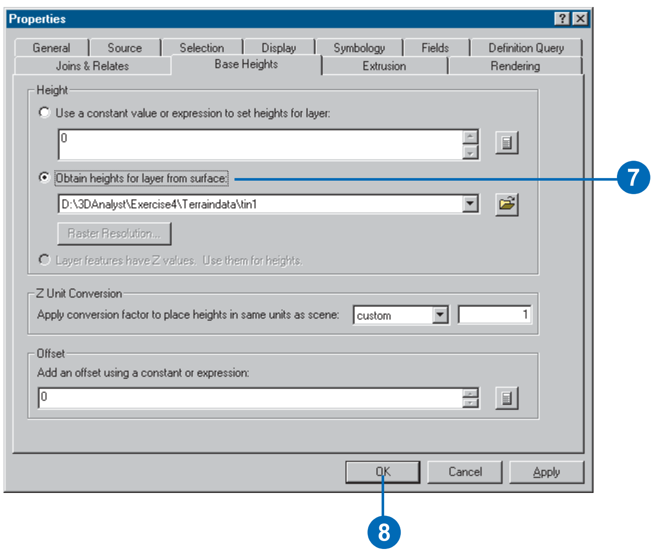
   The road features are now draped over the TIN surface that you created. Now you will drape the railroad features over the surface.

6. Right-click railroad and click Properties.

7. Click Obtain heights for layer from surface.



8. Click OK.

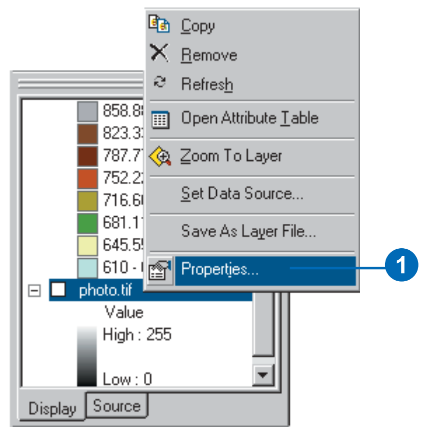


The railroad features are now draped over the TIN surface that you created. Next you'll drape the aerial photo over the TIN.

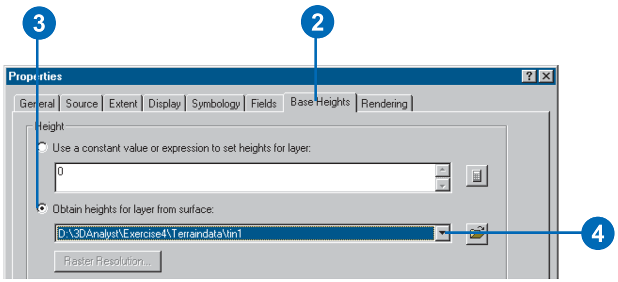
Setting raster base heights from the TIN

Including the aerial photo of the town in the scene makes the relationship between the cave and the town much more evident. You'll drape the raster over the TIN and make it partly transparent so that you'll be able to see the cave beneath the surface.

1. Right-click photo.tif and click Properties.



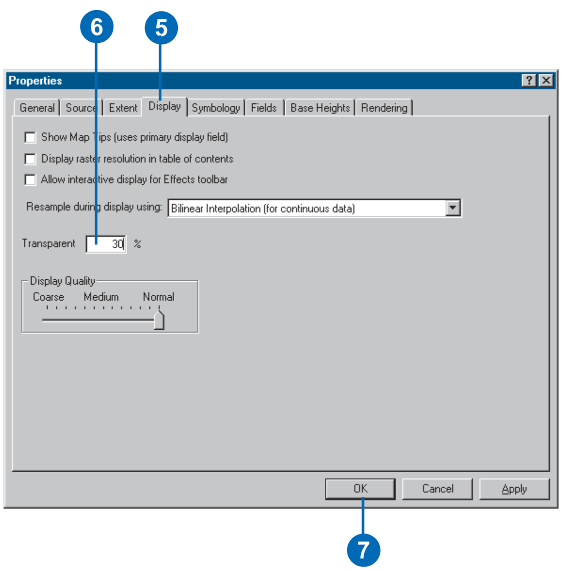
2. Click the Base Heights tab.



3. Click Obtain heights for layer from surface.

4. Click the dropdown arrow and click tin1.

5. Click the Display tab.



6. Type “30” in the Transparent text box.

7. Click OK.

8. Check photo.tif.

Now the aerial photo is 30 percent transparent. You can see large patches of the TIN over the photo because the TIN and the photo have the same drawing priority. If you wanted the TIN to be visible below the photo, you could change its drawing priority to 10 (lowest) on the Rendering tab of the TIN’s Layer Properties dialog box. You could also offset the base height of the TIN or the photo by a small amount.

Cleaning up the scene

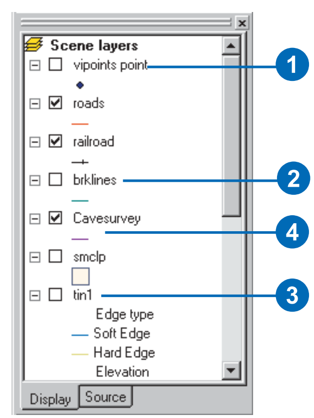
To clean up the scene you’ll turn off some layers that are no longer needed and make the cave line symbol larger.

1. Uncheck vipoints point.

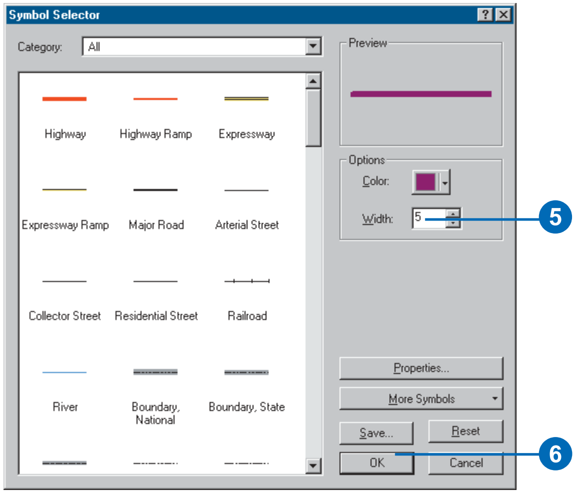
2. Uncheck brklines.

3. Uncheck tin1.

4. Double-click the line symbol for the Cavesurvey layer.



5. Type “5” in the Width box.



6. Click OK.

Now you can see the three-dimensional passages of the cave, symbolized by thick lines. The surface features and the aerial photo provide context, so you can easily see the relationship of the cave to the town.

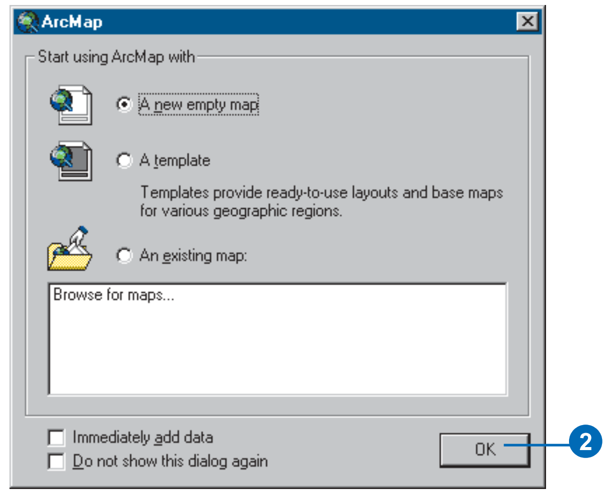
Creating a profile of the terrain

The cave follows the valley floor orientation. To get an understanding of the shape of the valley, you will create a profile across the TIN. In order to create a profile, you must first have a 3D line (feature or graphic). You will start ArcMap, copy the TIN to the map, and digitize a line to make your profile.

1. Click the Launch ArcMap button.

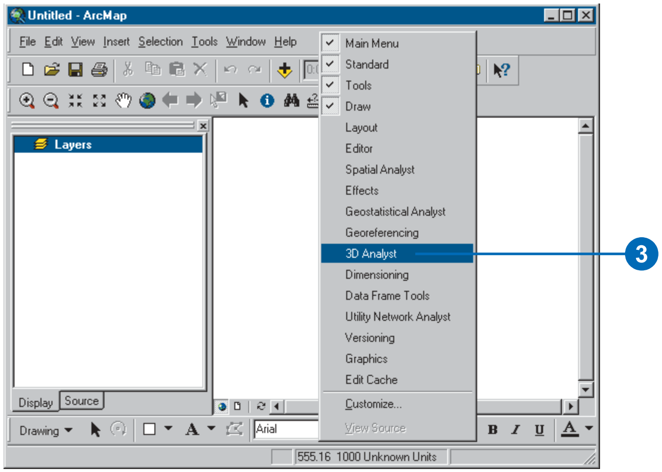
ArcMap starts.

2. Click OK.



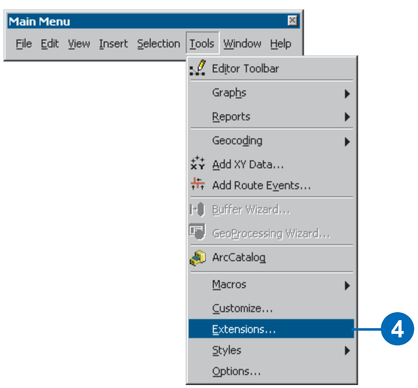
Now you will add the 3D Analyst toolbar to ArcMap. The ArcMap 3D Analyst toolbar contains a few tools that do not appear on the ArcScene 3D Analyst toolbar. Two of these are the Interpolate Line tool and the Create Profile Graph tool, which you will use to create your profile of the surface.

3. Right-click on one of the ArcMap toolbars and click 3D Analyst.



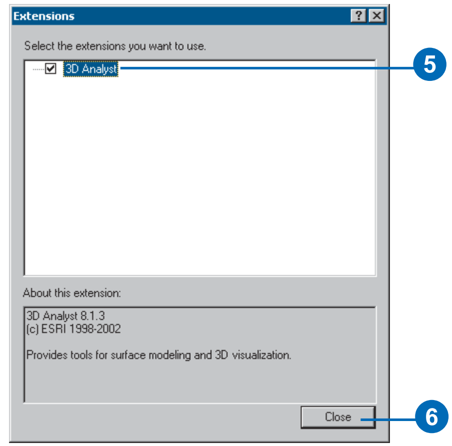
The 3D Analyst toolbar appears.

4. Click Tools and click Extensions.



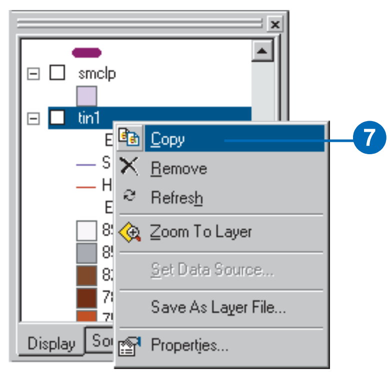
5. Check 3D Analyst.

6. Click Close.

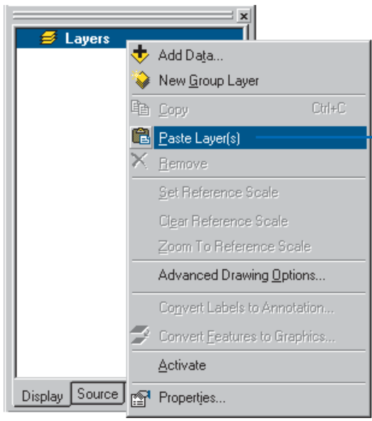


The 3D Analyst extension is enabled.

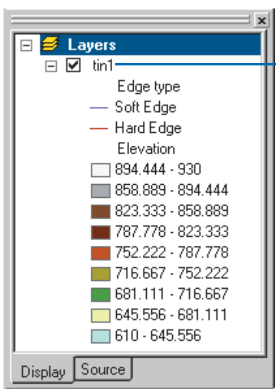
7. In the ArcScene table of contents, right-click tin1 and click Copy.



8. In the ArcMap table of contents, right-click Layers and click Paste Layer(s).

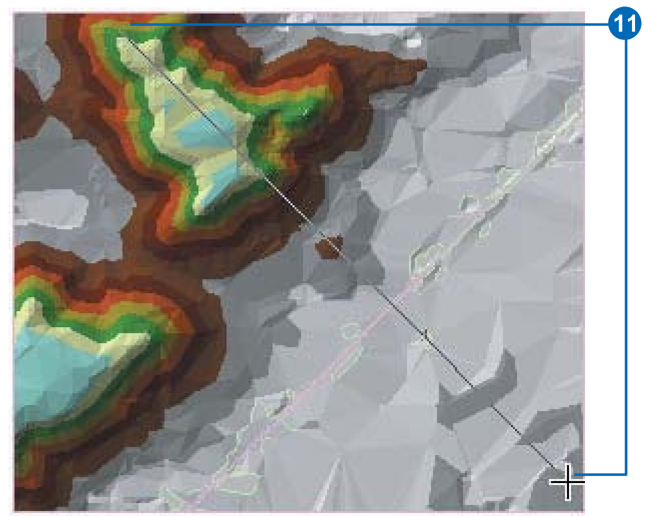


9. In the ArcMap table of contents, check tin1.



10. Click the Interpolate Line button.

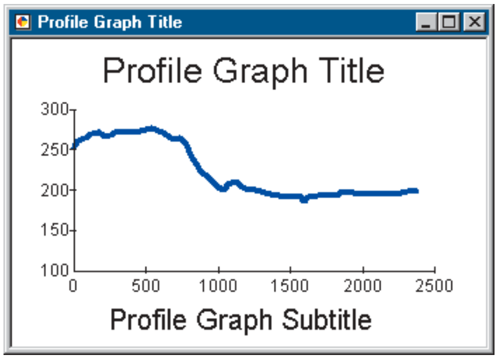
11. Click on the upper-left corner of the TIN, drag the line to the lower-right corner, and double-click to stop digitizing.



You can create a profile along a line with more than one segment, but in this case you'll just make one straight line.

12. Click the Create Profile Graph button.

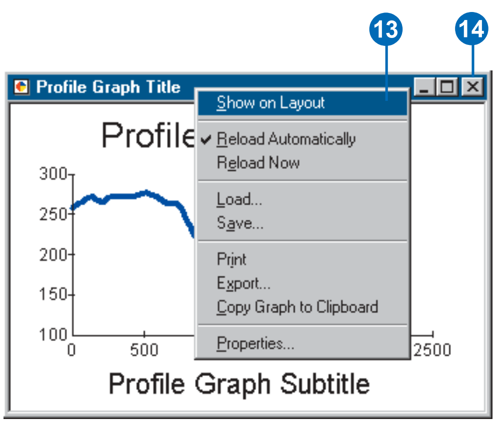
The profile graph is created.



You can edit the title, subtitle, and other properties of the graph; you can save, print, or export the graph; you can copy it to the clipboard; and you can show the graph on the layout. You can also simply close the graph.

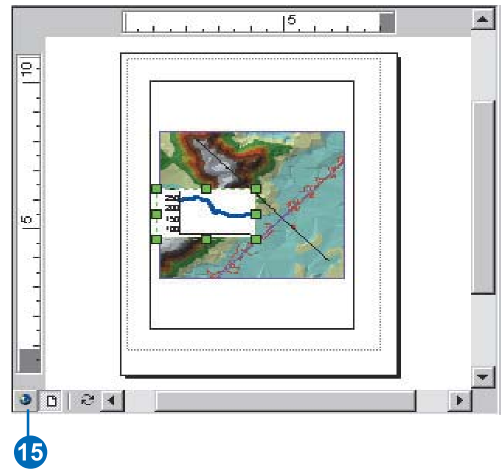
13. Right-click on the Profile Graph Title bar and click Show on Layout.

14. Close the profile graph window.



You can see the graph on the layout of the map.

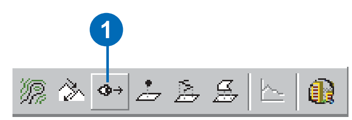
15. Click the Data View button to return to data view.



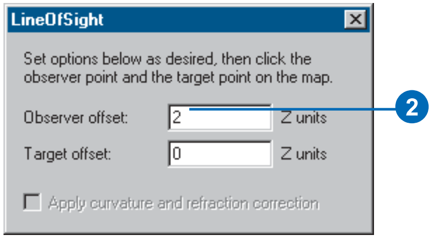
Creating a line of sight on the terrain

Another way of understanding the terrain is to create a line of sight. Lines of sight show what parts of a surface are visible and what parts are hidden along a line from an observer point to a target point.

1. Click the Create Line of Sight button.

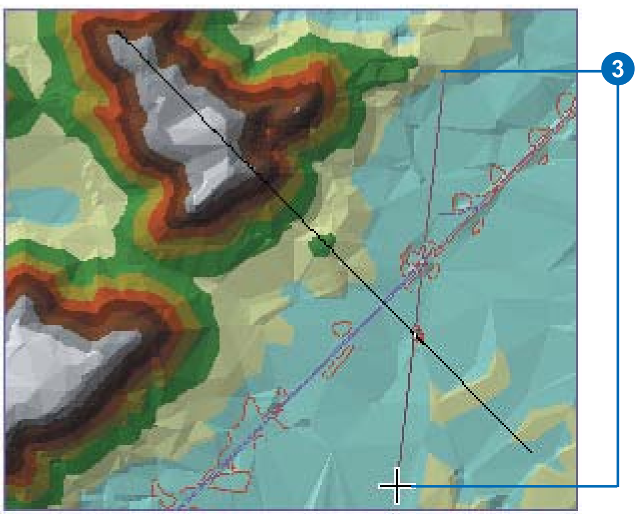


2. Type “2” in the Observer offset text box.

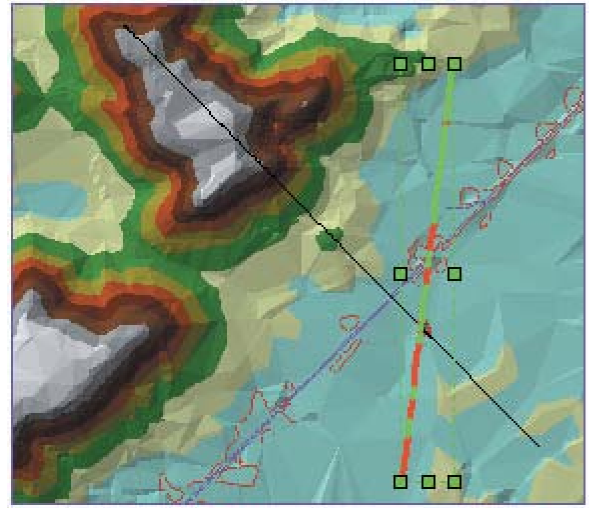


The line of sight will be calculated to show what is visible from the perspective of an observer two meters tall, as the z units for this scene are meters.

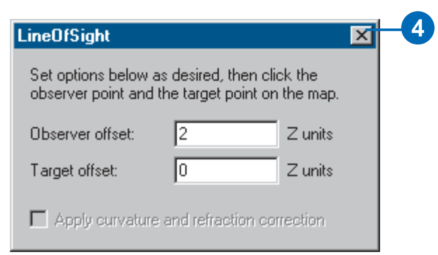
3. Click on the south slope of the higher land in the upper-right part of the TIN (the observer point), drag the line to the lower-right part, and release the mouse button (the target point).



The line of sight is calculated. The green segments show areas that are visible from the observer point; the red segments are hidden from the observer.

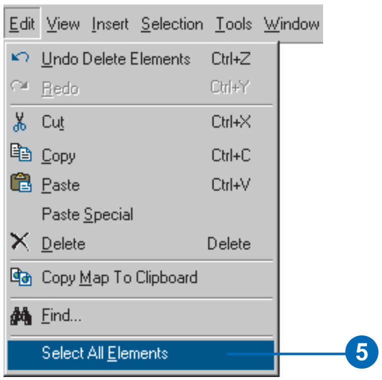


4. Close the LineOfSight dialog box.



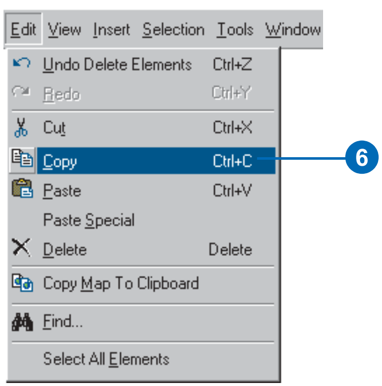
Lines of sight, like other graphic lines, can be copied from ArcMap to ArcScene. Now you will copy both the lines you've created into the scene.

5. Click Edit and click Select All Elements.

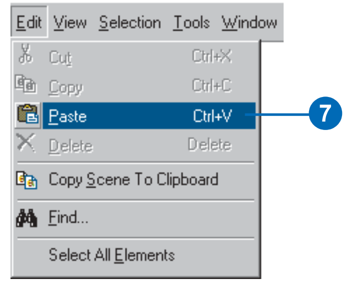


Both of the lines you created are selected.

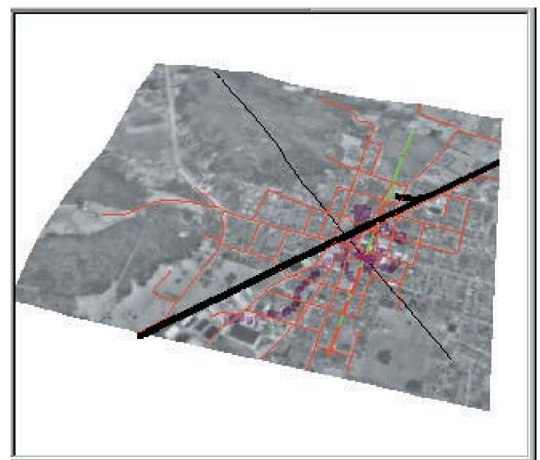
6. Click Edit and click Copy.



7. In ArcScene, click Edit and click Paste.

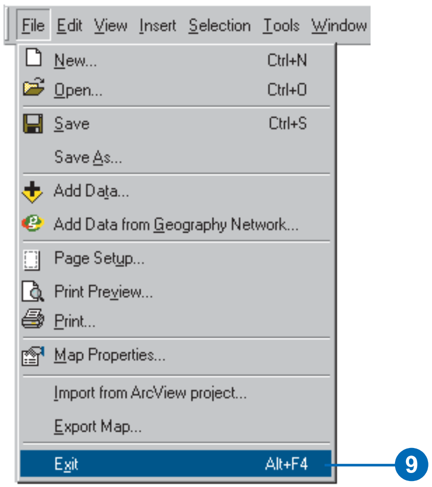


The lines are pasted into the scene.

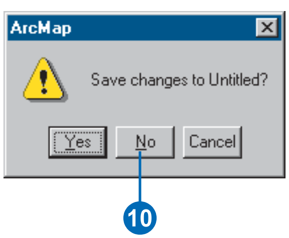


8. In ArcScene, click the Save button.

9. In ArcMap, click File and click Exit.



10. Click No.



This tutorial introduced you to a sample of the tasks you can do with 3D Analyst and showed how ArcCatalog, ArcMap, and ArcScene can be used together. The remaining chapters in this book discuss in detail the variety of tasks you can accomplish with 3D Analyst.

Exercise 5: Working with animations in ArcScene

Imagine that you wish to create an animated sequence showing the flight of an object over a landscape. You've created a TIN and have draped images over it to show the area. You also have some data pertaining to a strange phenomenon that has been occurring in the region. You are interested in displaying all the data in a dynamic way, making an animation to tour points of interest, and showing how you made the surface. You would also like to model the phenomenon by moving a layer in the scene.

The tutorial data has already been assembled in the scene document named Animation.sxd. You will use ArcScene animation tools to effectively convey the points you want to show.

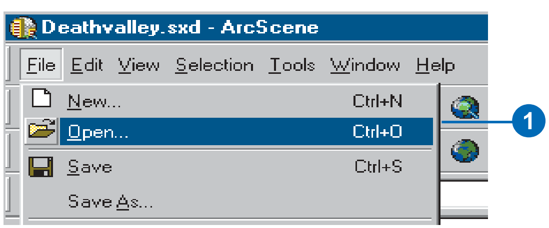
Data was supplied courtesy of MassGIS, Commonwealth of Massachusetts Executive Office of Environmental Affairs.

In this exercise, you will play an existing animation in the scene document, Final Animation_A.sxd, and perform the tasks typically used to create the animation.

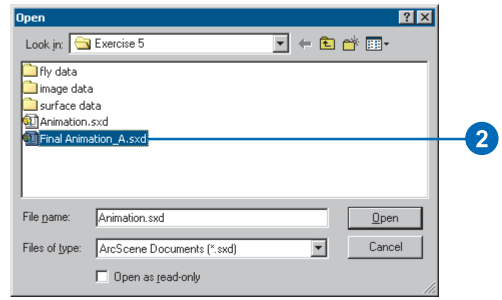
Opening the Final Animation_A scene document

In this section, you will play an animation that demonstrates some of the effects that you can create when you animate a scene.

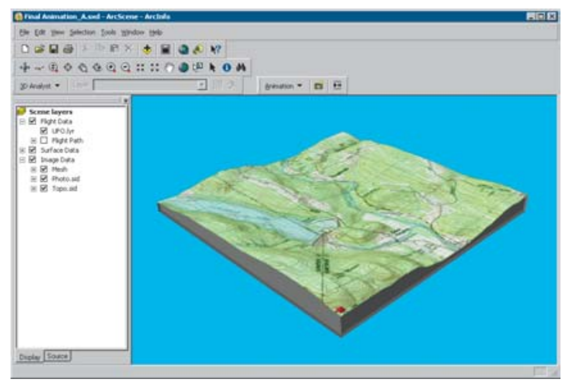
1. In ArcScene, click File and click Open.



2. Navigate to the Exercise5 folder and double-click Final Animation_A.sxd.



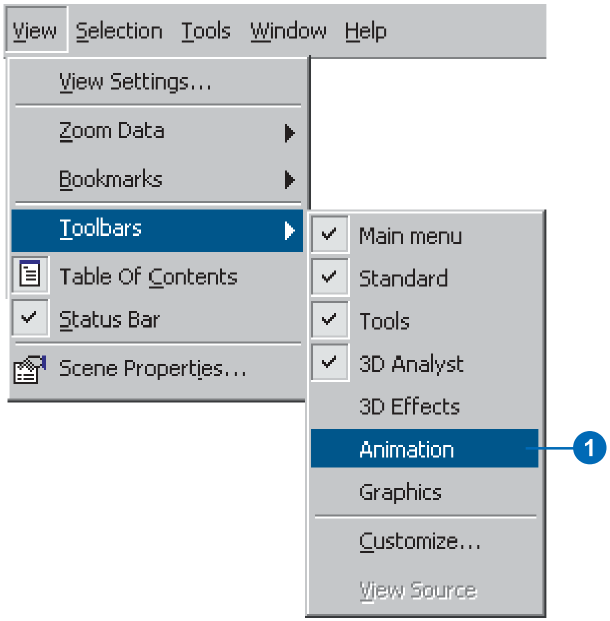
This scene contains geographic information and recorded special effects that have been combined to make an animation.



Playing the scene's animation

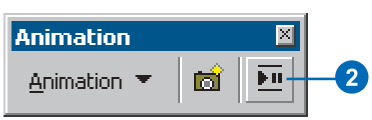
In order to view a scene's animation, you need to turn on the Animation toolbar.

1. Click View, point to Toolbars, and click Animation.



The Animation toolbar appears. Now you'll play the animation.

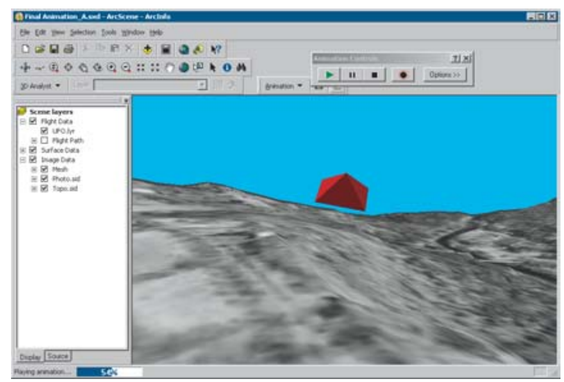
2. Click the Open Animation Controls button.



This animation shows the flight of a hypothetical unidentified flying object (UFO) over the terrain.

3. Click the Play button.

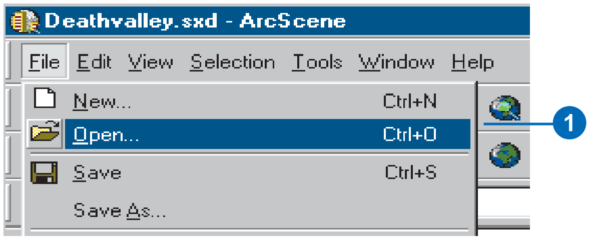
The animation plays, illustrating some of the effects you can use in an animated scene.



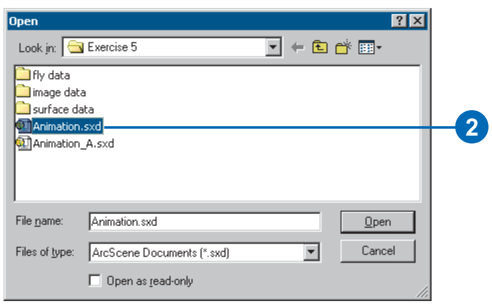
In the next section you will work through the steps used to make animations like this one.

Opening the Animation scene document

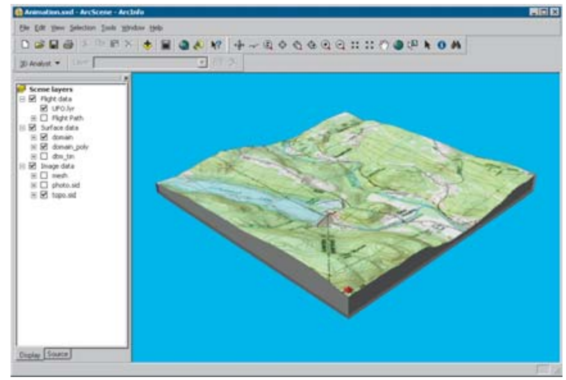
1. In ArcScene, click File and click Open.



2. Navigate to the Exercise 5 folder and double-click Animation.sxd.



The scene contains an ortho photo, a scanned topographic map, and other data you need to make your animation.



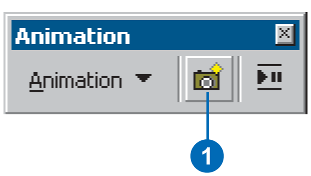
In this section, you'll use the animation tools to capture keyframes, import tracks, play back your animations, and save them to a scene document.

There are three types of keyframes that you can use to build animations. The first is a camera keyframe. A camera keyframe is a snapshot of the view you see in a scene. The second, a layer keyframe, is a snapshot of a layer's properties.The third type is a scene keyframe, which stores properties of a scene. In this section, you will create a simple animation from a set of camera keyframes.

Capturing Perspective views as keyframes to make an animation

The simplest way to make animations is by capturing views to be stored as keyframes. The captured views are snapshots of camera perspectives in a scene at a particular time. The most fundamental element of an animation is a keyframe. Keyframes are used as snapshots to interpolate between in a track. You'll create a set of keyframes to make a camera track that will show an animation between points of interest in your study area.

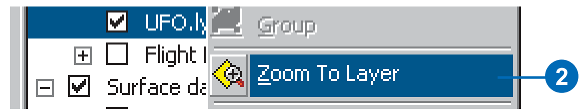
1. Click the Capture View button to create a camera keyframe showing the full extent of the scene.



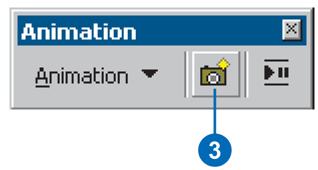
For a camera keyframe, the object is the virtual camera through which you view the scene. Navigating the scene changes camera properties that determine its position.

ArcScene interpolates a camera path between keyframes, so you'll need to capture more views to make a track that shows animation.

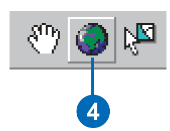
2. Right-click on UFO.lyr and click Zoom To Layer.



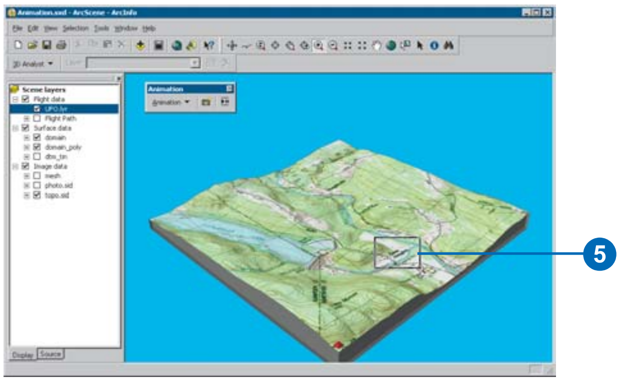
3. Click the Capture View button to create a camera keyframe showing the UFO layer.



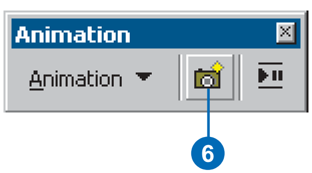
4. Click the Full Extent button to view all the data.



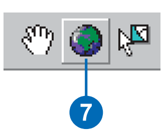
5. Click Zoom In on the Navigation toolbar and zoom to Goss Heights, located near the center of your view.



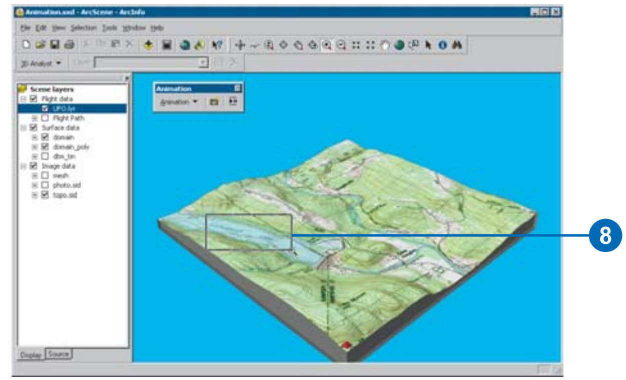
6. Click the Capture View button to create a camera keyframe of Goss Heights.



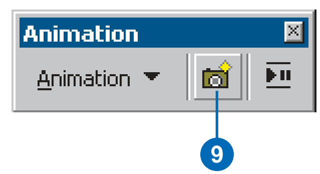
7. Click the Full Extent button.



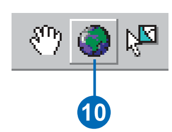
8. Zoom to Littleville Lake.



9. Click the Capture View button to capture a view of Littleville Lake.



10. Click the Full Extent button.

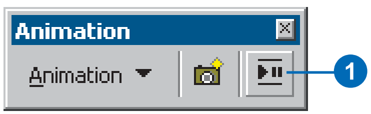


The captured views you just made are stored as a set of camera keyframes in a camera track. When the track is played, it shows a smooth animation between the keyframes. Next, you'll play your animation track.

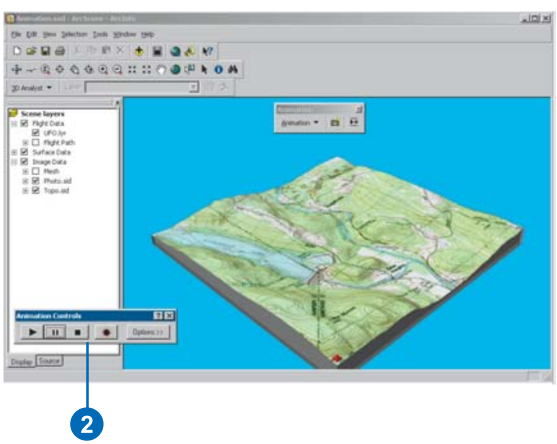
Playing back your animation

You will play back animations using simple tools that resemble the controls of a video cassette recorder (VCR).

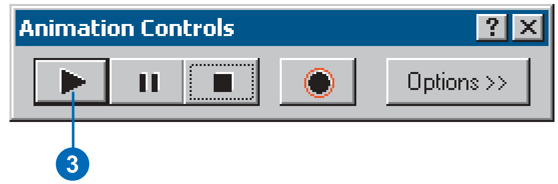
1. Click the Open Animation Controls button.



2. Click the Animation toolbar and drag it to the lower-left corner of the scene so it won't block your view of the tools or data.



3. Click the Play button.



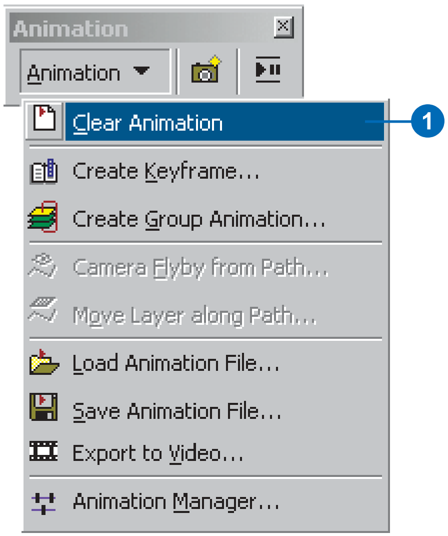
An animation is played back by interpolating the camera position between the keyframes in the track. In this case, the animation shows a virtual tour through the views you captured.

Clearing an animation

If you want to start over, you can erase all the tracks you created. In this section, you'll remove the tracks you just created so you can improve your animation.

1. Click Animation and click Clear Animation.

   All animation tracks are removed from the scene.

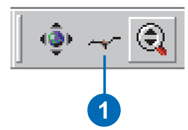


Recording navigation

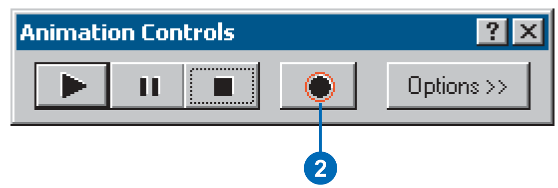
Another way to create a camera track for an animation is to record in real time while you navigate in a scene. In this section, you will record your view of the scene while you navigate using the Fly tool.

1. Click the Fly tool on the navigation toolbar.

   The Fly tool allows you to fly through your scenes.

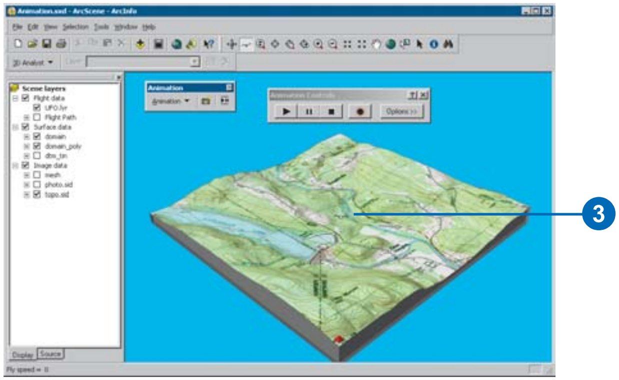


2. Click the Record button to start recording your navigation.



   ArcScene begins recording as soon as you click the Record button. If you don't navigate right away, your track will reflect this.

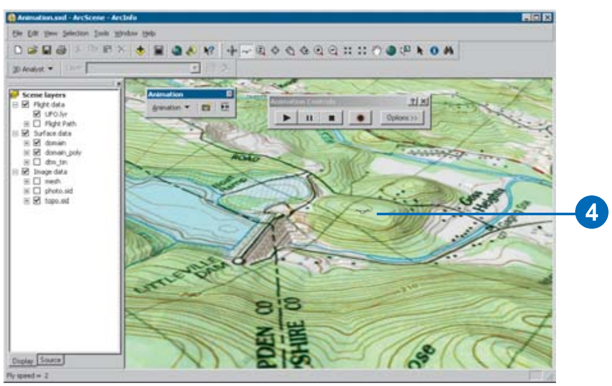
3. Click once in the center of the scene to activate the Fly tool. You start flying by entering into hovering mode.



In this mode your viewpoint follows the cursor. Point in the direction you wish to look.

4. Click once more to begin flying through the scene.

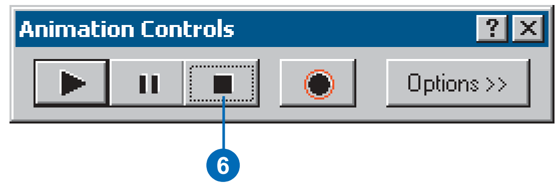
Point in the direction you want to move. Click again to increase your speed and right-click to decrease your speed. Your speed is indicated in the status bar in the lower-left corner of the ArcScene window.



5. Press Esc to stop flying.

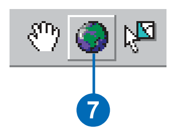
You can also stop flying by clicking until your speed is zero.

6. Click the Stop button to finish recording.

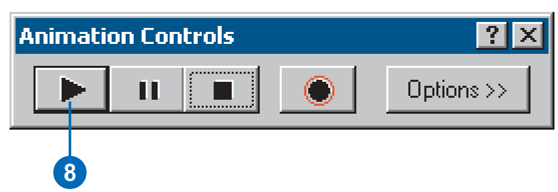


You have recorded your flight path through the scene as a new camera track that began when you clicked the Record button and ended when you clicked the Stop button.

7. Click the Full Extent button.

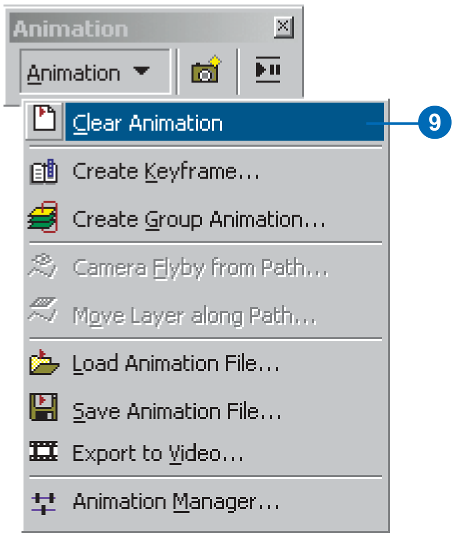


8. Click the Play button to see the animation you recorded.



When you are done viewing the animation you recorded, clear the track so you can make a better one in the next section.

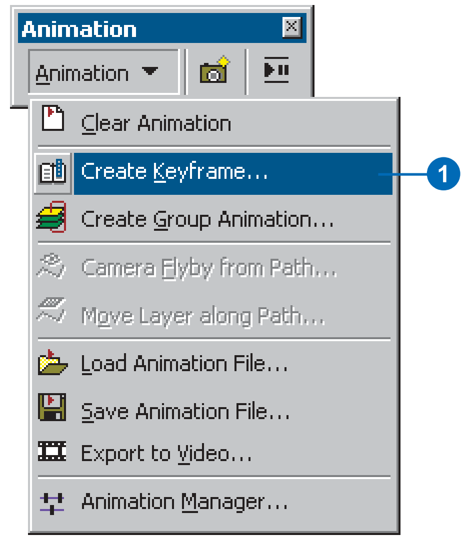
9. Click Animation and click Clear Animation.



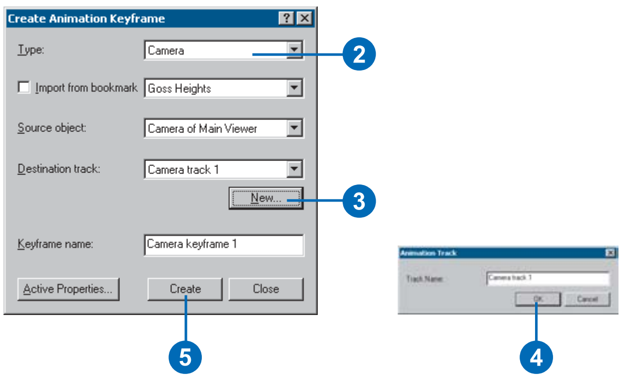
Making a camera track from 3D bookmarks

In the previous sections, you navigated in a scene and created keyframes to build a camera track. Another way to create the keyframes for a camera track is to import bookmarked perspective views of a scene. In this section, you'll create keyframes from 3D bookmarks.

1. Click Animation and click Create Keyframe.

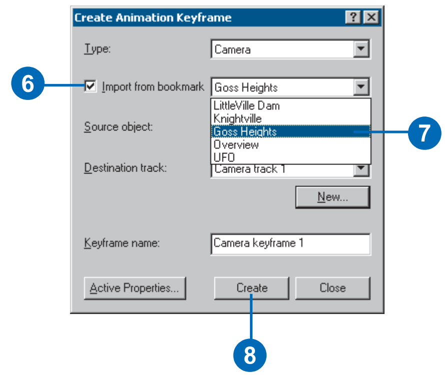


2. Click the Type dropdown arrow and choose Camera.
3. Click New to create a new track.
4. Click OK.
5. Click Create.



You've now created a camera track with one camera keyframe showing the full extent of your scene. You'll need to add more keyframes to your track so that it will show animation. Now, you'll import bookmarks to create the keyframes for the rest of the animation.

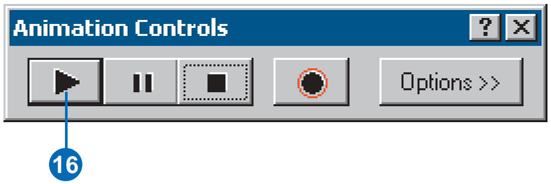
6. Check Import from bookmark.
7. Click the Import from bookmark dropdown arrow and click Goss Heights.



8. Click Create to make the second keyframe in your track.
9. Click the Import from bookmark dropdown arrow and choose LittleVille Dam.
10. Click Create to import this bookmark as a keyframe.
11. Click the Import from bookmark dropdown arrow and click Knightville.
12. Click Create to make the Knightville keyframe.
13. Click the Import from bookmark dropdown arrow once more and click Overview.
14. Click Create to create a keyframe showing all the data.

15. Click Close.

16. Click the Play button.

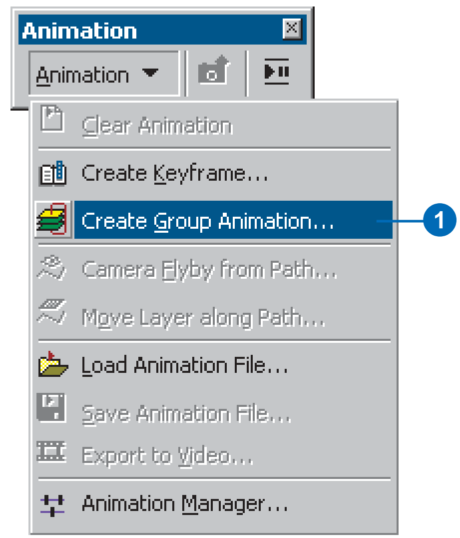


The camera track plays, moving the camera through the set of keyframes you imported from existing 3D bookmarks.

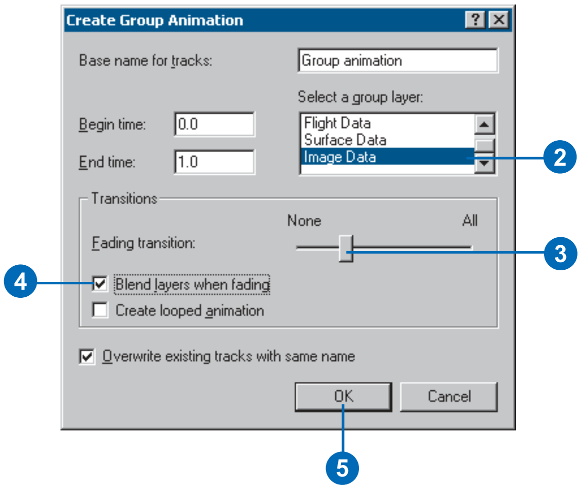
Switching the visibility between layers using a group animation

Now that you have explored some of the ways to create an animated camera track from keyframes, you'll learn how to change the way layers in a scene are displayed during animation. In this section, you will switch the layer that is draped over the terrain model to show different ways of representing the terrain.

1. Click Animation and click Create Group Animation.

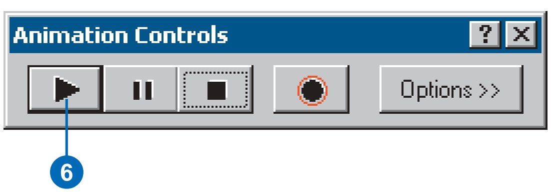


2. Select the group layer named Image Data.
3. Slide the Fading transition bar about a quarter of the way to the right.
4. Check Blend layers when fading.
5. Click OK.



The layer track you just created toggles the visibility of successive layers to animate a progression between them. The transition settings you modified will show a smooth blending between the layers in the progression.

6. Click the Play button to watch your animation.

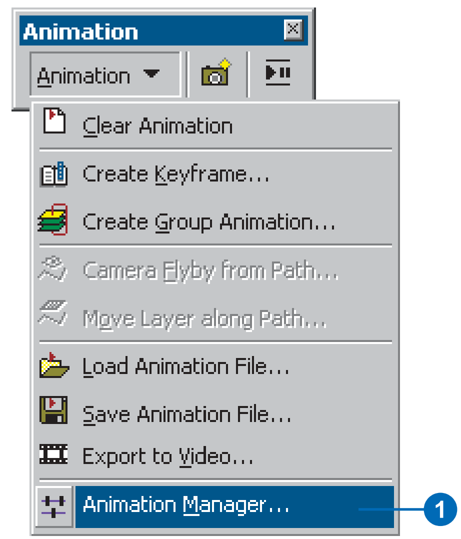


Since you didn't clear the animation track you made from the keyframes, it plays in addition to the layer tracks you just created. However, you can stop the camera track from playing. You'll do this next.

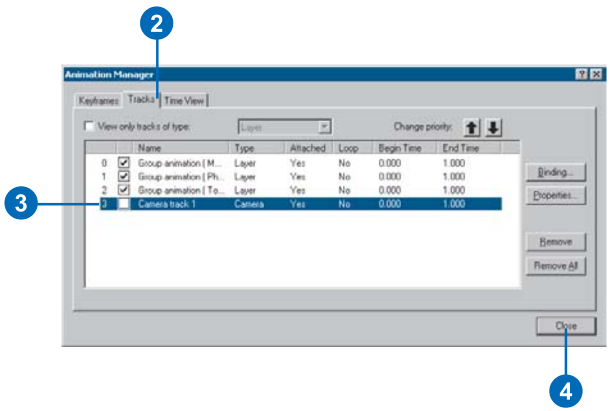
Using the Animation Manager to disable a track from playing

The Animation Manager allows you to control many properties of an animation. In this section, you'll use the Animation Manager to stop a camera track from playing.

1. Click Animation and click Animation Manager.

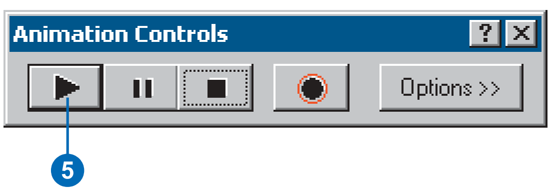


2. Click the Tracks tab.
3. Uncheck Camera track 1.
4. Click Close.



You have disabled the camera track. Now it will not play as part of the animation.

5. Click the Play button.



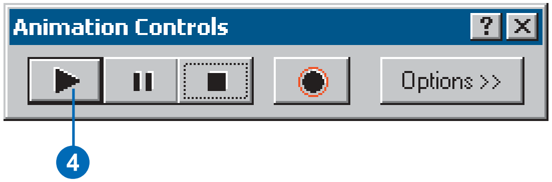
The animation plays again, only showing the layer tracks this time. It may now seem that the duration of the animation is too long. You can control the amount of time in which an animation is played.

Using Animation Controls to adjust the playback duration

1. Click Options.

2. Type "10" in the Duration text box.
3. Click Options to close this portion of the dialog box.

4. Click the Play button.



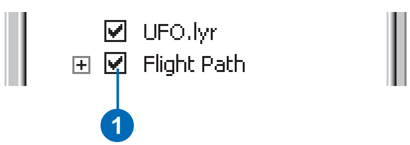
The animation now plays more quickly.

Moving an object along a predefined path

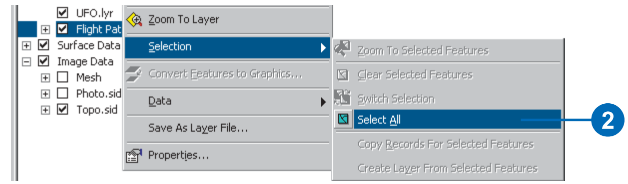
One of the things an animation allows you to do is move an object through a scene. You can add a layer containing a model vehicle and move it through the scene along a specified track.

You can choose to move a layer along a selected line feature or graphic. The scene contains a graphic layer with a model UFO that was created using Visual Basic code. In the next set of steps, you'll fly the model UFO along a shapefile that shows its flight path.

1. Check Flight Path.

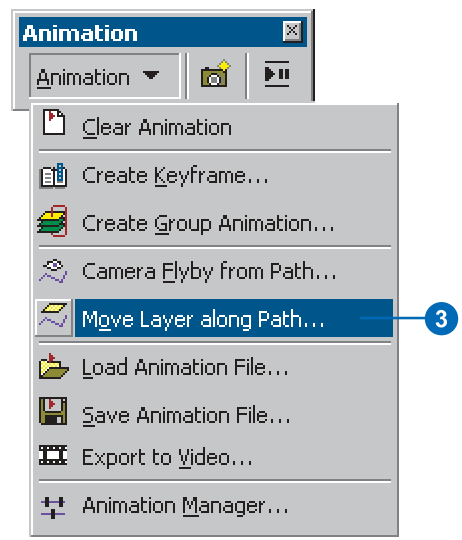


2. Right-click Flight Path, point to Selection, then click Select All.

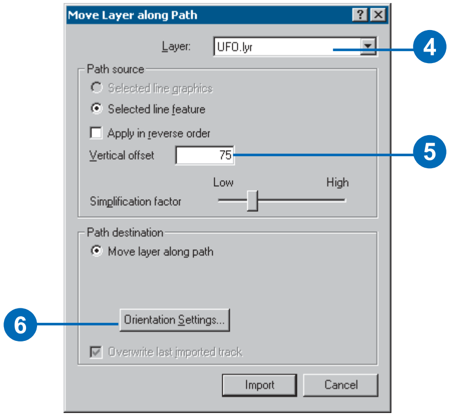


You can also use the Select Features or Select Graphics tools to select the path you want to use. A path is constructed from a single selected line feature or graphic.

3. Click Animation and click Move Layer along Path.



4. Click the Layer dropdown arrow and click UFO.lyr.



5. Type a Vertical offset of "75". This will make the object appear to fly above the surface.

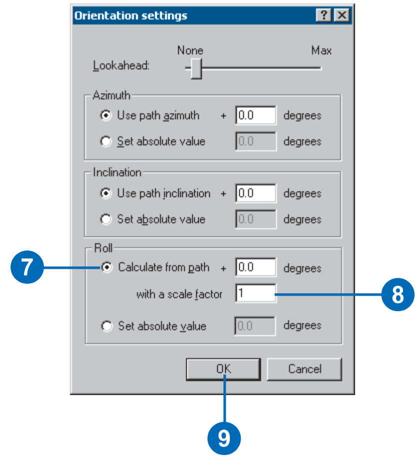
   ArcScene can improve the simulation of flight of an object, such as an airplane, along a path by making the object point in the direction it is moving and by rolling it from side to side as if it were banking. In the next steps, you'll define a roll for the UFO layer.

6. Click Orientation Settings to modify the layer's position while it's animated.

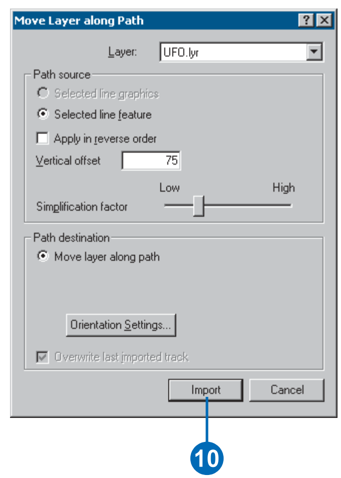
7. Click Calculate from path to calculate the layer's roll based on the path's shape.

8. Type "1" as a scale factor.

9. Click OK.

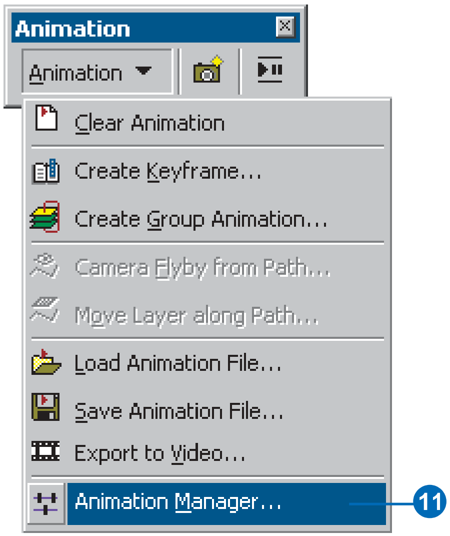


10. Click Import to import the selected line as a flight path.

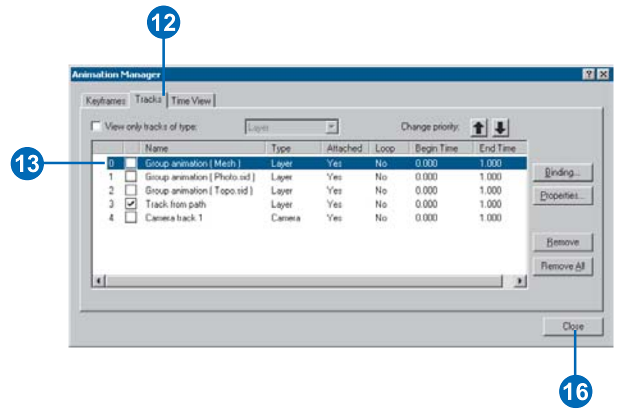


The UFO layer moves along the selected path. The movement is stored as a set of layer keyframes in a layer animation track. Now you'll disable the tracks you created previously so that just the UFO track is played.

11. Click Animation and click Animation Manager.

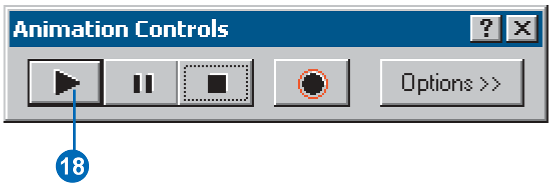


12. Click the Tracks tab.
13. Uncheck Group animation (Mesh).
14. Uncheck Group animation (Photo.sid).
15. Uncheck Group animation (Topo.sid).



16. Click Close.
17. Uncheck Flight Path to turn off the visibility of this layer.

18. Click the Play button.



The UFO flies along the path you indicated. Next, you'll move the camera along a predefined path.

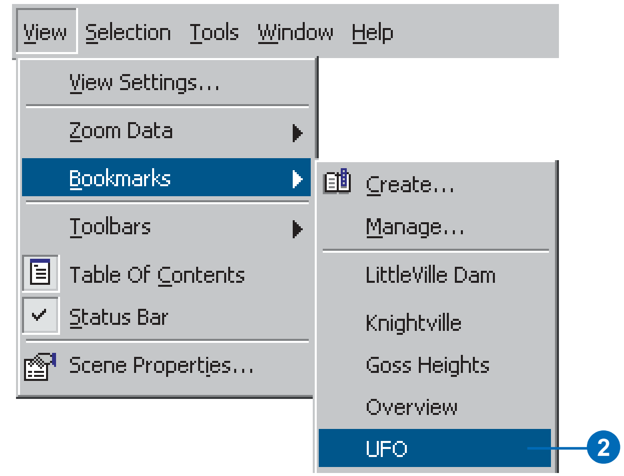
Creating a camera flyby from a path

You can move a camera along a flight path in the same way you just moved a layer along a path. Next, you'll combine the track you made in the last step with one that will point the camera at the UFO as it flies.

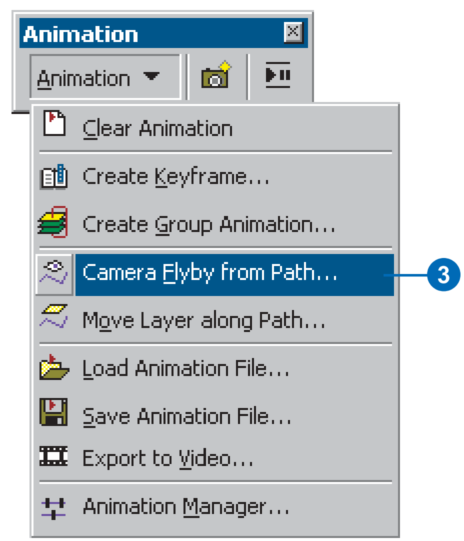
1. Check Flight Path, make sure the line you chose for your path is still selected, then uncheck Flight Path.

You'll move the camera location to a predefined location in the center of the scene that will give you a better vantage point from which to view the UFO layer as it is moved.

2. Click View, point to Bookmarks, then click UFO.

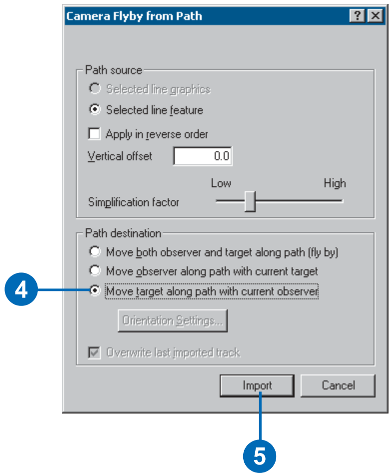


3. Click Animation and click Camera Flyby from Path.



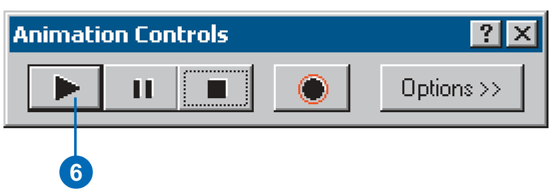
4. Click the third path destination option.

This option lets you observe the UFO as it moves along the path.



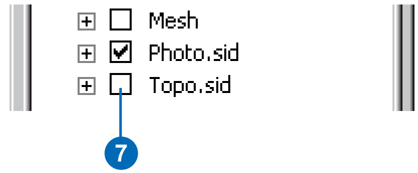
5. Click Import.

6. Click the Play button.



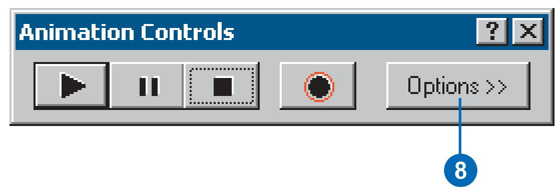
Both tracks play. The UFO layer moves, and the camera follows its movements.

7. Uncheck the Topo.sid check box to make the ortho photo visible.

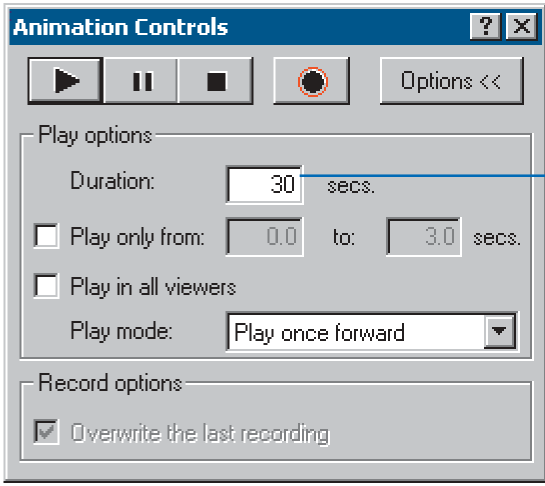


The animation is playing too quickly. Next, you'll learn how to adjust the duration that the animation is playing to enhance the visual effect.

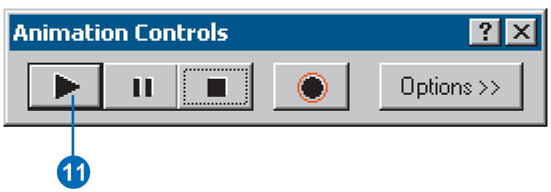
8. Click Options.



9. Type “30” in the Duration text box.



10. Click Options to minimize the dialog box.
11. Click the Play button.

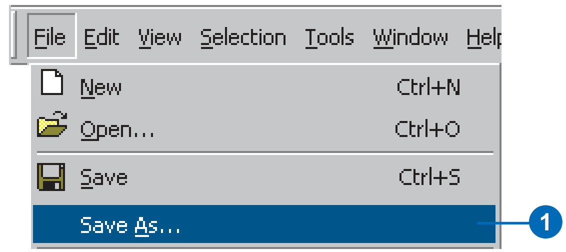


Now the animation plays more slowly as the UFO flies over the terrain.

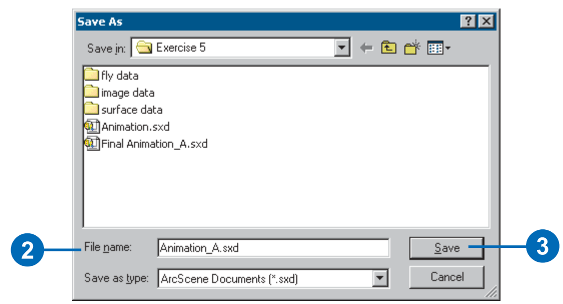
Saving an animation in a scene document

You can save animation tracks in a scene document. In the next step, you’ll save the animations you made in a scene document.

1. Click File and click Save As.



2. In the File name text box, type “Animation_A.sxd”.
3. Click Save.



The new scene document is created, storing the animation tracks.

In this exercise you learned how to create and save simple animations that help you better visualize 3D data. There are other more advanced ways of creating, saving, and sharing animations. For more information, see Chapter 8, ‘Animation’.

Creating surface models

3

IN THIS CHAPTER

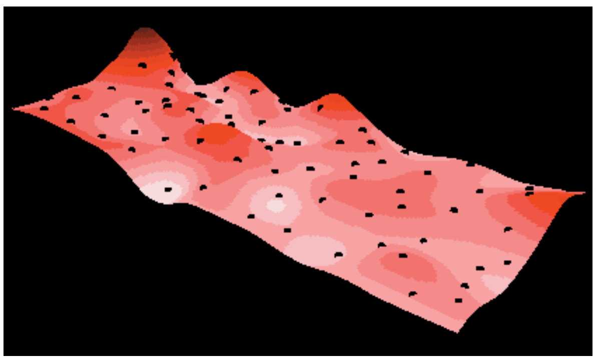
- **Representing surfaces with rasters and TINs**
- **Creating a raster from points**
- **Interpolator assumptions**
- **Raster output controls**
- **Creating a TIN from vector data**
- **Modifying a TIN**
- **Creating a TIN from a raster**
- **Creating a raster from a TIN**

3D Analyst lets you work with real or hypothetical surfaces with two types of surface models: rasters and TINs. You can use existing models, or you can create your own from a variety of data sources. The two main methods of creating surface models are by interpolation and triangulation. You can choose from several interpolation methods, including Inverse Distance Weighted, Spline, Kriging, and Natural Neighbors interpolation. You can build triangulated surfaces by creating a TIN or by adding elements to an existing TIN. You can also convert between TIN and raster surface models.

What are surfaces and surface models?

A surface is a continuous field of values that may vary over an infinite number of points. For example, points in an area on the earth's surface may vary in elevation, proximity to a feature, or concentration of a particular chemical. Any of these values may be represented on the z-axis in a three-dimensional x,y,z coordinate system, so they are often called *z-values*.

Because a surface contains an infinite number of points, it is impossible to measure and record the z-value at every point. Surface models allow you to store surface information in a GIS. A surface model approximates a surface by taking a sample of the values at different points on the surface and then interpolating the values between these points.

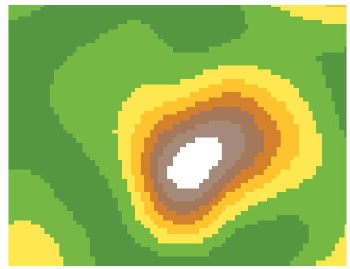


Surface model of chemical concentration across an area with points showing where the concentration was sampled

3D Analyst uses two types of surface models: rasters and TINs. Rasters represent a surface as a regular grid of locations with sampled or interpolated values. TINs represent a surface as a set of irregularly located points linked to form a network of triangles with z-values stored at the nodes.

Raster surfaces

Raster surfaces are usually stored in grid format. A grid consists of a rectangular array of uniformly spaced cells with z-values. The smaller the cells, the greater the locational precision of the grid.

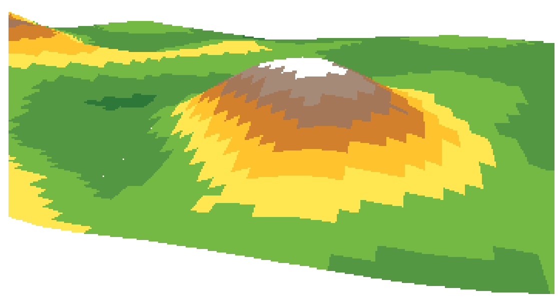


Higher precision grid

Lower precision grid

You cannot locate individual features—for example, the summit of a mountain—any more precisely than the size of the grid cells.

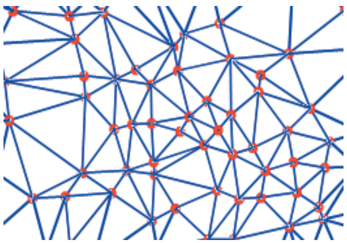
Rasters are also used to store images and thematic grid data.



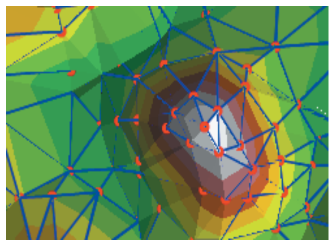
Grid in perspective view

TIN surfaces

TINs consist of nodes that store z-values, connected by edges to form contiguous, nonoverlapping triangular facets. The edges in TINs can be used to capture the position of linear features that play an important role in the surface, such as ridgelines or stream courses.

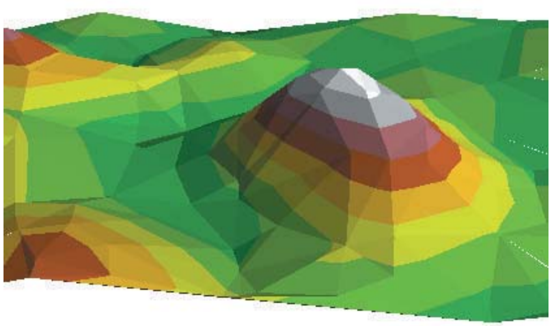


Nodes and edges of a TIN



Nodes, edges, and faces

Because the nodes can be placed irregularly over the surface, TINs can have a higher resolution in areas where a surface is highly variable or where more detail is desired and a lower resolution in areas that are less variable or of less interest.



TIN in perspective view

The input features used to create a TIN remain in the same position as nodes or edges in the TIN. This allows a TIN to preserve all of the precision of the input data while simultaneously modeling the values between known points. You can include precisely located features on a surface—such as mountain peaks, roads, and streams—by using them as input features to the TIN.

TIN models are less widely available than raster surface models and tend to be more expensive to build and process. The cost of obtaining good source data can be high, and processing TINs tends to be less efficient than processing raster data because of their complex data structure.

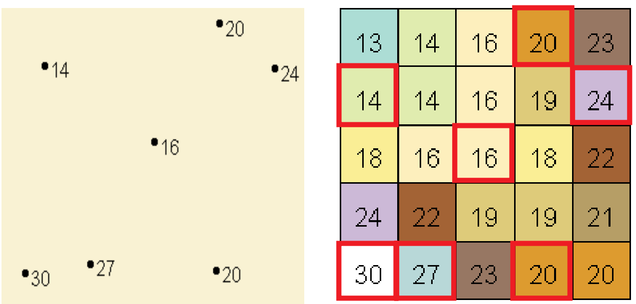
TINs are typically used for high precision modeling of smaller areas, such as in engineering applications, where they are useful because they allow calculations of planimetric area, surface area, and volume.

Creating raster surfaces from points

Surfaces of continuous data are usually generated from samples taken at points across the area. For example, the irregularly spaced weather stations in a region can be used to create raster surfaces of temperature or air pressure. The resulting surface is a regular grid of values.

What is interpolation?

Interpolation predicts values for cells in a raster from a limited number of sample data points. It can be used to predict unknown values for any geographic point data: elevation, rainfall, chemical concentrations, noise levels, and so on.



On the left is a point dataset of known values. On the right is a raster interpolated from these points. Unknown values are predicted with a mathematical formula that uses the values of nearby known points.

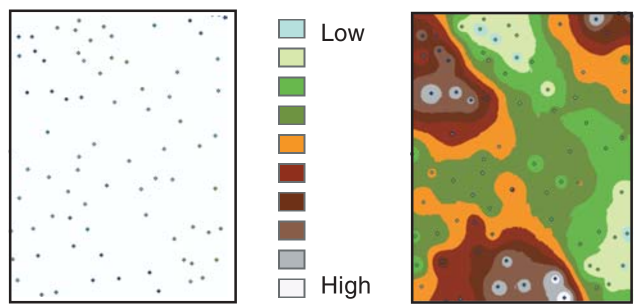
In this example the input points happen to fall on cell centers—this is unlikely in practice. One problem with creating rasters by interpolation is that the original information is degraded to some extent—even when a data point falls within a cell, it is not guaranteed that the cell will have exactly the same value.

Interpolation is based on the assumption that spatially distributed objects are spatially correlated; in other words, things that are close together tend to have similar characteristics. For instance, if it is snowing on one side of the street, you can predict with a high level of confidence that it is also snowing on the other side of the street. You would be less sure if it was snowing across town and less confident still about the state of the weather in the next county.

Why interpolate?

Visiting every location in a study area to measure the height, magnitude, or concentration of a phenomenon is usually difficult or expensive. Instead, dispersed sample input point locations can be selected, and a predicted value can be assigned to all other locations. Input points can be either randomly, strategically, or regularly spaced points containing height, concentration, or magnitude measurements.

A typical use for point interpolation is to create an elevation surface from a set of sample measurements. Each point represents a location where the elevation has been measured. The values between these input points are predicted by interpolation.



The resulting grid is a prediction of what the elevation is at any location on the actual surface.

Interpolation methods

There are several ways to create raster surfaces from point data. You can use the IDW, Natural Neighbors, Spline, and Kriging methods to create surfaces through the user interface of 3D Analyst. Trend surface interpolation is available through customization.

Each interpolation method makes assumptions about how to determine the estimated values. Depending on the phenomenon you are modeling and the distribution of sample points, different interpolators produce better models of the actual surface. Regardless of the interpolator, the more input points and the more even their distribution, the more reliable the results.

Inverse Distance Weighted: This interpolation method assumes that each sample point has a local influence that diminishes with distance. It weights the points closer to the processing cell more heavily than those farther away. Either a specified number of points or all of the points within a given radius can be used to determine the value of each output cell. This method is appropriate when the variable being mapped decreases in influence with distance from the sampled location. For example, when interpolating a surface of consumer purchasing power for a retail site analysis, the purchasing power of a more distant location will have less influence because people are more likely to shop closer to home.

Natural Neighbors: Like IDW, this interpolation method is a weighted-average interpolation method. However, instead of finding an interpolated point's value using all of the input points weighted by their distance, Natural Neighbors interpolation creates a Delauney Triangulation of the input points and selects the closest nodes that form a convex hull around the interpolation point, then weights their values by proportionate area. This method is most appropriate where sample data points are distributed with uneven density. It is a good general-purpose interpolation technique and has the advantage that you do not have to specify parameters, such as radius, number of neighbors, or weights.

Spline: This general-purpose interpolation method fits a minimum-curvature surface through the input points. Conceptually, this is like bending a sheet of rubber to pass through the points while minimizing the total curvature of the surface. It fits a mathematical function—a minimum curvature, two-dimensional, thin-plate spline—to a specified number of the nearest input points while passing through all input points. This method is best for gradually varying surfaces such as elevations, water-table depths, or pollution concentrations. It is not appropriate when there are large changes within a short horizontal distance because it can overshoot estimated values.

Kriging: This interpolation method assumes that the distance or direction between sample points reflects a spatial correlation that can be used to explain variation in the surface. Kriging fits a mathematical function to a specified number of points, or all points within a specified radius, to determine the output value for each location. Kriging is a multistep process; it includes exploratory statistical analysis of the data, variogram modeling, creating the surface, and, optionally, exploring a variance surface. This function is most appropriate when you know there is a spatially correlated distance or directional bias in the data. It is often used in soil science and geology.

Trend: This interpolation method fits a mathematical function—a polynomial of specified order—to all input points. This method uses a least-squares regression fit, which results in a surface that minimizes the variance of the surface in relation to the input values. The surface is constructed so that for every input point, the total of the differences between the actual values and the estimated values (the variance) will be the smallest possible. The resulting surface rarely goes through the input points. The Trend method is available through customization but not through the 3D Analyst user interface.

Details on the interpolation methods

Of the available interpolation methods, IDW, Spline, and Kriging, each has some parameters that influence how the interpolation is done. The Natural Neighbors method only requires that you specify the output cell size.

Inverse Distance Weighted

IDW estimates grid cell values by averaging the values of sample data points in the vicinity of each cell. The closer a point is to the center of the cell being estimated, the more influence, or weight, it has in the averaging process. There are several parameters you can set to control the IDW interpolation. These are the Power, the Radius Type, and Barriers.

Power: With IDW you can control the significance of known points on the interpolated values based on their distance from the output point. By defining a high power, more emphasis is placed on the nearest points, and the resulting surface will have more detail (be less smooth). Specifying a lower power will give more influence to the points that are further away, resulting in a smoother surface. A power of 2 is most commonly used and is the default.

The characteristics of the interpolated surface can also be controlled by limiting the number of input points used for calculating each interpolated point:

Radius Type: Fixed

With a fixed radius, the distance of the radius is constant, and for each interpolated point, all of the points within the circle will be used. By specifying a minimum count, you can ensure that if in a certain area the number of points is less than the count specified, the radius of the circle will be increased until the count is met.

Radius Type: Variable

With a variable radius, the number of points used in calculating the value of the interpolated point, or count, is specified. This makes the radius vary for each interpolated point, depending on how far it has to search around each interpolated point to reach the specified number of input points. The variable radius approach may produce better surfaces when the density of input points varies significantly from one area to another. If you know you have areas with sparsely distributed input points, you can specify a maximum distance to limit the potential radius of the circle. In this case, if the number of points is not reached before the maximum distance of the radius is reached, fewer points will be used in the calculation of the interpolated point.

Barrier: A line or polygon dataset can be used as a break that limits the search for input sample points. A line can represent a cliff, ridge, or some other interruption in a landscape. Only those input sample points on the same side of the barrier as the current processing cell will be considered. A choice of No Barriers will use all points within the identified radius.

Spline

Spline estimates values using a mathematical function that minimizes overall surface curvature, resulting in a smooth surface that passes exactly through the input points.

There are two Spline methods: Regularized and Tension. They are described below, as well as the meaning of the weight parameter, which changes depending on the Spline method. Both methods share a number of points parameter.

Number of points: For both methods, this parameter defines the number of points used in the calculation of each interpolated point. The more input points you specify, the more each cell is influenced by distant points, and the smoother the surface.

Regularized spline

The Regularized method creates a smooth, gradually changing surface with values that may lie outside the sample data range.

Weight: Defines the weight of the third derivatives of the surface in the curvature minimization expression. The higher the weight, the smoother the surface.

The values entered for this parameter must be equal to or greater than zero. The typical values that may be used are 0, .001, .01, .1, and .5.

Tension spline

The Tension method tunes the stiffness of the surface according to the character of the modeled phenomenon. It creates a less smooth surface with values more closely constrained by the sample data range.

Weight: Defines the weight of tension. The higher the weight, the coarser the surface.

The values entered have to be equal to or greater than zero. The typical values are 0, 1, 5, and 10.

Kriging

IDW and Spline (discussed earlier) are referred to as deterministic interpolation methods because they are directly based on the surrounding measured values or on specified mathematical formulas that determine the smoothness of the resulting surface. A second family of interpolation methods consists of geostatistical methods (such as kriging), which are based on statistical models that include autocorrelation (the statistical relationship among the measured points). Because of this, not only do these techniques have the capability of producing a prediction surface, but they can also provide some measure of the certainty or accuracy of the predictions.

Kriging is similar to IDW in that it weights the surrounding measured values to derive a prediction for an unmeasured location. The general formula for both interpolators is formed as a weighted sum of the data:

$$\hat{Z}(\mathbf{s}_0) = \sum_{i=1}^{N} \lambda_i Z(\mathbf{s}_i)$$

where

$Z(\mathbf{s}_i)$ is the measured value at the ith location;

λ_i is an unknown weight for the measured value at the ith location;

$\mathbf{s}_0$ is the prediction location;

N is the number of measured values.

In IDW, the weight, λ_i, depends solely on the distance to the prediction location. However, in Kriging, the weights are based not only on the distance between the measured points and the prediction location but also on the overall spatial arrangement among the measured points. To use the spatial arrangement in the weights, the spatial autocorrelation must be quantified. Thus, in Ordinary Kriging, the weight, λ_i, depends on a fitted model to the measured points, the distance to the prediction location, and the spatial relationships among the measured values around the prediction location.

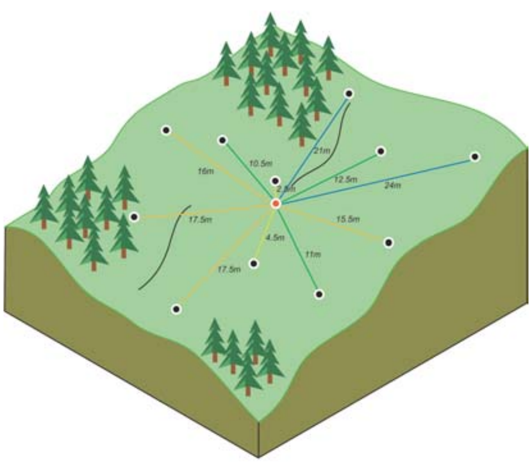
To make a prediction with Kriging, two tasks are necessary: (1) to uncover the dependency rules and (2) to make the predictions. To realize these two tasks, Kriging goes through a two-step process: (1) the creation of variograms and covariance functions to estimate the statistical dependence—called spatial autocorrelation—values, which depends on your model of autocorrelation—fitting a model—and (2) actually predicting the unknown values—making a prediction. It is because of these two distinct tasks that it has been said that Kriging uses the data twice: the first time to estimate the spatial autocorrelation of the data and the second time to make the predictions.

Variography

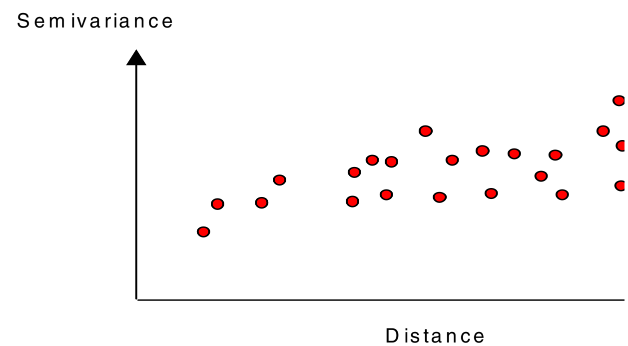
Fitting a model or spatial modeling is also known as structural analysis or variography. In spatial modeling of the structure of the measured points, you will begin with a graph of the empirical semivariogram, computed as:

Semivariogram(distance h) = 0.5 * average[(value at location i – value at location j)^2]

for all pairs of locations separated by distance h. The formula involves calculating the difference squared between the values of the paired locations. The image below shows the pairing of one point (the red point) with all other measured locations. This process continues for each measured point.



Often each pair of locations has a unique distance, and there are often many pairs of points. To plot all pairs quickly becomes unmanageable. Instead of plotting each pair, the pairs are grouped into *lag bins*. For example, compute the average semivariance for all pairs of points that are greater than 40 meters apart but less than 50 meters. The empirical semivariogram is a graph of the averaged semivariogram values on the y-axis and the distance, or lag, on the x-axis (see diagram below).



Spatial autocorrelation quantifies a basic principle of geography; things that are closer are more alike than things farther apart. Thus, pairs of locations that are closer—far left on the x-axis of the semivariogram cloud—should have more similar values—low on the y-axis of the semivariogram cloud. As pairs of locations become farther apart—moving to the right on the x-axis of the semivariogram cloud—they should become more dissimilar and have a higher squared difference—move up on the y-axis of the semivariogram cloud.

Fitting a model to the empirical semivariogram

The next step is to fit a model to the points forming the empirical semivariogram. Semivariogram modeling is a key step between spatial description and spatial prediction. The main application of Kriging is the prediction of attribute values at unsampled locations. You have seen how the empirical semivariogram provides information on the spatial autocorrelation of datasets. However, it does not provide information for all possible directions and distances. For this reason, and to ensure that Kriging predictions have positive Kriging variances, it is necessary to fit a model (i.e., a continuous function or curve) to the empirical semivariogram. Abstractly, this is similar to regression analysis, where a continuous line or curve is fitted.

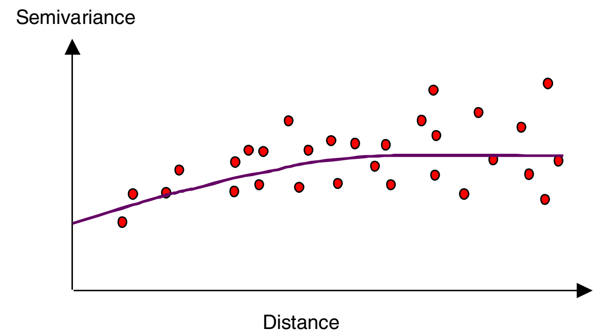
Select some function that serves as your model, for example, a spherical type that rises at first and then levels off for larger distances beyond a certain range (see previous image). There are deviations of the points on the empirical semivariogram from the model; some points are above the model curve, and some points are below. But if you add the distance each point is above the line and add the distance each point is below the line, the two values should be similar. There are a lot of different semivariogram models to choose from.

Different types of semivariogram models

3D Analyst provides the following functions to choose from to model the empirical semivariogram: Circular, Spherical, Exponential, Gaussian, and Linear. The selected model influences the prediction of the unknown values, particularly when the shape of the curve near the origin differs significantly. The steeper the curve is near the origin, the more influence the closest neighbors will have on the prediction. As a result, the output surface will be less smooth. Each model is designed to fit different types of phenomenon more accurately.

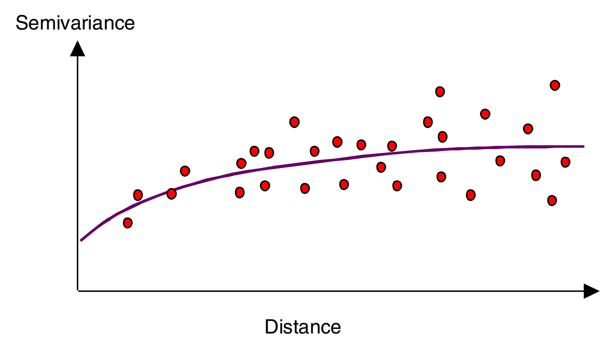
The diagrams below show two common models and identify how the functions differ:

- The Spherical Model



This model shows a progressive decrease of spatial autocorrelation—equivalently, an increase of semivariance—until some distance, beyond which autocorrelation is zero. The spherical model is one of the most commonly used models.

- The Exponential Model



This model is applied when spatial autocorrelation decreases exponentially with increasing distance. Here the autocorrelation disappears completely only at an infinite distance. The exponential model is also a commonly used model.

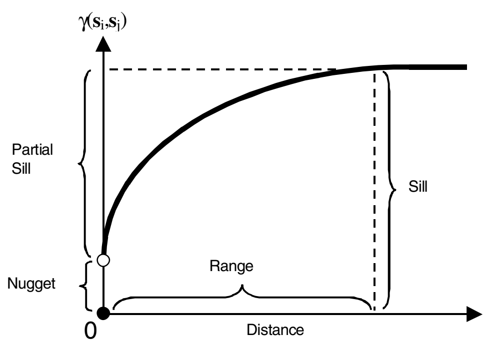
The choice of which model to use in 3D Analyst is based on the spatial autocorrelation of the data and on prior knowledge of the phenomenon.

Understanding a semivariogram—the range, sill, and nugget

As previously discussed, the semivariogram depicts the spatial autocorrelation of the measured sample points. Because of a basic principle of geography—things that are closer are more alike—measured points that are close will generally have a smaller difference squared than those farther apart. Once each pair of locations is plotted—after being binned—a model is fit through them. There are certain characteristics that are commonly used to describe these models.

The range and sill

When you look at the model of a semivariogram, you will notice that at a certain distance the model levels out. The distance where the model first flattens out is known as the range.



Sample locations separated by distances closer than the range are spatially autocorrelated, whereas locations farther apart than the range are not.

The value at which the semivariogram model attains the range—the value on the y-axis—is called the sill. The partial sill is the sill minus the nugget (see the following section).

The nugget

Theoretically, at zero separation distance (i.e., lag = 0), the semivariogram value is zero. However, at an infinitely small separation distance, the semivariogram often exhibits a nugget effect, which is some value greater than zero. If the semivariogram model intercepts the y-axis at 2, then the nugget is 2.

The nugget effect can be attributed to measurement errors or spatial sources of variation at distances smaller than the sampling interval, or both. Measurement error occurs because of the error inherent in measuring devices. Natural phenomena can vary spatially over a range of scales (i.e., micro or macro scales). Variation at micro scales smaller than the sampling distances will appear as part of the nugget effect. Before collecting data, it is important to gain some understanding of the scales of spatial variation that you are interested in.

Making a prediction

The first task of uncovering the dependence (autocorrelation) in your data has been accomplished. You have also finished with the first use of the data, where the spatial information in the data—to compute distances—is used to model the spatial autocorrelation. Once you have the spatial autocorrelation, you proceed with prediction using the fitted model; thereafter, the empirical semivariogram is set aside.

For the second task, you use the data again to make predictions. Like IDW interpolation, Kriging forms weights from surrounding measured values to predict at unmeasured locations. As with IDW interpolation, the measured values closest to the unmeasured locations have the most influence. However, the Kriging weights for the surrounding measured points are more sophisticated than those of IDW. IDW uses a simple algorithm based on distance, but kriging weights come from a semivariogram that was developed by looking at the spatial nature of the data. To create a continuous surface or map of the phenomenon, predictions are made for each location (cell centers) in the study area based on the semivariogram and the spatial arrangement of measured values that are nearby.

Search radius

You can assume that as the locations get farther from the prediction location, the measured values will have less spatial autocorrelation with the unknown value for the location you are predicting. Thus, you can eliminate those farther locations with little influence. Not only is there less relationship with farther locations, but it is also possible that the farther locations may have a negative influence if they are located in an area much different than the prediction location. Another reason to use search neighborhoods is for computational speed. The smaller the search neighborhood, the faster the predictions can be made. As a result, it is common practice to limit the number of points that are used when making a prediction by specifying a search neighborhood. The specified shape of the neighborhood restricts how far and where to look for the measured values to be used in the prediction. Other neighborhood parameters restrict the locations that will be used within that shape so, for example, you can define the maximum and minimum number of measured points to use within the neighborhood.

Using the configuration of the valid points within the specified neighborhood around the prediction location in conjunction with the model fit to the semivariogram, the weights for the measured locations can be determined. From the weights and the values, a prediction can be made for the unknown value at the prediction location.

3D Analyst has two neighborhood types, fixed and variable.

Fixed search radius

A fixed search radius requires a distance and a minimum number of points. The distance dictates the radius of the circle of the neighborhood (in map units). The distance of the radius is constant, so for each interpolated cell, the radius of the circle used to find input points is the same. The minimum number of points indicates the minimum number of measured points to use within the neighborhood. All the measured points that fall within the radius will be used in the calculation of each interpolated cell. When there are fewer measured points in the neighborhood than the specified minimum, the search radius will increase until it can encompass the minimum number of points. The specified fixed search radius will be used for each interpolated cell (cell center) in the study area, thus if your measured points are not spread out equally, which they rarely are, there likely will be a different number of measured points used in the different neighborhoods for the various predictions.

Variable search radius

With a variable search radius, the number of points used in calculating the value of the interpolated cell is specified, which makes the radius distance vary for each interpolated cell, depending on how far it has to search around each interpolated cell to reach the specified number of input points. Thus, some neighborhoods can be small and others can be large depending on the density of the measured points near the interpolated cell. You can also specify a maximum distance—in map units—that the search radius cannot exceed. If the radius for a particular neighborhood reaches the maximum radius before obtaining the specified number of points, the prediction for that location will be performed on the number of measured points within the maximum radius.

Kriging methods

3D Analyst provides two kriging methods: Ordinary and Universal Kriging.

Ordinary Kriging

Ordinary Kriging is the most general and widely used of the kriging methods. It assumes the constant mean is unknown. This is a reasonable assumption unless there is some scientific reason to reject this assumption.

Universal Kriging

Universal Kriging assumes that there is an overriding trend in the data (i.e., a prevailing wind) and it can be modeled by a deterministic function, a polynomial. This polynomial is subtracted from the original measured points, and the autocorrelation is modeled from the random errors. Once the model is fit to the random errors, before making a prediction, the polynomial is added back to the predictions to give you meaningful results. Universal Kriging should only be used when you know there is a trend in your data and you can give a scientific justification to describe it.

Interpolating a raster surface

You can create a new raster surface from input point data using four different interpolation methods. Each method has parameters that you can modify to create a raster that suits your needs.

Tip

What is the power?

The power is the exponent of the distance used in IDW interpolation. Higher numbers result in a lower influence of distant points on each processing cell. Lower numbers result in more smoothing of the surface. Reasonable power values range between 0.5 and 3.

Tip

What is a variable radius?

A variable radius interpolation uses the closest n points that it finds within the maximum distance of the output cell as input. In contrast, a fixed radius uses all of the points within the specified distance.

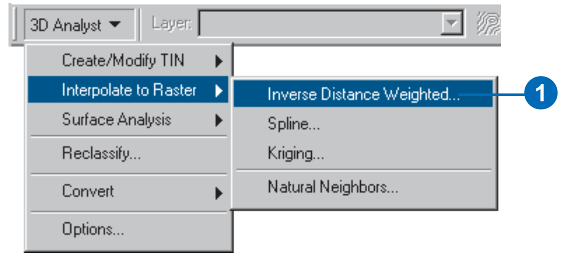
Tip

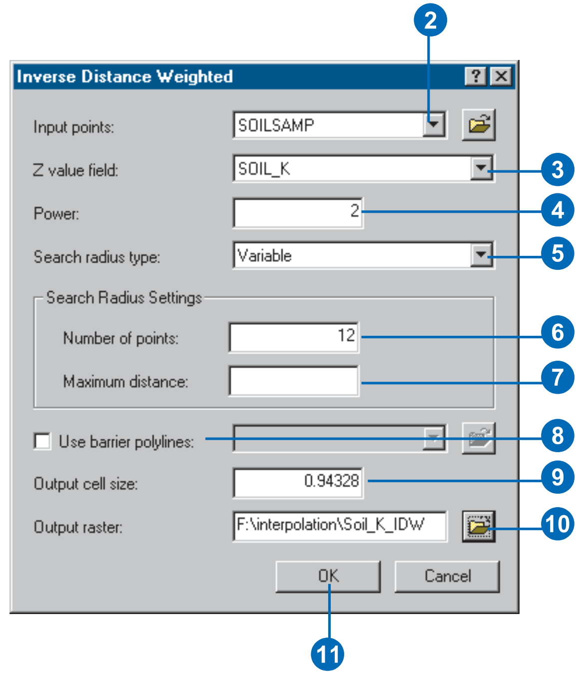
What are barriers?

A feature class of linear features, such as faults or cliffs, can be used to limit the search for input points for the interpolation of each output cell.

Interpolating using Inverse Distance Weighted with a variable search radius

1. Click 3D Analyst, point to Interpolate to Raster, and click Inverse Distance Weighted.
2. Click the dropdown arrow and choose the input point data source.
3. Click the field that contains the attribute data that you want to interpolate.
4. Type the power.
5. Click the dropdown arrow and click Variable.
6. Type the number of points to use as input from within the maximum distance.
7. Type the maximum distance to search for points to use in the interpolation.
8. Optionally, check the box and select or navigate to a feature class to use as barriers to interpolation.
9. Type a cell size for the output raster.
10. Optionally, type a name for the Output raster. If you do not, the raster will be temporary.
11. Click OK.





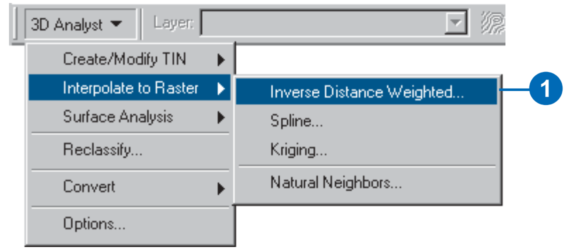
Tip

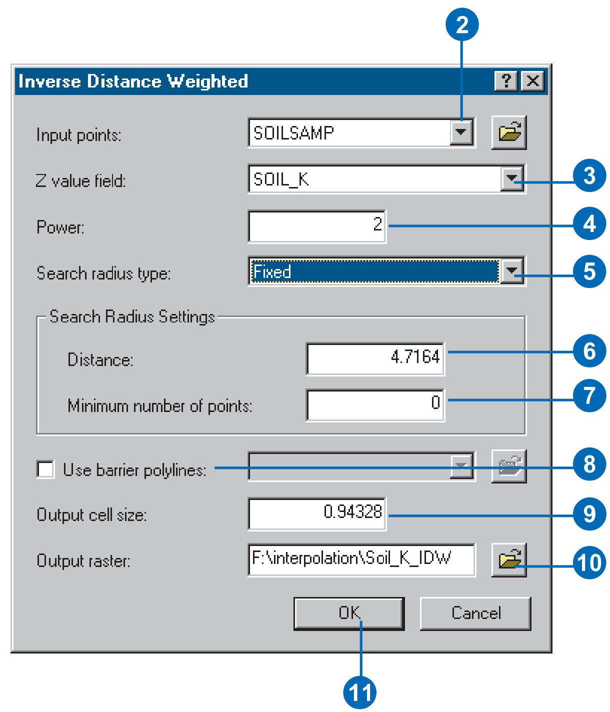
What is the Minimum Count?

A fixed radius interpolation uses all of the points within the specified distance. If no points are found within the search radius, the search radius will increase for the cell until the specified minimum number of points is found.

Interpolating using Inverse Distance Weighted with a fixed search radius

1. Click 3D Analyst, point to Interpolate to Raster, and click Inverse Distance Weighted.
2. Click the dropdown arrow and choose the input point data source.
3. Click the dropdown arrow and choose the field that contains the attribute data that you want to interpolate.
4. Type the power.
5. Click the dropdown arrow and click Fixed.
6. Type the distance within which to search for input points.
7. Type the minimum number of points that must be included in the interpolation for a cell.
8. Optionally, check the box and select or navigate to a feature class to use as barriers to interpolation.
9. Type a cell size for the output raster.
10. Optionally, type a name for the Output raster. If you do not, the raster will be temporary.
11. Click OK.





Tip

What is a Tension spline?

A spline interpolation is a method of fitting a surface to a set of points. Think of the surface as a thin plate of slightly elastic material deformed to fit the points. In a Tension spline, you can control the elasticity of the surface.

Tip

What is the weight value?

The weight value in a Tension spline adjusts the elasticity of the surface. A weight of 0 results in a basic thin-plate spline interpolation. Larger values increase the elasticity of the surface. Typical weight values are 0, 1, 5, and 10.

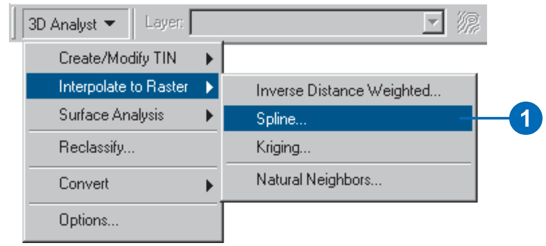
Tip

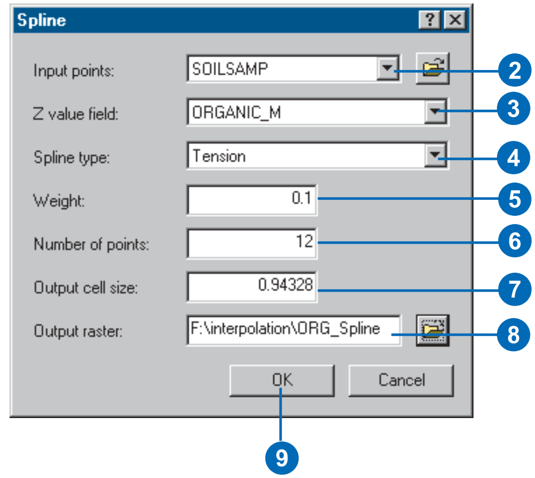
What is the number of points?

The number of points controls the average number of points to be contained in each region used in computing the surface. The regions are rectangles of equal size, and the number of regions is determined by dividing the total number of input points by the number of points. When the data is not uniformly distributed, the actual number of points in a region may be different from the number of points. If the number of points in a region is less than eight, the region is expanded until it contains eight points.

Interpolating using a Tension spline

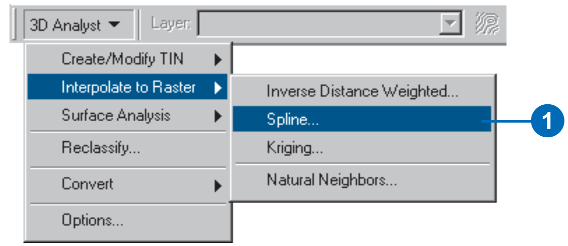
1. Click 3D Analyst, point to Interpolate to Raster, and click Spline.
2. Click the dropdown arrow and choose the input point data source.
3. Click the dropdown arrow and choose the field that contains the attribute data that you want to interpolate.
4. Click the dropdown arrow and click Tension.
5. Type a weight.
6. Type the minimum number of points that must be included in the interpolation for a cell.
7. Type a cell size for the output raster.
8. Optionally, type a name for the Output raster. If you do not, the raster will be temporary.
9. Click OK.

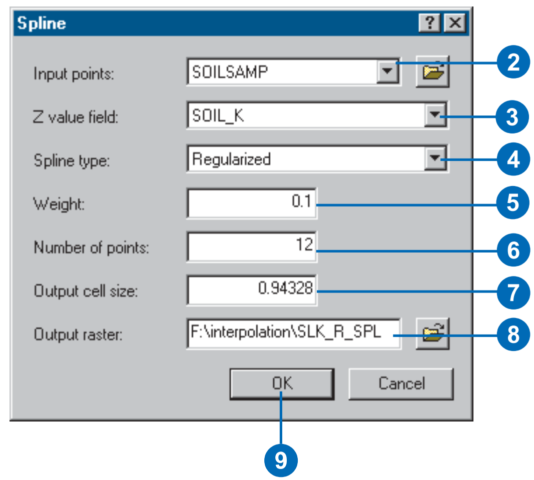




Interpolating using a Regularized spline

1. Click 3D Analyst, point to Interpolate to Raster, and click Spline.
2. Click the dropdown arrow and choose the input point data source.
3. Click the dropdown arrow and choose the field that contains the attribute data that you want to interpolate.
4. Click the dropdown arrow and click Regularized.
5. Type a weight.
6. Type the minimum number of points that must be included in the interpolation for a cell.
7. Type a cell size for the output raster.
8. Optionally, type a name for the Output raster. If you do not, the raster will be temporary.
9. Click OK.





Tip

What is a Regularized spline?

A Regularized spline lets you control the smoothness of the surface. Regularized splines are useful if you will need to calculate the second derivative of the interpolated surface.

Tip

What is the weight?

The weight value in a Regularized spline adjusts the smoothness of the surface. The weight specifies the weight (tau) attached to the third derivative term, used in minimizing the curvature of the surface. Larger values result in smoother surfaces and smooth first-derivative (slope) surfaces. Values between 0 and 0.5 are suitable.

Tip

What is the number of points?

The number of points controls the average number of points contained in each region used in computing the surface. The regions are rectangles of equal size; the number of regions is determined by dividing the total number of input points by the number of points. When the data is not uniformly distributed, the actual number of points in a region may be different from the number of points. If the number of points in a region is less than eight, the region is expanded until it contains eight points.

Kriging interpolation

There are two kriging methods: Ordinary and Universal.

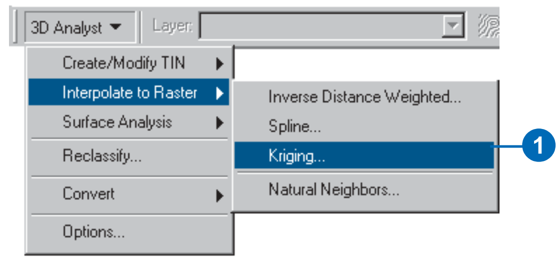
Ordinary Kriging is the most general and widely used of the kriging methods and is the default. It assumes the constant mean is unknown. Universal Kriging should only be used when you know there is a trend in your data and you can give a scientific justification to describe it.

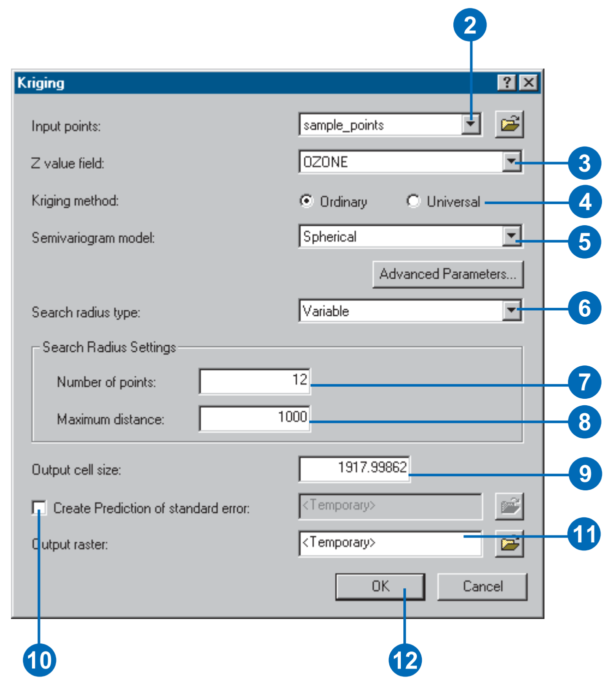
By using a variable search radius, you can specify the number of points to use in calculating the value of the interpolated cell. This makes the search radius variable for each interpolated cell, depending on how far it has to stretch to reach the specified number of input points.

Specifying a maximum distance limits the potential size of the radius of the circle. If the number of points is not reached before the maximum distance of the radius is reached, fewer points will be used in the calculation of the interpolated cell. ▸

Creating a surface using Kriging interpolation with a variable radius

1. Click the 3D Analyst dropdown arrow, point to Interpolate to Raster, and click Kriging.
2. Click the dropdown arrow and click the Input points dataset.
3. Click the Z value field dropdown arrow and click the field you wish to use.
4. Click the Kriging method you wish to use.
5. Click the Semivariogram model dropdown arrow and click the model you wish to use.
6. Click the Search radius type dropdown arrow and click Variable.
7. Optionally, change the default Number of points.
8. Optionally, specify a Maximum distance.
9. Optionally, change the default Output cell size.
10. Optionally, check the Create Prediction of standard error check box.
11. Specify a name for the outputs or leave the default to create a temporary dataset in your working directory.
12. Click OK.





With a Fixed radius, the radius of the circle used to find input points is the same for each interpolated cell. The default radius is five times the cell size of the output grid. By specifying a minimum number of points, you can ensure that within the fixed radius, at least a minimum number of input points will be used in the calculation of each interpolated cell.

Tip

Deciding on the radius or the number of points

Use the measure tool on the Tools toolbar to measure distance between points to get an idea of the radius and number of points to use.

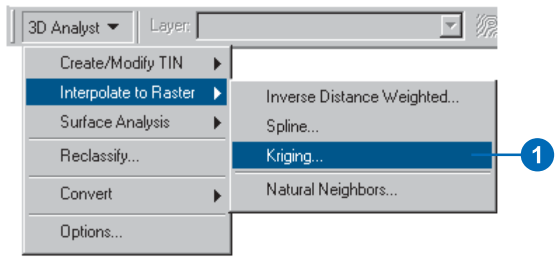
Tip

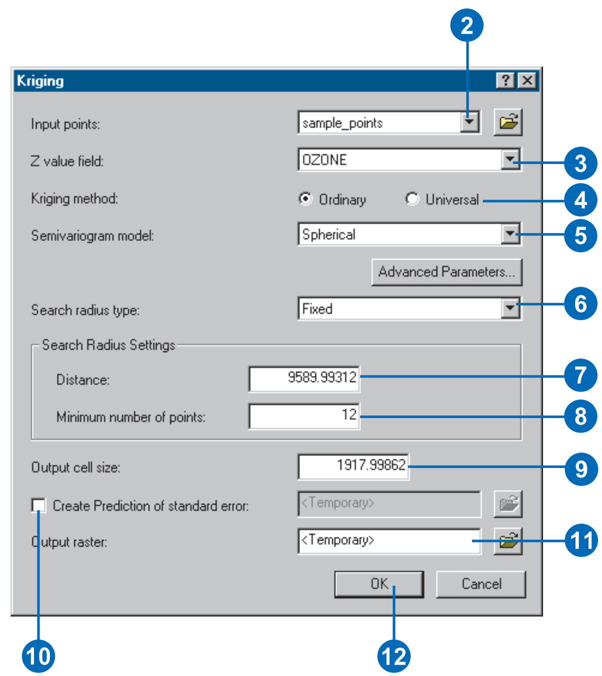
Changing the lag size, major range, partial sill, and nugget

Click Advanced Parameters on the Kriging dialog box to modify these parameters.

Creating a surface using Kriging interpolation with a fixed radius

1. Click the 3D Analyst dropdown arrow, point to Interpolate to Raster, and click Kriging.
2. Click the dropdown arrow and click the Input points dataset.
3. Click the Z value field dropdown arrow and click the field you wish to use.
4. Click the Kriging method you wish to use.
5. Click the Semivariogram model dropdown arrow and click the model you wish to use.
6. Click the Search radius type dropdown arrow and click Fixed.
7. Optionally, change the default Distance for the search radius setting.
8. Optionally, change the Minimum number of points.
9. Optionally, change the default Output cell size.
10. Optionally, check the Create Prediction of standard error check box.
11. Specify a name for the outputs or leave the default to create a temporary dataset in your working directory.
12. Click OK.





Tip

What is a Natural Neighbors interpolation?

A Natural Neighbors interpolation combines some TIN functionality with the raster interpolation process. The raster surface is interpolated using the input data points that are natural neighbors of the cell.

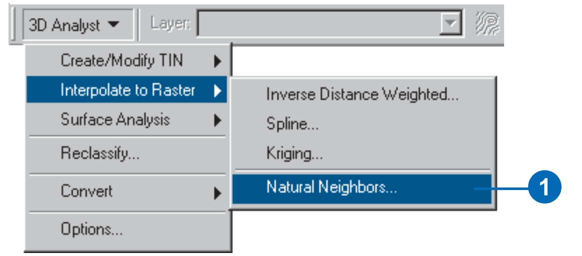
The interpolation method first creates a Delauney Triangulation of the input points (part of the TIN creation process); the points are nodes in a triangulation that follows the rule that a circle around each triangle can contain no other nodes. This means that, collectively, the triangles are constrained to be as close to equilateral triangles as possible.

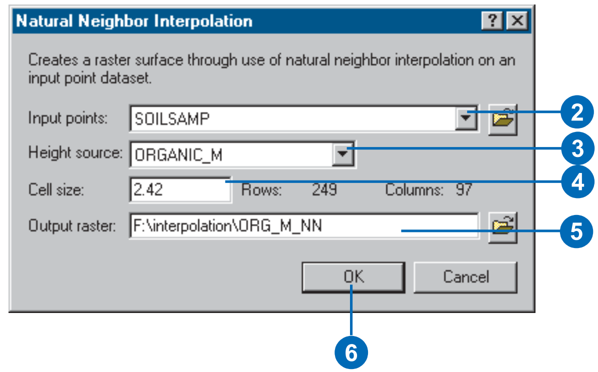
For each data point, the natural neighbors are the minimum number of nodes in the triangulation that connect to form a convex hull around the point.

The weight of each neighbor point is determined through the use of a Thiessen/Voroni technique that evaluates its area of influence.

Interpolating using Natural Neighbors

1. Click 3D Analyst, point to Interpolate to Raster, and click Natural Neighbors.
2. Click the dropdown arrow and choose the Input points data source.
3. Click the dropdown arrow and click the height source.
4. Type a cell size for the output raster.
5. Type a name for the Output raster.
6. Click OK.





Saving all rasters in a specified location

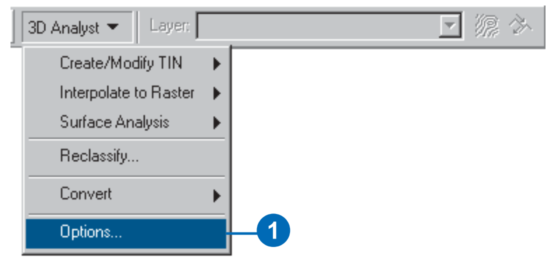
You can specify a folder where 3D Analyst will save output temporary rasters. Temporary rasters are automatically deleted when you exit the application where they were created, unless you save a layer file, scene, or map document that refers to the raster.

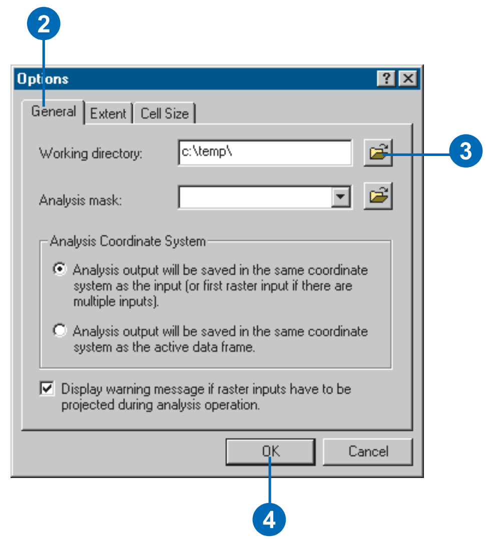
There are two ways to specify where your analysis results should go. The first way is to specify a location on disk for your results using the Analysis Options dialog box before you perform any analysis. This way all your analysis results will go to this directory. The second way is to specify a location on disk each time you perform analysis in each of the function dialog boxes. This is useful if you want to sort your analysis results into different folders.

By default, temporary rasters are saved to your system's temporary directory, usually C:\temp.

1. Click 3D Analyst and click Options.
2. Click the General tab.
3. Type the name of the new working directory.

   Alternatively, you can browse to your new working directory.
4. Click OK.





Setting an analysis mask

An analysis mask is a raster that allows an analysis operation to be performed only on certain cells of interest in another raster. It contains selected cells that are of interest for future processing and other cells that are to be masked out. The masked-out cells should have a value of nodata.

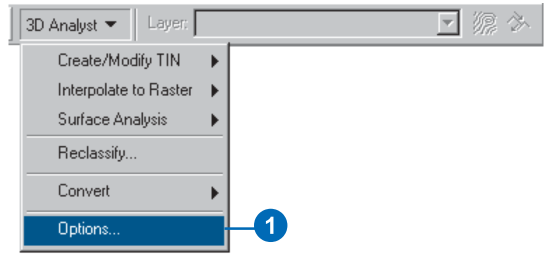
Setting an analysis mask means that processing will only occur on selected cells and that all other cells will be assigned values of NoData.

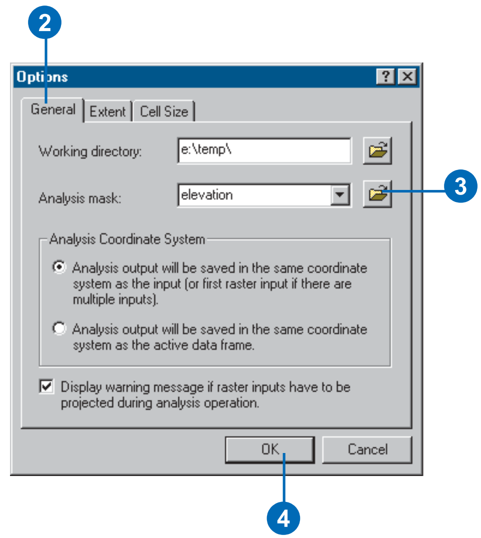
Setting an analysis mask is a two-step process:

1. The analysis mask must first be created using the Reclassify dialog box. See the section on reclassifying rasters for more information about creating an analysis mask.
2. The analysis mask must then be specified in the General tab of the Analysis Options dialog box in order for it to be used in all subsequent analysis.

Analysis masks will work on the output of the surface analysis tools in 3D Analyst.

1. Click 3D Analyst and click Options.
2. Click the General tab.
3. Browse to the raster that will be the analysis mask.
4. Click OK.

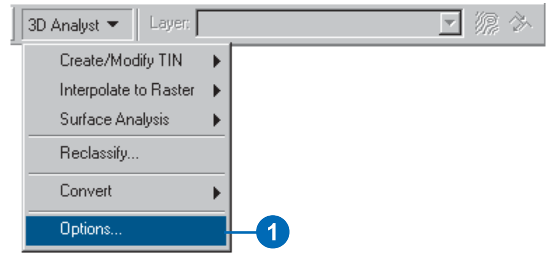


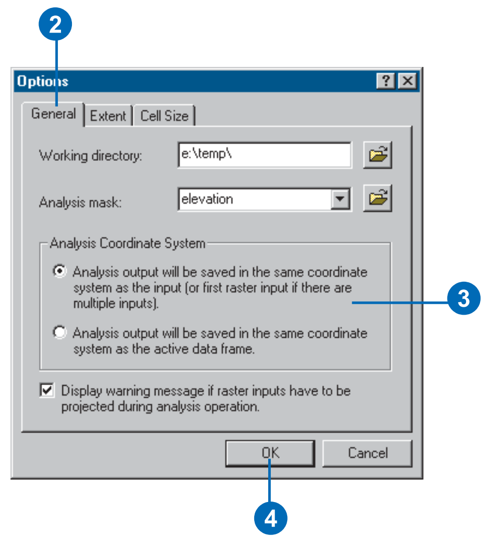


Setting the coordinate system for your analysis results

Rasters with different coordinate systems can be projected on the fly into a single data frame with a given coordinate system for display and analysis in ArcMap. When you have rasters in more than one coordinate system in your scene or map, you need to specify the coordinate system for the output of the analysis tools.

1. Click 3D Analyst and click Options.
2. Click the General tab.
3. Click the Analysis Coordinate System option you wish to use to save your analysis results into either the coordinate system of the input raster or the coordinate system of the data frame.
4. Click OK.





Setting the output extent

The extent of a layer is the x,y coordinates for the bottom-left and the top-right corners.

You can control how the output raster extent is determined for the Interpolation and Analysis tools. By default, the output extent is the same as the input dataset's extent.

You may only wish to perform analysis on the area visible in the map ('Same As Display'), or you may wish to make the extent the same as another layer in the table of contents ('Same As Layer'). Alternatively, you can specify a custom extent ('As Specified Below').

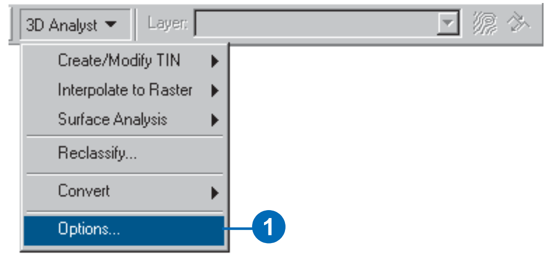
Tip

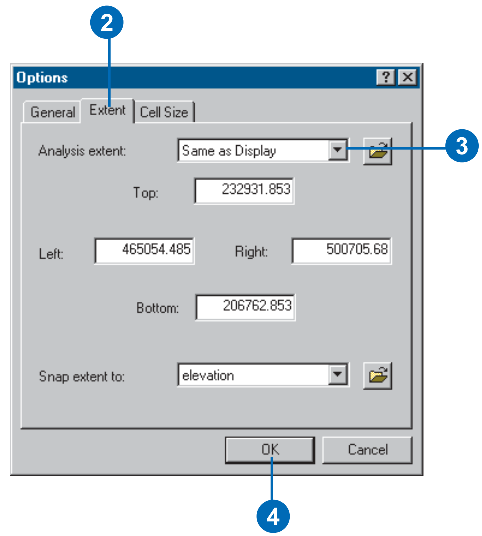
Setting the Snap extent

Setting the Snap extent to a specific grid will snap all layers to the cell registration of the specified grid. All layers will share the lower left corner and cell size of the specified grid. Use this to resample layers to the same registration and cell size in order to perform analysis.

Specifying an extent for analysis results

1. Click 3D Analyst and click Options.
2. Click the Extent tab.
3. Click the Analysis extent dropdown arrow and choose an option to specify the extent for all subsequent analysis results.
4. Click OK.





Setting the output cell size

You can control the default size of the cells in your output rasters. By default, the output cell size is the same as the input datasets cell size. You can change the output cell size to be the cell size of a dataset in the current scene or map or a cell size that you define.

Exercise caution when specifying a cell size finer than the input grids. No new data is created; cells are interpolated using nearest-neighbor resampling. The result is as precise as the coarsest input.

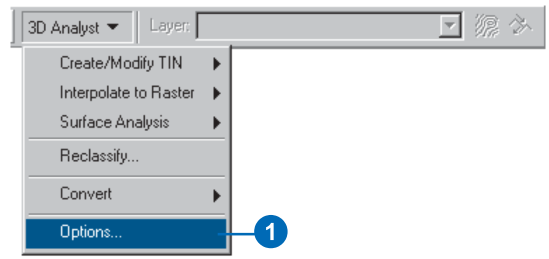
Tip

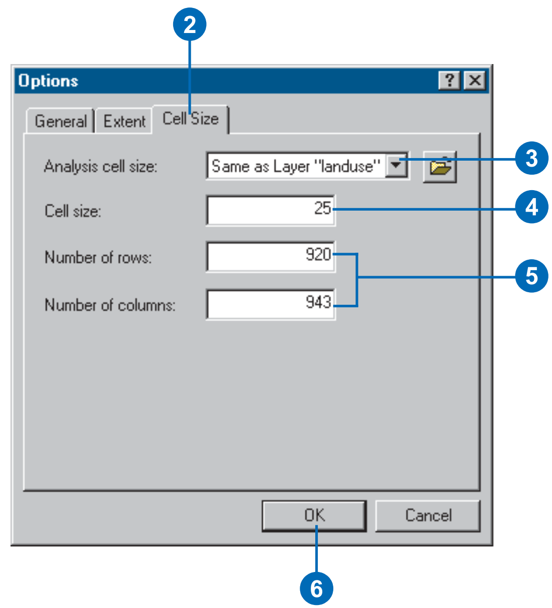
Overriding the default cell size

Even if you've set a particular cell size on the Options dialog box, you can override the cell size for a particular analysis tool by setting the cell size on the tool.

Setting the cell size

1. Click 3D Analyst and click Options.
2. Click the Cell Size tab.
3. Click the dropdown arrow and choose the method of determining the cell size.
4. Optionally, type the number of map units to define the cell size.
5. Optionally, type the number of rows and the number of columns to define a cell size.
6. Click OK.



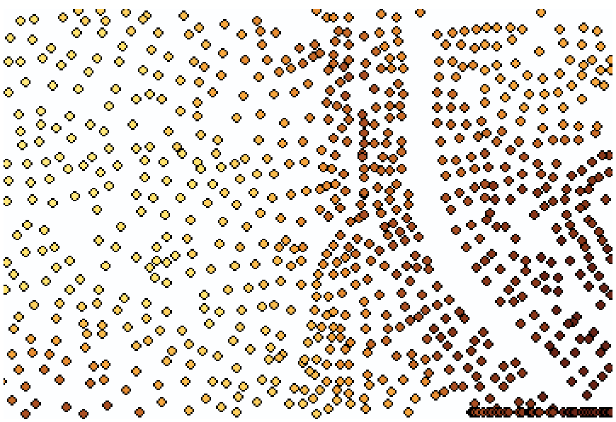


Creating TIN surfaces from vector data

TINs are usually created from a combination of vector data sources. You can use point, line, and polygon features as input data for a TIN. Some of these input features should have z-values, though not all of the features need z-values. The input features to a TIN may also contain integer attribute values that are preserved in the resulting TIN features. These may be used to indicate the relative accuracy of different input data or to identify features, such as roads or lakes.

Building a TIN

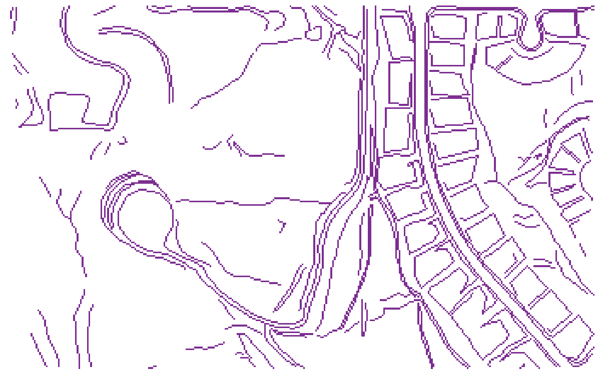
You can create a TIN all at once from one or more kinds of input data, you can create it in stages, or you can add data to refine an existing TIN. TINs are made from mass points, breaklines, and polygons. Mass points are point height measurements; they become nodes in the network. Mass points are the primary input into a TIN; they determine the overall shape of the surface.



Mass points, categorized by height attribute

TINs allow you to model heterogeneous surfaces efficiently by including more mass points in areas where the surface is highly variable and fewer in places where the surface is less variable.

Breaklines are lines with or without height measurements. They become sequences of one or more triangle edges.



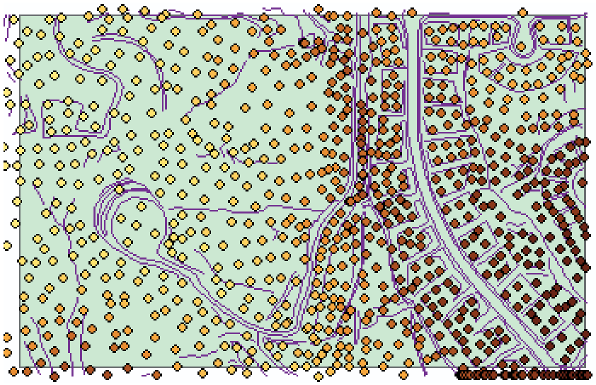
Breaklines

Breaklines typically represent either natural features, such as ridgelines or streams, or built features, such as roadways. There are two kinds of breaklines: hard and soft.

Hard breaklines represent a discontinuity in the slope of the surface. Streams and road cuts could be included in a TIN as hard breaklines. Hard breaklines capture abrupt changes in a surface and improve the display and analysis of TINs.

Soft breaklines let you add edges to a TIN to capture linear features that do not alter the local slope of a surface. Study area boundaries could be included in a TIN as soft breaklines to capture their position, without affecting the shape of the surface.

Polygons represent surface features with area—such as lakes—or boundaries—also called hulls—of separately interpolated areas.



Clip polygon

Hulls could define the shores of individual islands in an archipelago, or the boundary of a study area.

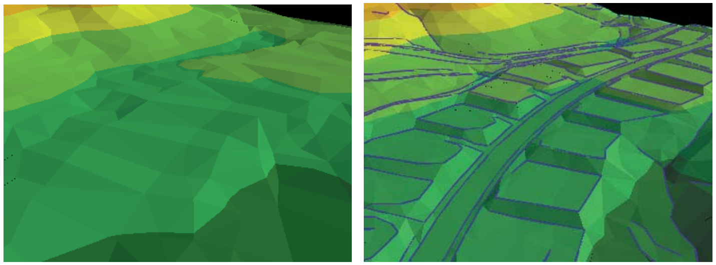
There are four polygon surface feature types: clip polygons, erase polygons, replace polygons, and fill polygons.

Clip polygons define a boundary for interpolation. Input data that falls outside of the clip polygon is excluded from the interpolation and analysis operations—for example, contouring or volume calculations.

Erase polygons define a boundary for interpolation. Input data that falls within the erase polygon is excluded from the interpolation and analysis operations—for example, contouring or volume calculations.

Replace polygons set the boundary and all interior heights to the same value. A replace polygon could be used to model a lake or an area on a slope excavated to a level surface.

Fill polygons assign an integer attribute value to all triangles that fall within the fill polygon. The surface height is unaffected, and no clipping or erasing takes place.



The graphic on the left shows a TIN created from mass points; the graphic on the right shows a TIN of the same area created from mass points and breaklines.

Polygon features are integrated into the triangulation as closed sequences of three or more triangle edges.

Including breaklines and polygons in a TIN gives you more control over the shape of the TIN surface.

To get a sense of the difference that breaklines can make in a TIN, compare the surface created from mass points alone to the surface created from mass points and breaklines.

Building a TIN

TINs are often created from a group of vector data sources. You can use point, line, and polygon features as input data for a TIN. Some of these input features should have z-values, though not all of the features need z-values.

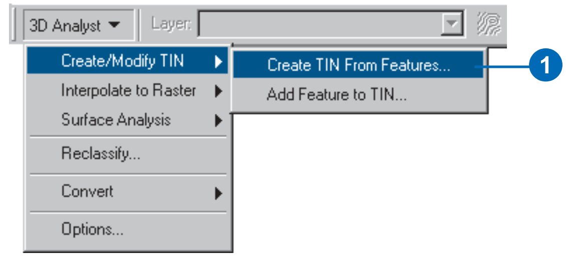
You can create a new TIN in ArcMap or ArcScene. You can use features that are in the map or scene, or you can navigate to feature classes that aren't on the map to include them in a TIN.

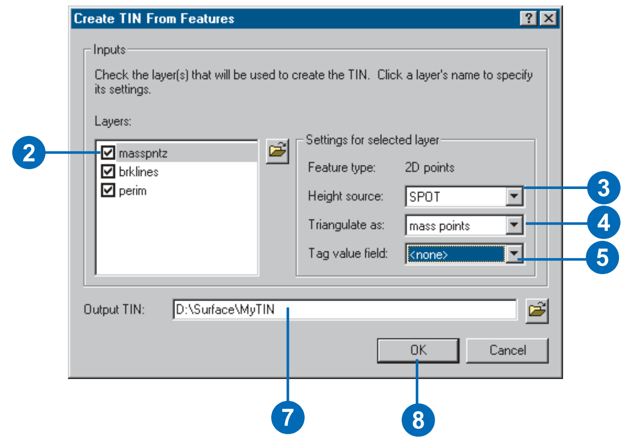
Creating a TIN

1. Click 3D Analyst, point to Create/Modify TIN, and click Create TIN From Features.
2. Check the features that you want to use in building your TIN.

   You can use the browse button beside the Layers list to navigate to other feature classes to include in the TIN.
3. Click the dropdown arrow and choose the Height source field.

   You can choose shape geometry if the features have 3D geometry.
4. Click the dropdown arrow and choose how the features should be incorporated into the TIN—as mass points, breaklines, or polygons.
5. Optionally, click the dropdown arrow and choose the Tag value field if you wish to tag the TIN features with a value from the input features.
6. Repeat steps 2 through 5 for each input feature class.
7. Type a name for the TIN.
8. Click OK.





Tip

Adding the selected features to a TIN

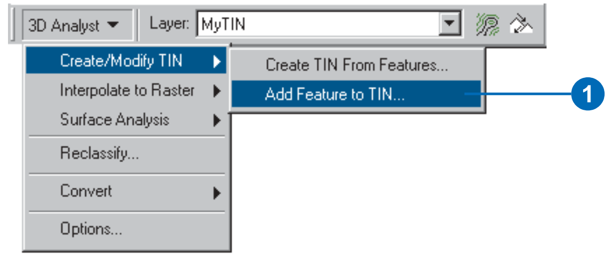
You can add features to refine a TIN that has already been created, rather than re-creating the TIN from scratch. If a feature class has some features selected, only those features will be included in the TIN.

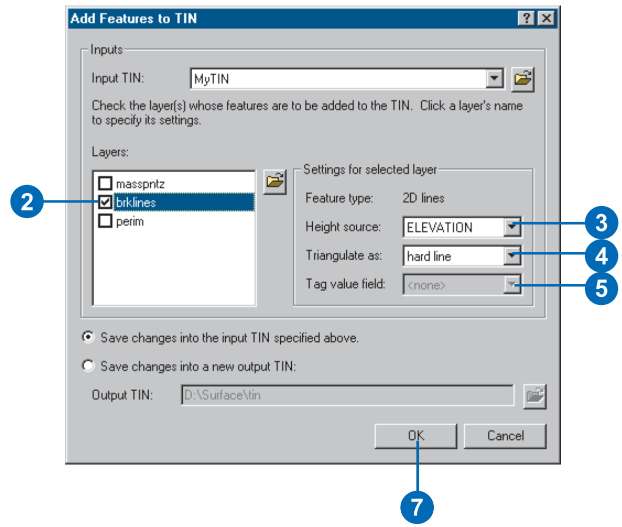
Adding features to a TIN

1. Click 3D Analyst, point to Create/Modify TIN, and click Add Feature to TIN.
2. Check the features that you want to add to the TIN.

   You can use the Browse button beside the Layers list to navigate to other feature classes to include in the TIN.
3. Click the dropdown arrow and choose the Height source field.

   You can choose shape geometry if the features have 3D geometry.
4. Select how the features should be incorporated into the TIN—as mass points, breaklines, or polygons.
5. Optionally, click the dropdown arrow and choose the Tag value field if you wish to tag the TIN features with a value from the input features.
6. Repeat steps 2 through 5 for each input feature class.
7. Click OK.





Creating a TIN from a raster

You can convert a raster surface to a TIN for use in surface modeling or to simplify the surface model for visualization. Converting to a TIN can also allow you to enhance your surface model by adding features such as streams and roads that are not represented in the original raster.

When you convert a raster to a TIN, you need to specify the vertical accuracy of the output TIN with respect to the original raster. 3D Analyst will select the subset of points needed to achieve this level of accuracy.

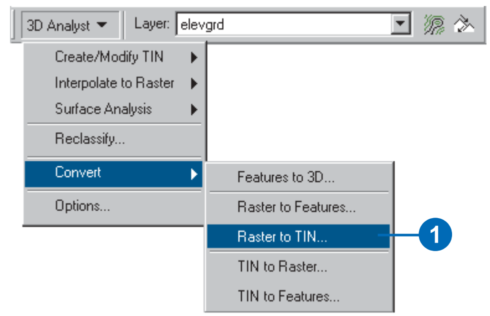
Tip

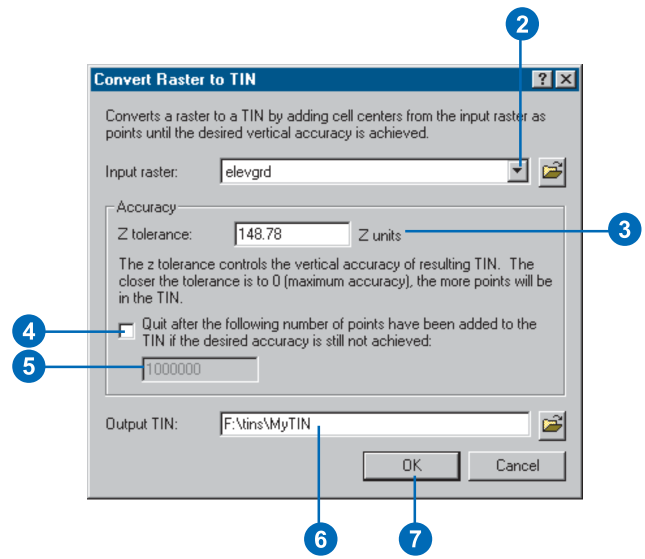
What is the vertical accuracy?

The vertical accuracy is the maximum number of units that the TIN surface may differ from the cell center heights of the input raster. Specifying a low number results in a TIN that preserves more of the detail of the raster surface. A higher number will result in a more generalized representation of the surface.

Converting a raster to a TIN

1. Click 3D Analyst, point to Convert, and click Raster to TIN.
2. Click the dropdown arrow and click the raster that you want to convert to a TIN.
3. Type a vertical accuracy for the TIN.
4. Optionally, check the box to limit the number of points added to the TIN in pursuit of accuracy.
5. Optionally, type the maximum number of points to add to the TIN in pursuit of accuracy.
6. Type a name for the Output TIN.
7. Click OK.





Creating a raster from a TIN

You can convert a TIN to a raster to use in surface modeling or to extract slope or aspect information from the TIN.

Tip

What is the Z factor?

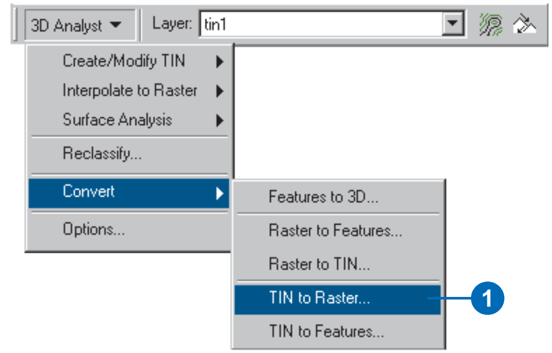
The Z factor is used to convert the z units to the same scale as the x and y units, if they are different.

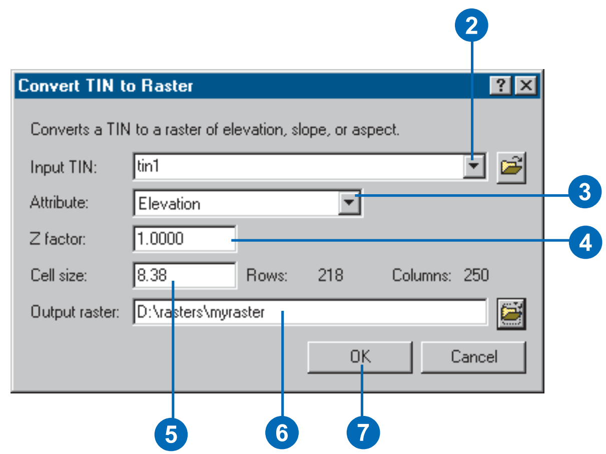
Converting a TIN to a raster

1. Click 3D Analyst, point to Convert, and click TIN to Raster.
2. Click the dropdown arrow and click the TIN that you want to convert to a raster.
3. Click the Attribute dropdown arrow and click the attribute of the TIN that you want to save as a raster.

   You can choose from elevation, aspect, slope in percentage, and slope in degrees.

   The TIN must have already been symbolized by the attribute in your scene or map.
4. Optionally, set a Z factor.
5. Optionally, type the Cell size for the raster.
6. Type a name for the Output raster.
7. Click OK.





Managing 3D data

4

IN THIS CHAPTER

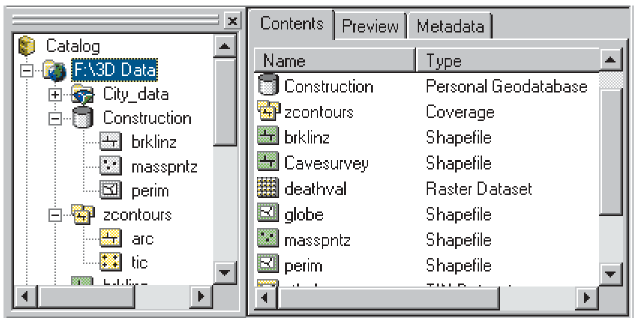
- **ArcCatalog basics**
- **Previewing 3D data**
- **The camera, the observer, and the target**
- **Displaying 2D data in 3D**
- **Starting ArcScene from ArcCatalog**
- **Creating a new 3D feature class**

ArcCatalog allows you to manage your 3D data in the same way that you manage all of your other GIS data. You can use ArcCatalog to organize your work—copy, paste, move, and delete 3D data in the Catalog tree using drag and drop, commands, or shortcut keys. You can use the Preview tab to see a planimetric preview of your 3D data with the Geography option or preview it in 3D using the 3D View option. You can create 3D layers of 3D and 2D data and assign 3D viewing properties—for example, you can set the base heights and extrusion expressions for features with z-values stored in an attribute. You can also control how a 3D layer is rendered during scene navigation, and you can specify whether or not features are shaded based on their position relative to a light source. In addition, you can create metadata, including perspective view thumbnail graphics, for your data.

ArcCatalog basics

ArcCatalog lets you organize your data in an expandable tree view. You can view the selected item in the Catalog tree in different ways, using the tabbed view area to the right of the tree.

The Contents tab lets you see the contents of an item in the tree. The Preview tab lets you see the data contained in the selected item. The Metadata tab lets you see the selected item's metadata.

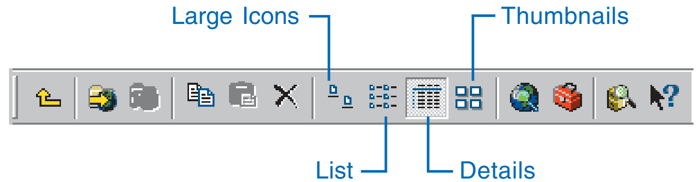


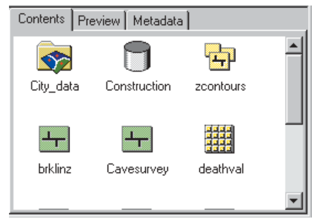
Catalog tree and contents of a folder connection

Browsing the Catalog's contents

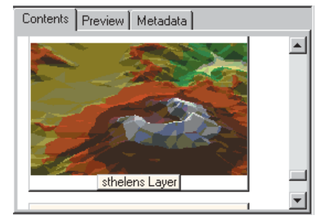
When you select items—for example, folders, databases, or feature datsets—in the Catalog tree, the Contents tab lists the contents of each item. You can expand items that contain other items—for example, folders—and see their contents in the tree as well. Items such as scene documents, maps, layers, and tables don't contain other items; when you select them in the Catalog tree, the Contents shows the name and type of the object and a thumbnail if one has been created.

You can display the Contents list in several ways. To change its appearance, use buttons on the Standard toolbar.

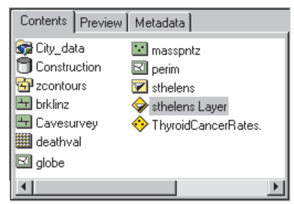




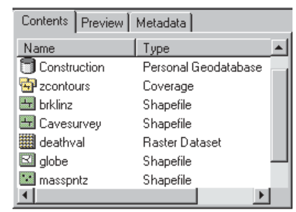
Large Icons



Thumbnails



List



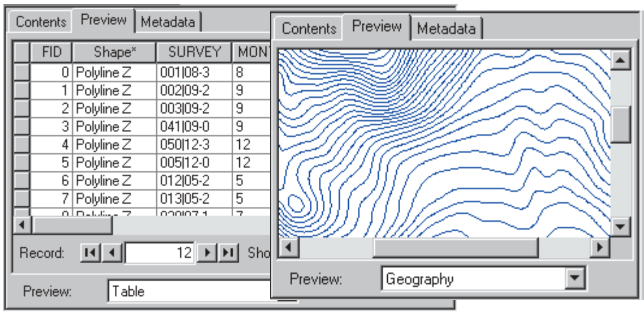
Details

Use the Contents tab and the Catalog tree to browse and manage your data.

Previewing data

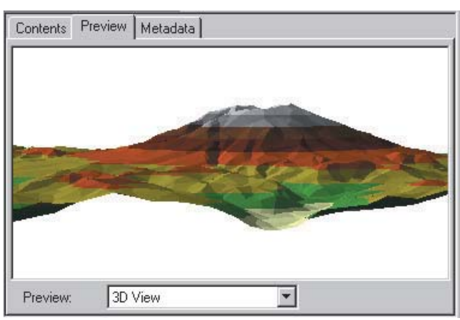
Often, viewing your data in the Contents tab shows you all you need to find the right data for a scene or map. Sometimes, though, you need more information. You can explore your data in the Preview tab. As with the Contents tab, you can control how you preview data.

You can draw geographic data with the Geography view. You can also examine the attributes of feature data or the contents of any other table with the Table view. Use the dropdown list at the bottom of the Preview tab to choose how you want to preview your data.



Geography and table previews—the Z in the Shape field indicates a geometry that contains z-values.

3D Analyst also gives you another way to preview your data—the 3D View preview—and a new toolbar that you can add to ArcCatalog to navigate the 3D preview.



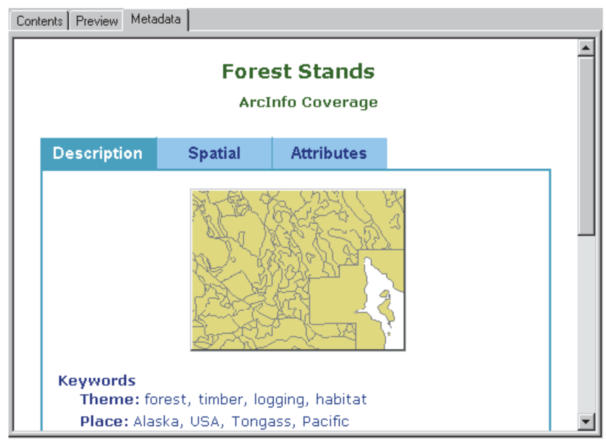
3D View preview

This set of tools enables you to navigate around your data in 3D, query the features in your data, and create perspective view thumbnail snapshots of your data.

Using metadata

Before you decide to use a data source in a map, you may want to learn more about it. Metadata is background information that lets you make better decisions about what data to use and how to use it. You can view metadata in ArcCatalog using the Metadata tab.

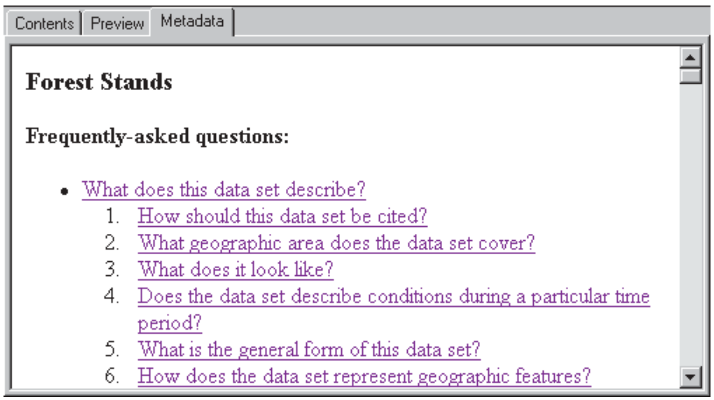
Metadata usually contains documentary information about why the data was created, who created it, and how accurate the data



Metadata view

is. This kind of information can be edited using a metadata editor in ArcCatalog. Metadata also contains information about the properties of the data—for example, the attributes stored for features or the coordinate system of the data. ArcCatalog automatically maintains this information for you.

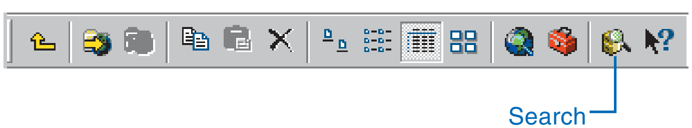
You can change the way you view an item's metadata by choosing a different stylesheet from the dropdown list on the Metadata toolbar.

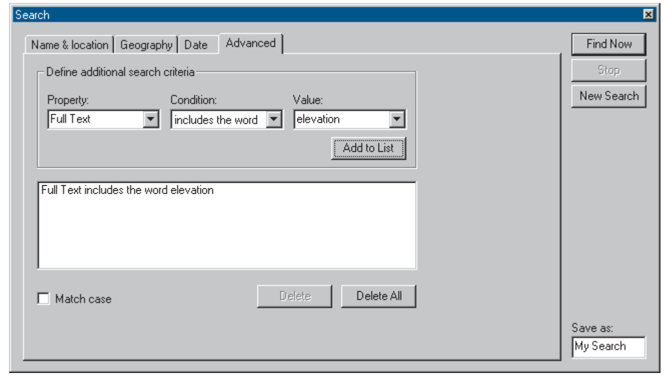


Metadata viewed with the FGDC FAQ stylesheet

You can also use the ArcCatalog Search tool to search for data based on information in the metadata.

The Search tool dialog box contains four tabs that allow you to search for data that satisfies various conditions.





Search for data with metadata containing the word elevation.

When you've found the data you want to use, ArcCatalog lets you drag it directly onto a map or scene document or create a layer that specifies how the data should be symbolized and rendered in 2D or 3D.

Previewing 3D data

When the selected item in the Catalog tree contains geographic data, you can preview that data without having to create a map or 3D Scene. For a planimetric view of your data, choose Geography from the Preview dropdown list on the Preview tab. For a perspective view, choose 3D View. In order to view data in 3D, you will need to enable the 3D Analyst extension.

The 3D View tools allow you to navigate around your data in 3D. Some of these tools are like tools that you use in ArcMap and ArcCatalog to pan and zoom in 2D. Other tools are ▸

Tip

Why does my data look flat in 3D View preview?

TINs and feature classes with z-values embedded in their geometry—for example, PointZ, PolyLineZ, PolygonZ, and Multipatch shapes—will be automatically rendered in 3D when you select 3D View. Rasters and 2D feature classes will be rendered as though they rest on a flat surface. You can create layers that specify 3D rendering properties for all types of data, and the layers will be rendered in 3D.

Navigating in 3D

1. Click the Navigate button on the 3D View toolbar.

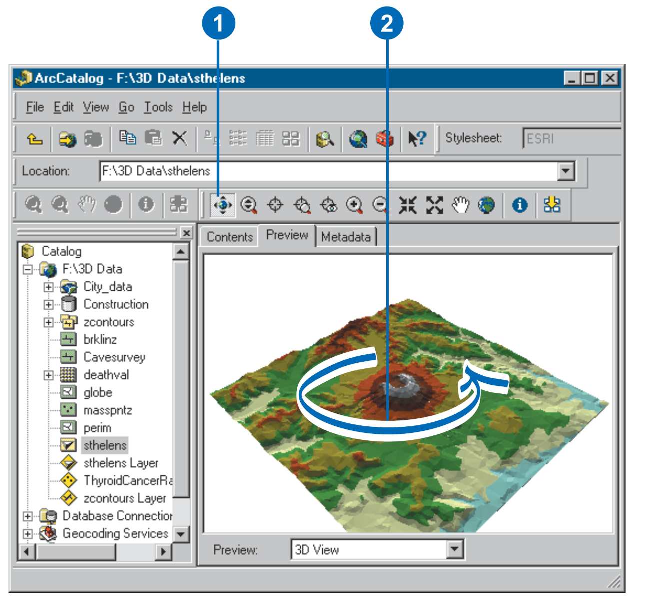
   The Navigate tool allows you to rotate the data in 3D, to zoom in and out, and to pan the data.

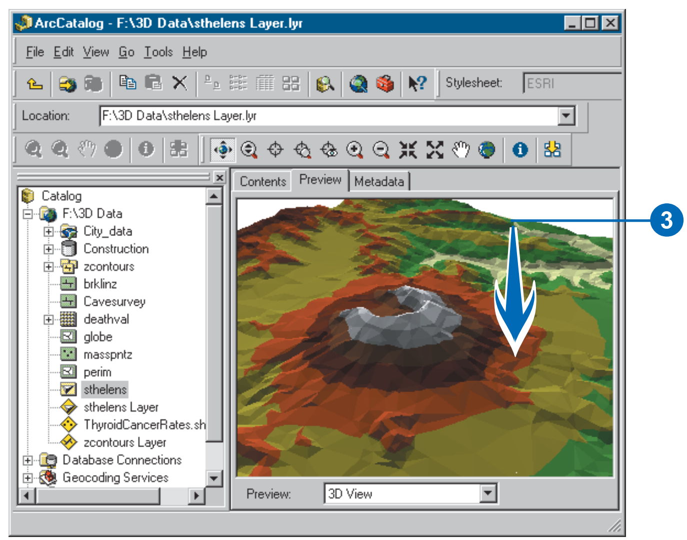
2. Click on the data and drag it to the right.

   When you click on the data and drag to the right, you rotate the data counterclockwise around the z-axis.

3. Right-click on the 3D data and drag down.

   When you right-click on the data and drag down, you zoom in to the data. ▸





specialized for 3D perspective viewing.

You use the same tools for navigating in ArcScene.

The Navigate tool combines the functions of several of the other tools.

Clicking the left, right, and center mouse buttons and dragging up, down, left, or right lets you rotate the 3D view, zoom in and out, and pan across the view.

You can also use the 3D preview to preview a scene document before you open it in ArcScene.

Tip

Seeing the entire dataset

If you want to see a dataset's entire contents after zooming and panning around it, click the Full Extent button on the 3D View toolbar.

4. Click both mouse buttons—or the center button on a three-button mouse—and drag the data to the right.

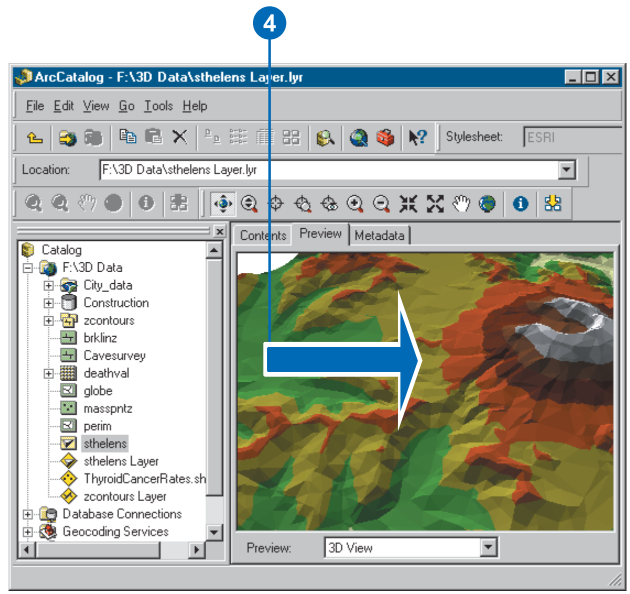
   Clicking both buttons and dragging the data pans the data.

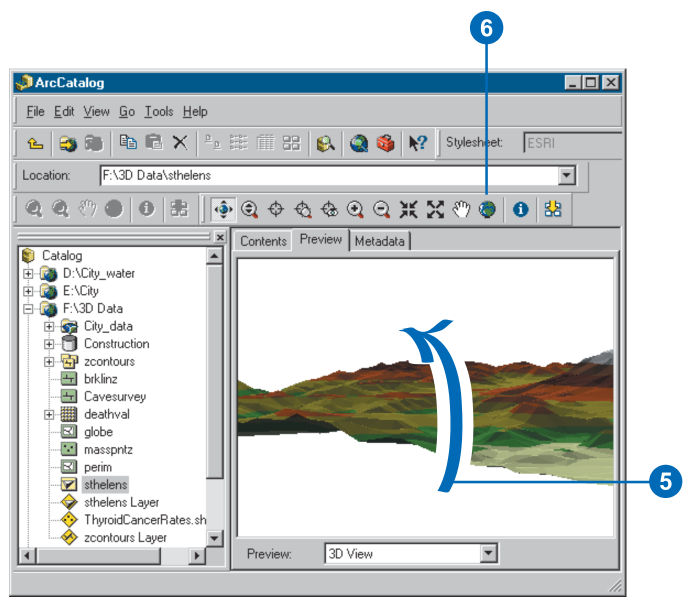
5. Click on the data and drag up.

   Clicking and dragging the data up lowers your viewing position relative to the data.

6. Click the Full Extent button.

   The data returns to the original extent and view position.





The camera, the observer, and the target

You view a scene through a viewer's *camera*. A camera's location in a scene is defined by the *observer* property. The point at which the camera is directed is called the *target*. When you navigate through a scene, you are actually moving the observer in conjunction with the target. ►

Tip

Why change the center of the view?

By default, the center of the view is the center of the data. The Navigation tool, the Zoom In and Zoom Out tools, and the field of view tools all change the view of the data relative to the current center of the view. If you want to zoom in to see a particular area, centering on it first will prevent it from zooming out of your field of view. Centering on a point also allows you to rotate the view around it.

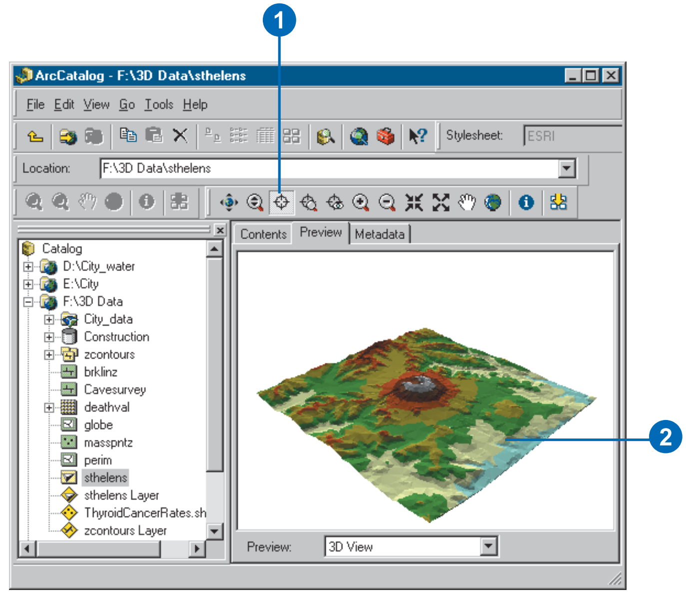
Tip

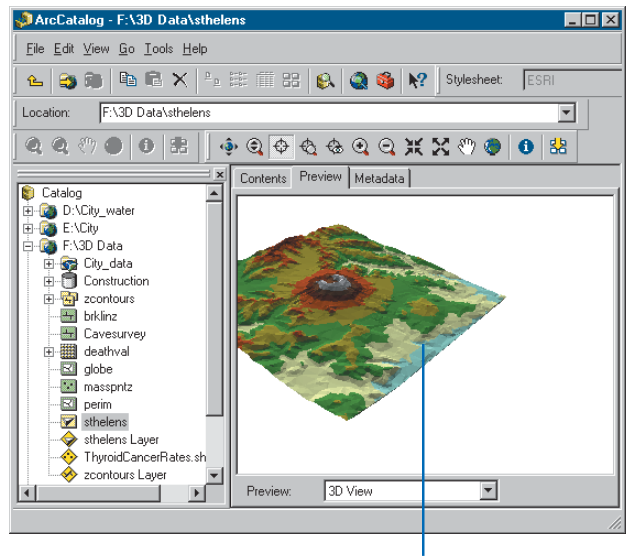
Centering shortcut

You can center on a target while using the Navigate tool. Press the Ctrl key and click at the location where you want to center the scene.

Centering

1. Click the Center on Target tool.
2. Click the location that you want to occupy the center of the 3D view.





The point you clicked is the new center of the view.

When you center on the camera target, the point you click moves to the center of the view. When you zoom to the target, the view zooms to the point you click. ▸

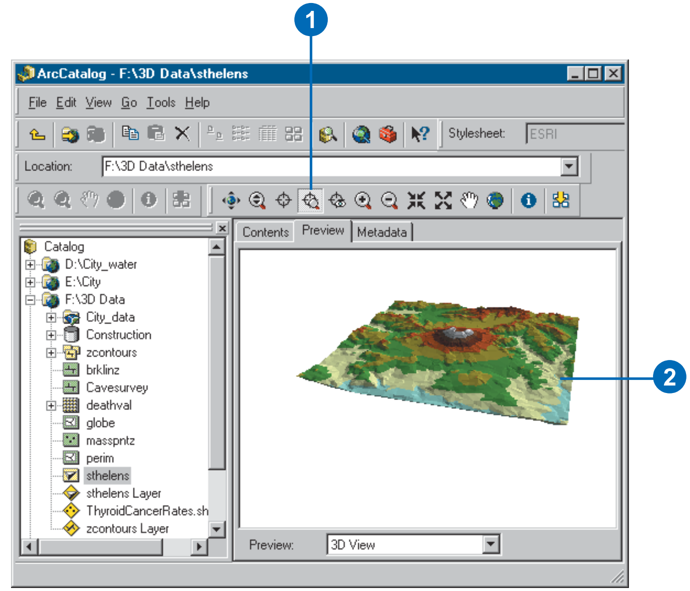
Tip

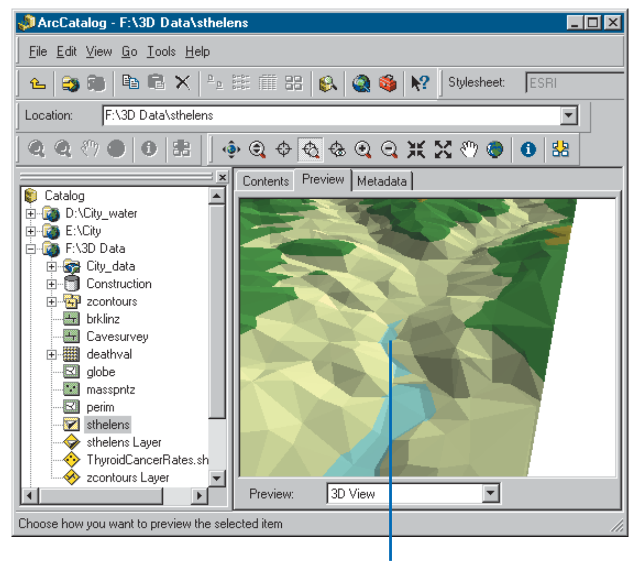
Zooming shortcut

You can zoom to a target while using the Navigate tool. Press the Ctrl key and right-click to zoom to a location.

Zooming to a target

1. Click the Zoom to Target tool.
2. Click the location that you want to zoom to.





The view zooms in to the point you clicked.

When you set the camera observer, the view zooms to the point you click while pointing at the target. You can use the Set Observer and the Center on Target tools to define how you want this to happen. Anytime you use the Set Observer tool, the camera will point at the current target. To point the camera at a target that you choose, use the Center on Target tool first. Then when you set the camera observer, the camera will point toward the target you defined. The camera, the observer, and the target are used in the 3D view of ArcCatalog, as well as in the perspective view of ArcScene.

Tip

Set observer shortcut

You can set the observer while using the Navigate tool. Press the Ctrl key and middle-click to zoom to a location.

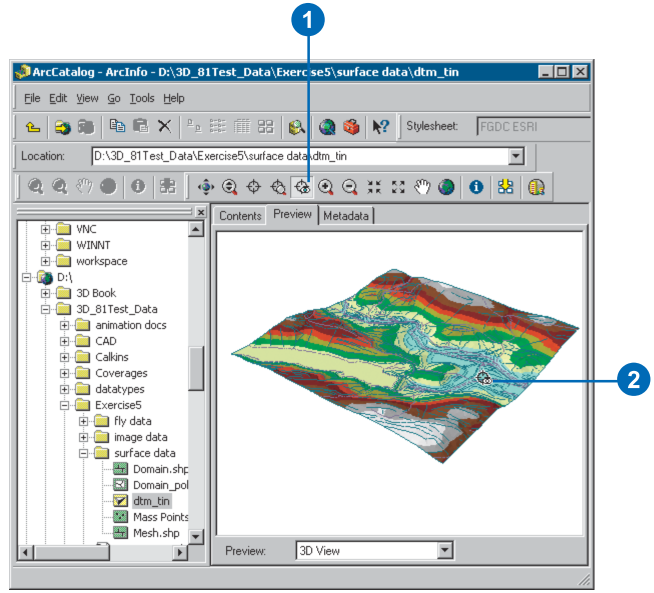
Tip

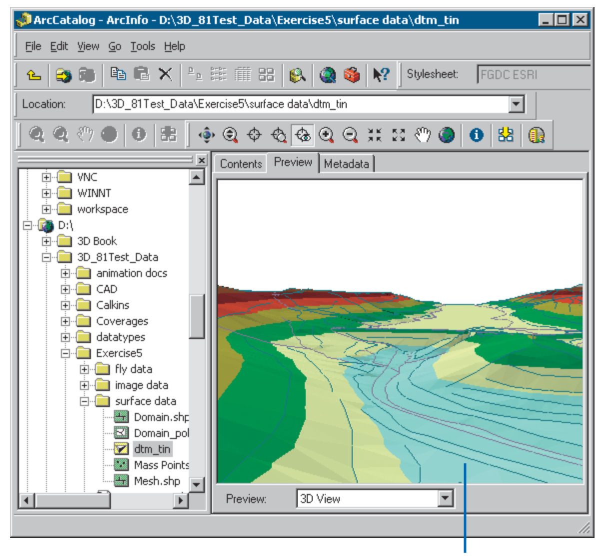
Using the camera tools in ArcScene

The Center on Target, Zoom to Target, and Set Observer tools are also available on the Navigation toolbar in ArcScene.

Setting the camera observer

1. Click the Set Observer tool.
2. Click the location that you want to zoom to.





The view zooms to the point you clicked, honoring the target.

Tip

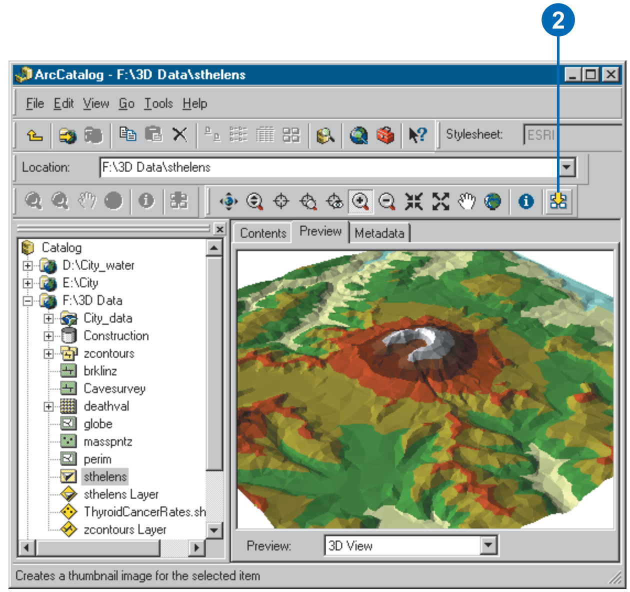
Viewing thumbnails in the Contents tab

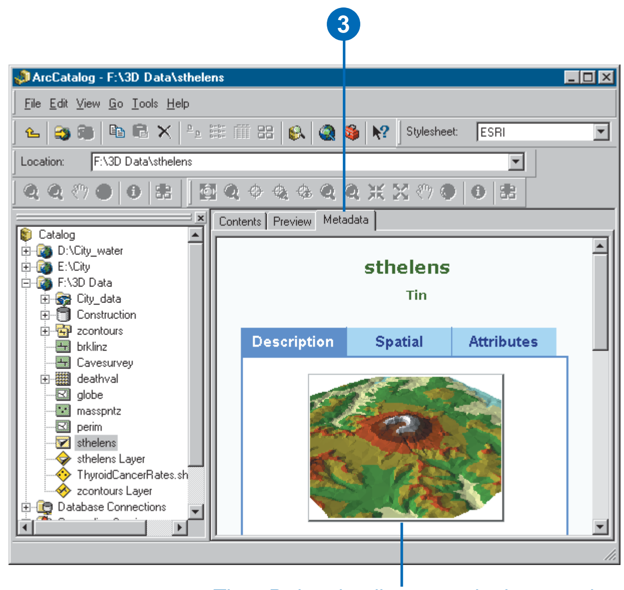
If you create a thumbnail image of your data, it will be shown in the Contents tab when you choose Thumbnail view.

Creating a 3D thumbnail

1. Navigate the 3D view to the perspective that you'd like to show in a thumbnail snapshot.
2. Click the Create Thumbnail button.
3. Click the Metadata tab.

   The 3D thumbnail appears.





The 3D thumbnail appears in the metadata.

Displaying 2D data in 3D

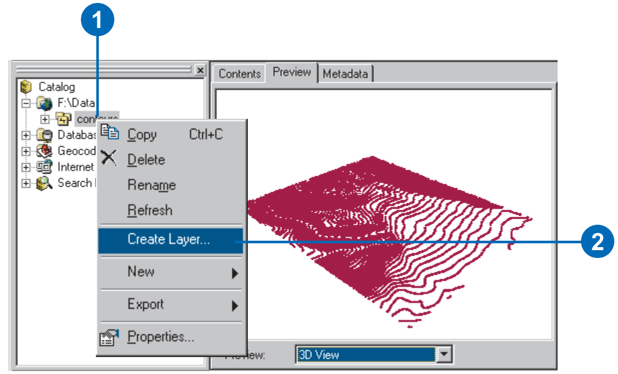
2D data can be viewed in 3D if you create a layer and set its 3D viewing properties. You can assign z-values to features based on an attribute, a constant, or an expression. You can also specify a surface from which to derive the z-values of the features.

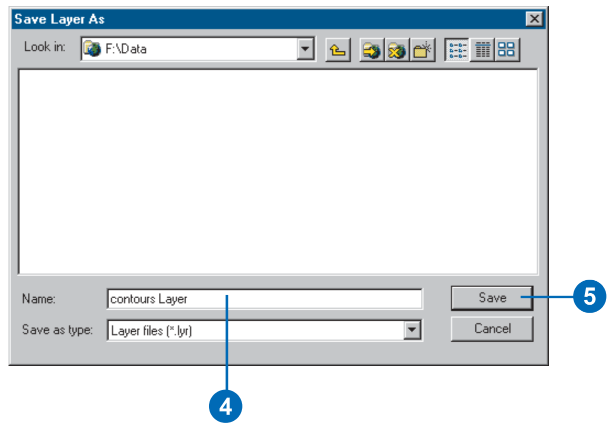
3D Analyst adds three tabs to the Layer Properties dialog box; these tabs allow you to modify how a layer is rendered in 3D. The tabs are called Base Heights, Extrusion, and Rendering. Most of the time, if you want to create a layer that you can preview in 3D, you'll set the height of the features in the layer using the Base Heights tab. Sometimes you may want to extrude features up or down from the base height—for example, if you want to create lines that show the depths of well point features. You can set how features are extruded on the Extrusion tab. If you want to shade features in your layer based on a light source, you can set that property of a layer from the Rendering tab. This tab also lets you fine-tune how your data is displayed while you are navigating.

Creating a layer

1. Right-click the data source from which you want to create the layer.
2. Click Create Layer.
3. Navigate to the folder in which you want to save the layer.
4. Type a name for the layer file.
5. Click Save.

   The layer file is now an item in the Catalog.

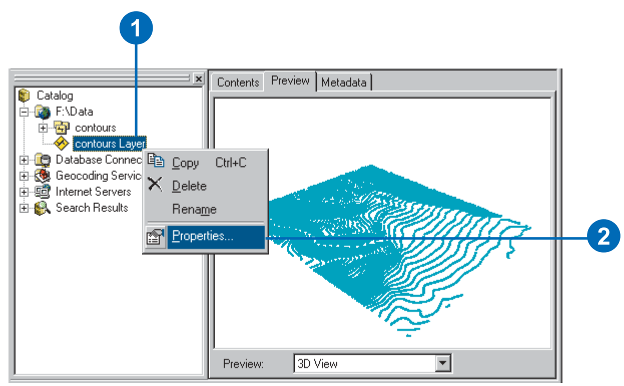




Setting layer properties

1. Right-click the layer.
2. Click Properties.

   You can use the Layer Properties dialog box to set many properties of the layer in addition to the 3D Base Height, Extrusion, and Rendering properties.

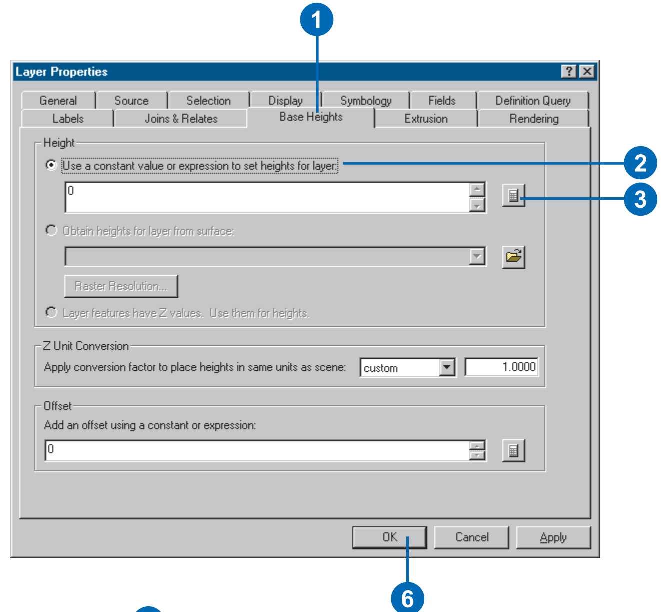


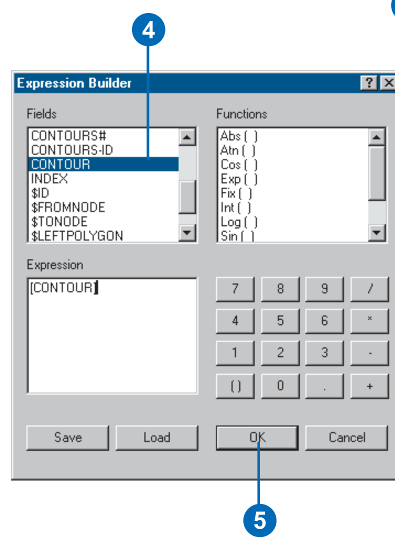
Setting base heights using an attribute

1. Click the Base Heights tab.
2. Click the option to Use a constant value or expression to set heights for layer.
3. Click the Calculate button.
4. Double-click the field that will provide the z-value for the features.

   You can create an expression, such as [contour] * 3.28, to set the z-values as well.
5. Click OK.
6. Click OK on the Layer Properties dialog box.

   The 2D features are now drawn in 3D using the attribute that you selected as the z-value.

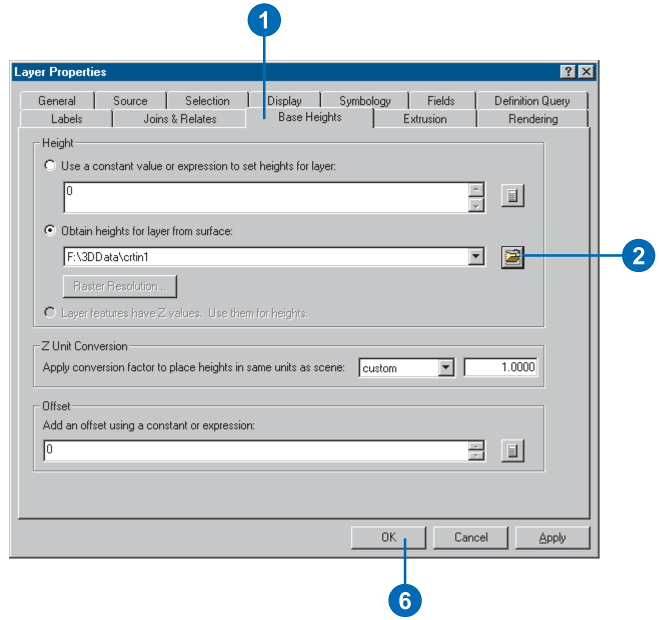


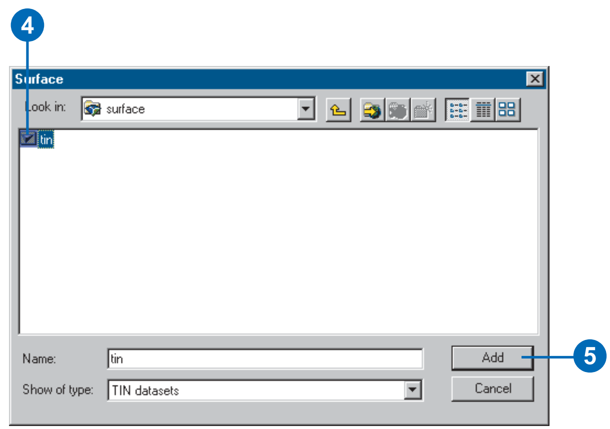


Setting base heights using a surface

1. Click the Base Heights tab.
2. Click the Browse button.
3. Navigate to a surface.
4. Click the surface.
5. Click Add.
6. Click OK on the Layer Properties dialog box.

   The 2D features are now drawn in 3D using the surface that you selected to provide the z-value.





Tip

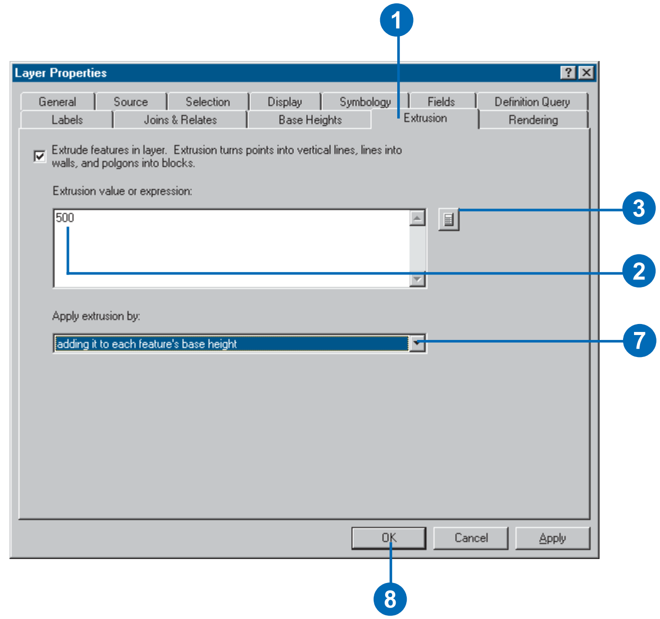
What is extrusion?

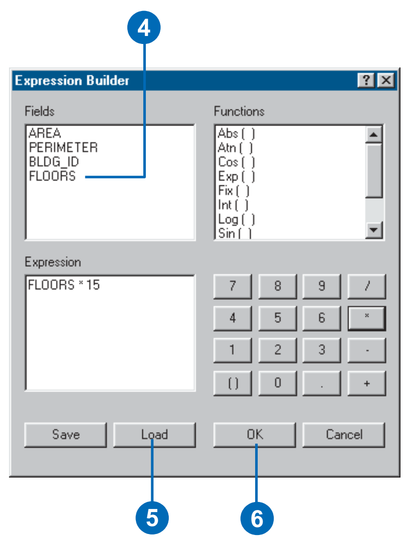
Extrusion extends features vertically from a given base height to the height that you specify. Points become vertical lines, lines become walls, and polygons become blocks.

Extruding features

1. Click the Extrusion tab.
2. Optionally, type a constant height to which all features will be extruded.
3. Optionally, click the Calculate Extrusion Expression button.
4. Optionally, click the field that you want to use in the extrusion expression. You can build complex extrusion expressions using functions and algebraic expressions.
5. Optionally, click Load and browse to a previously saved extrusion expression.
6. Click OK.
7. Click the dropdown list to choose how you want the extrusion expression applied.
8. Click OK.

   The 2D features are now extruded using the constant or expression.





Starting ArcScene from ArcCatalog

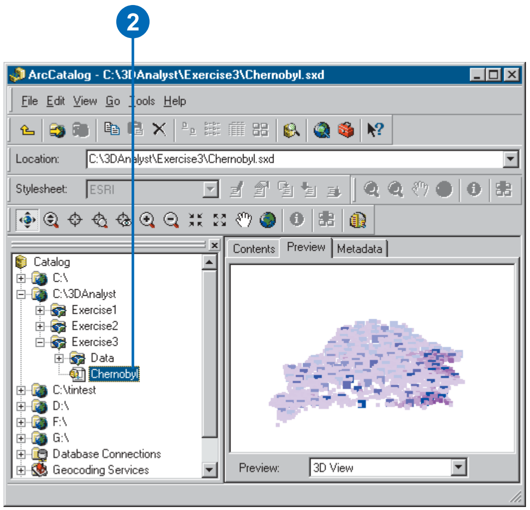
You can start ArcScene from ArcCatalog by double-clicking a scene document or by clicking the Launch ArcScene button on the 3D View Tools toolbar.

Opening an existing scene from ArcCatalog

1. Navigate to the scene that you want to open.

   You can use the Preview tab and 3D View tools to preview the scene before you open it.

2. Double-click the scene in the Catalog tree.



Opening a new scene from ArcCatalog

1. Click the ArcScene button on the 3D View Tools toolbar.

Creating a new 3D feature class

You can create new empty feature classes in ArcCatalog to hold the 3D feature data that you create with the 3D Analyst geometry interpolation tools in ArcMap. These tools will create 3D graphics by default, but you can use them while editing a 3D feature class to create 3D features. For more information about creating feature classes and shapefiles, see *Using ArcCatalog*.

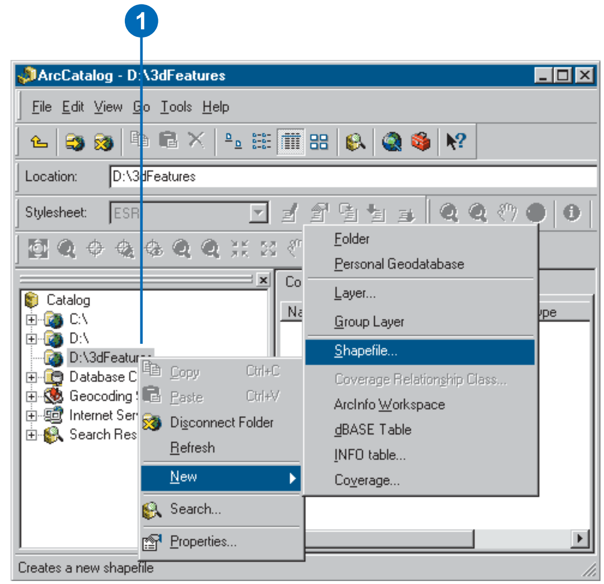
Creating an empty 3D feature class or shapefile

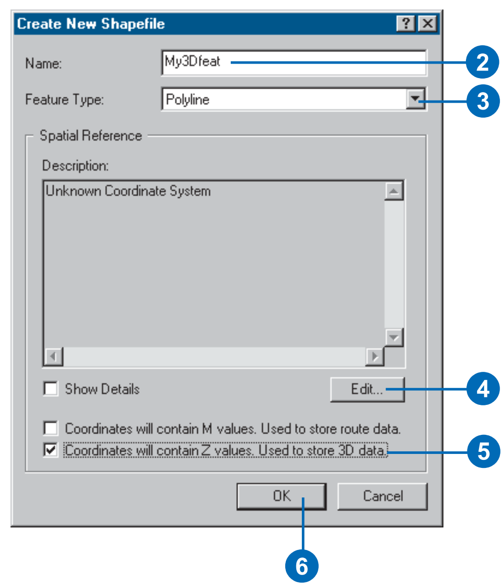
1. Right-click in the location where you want to create the new shapefile. Point to New and click Shapefile.

   If you want to create a geodatabase feature class, navigate to an existing geodatabase and click Feature class.

2. Type a name for the new shapefile.
3. Click the Feature Type dropdown arrow and click the type (geometry) of feature that you want to create—for example, point, polyline, polygon.
4. Optionally, click Edit to define the spatial reference for the features.
5. Click the check box to indicate that the feature coordinates will contain z-values.
6. Click OK.

   The shapefile is created. You can right-click it in the Catalog tree, click Properties, and add attributes if you want to store attributes for the 3D features. You do not need to include a height attribute, as the z-values will be stored in the feature geometry.





Displaying surfaces

5

IN THIS CHAPTER

- **Displaying raster surfaces in 3D**
- **Displaying raster surfaces**
- **Symbolizing areas with unknown values**
- **Displaying TIN surfaces**
- **Making a layer transparent**
- **Shading a layer**

ArcScene and ArcMap give you a wide variety of ways to symbolize and display surfaces. Because of the differences between rasters and TINs, you have different options for symbolizing them.

You can classify the data in continuous rasters, stretch them to improve contrast, and display categorical rasters with colors to represent their unique values. For multiband rasters, such as satellite images and some aerial photographs, you can display the raster as a red–green–blue (RGB) composite or as a single-band stretched image. You can render cells with no data and background cells in different ways.

ArcScene and ArcMap let you show the elevation of TIN surfaces or the aspect or slope of each TIN facet. You can also show the nodes and edges of the TIN in several different ways.

You can make all surfaces transparent and add depth and realism to a surface by shading it based on its position relative to a light source.

Displaying raster surfaces in 3D

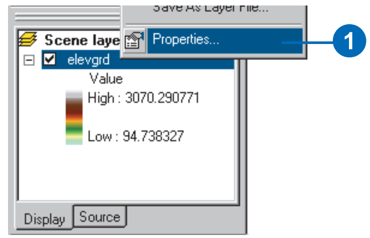
In ArcScene, you can create 3D representations of raster surfaces by setting the base height from a raster or TIN surface. You can use this technique to visualize terrain surfaces, show remote-sensing images in perspective, or create abstract visualizations of other surface data such as chemical concentrations or population densities.

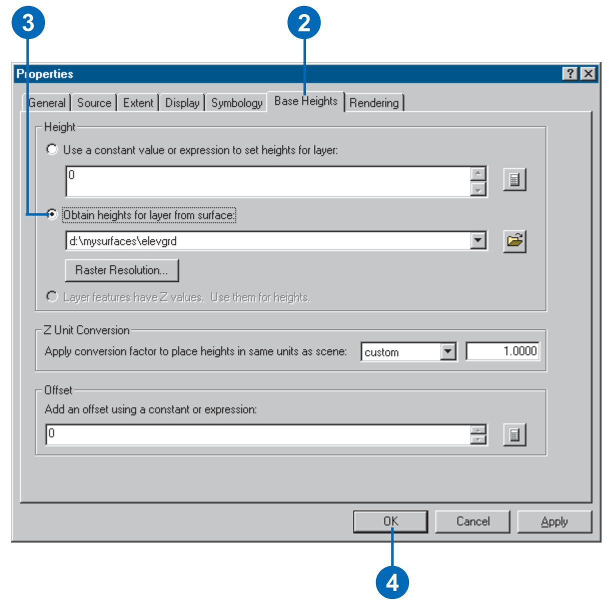
If you drape a raster over a raster surface, you can change the resolution of the base surface; the base surface is usually resampled to a lower resolution in order to increase the drawing speed. Click Raster Resolution to set the resolution of the base surface.

Draping a raster on a surface

1. In the table of contents, right-click the raster layer that you want to drape over a surface and click Properties.
2. Click the Base Heights tab.
3. Click the button to Obtain heights for layer from surface.

   If you want to use a surface other than the current raster layer, either select one of the surfaces in the scene by clicking the dropdown arrow or click the Browse button and navigate to another surface.
4. Click OK.





Displaying raster surfaces

You can display rasters by grouping their values into a number of classes, by stretching the values to enhance contrast, or by assigning each unique value in the raster to a color.

How you display a raster depends on the type of data that it contains and what you want to show. Some rasters have a predefined color scheme, but for others, ArcMap and ArcScene will choose an appropriate display method that you can adjust as needed. You can change display colors, group data values into classes, or stretch values to increase visual contrast. For multiband rasters, you can choose three bands to display together in an RGB composite. This drawing method often improves your ability to distinguish features in multispectral images.

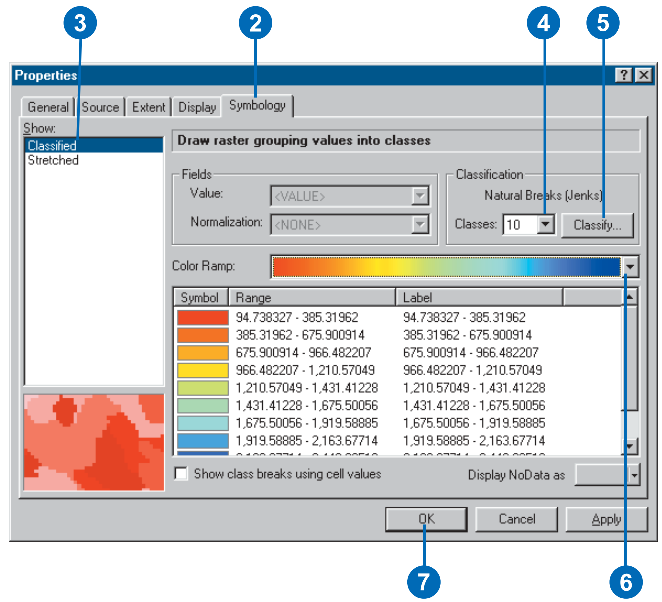
Tip

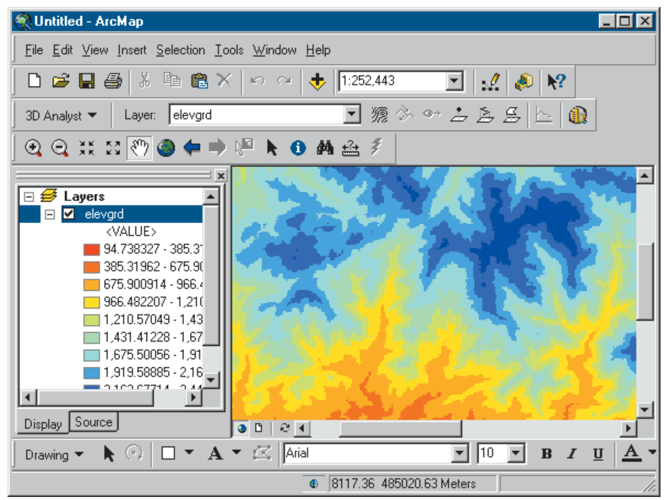
Why don't I see the option to use unique values?

You can only draw data in integer rasters with unique values. Continuous data can be classified, drawn using a stretch, or—for multiband rasters—drawn as an RGB composite image.

Drawing rasters that represent continuous surface data with separate classes

1. In the table of contents, right-click the raster layer that you want to show by grouping values into classes and click Properties.
2. Click the Symbology tab.
3. Click Classified.
4. Click the Classes dropdown arrow and click the number of classes you want.
5. Optionally, click Classify and choose the classification method you want to use.
6. Click the Color Ramp dropdown arrow and click a color ramp.
7. Click OK.





The continuous surface is drawn with different colors for each class.

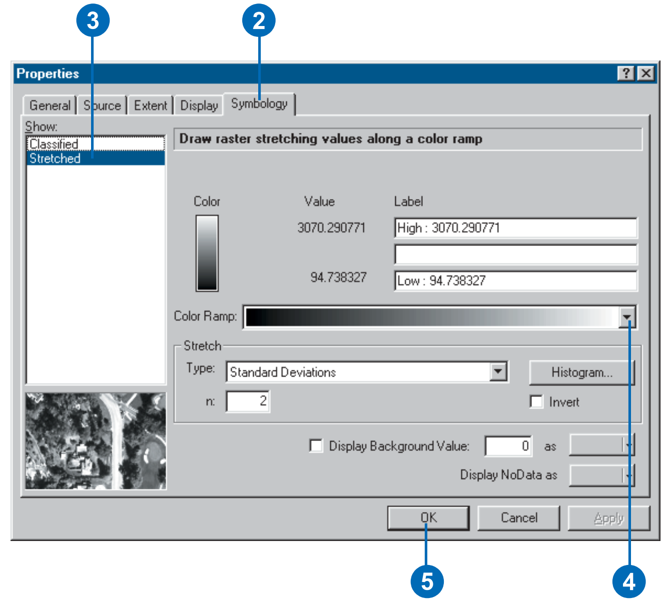
Tip

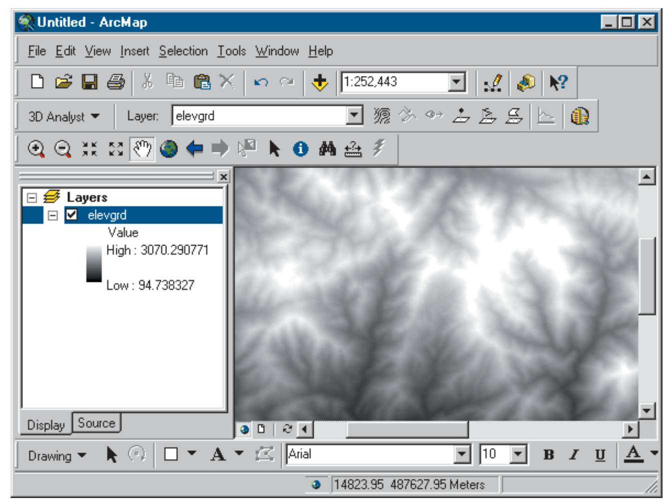
Continuous rasters

Most rasters that represent surfaces contain continuous values, whether the surface values represent elevation, temperature, or concentration of a chemical. It is often useful to display continuous surfaces with a continuous grayscale or color ramp. The Stretched option maps the low and high values in the raster to a 0–255 intensity scale. You can change the way that the values are mapped by changing the type of stretch used.

Drawing rasters that represent continuous surface data with a continuous color ramp

1. In the table of contents, right-click the raster layer that you want to show with a continuous color ramp.
2. Click the Symbology tab.
3. Click Stretched.
4. Optionally, click the Color Ramp dropdown arrow and click a color ramp.
5. Click OK.





The continuous surface is drawn with a continuous color ramp.

Tip

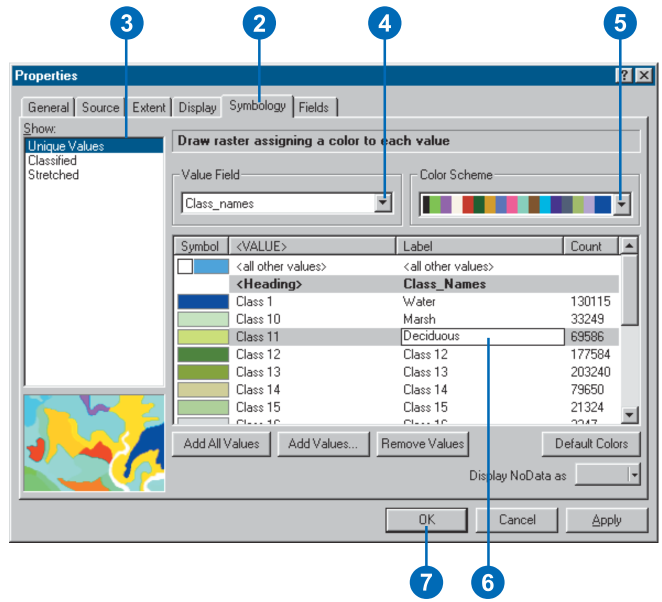
Categorical rasters

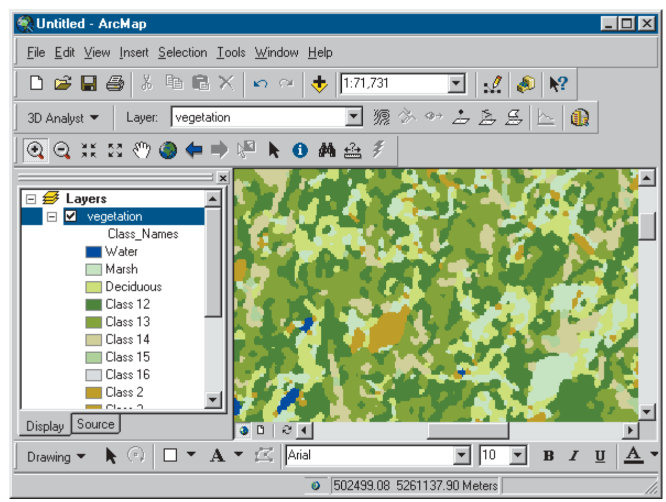
Classifying remotely sensed imagery, converting vector data to raster, and running certain analysis procedures can produce rasters that contain categorical values. In these cases, the values may be highly discontinuous and patchy, in comparison to the continuous and, generally, smoothly varying values of surfaces. Categorical rasters are usually drawn with Unique Values.

Drawing thematic rasters that represent unique categories, such as land cover

1. In the table of contents, right-click the raster layer that you want to show using unique categories and click Properties.
2. Click the Symbology tab.
3. Click Unique Values.
4. Click the Value Field dropdown arrow and click the field you want to map.
5. Click the Color Scheme dropdown arrow and click a color scheme.

   If your raster has a colormap, click Default Colors to revert to colors specified in the colormap.
6. Optionally, click a label and type in a more descriptive name.
7. Click OK.





The categorical raster is drawn with unique colors for each value.

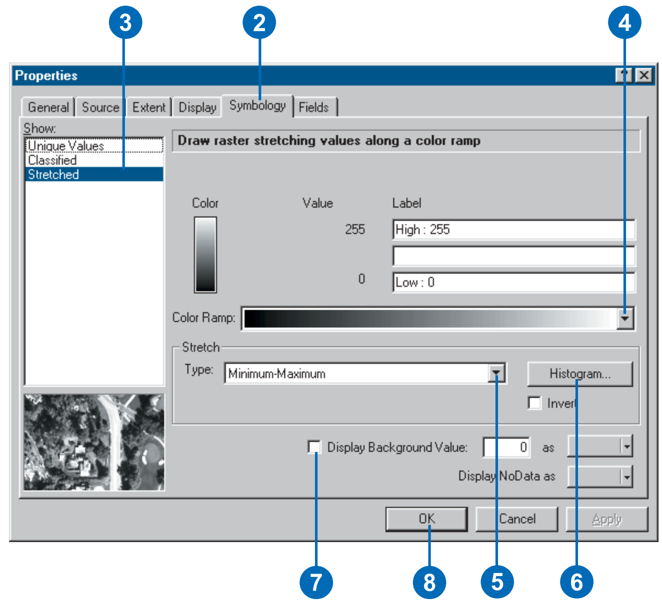
What is a stretch?

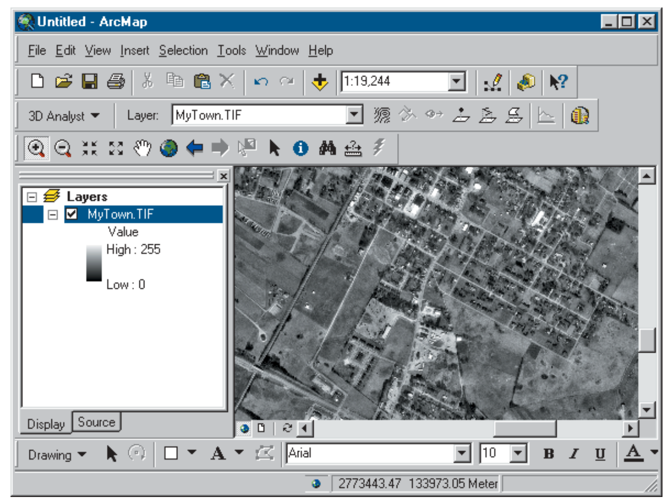
A stretch increases the visual contrast of a raster. You might apply a stretch when your raster appears too dark or has little contrast. Different stretches will produce different results in the raster display. You can experiment to find the best one for a particular raster.

Stretching a raster to improve the visual contrast

1. In the table of contents, right-click the raster layer that you want to increase the visual contrast of and click Properties.
2. Click the Symbology tab.
3. Click Stretched.
4. Click the Color Ramp dropdown arrow and click a color ramp.
5. Click the Stretch Type dropdown arrow and click the stretch you want to apply.
6. Optionally, click Histogram to modify the stretch settings.
7. Optionally, if the raster contains a background or border around the data that you want to hide, check Display Background Value and set the color to No Color.

   The cells will display transparently.
8. Click OK.





The image raster is drawn with a contrast stretch.

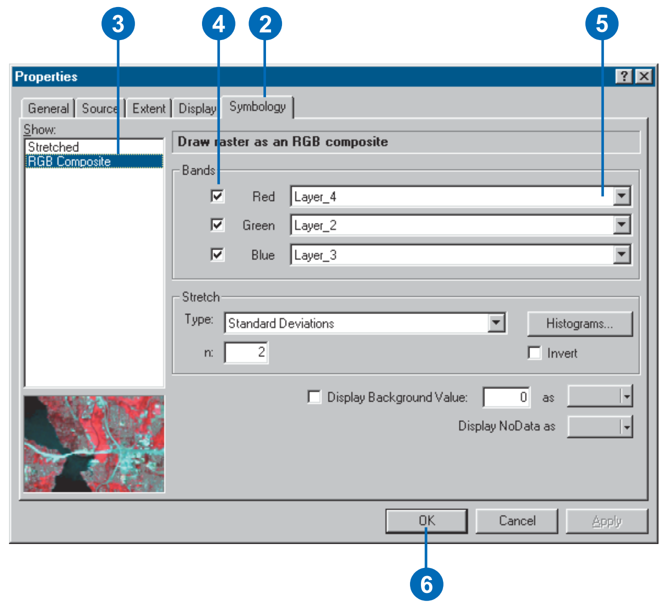
Tip

Why do I only see three bands when I added a five-band raster?

When you add a multiband raster to a map or scene, you see an RGB composite image made from three of the bands. You can select which three or fewer of the bands will be drawn using the Red, Green, and Blue color elements. By default, the first three bands are mapped to Red, Green, and Blue, but you can change the band mapping for a given image or for all images that you add.

Changing the band color assignments for a multiband raster

1. In the table of contents, right-click the raster layer and click Properties.
2. Click the Symbology tab.
3. Click RGB Composite.
4. Optionally, uncheck bands to turn them off.
5. Click the Red, Green, and Blue dropdown arrows and select the bands of the raster that will be displayed in red, green, and blue.
6. Click OK.



Tip

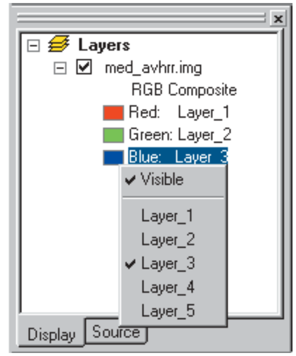
Changing band colors

You can quickly change the color assignments of the bands in a multiband raster. In the table of contents, click the color square—Red, Green, or Blue—and click the band in the multiband raster that you want to display using that color.

Turning bands on and off in a multiband raster

1. Click the color square beside the band you want to turn off.
2. Click Visible to uncheck it.

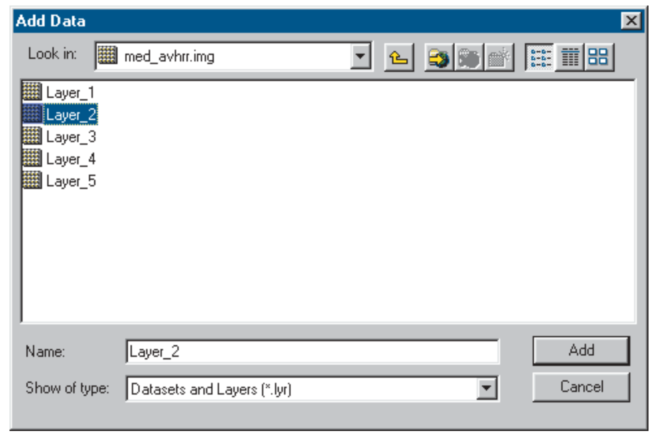
   The band is turned off.



Adding a single band of a multiband raster

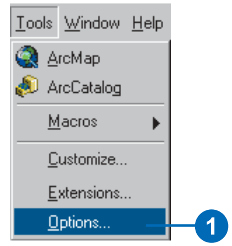
1. Navigate to the raster.
2 Expand the raster to view the individual bands.
3. Select a single band and click Add.

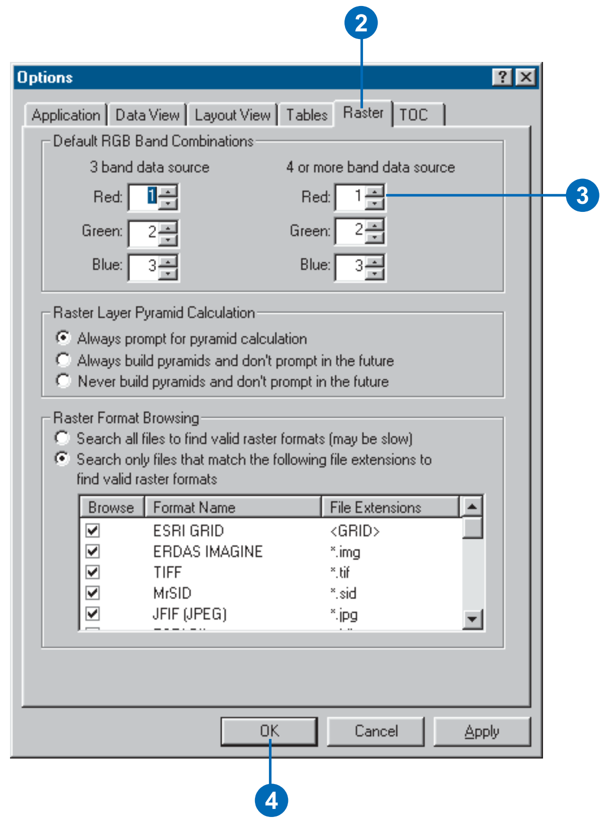
   The band is added as a single-band raster.



Setting the default color band assignments for multiband rasters

1. Click Tools and click Options.
2. Click the Raster tab.
3. Click the up and down arrows to select the bands in the raster that will be rendered in Red, Green, and Blue.
4. Click OK.





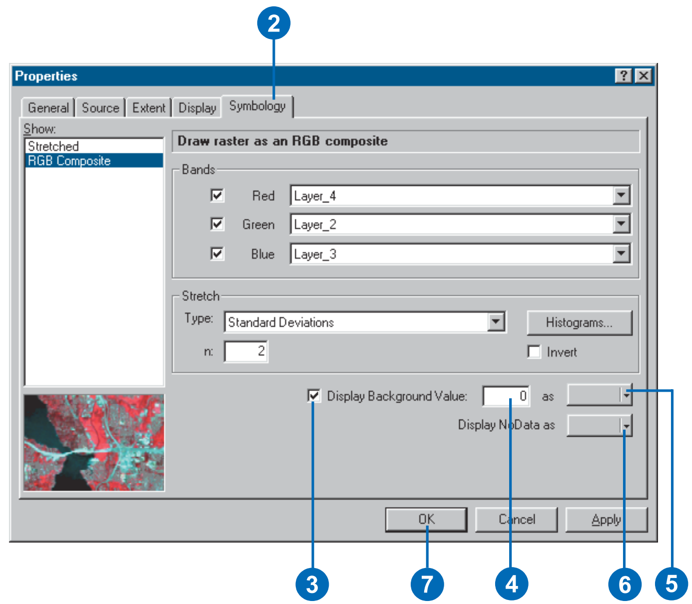
Symbolizing areas with unknown values

Sometimes rasters contain cells where there is no measurement of the surface. In grids these are usually represented as NoData values. In remote-sensing images there are often triangular or linear border areas within the raster but outside the area for which image values were recorded. These are often black and are called Background values.

You can control the way that areas of NoData and Background are displayed. The default is to display NoData with no color.

Setting the NoData or Background display color

1. In the table of contents, right-click the raster layer and click Properties.
2. Click the Symbology tab.
3. Check the box to Display Background Value.
4. Optionally, type a value to define the raster's Background color.
5. Click the Background color selector dropdown arrow and click a color—or no color.
6. Click the Display NoData as color selector dropdown arrow and click a color—or no color.
7. Click OK.



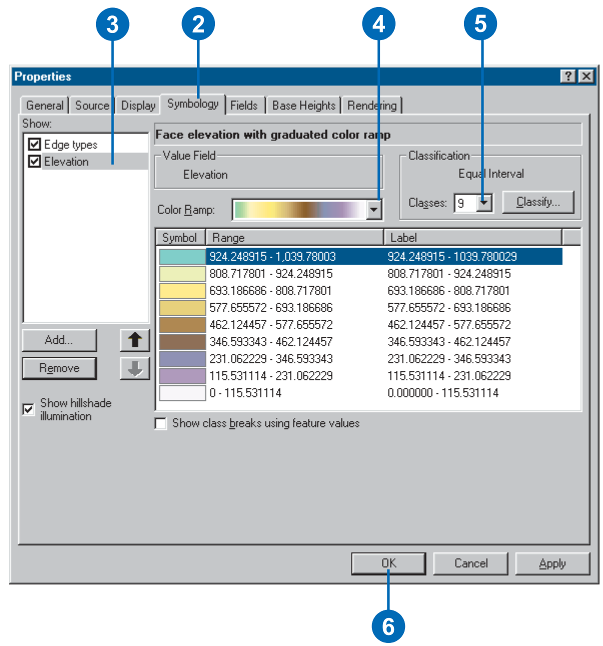
Displaying TIN surfaces

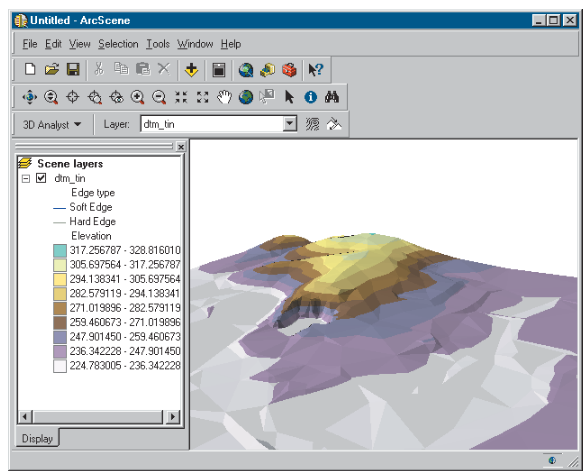
TINs are made up of triangular facets and the nodes and edges that make up the triangles. They may also contain breaklines—lines that follow sets of edges that play important roles in defining the shape of the surface—such as ridgelines, roads, or streams.

You can display one type of TIN feature in a map or scene—for example, just the triangles—or all of the TIN features. You can also symbolize each type of feature in different ways.

Drawing TIN faces by elevation

1. In the table of contents, right-click the TIN.
2. Click the Symbology tab.
3. Click Elevation.
4. Optionally, click the Color Ramp dropdown arrow and click a new color ramp.
5. Optionally, click the Classes dropdown arrow and choose the number of classes.
6. Click OK.

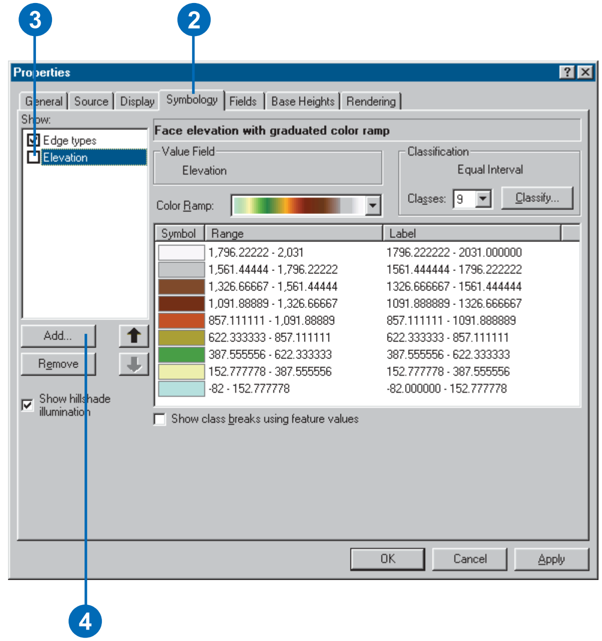


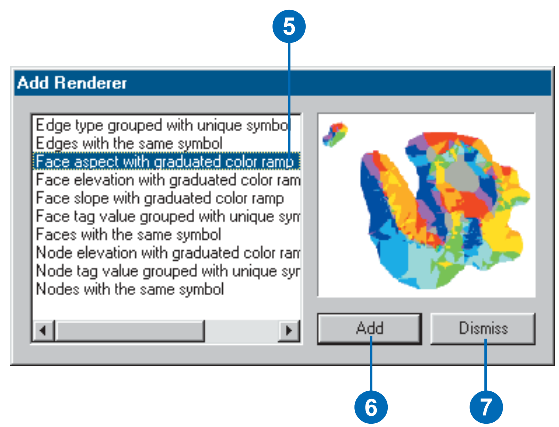


The TIN is drawn with different colors for each elevation class.

Drawing TIN faces by aspect

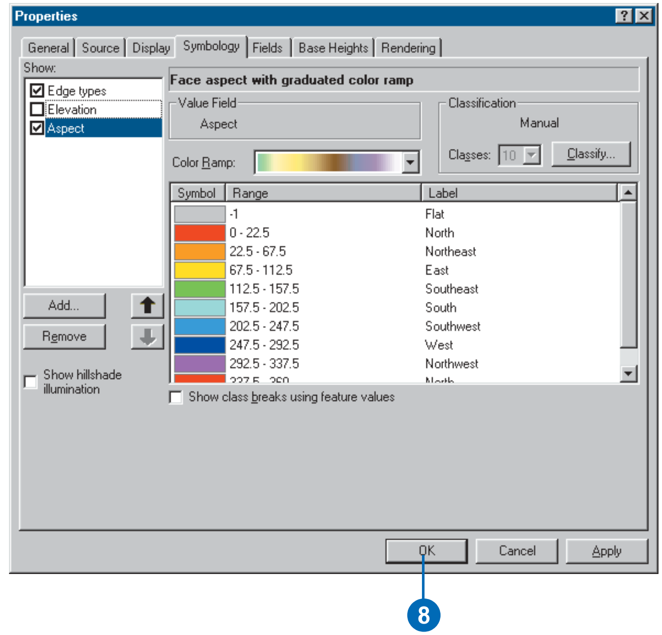
1. In the table of contents, right-click the TIN.
2. Click the Symbology tab.
3. Uncheck Elevation—or any other TIN face renderer that is checked.
4. Click Add.
5. Click Face aspect with graduated color ramp.
6. Click Add.
7. Click Dismiss. ▸

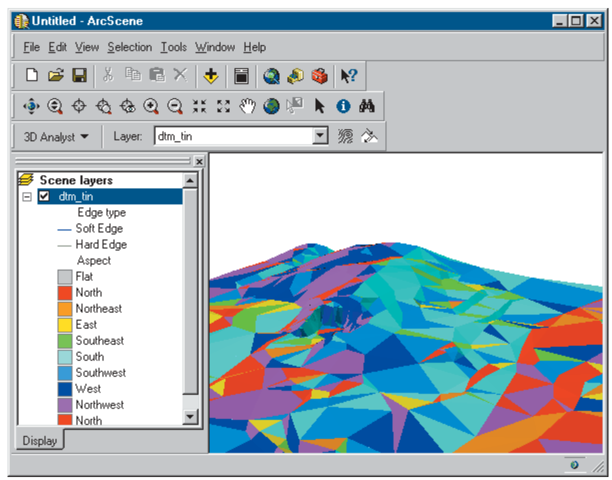




8. Click OK.

The TIN faces are rendered with colors to indicate the direction that they face.





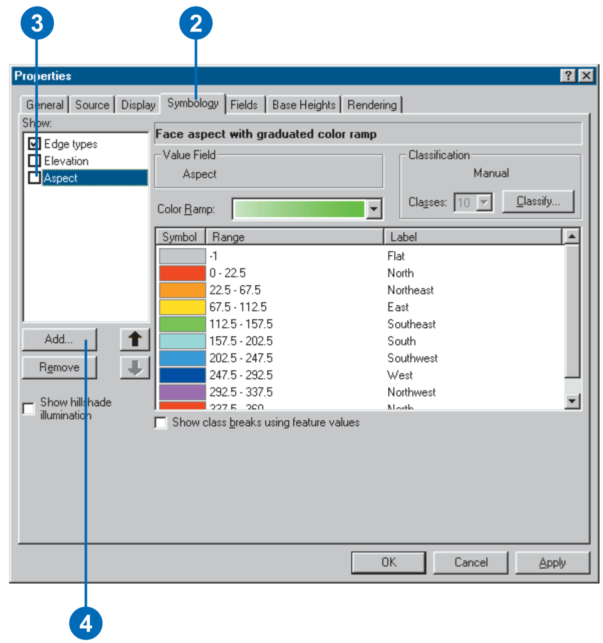
The TIN is drawn with different colors for each triangle's aspect class.

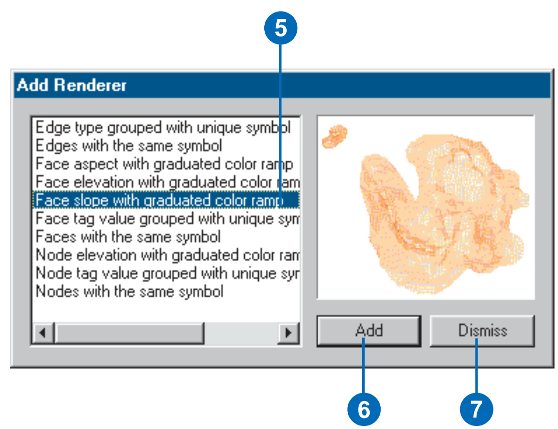
Changing the slope of a TIN surface

TIN triangle slope values are calculated, not stored in the TIN. If you change the z unit conversion factor, the slopes of each TIN face will be recalculated.

Drawing TIN faces by slope

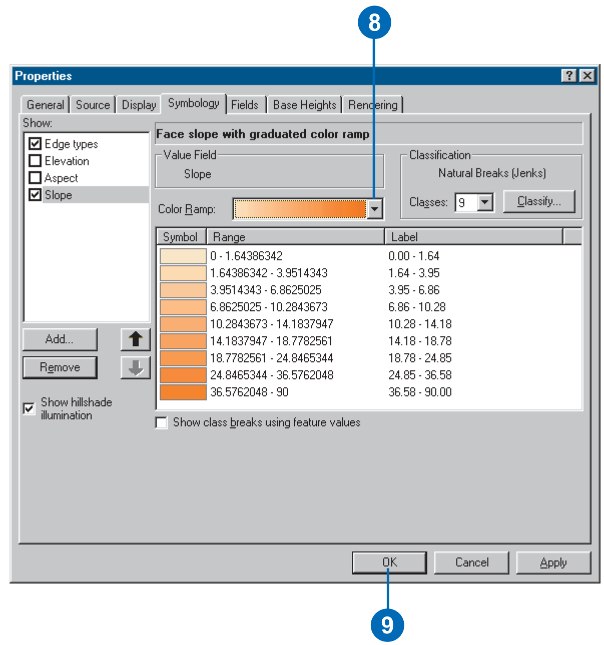
1. In the table of contents, right-click the TIN.
2. Click the Symbology tab.
3. Uncheck Aspect—or any other TIN face renderer that is checked.
4. Click Add.
5. Click Face slope with graduated color ramp.
6. Click Add.
7. Click Dismiss. ▸

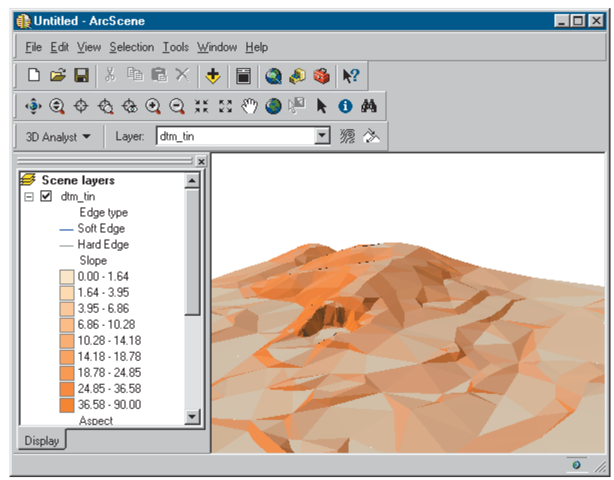




8. Click the Color Ramp dropdown arrow and select a color ramp.
9. Click OK.

   The TIN faces are rendered with colors to indicate the slope of the terrain.





The TIN is drawn with graduated colors for each triangle's slope class.

Tip

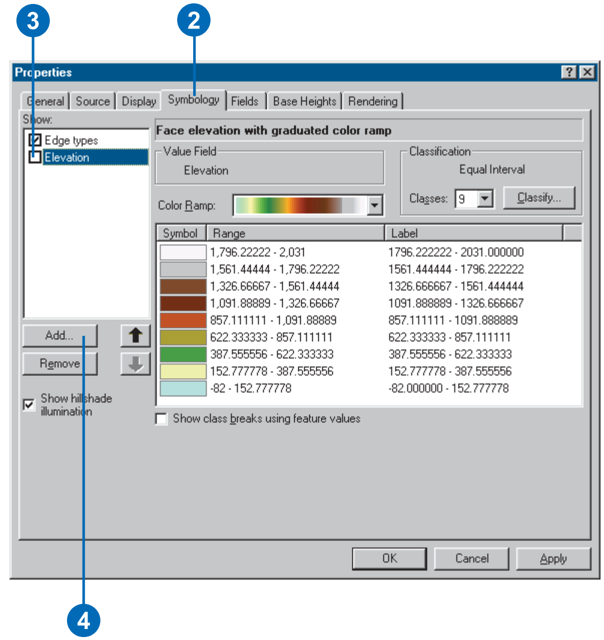
What are tag values?

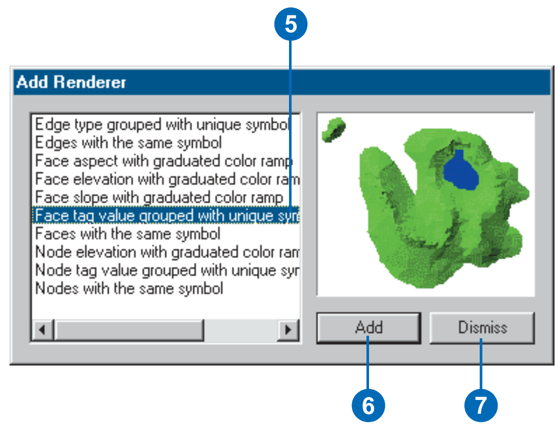
TIN nodes and triangles can be tagged with integer values to allow you to store additional information about them. These integer values can be used as lookup codes—for example, to indicate the accuracy of the input feature data source or the land use type code for areas on the surface.

The codes can be derived from fields in the input feature classes. You can symbolize tagged features with unique values.

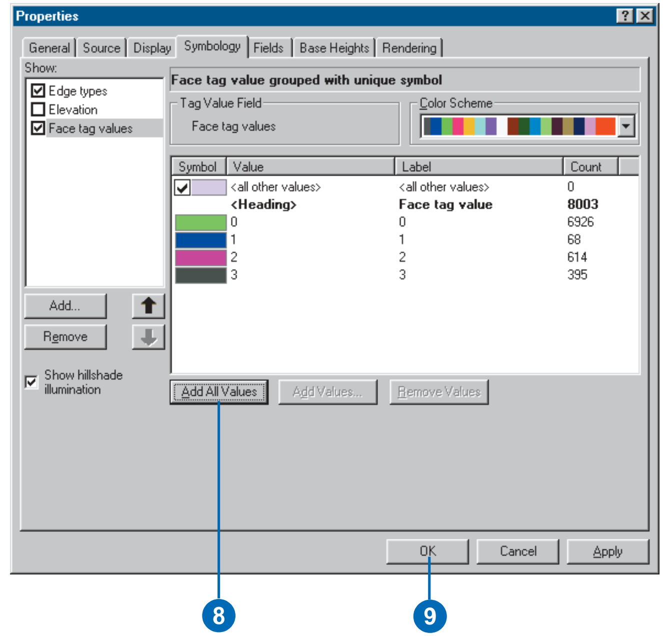
Drawing TIN faces by tag value

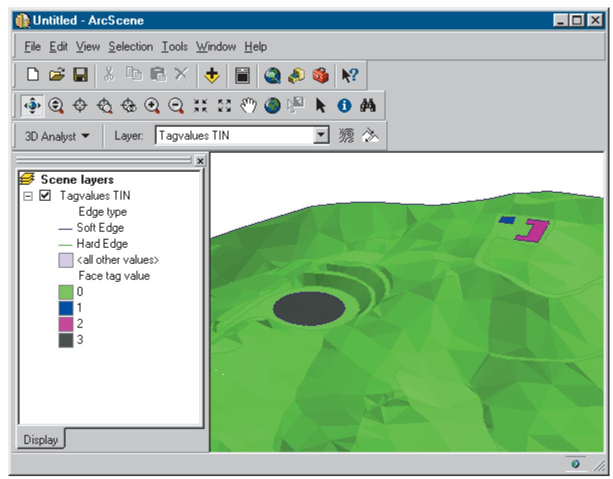
1. In the table of contents, right-click the TIN.
2. Click the Symbology tab.
3. Uncheck Elevation—or any other TIN face renderer that is checked.
4. Click Add.
5. Click Face tag value grouped with unique symbols.
6. Click Add.
7. Click Dismiss. ▸





8. Click Add All Values.
9. Click OK.





The TIN is drawn with different colors for each triangle's tag value.

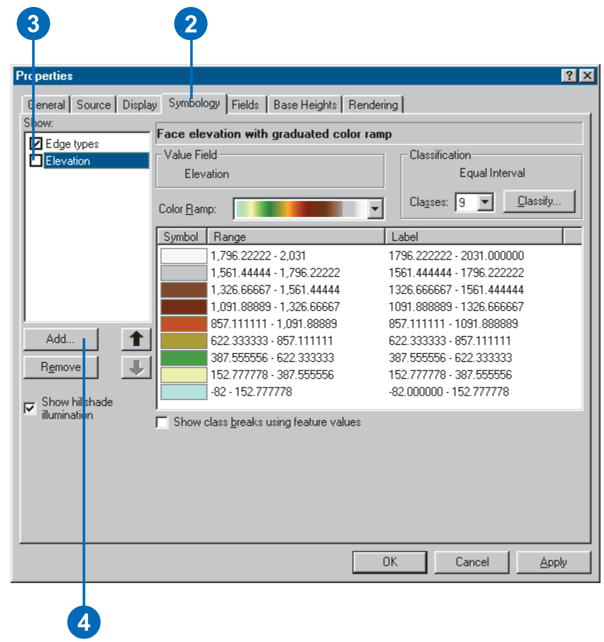
Tip

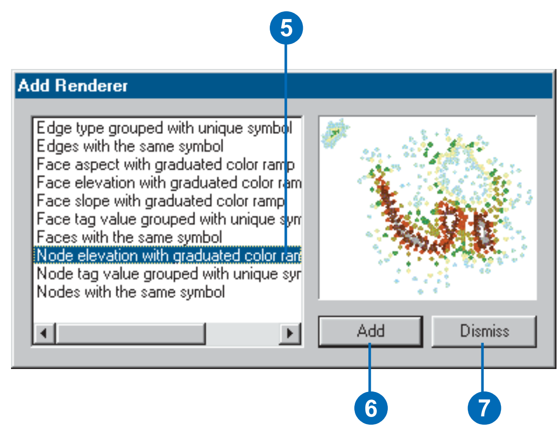
How else can I display TIN nodes?

You can also display TIN nodes by tag value or all nodes using the same symbol.

Drawing TIN nodes by elevation

1. In the table of contents, right-click the TIN.
2. Click the Symbology tab.
3. Uncheck Elevation—or any other TIN face renderer that is checked.
4. Click Add.
5. Click Node elevation with graduated color ramp.
6. Click Add.
7. Click Dismiss. ▸

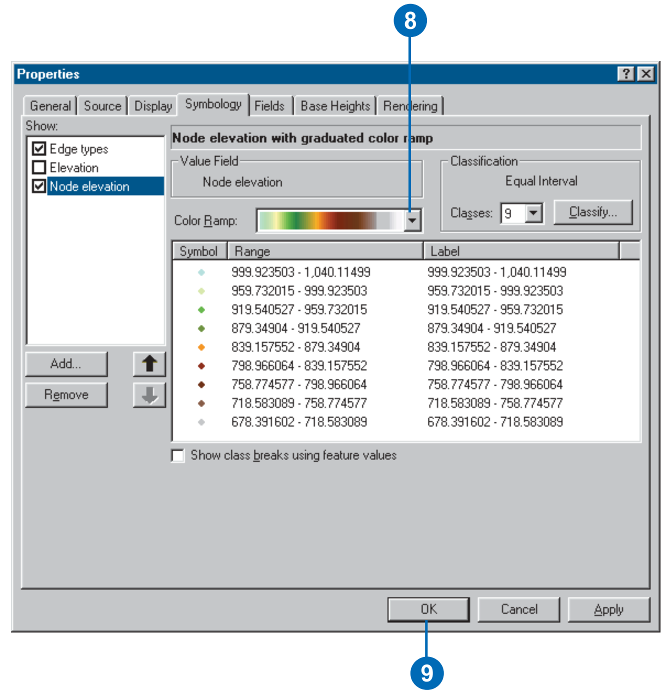


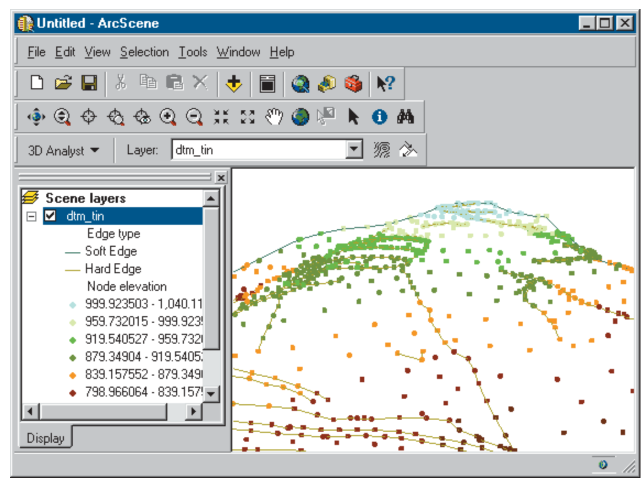


8. Click the Color Ramp dropdown arrow and select a color ramp.
9. Click OK.

The TIN nodes are rendered with colors to indicate their elevation.

In this example, you can also see the Soft and Hard edges, which are on by default, if the TIN has them.





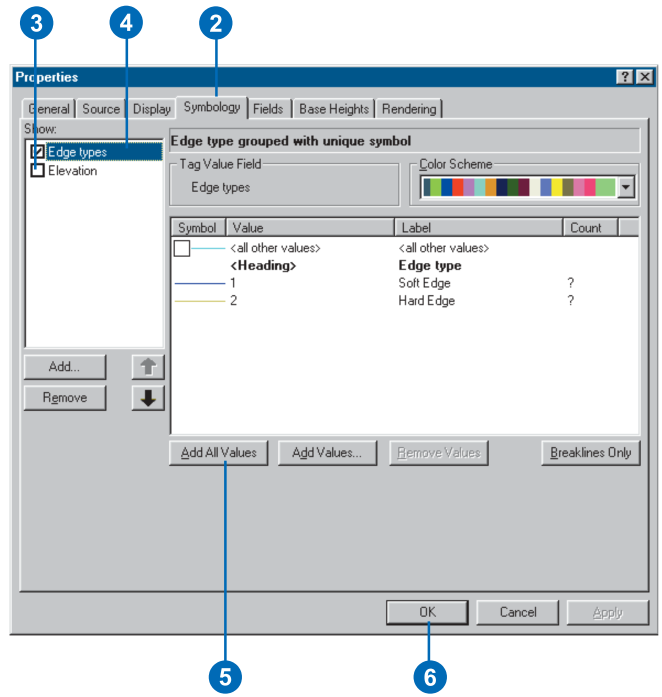
How else can I display TIN edges?

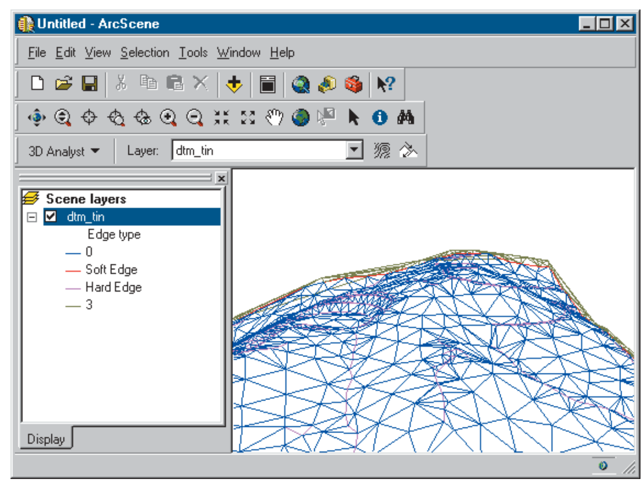
You can also display all TIN edges with the same symbol, and you can add and remove individual edge types from the display using the Add Values and Remove Values buttons.

Drawing TIN edges grouped with a unique symbol

1. In the table of contents, right-click the TIN.
2. Click the Symbology tab.
3. Uncheck Elevation.
4. Click Edge types.

   By default, only the type 1 and 2 (Hard and Soft) edges are displayed.
5. Click Add All Values.
6. Click OK.





Making a layer transparent

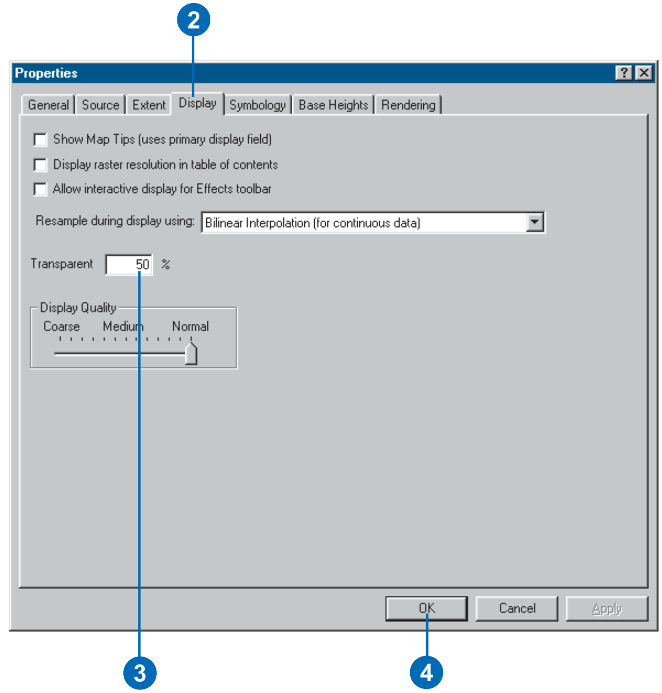
You can make raster, TIN, and feature layers transparent in scenes and maps. You can use transparent layers to visualize data that is scattered above and below a reference plane, compare two surfaces, and show terrain and subsurface features at the same time.

Tip

How else can I make a layer transparent?

You can use the Adjust Transparency tool on the Effects toolbar in ArcMap or the 3D Effects toolbar in ArcScene to set the transparency for a layer.

1. In the table of contents, right-click the layer and click Properties.
2. Click the Display tab.
3. Type a value in the Transparent box.
4. Click OK.



Shading a layer

You can shade raster, TIN, and feature layers relative to the position of a light source in scenes and maps. You can use shading to increase the perception of depth in the scene and to enhance details of the topography.

Tip

Changing the light source

You can change the light source for a scene or map to enhance the shading on a surface. Click the Illumination tab on the Scene Properties or Data Frame Properties dialog box for controls that let you change the light source position.

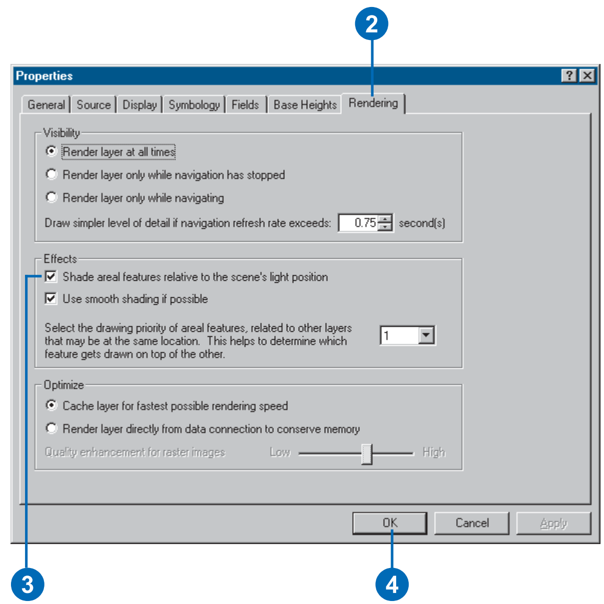
Tip

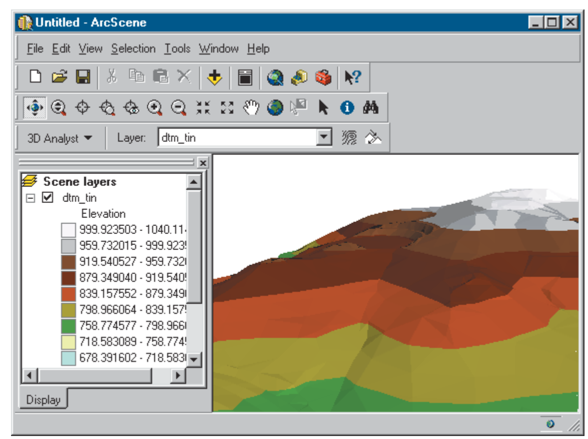
TIN face shading

You turn shading on or off for individual TIN face renderers by clicking the Symbology tab, clicking the TIN face rendered, and checking or unchecking the Show hillshade illumination check box.

1. In the table of contents, right-click the layer and click Properties.
2. Click the Rendering tab.
3. Check the box to Shade areal features relative to the scene's light position.

   The option to Use smooth shading if possible is checked by default. This minimizes the shading of small surface variations to create a smoother-looking shaded surface.
4. Click OK.





Analyzing surfaces

6

IN THIS CHAPTER

- **Querying surface values**
- **Understanding the shape of a surface**
- **Deriving slope and aspect information from a surface**
- **Creating contours**
- **Analyzing visibility**
- **Deriving hillshade from a surface**
- **Determining height along a profile**
- **Finding the steepest path**
- **Calculating area and volume**
- **Reclassifying your data**
- **Converting surfaces to vector data**
- **Creating 3D features**

Surfaces—fields that contain an infinite number of points—typically embody a great deal of information. You may want to simply view the surface, which is a great way to understand the surface in general, or you may be interested in specific information about parts of the surface. For example, you might want to know the altitude, temperature, atmospheric pressure, or concentration of a pesticide at a given point on the surface. You might want to know whether an observer at point A can see point B or how steep a proposed trail would be. You might be interested in the capacity of a proposed reservoir or the amount of material in a ridge. Or, you might be interested in general information about the surface's shape that is not immediately apparent by simply viewing the surface. For example, you may want to know what points are at the same elevation, what parts of the surface face the same direction, and where the concentration of a chemical or the land surface declines most precipitously.

In addition to letting you view surfaces in perspective and symbolize them in different ways, 3D Analyst gives you tools to get more information about specific points on a surface and lets you derive general information, such as slope, aspect, and contours, from the whole surface. 3D Analyst also gives you tools to convert surface data into vector data for analysis with other vector data sources.

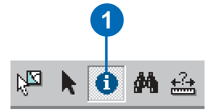
Querying surface values

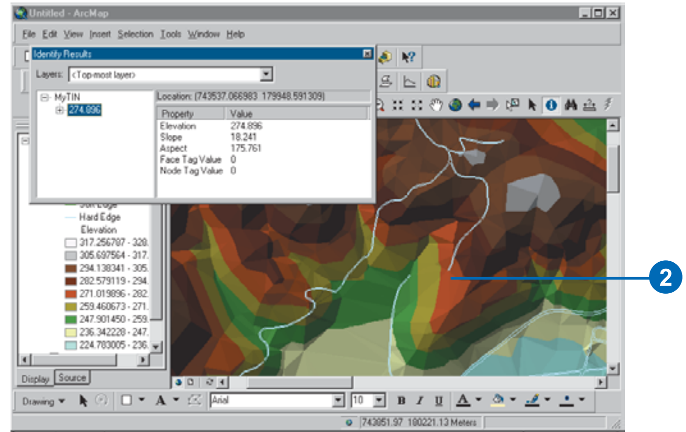
Sometimes just looking at 3D data is not enough. You often need to query data or derive new data to solve problems. 3D Analyst lets you explore the data on a map or in a scene and get the information you need.

You can click surfaces or features to find out what they are. When you click to get information about a TIN, you can find out the elevation, slope, and aspect of the point where you clicked. If the TIN feature has a tag value, it will be shown as well. When you click a raster surface, you see the elevation value at the point. When you click a feature, you see all of the attributes of the feature.

Identifying features or cells by clicking them

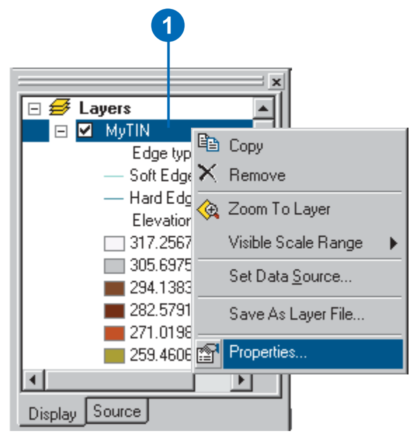
1. Click the Identify button on the Tools toolbar in ArcScene or ArcMap or on the 3D View Tools toolbar in ArcCatalog.
2. Click the mouse pointer over the feature or cell you want to identify.





Displaying Map Tips for surfaces

1. In the ArcMap table of contents, right-click the layer for which you want to display Map Tips and click Properties. ▸



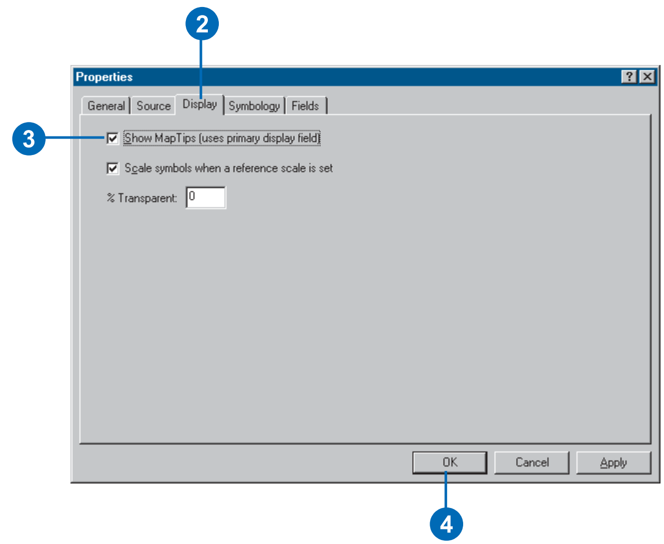
Tip

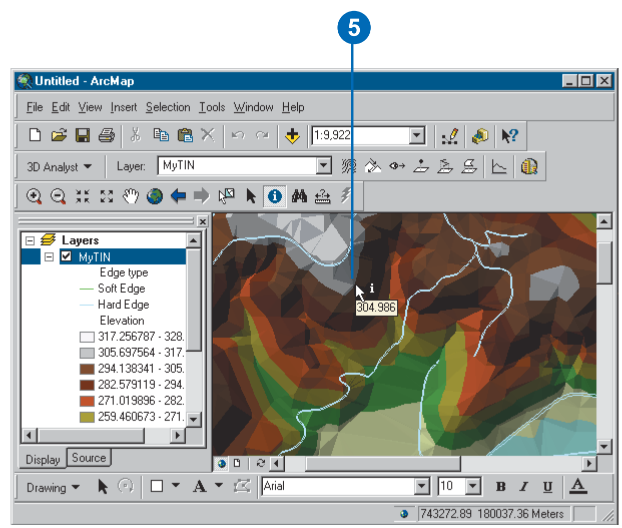
Displaying Map Tips for features

You can display Map Tips for feature data layers. The Map Tip shows the contents of a feature's primary display field.

You can change the primary display field by clicking the Fields tab in the Layer Properties dialog box, clicking the Primary display field dropdown arrow, then clicking a field.

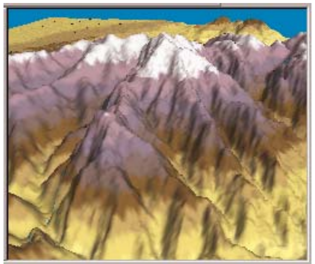
2. Click the Display tab.
3. Check Show Map Tips.
4. Click OK.
5. Move the mouse pointer over a TIN facet or raster cell to see the Map Tip.

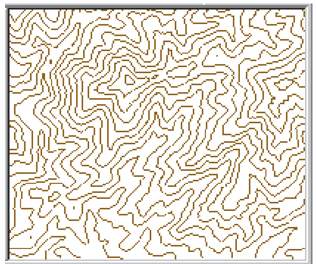




Understanding the shape of a surface

One good way to get a general understanding of the shape of a surface is to view it in 3D. You can zoom in and out and rotate the surface to see it from different angles. When you need to show a surface on a printed map, it can be useful to create contour lines to represent the surface.



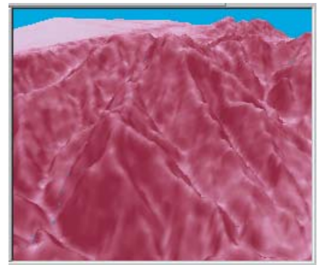


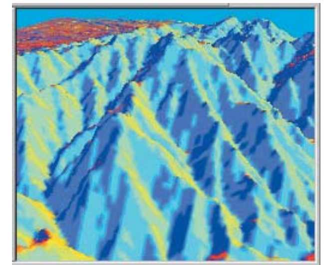
3D perspective hillshaded view of terrain and planimetric contour line representation of the same area

3D Analyst gives you tools that let you create individual contours or contours for a whole surface. An experienced map reader can tell that the surface is steeper where the lines are close together and identify ridgelines and streams from the shape of the contours. Contour lines can give you a feel for the shape of a surface, but they are not very useful as input for an analysis.

3D Analyst also has tools that allow for more quantitative analysis of the shape of a surface. Slope and aspect are two ways of quantifying the shape of a surface at a particular location.

Slope is the incline, or steepness, of a surface. It is often used in analyses to find areas with low slopes for construction or areas with high slopes, which may be prone to erosion or landslides.





Terrain slope and aspect surfaces. In the picture on the left, darker shades of red indicate steeper slopes. In the picture on the right, west-facing slopes are dark blue, and southeast-facing slopes are yellow.

Aspect is the direction a slope faces. Aspect is often used to determine how much sun a slope will receive—for instance, to model how vegetation will grow, how snow will melt, or how much solar heating a building will receive.

Slope and aspect in rasters and TINs

Rasters and TINs model a surface's slope and aspect in different ways. In a raster, slope and aspect are calculated for each cell by fitting a plane to the z-values of each cell and its eight surrounding neighbors. The slope or the aspect of the plane becomes the slope or aspect value of the cell in a new raster. In a TIN, each triangle face defines a plane with a slope and an aspect. These values are quickly calculated, as needed, when you query or render the faces.

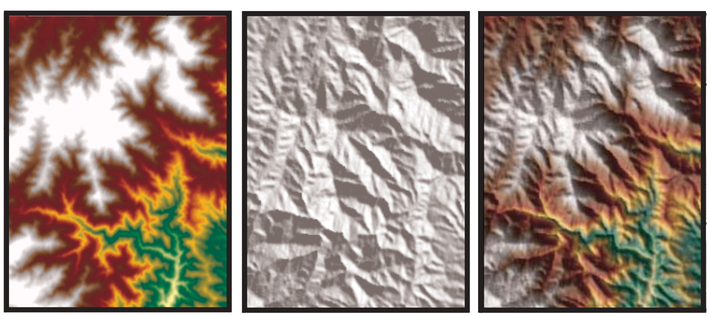
Hillshading surfaces

Hillshades are the patterns of light and dark that a surface would show when illuminated from a particular angle. Hillshades are useful for increasing the perception of depth in a 3D surface and for analysis of the amount of solar radiation available at a location.

There are two ways to hillshade a surface. One technique, which is most useful for adding depth for 3D visualization, is to turn on shading for the layer in ArcScene using the Rendering tab in the Layer Properties dialog box. This shades the layer—any surface or areal 3D feature layer—on the fly, using the scene's Illumination settings. Hillshading is on by default for TINs. You must turn it on for rasters because not all rasters are surfaces.

3D perspective view of terrain without and with hillshading

The second technique, which is useful as input for analyses and for enhancing depth in 2D surfaces in ArcMap, is to use the Hillshade surface analysis command in the 3D Analyst toolbar. This creates a new hillshade raster that you can make partially transparent and display in 2D, or 3D, with an elevation layer.

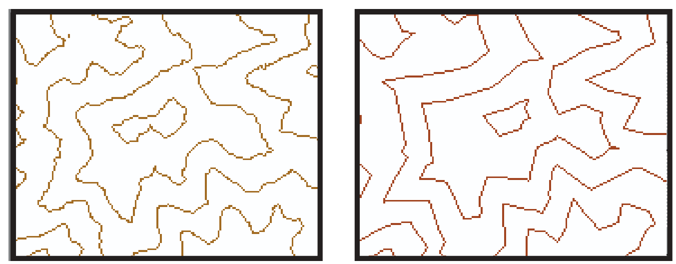


2D elevation raster, transparent hillshade raster, and shaded relief map

Deriving contour lines from rasters and TINs

Contours are lines that connect all contiguous locations with the same height—or other—value in the input grid or TIN. There are two ways to create contours with 3D Analyst. One way is to click the surface with the contour tool. This creates a single contour line, which exists as a 3D graphic in a scene or map. The other way to create contours is to use the Contour surface analysis command. This creates a series of contours with a given contour interval for the whole surface. These contours are saved as a feature class with a height attribute.

When you create contours from a grid, the contouring function interpolates lines between the cell centers. The lines seldom pass through the cell centers and do not follow the cell boundaries. In contrast, when you create contours from a TIN, the function interpolates straight lines across each triangle that spans the contour value, using linear interpolation between the edge endpoints to determine where the contour crosses the face.



Contours created from a raster surface and from a TIN surface

Calculating slope

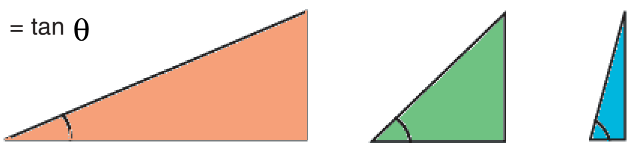
What is slope?

Slope identifies the steepest downhill slope for a location on a surface. Slope is calculated for each triangle in TINs and for each cell in rasters. For a TIN this is the maximum rate of change in elevation across each triangle. For rasters it is the maximum rate of change in elevation over each cell and its eight neighbors.

The slope command takes an input surface raster and calculates an output raster containing the slope at each cell. The lower the slope value, the flatter the terrain; the higher the slope value, the steeper the terrain. The output slope raster can be calculated as percent slope or degree of slope.

Degree of slope = θ Percent of slope = $\frac{\text{rise}}{\text{run}} * 100$

$\frac{\text{rise}}{\text{run}} = \tan \theta$

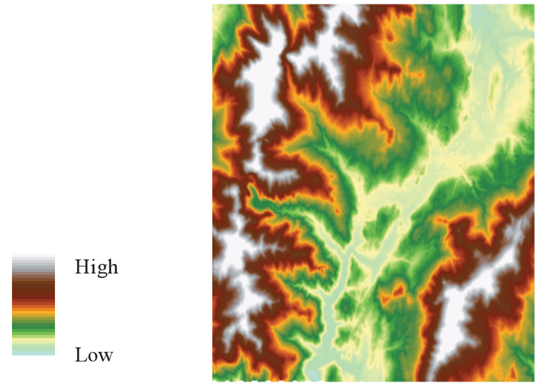


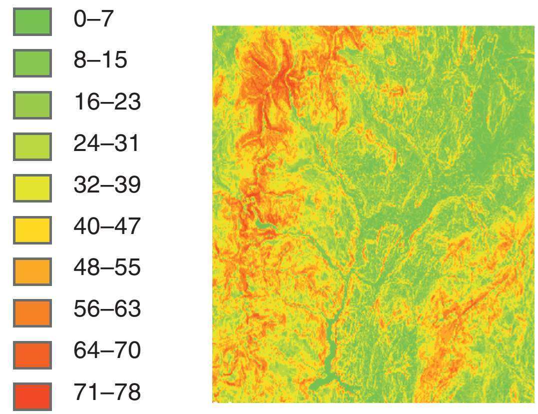
Degree of slope =	30	45	76
Percent of slope =	58	100	375

When the slope angle equals 45 degrees, the rise is equal to the run. Expressed as a percentage, the slope of this angle is 100 percent. Note that as the slope approaches vertical (90°), the percentage slope approaches infinity.

The slope function is most frequently run on an elevation grid, as the following diagrams show. Steeper slopes are shaded red on the output slope map.

Elevation grid





Slope map (in degrees)

Deriving slope from a raster surface

Sometimes the elevation of a point on a surface is less interesting than the slope of the surface at that point.

You can derive new rasters that show the slope of surfaces to get this information.

Slope is rapidly calculated on the fly for each triangle face in a TIN, though you can also create rasters of slope from TIN surfaces.

Tip

Degree and percent slope

Slope can be measured in degrees from horizontal (0–90) or percent slope, which is the rise divided by the run, times 100. As slope angle approaches vertical (90 degrees), the percent slope approaches infinity.

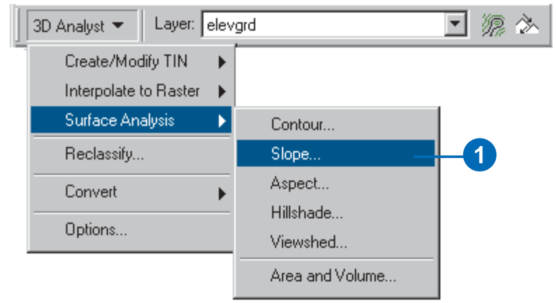
Tip

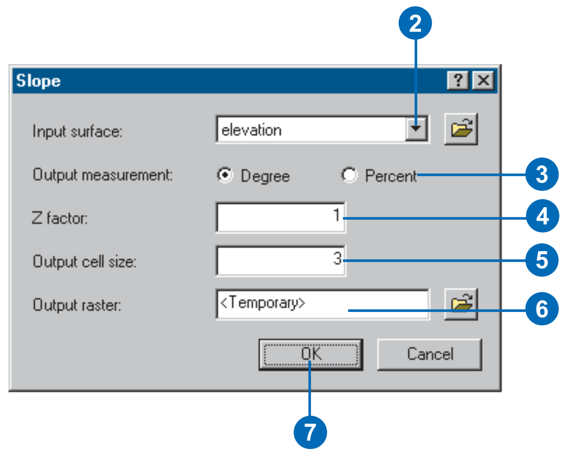
Why use a Z factor?

To get accurate slope results, the z units must be the same as the x,y units. If they are not the same, use a Z factor to convert z units to x,y units. For example, if your x,y units are meters, and your z units are feet, you could use a Z factor of 0.3048 to convert feet to meters.

Deriving slope

1. Click 3D Analyst, point to Surface Analysis, and click Slope.
2. Select the surface from which you want to derive a raster of slope values.
3. Click Degree or Percent.
4. Type a Z factor. This is calculated automatically if the input has a defined spatial reference that includes z unit information.
5. Type an Output cell size.
6. Type the name of the Output raster.
7. Click OK.



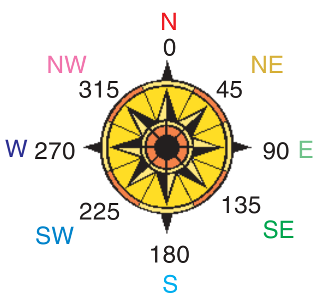


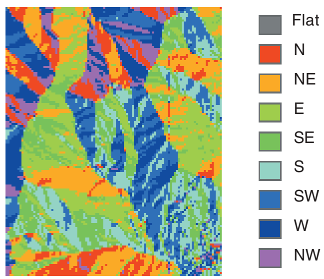
Calculating aspect

What is the aspect?

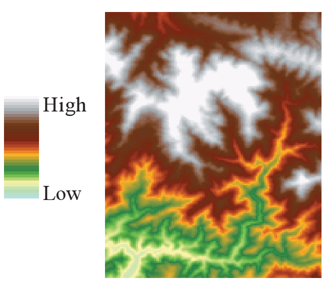
Aspect is the direction that a slope faces. It identifies the steepest downslope direction at a location on a surface. It can be thought of as slope direction or the compass direction a hill faces. Aspect is calculated for each triangle in TINs and for each cell in rasters.

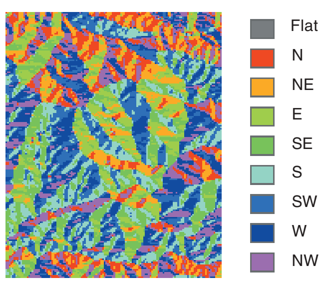
Aspect is measured counterclockwise in degrees from 0—due north—to 360—again due north, coming full circle. The value of each cell in an aspect grid indicates the direction in which the cell's slope faces. Flat slopes have no direction and are given a value of -1.





The diagram below shows an input elevation grid and the output aspect grid.





Why use the aspect function?

There are many different reasons to use the aspect function. For instance, you may want to:

- Find all north-facing slopes on a mountain as part of a search for the best slopes for ski runs.
- Calculate the solar illumination for each location in a region as part of a study to determine the diversity of life at each site.
- Find all southerly slopes in a mountainous region to identify locations where the snow is likely to melt first, as part of a study to identify those residential locations that are likely to be hit by meltwater first.
- Identify areas of flat land to find an area for a plane to land in case of emergency.

Deriving aspect from a raster surface

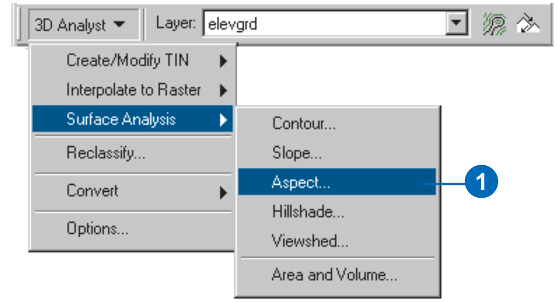
Sometimes the elevation of a point on a surface is less interesting than the aspect of the surface at that point.

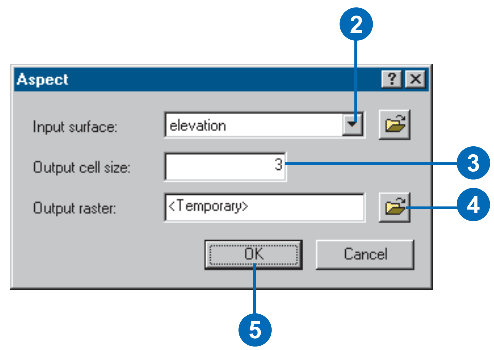
To get this information, you can derive new rasters that show the aspect of surfaces.

Aspect is rapidly calculated on the fly for each triangle face in a TIN, though you can also create rasters of aspect from TIN surfaces.

Deriving aspect

1. Click 3D Analyst, point to Surface Analysis, and click Aspect.
2. Click the dropdown arrow and click the surface from which you want to derive a raster of aspect values.
3. Type an Output cell size.
4. Type the name of the Output raster.
5. Click OK.





Mapping contours

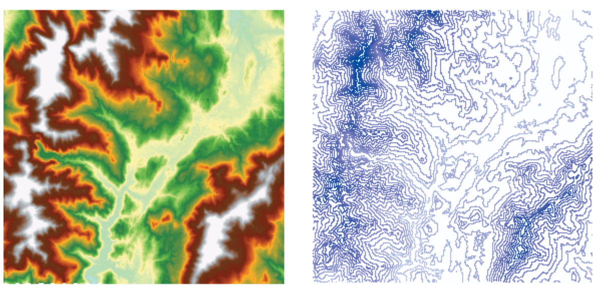
What are contours?

Contours are lines that connect points of equal value, such as elevation, temperature, precipitation, pollution, or atmospheric pressure. The distribution of the lines shows how values change across a surface. Where there is little change in a value, the lines are spaced farther apart. Where the value rises or falls rapidly, the lines are closer together.

Why map contours?

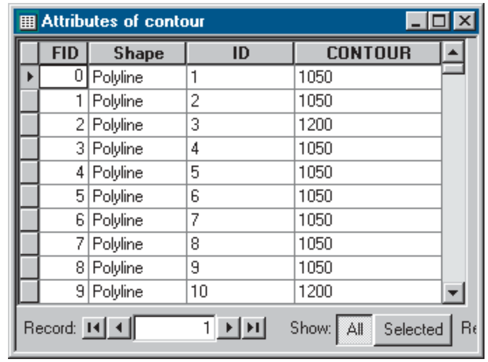
By following the line of a particular contour, you can identify which locations have the same value. By looking at the spacing of adjacent contours, you can gain a general impression of the gradation of values.

The example below shows an input elevation grid and the output contour map. The areas where the contours are closer together indicate the steeper locations. They correspond with the areas of higher elevation (in white on the input elevation grid).



Elevation grid and derived contour map

The attribute table of the contour features contains an elevation attribute for each contour line.



Attributes of contour

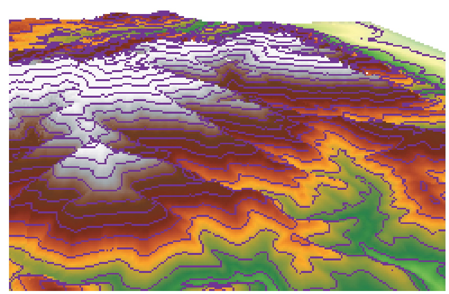
FID	Shape	ID	CONTOUR
0	Polyline	1	1050
1	Polyline	2	1050
2	Polyline	3	1200
3	Polyline	4	1050
4	Polyline	5	1050
5	Polyline	6	1050
6	Polyline	7	1050
7	Polyline	8	1050
8	Polyline	9	1050
9	Polyline	10	1200

Record: 1 Show: All Selected Re

Feature attribute table for contours

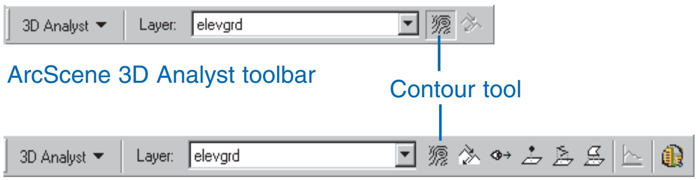
Using contours in a scene

Contours are a familiar surface representation for many people, and they have many uses in a scene. You can display contour features in a 3D scene by setting the base height of the contours from their value in the feature table. Contours in a scene can enhance terrain visualization.



Contours superimposed on terrain surface model

You can also create individual 3D graphic contour lines if you don't want contours for the whole surface. The Contour tool on the 3D Analyst toolbar lets you create 3D graphic contour lines by clicking a surface in a scene or map.

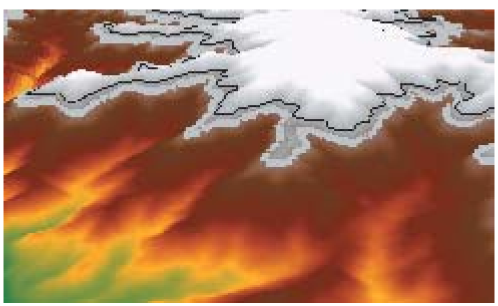


ArcScene 3D Analyst toolbar

Contour tool

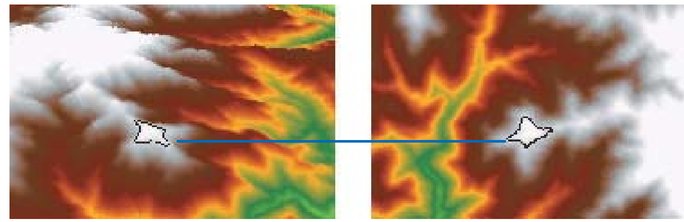
ArcMap 3D Analyst toolbar

You can use individual graphic contours to quickly find points of equal value in a scene. For example, you can use them to mark thresholds in the concentration of a chemical or fill lines in a reservoir.



3D graphic contours marking lines of equal elevation

You can copy 3D contour graphics between ArcScene and ArcMap to establish a visual correspondence between a 3D terrain representation in a scene and a 2D representation on a map.



3D graphic contour copied from a scene into a map

Deriving contour lines from a surface

Contour lines are a familiar way of representing surfaces on maps. A contour is a line through all contiguous points with equal height or other values. While contours may be readily interpreted by people, they are a poor surface model for computers.

You can create contour lines for a whole surface, or you can click a point and create a single contour that passes through it.

Tip

What is the offset?

The offset lets you control the position of the minimum contour. By default, the contour tool uses the minimum Input height to calculate the elevation for the minimum contour. If the default minimum contour was 298 meters, you could use an offset of 2 to place the minimum contour at 300.

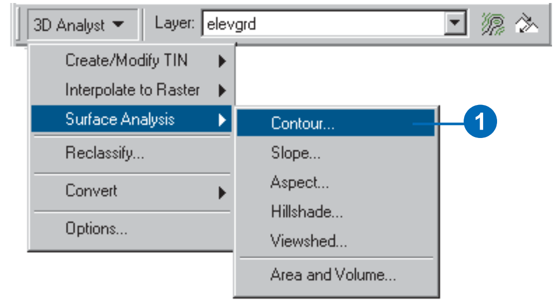
Tip

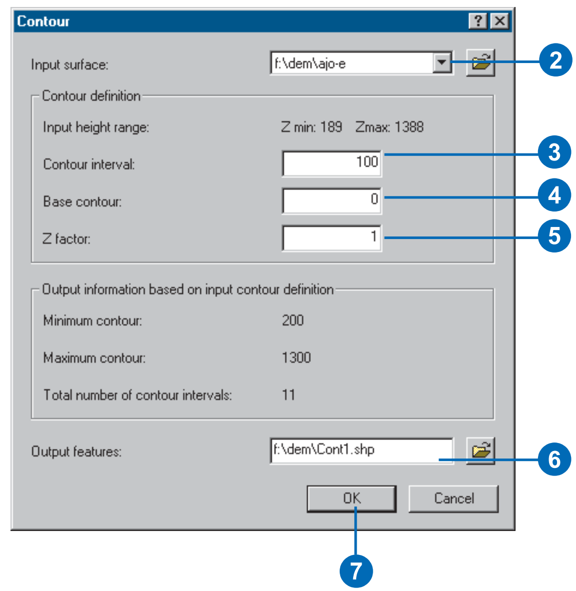
What is the Z factor?

The Z factor is used to adjust the units of the data. For example, if you have data in meters, and you want to produce contours in feet, you could use a Z factor of 3.28.

Deriving contours

1. Click 3D Analyst, point to Surface Analysis, and click Contour.
2. Click the dropdown arrow and click the surface from which you want to derive contours.
3. Type the Contour interval.
4. Optionally, specify a Base contour.
5. Optionally, specify a Z factor.
6. Browse to the location where you want to save the contours and type a name for them.
7. Click OK.

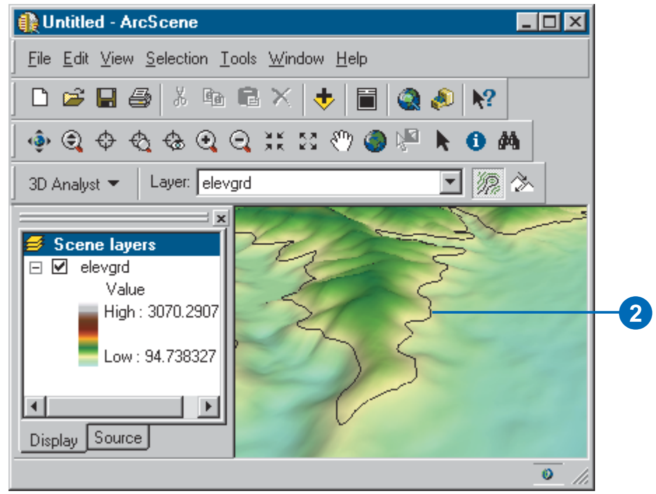




Creating a single contour

1. Click the Contour button.
2. Click the surface at the point where you want the contour.

   The contour is added as a 3D polyline graphic. The height of the contour is written to the status bar.

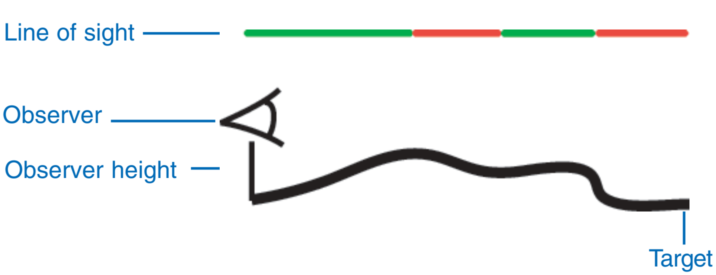


Analyzing visibility

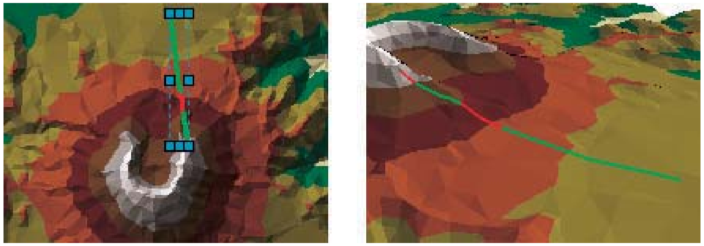
The shape of a terrain surface dramatically affects what parts of the surface someone standing at a given point can see. What is visible from a location is an important element in determining the value of real estate, the location of telecommunication towers, or the placement of military forces. 3D Analyst allows you to determine visibility on a surface from point to point along a given line of sight or across the entire surface in a *viewshed*.

What is a line of sight?

A *line of sight* is a line between two points that shows the parts of the surface along the line that are visible to or hidden from an observer. Creating a line of sight lets you determine whether a given point is visible from another point. If the terrain hides the target point, you can see where the obstruction is and what else is visible or hidden along the line of sight. The visible segments are shown in green, and the hidden segments are shown in red.



When you create a line of sight, you first set the offset of the observer and target points above the surface—the observer should always be set a little above the surface—then you click the observer and target points, and a graphic line appears between them.



Line of sight created in ArcMap and copied and pasted into a 3D scene

While you cannot create a line of sight in ArcScene, you can create one in ArcMap and copy and paste it into a scene. The line of sight is shown in the scene as a green and red 3D line graphic that follows the surface.

What is the viewshed?

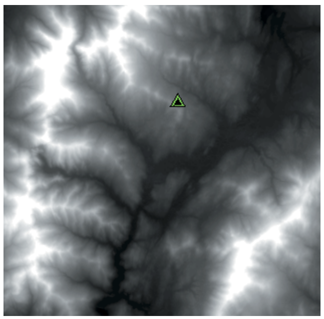
The viewshed identifies the cells in an input raster that can be seen from one or more observation points or lines. Each cell in the output raster receives a value that indicates how many observer points can see the location. If you have only one observer point, each cell that can be seen from the observer point is given a value of 1. All cells that cannot be seen from the observer point are given a value of 0.

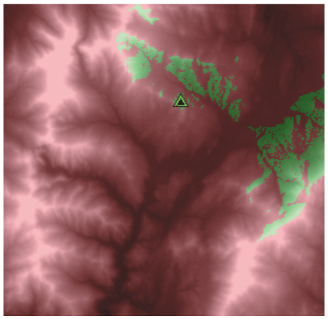
The Observer Points feature class can contain points or lines. The nodes and vertices of lines will be used as observation points.

Why calculate the viewshed?

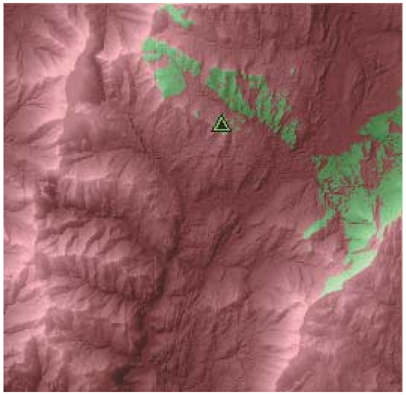
The viewshed is useful when you want to know how visible objects might be—for example, you may need to know "From which locations on the landscape will the landfill be visible if it is placed in this location?", "What will the view be like from this road?", or "Would this be a good place for a communications tower?".

In the example below, the viewshed from an observation point is identified. The elevation grid displays the height of the land (darker locations represent lower elevations), and the observation point is marked as a green triangle. Cells in green are visible from the observation point, while cells in red are not visible.





Displaying a hillshade underneath your elevation and the output from the Viewshed function gives a very realistic impression of the landscape and clearly indicates the locations that an observer can see from the observation point.



Creating a line of sight

A line of sight is a graphic line between two points on a surface that shows where along the line the view is obstructed. The color of the line indicates the locations where the surface is visible and where the surface is hidden. The status bar indicates if the target is visible or hidden.

The Line of Sight tool is only available on the 3D Analyst toolbar in ArcMap. Once you've created a line of sight in ArcMap, you can copy and paste it into a scene. The line of sight appears in a scene as a 3D line that follows the shape of the surface, with obstructed areas shown in red and clear areas shown in green.

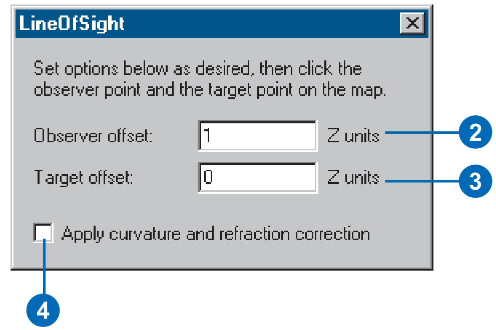
Tip

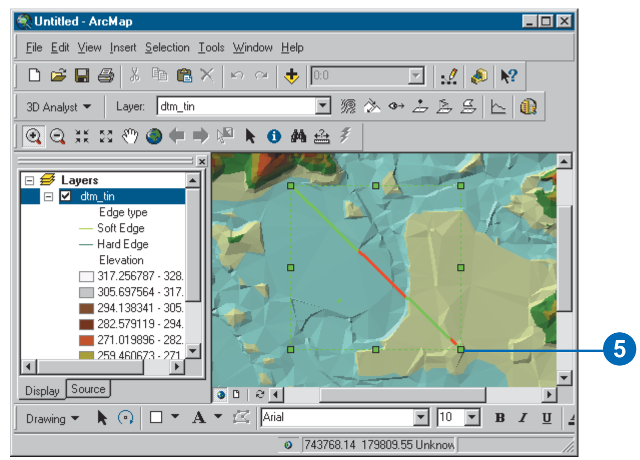
What are the Observer offset and Target offset?

The Observer offset is the eye level of the observer used in determining what is visible from the observer's location. An observer with a height of 0—the units are the same as the surface's z-units—will usually have a more obstructed view than one with a height of 1 or 10.

The Target offset is the height of the target point above the surface. Targets with a height of 0 are less visible than taller ones.

1. In ArcMap, click the Line of sight button on the 3D Analyst toolbar.
2. Optionally, type an Observer offset.
3. Optionally, type a Target offset.
4. Optionally, check the box to model curvature and refraction. For this option to be enabled, the surface must have a defined spatial reference in projected coordinates and defined z-units.
5. Click the surface at the observer location, then click the target location.





Deriving a viewshed

The Viewshed tool lets you find the places that can be seen from one or more observation points or lines. If lines are used as input, the observation points occur at the vertices of the lines.

The Viewshed tool creates a raster that contains cells coded to indicate whether they are visible to or hidden from the observer. If there are more than one observer points, each visible cell in the raster shows the number of points from which it is visible.

The Input surface can be a grid or TIN.

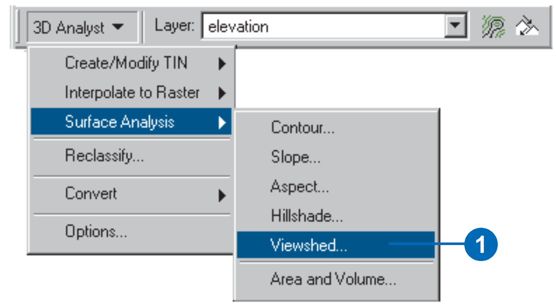
Tip

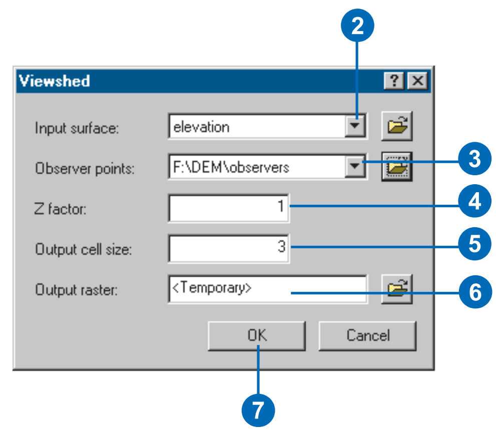
Using optional parameters

Optional viewshed parameters—SPOT, OFFSETA, OFFSETB, and so on—will be used if they are present in the feature observer attribute table. For more information, search ArcGIS Desktop Help for viewshed.

Creating a map displaying the viewshed

1. Click 3D Analyst, point to Surface Analysis, and click Viewshed.
2. Click the Input surface dropdown arrow and click the input surface you want to calculate the Viewshed from.
3. Click the Observer points dropdown arrow and click the feature layer to use as observer points.
4. Specify a Z factor. The default is 1. The Z factor is automatically calculated if the input surface has a spatial reference with z-units defined.
5. Specify an Output cell size.
6. Specify a name for the Output raster or leave the default to create a temporary dataset in your working directory.
7. Click OK.



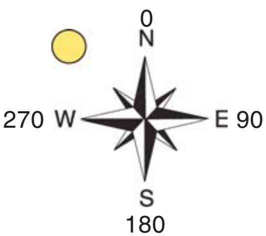


Computing hillshade

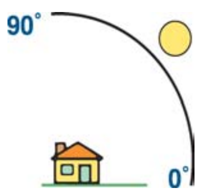
What is the hillshade function?

The hillshade function obtains the hypothetical illumination of a surface by determining illumination values for each cell in an elevation grid. It does this by evaluating the relationship between the position of the light source and the direction and steepness of the terrain. It can greatly enhance the visualization of a surface for analysis or graphical display.

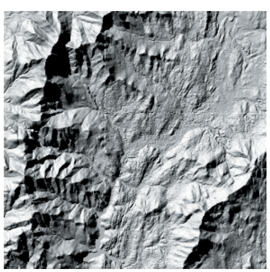
By default, shadow and light are shades of gray associated with integers from 0 to 255, increasing from black to white.



The azimuth is the angular direction of the sun, measured from north in clockwise degrees from 0 to 360. An azimuth of 90 degrees is due east. The default is 315 degrees (NW).



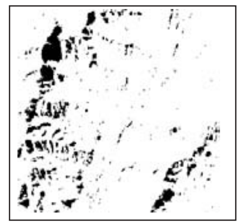
The altitude is the slope or angle of the illumination source above the horizon. The units are in degrees, from 0—on the horizon—to 90 degrees—overhead. The default is 45 degrees.



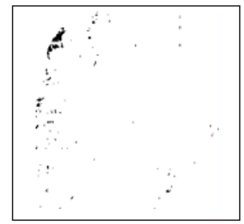
The light source of the hillshade to the left has an azimuth of 315 degrees and an altitude of 45 degrees.

Using hillshading in analysis

By modeling shadow, you can identify those cells that will be in the shadow of another cell at a particular time of day. Cells that are in the shadow of another cell are coded 0; all other cells are coded with integers from 1 to 255. You can reclassify values greater than 1 to 1, producing a binary raster. In the example below, the black areas are in shadow. The azimuth is the same, but the sun angle (altitude) has been modified.



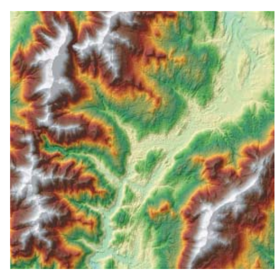
Sun angle: 45 degrees



Sun angle: 60 degrees

Using hillshading for display

By placing an elevation grid on top of a created hillshade grid and making the elevation grid transparent, you obtain a very realistic image of the elevation of the landscape. You can add other layers, such as roads or streams, to further increase the informational content in the display.



Deriving hillshade from a surface

Hillshade rasters show the hypothetical illumination of a surface, given a specified light source. Hillshades can be used to analyze the intensity and duration of sunlight received at a location on a surface. They are also used in conjunction with elevation layers to give depth to terrain maps. Hillshade works on raster and TIN layers.

You can accentuate topographic features by changing the altitude and azimuth values. Altitude is the slope or angle of the illumination source above the horizon; azimuth is its angular direction.

Tip

Modeling shadows

Check Model shadows to code cells 0 if they are in the shadow of other cells.

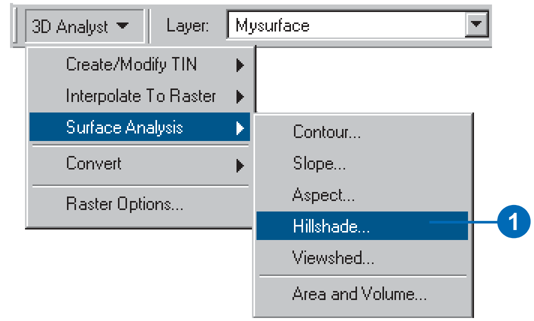
Tip

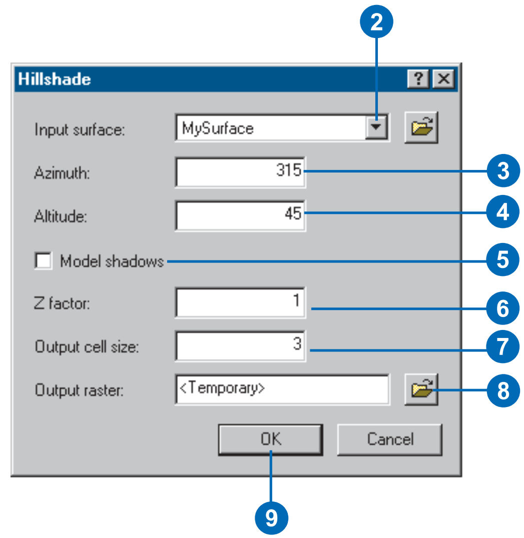
Specifying a Z factor

Use the Z factor to make the z-value units the same as the x,y units. If your x and y units are in meters, and your z units are in feet, you would specify a Z factor of 3.28 to convert feet to meters.

Creating a hillshade raster

1. Click 3D Analyst, point to Surface Analysis, and click Hillshade.
2. Click the dropdown arrow and click the surface from which you want to derive a hillshade raster.
3. Specify the azimuth you wish to use. The default is 315 degrees.
4. Specify an altitude. The default is 30 degrees.
5. Optionally, check Model shadows if you wish to include the shading effect of surrounding cells. Cells in the shadow of other cells will be coded 0.
6. Specify a Z factor. The default is 1. The Z factor is automatically calculated if the input surface has a spatial reference with z-units defined.
7. Specify an Output cell size.
8. Specify a name for the Output raster.
9. Click OK.





Tip

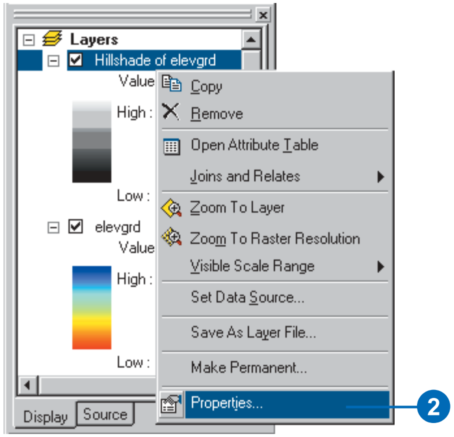
Adding depth in 2D

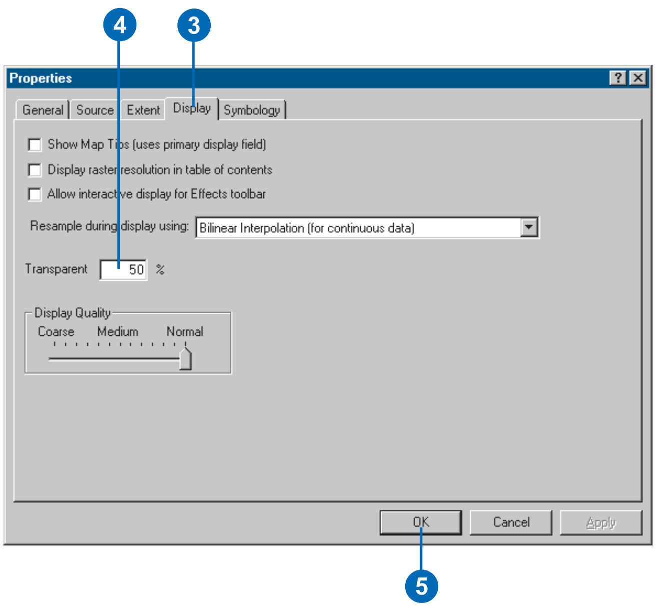
You can use a hillshade raster to enhance the perception of depth in a 2D raster representation of terrain. TINs support colored and shaded relief directly, so you don't need to create a hillshade for them.

Shading a raster with a hillshade

1. Create a hillshade raster from the surface raster that you want to shade (see 'Creating a hillshade raster' in this chapter).
2. In the table of contents, right-click the hillshade layer and click Properties.
3. Click the Display tab.
4. Type a value in the Transparent box to set the percent transparency. 50% is a reasonable value.
5. Click OK.

   You should now see the elevation raster through the hillshade raster.





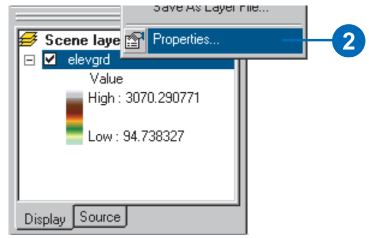
Shading 3D surfaces in a scene

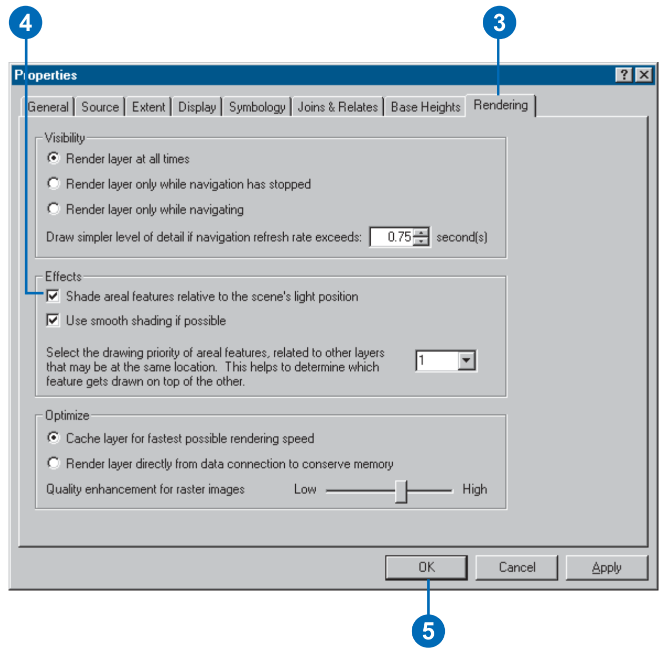
Shading increases the realism of a 3D scene and improves your ability to distinguish details of the surface. By default, TIN surfaces are drawn using shading, though you can turn it off. Raster surfaces can easily be shaded, too—without the need for creating a hillshade raster.

Shading a raster surface in a scene

1. In ArcScene, set the base height of the raster to itself (see 'Displaying raster surfaces in 3D' in Chapter 5).
2. Right-click the raster layer in the table of contents and click Properties.
3. Click the Rendering tab.
4. Check the box to Shade areal features relative to the scene's light position.

   The option to Use smooth shading if possible is checked by default. This minimizes the shading of small surface variations, which creates a smoother-looking shaded surface.
5. Click OK.





Determining height along a profile

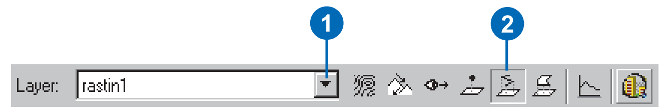
Profiles show the change in elevation of a surface along a line. They can help you assess the difficulty of a trail or evaluate the feasibility of placing a rail line along a given route.

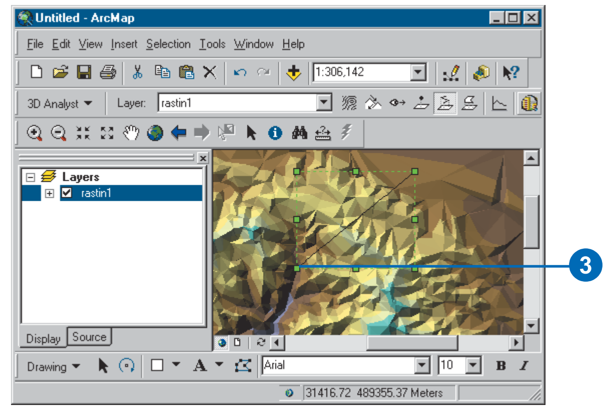
Tip

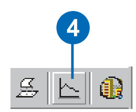
Profiling a 3D line feature

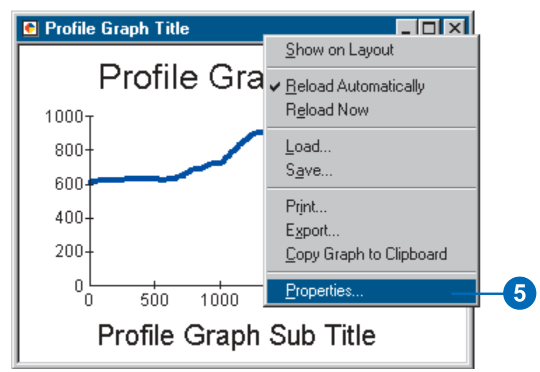
You can also create a profile graph for 3D line features, as well as for 3D graphics. Select a 3D line feature and click Profile Graph.

1. In ArcMap, click the Layer dropdown arrow and click the surface that you want to profile.
2. Click the Interpolate Line button.
3. Click the surface and digitize a line. When you are finished, double-click to stop digitizing.
4. Click the Profile Graph button.
5. Optionally, to change the layout of the profile graph, right-click on the title bar of the profile graph and click Properties.







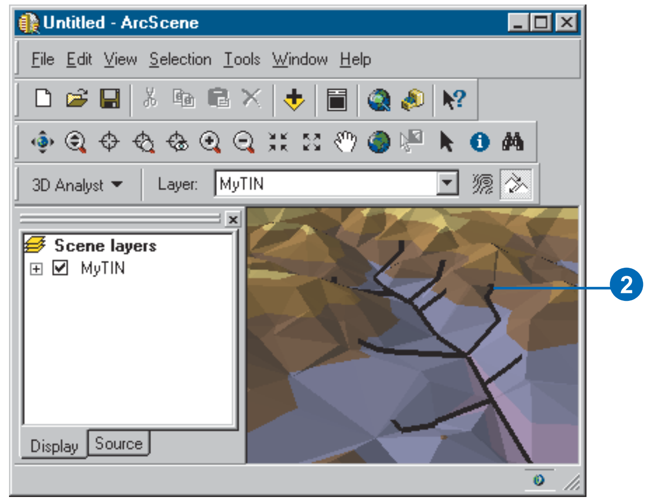


Finding the steepest path

You can evaluate runoff patterns on a TIN surface model by using the Steepest Path tool. The Steepest Path tool calculates the direction a ball would take if released at a given point on the surface. The ball will take the steepest path downhill until it reaches the perimeter of the surface model or it reaches a pit—a point all surrounding areas flow into. The result is a 3D graphic line added to the map or scene.

You can use the Steepest Path tool to evaluate the integrity of a TIN surface model, for example, to find paths that end unaccountably or meander off in a direction different from runoff on the actual site. If you're going to use the model for runoff studies, you'll need to correct such anomalies.

1. Click the Steepest Path tool.
2. Click the surface at the location where you want the path to begin.



Calculating area and volume

Use the Area and Volume Statistics tool to calculate 2D area, surface area, and volume. All are relative to a given reference plane.

The 2D area of a rectangular patch of surface model is simply its length times its width. The surface area is measured along the slope of the surface; it takes the variation in the height of the surface into account. Unless the surface is flat, the surface area will always be greater than the 2D area. You can compare the values for the 2D area and surface area to get an indication of the roughness or slope of the surface—the larger the difference between the values, the rougher or steeper the surface.

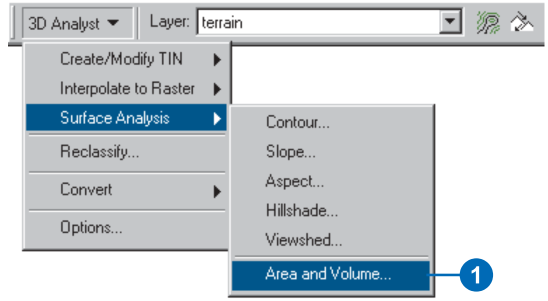
The volume is the space—in cubic map units—between the surface and a reference plane set at a particular height. You can determine the volume above the plane or below it, so you can answer questions such as “how much material is between the 1,200-foot contour and the top of this hill?” or “how much water could be stored in this reservoir if the top of the dam is at 300 meters?”

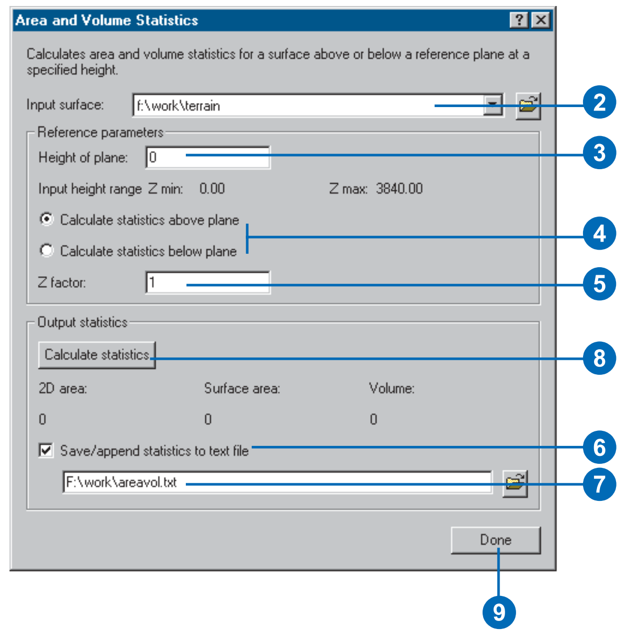
1. Click 3D Analyst, point to Surface Analysis, and click Area and Volume.
2. Click the dropdown arrow and click the surface for which you want to calculate area and volume statistics.
3. Type a height for the reference plane.
4. Click to calculate above or below the reference plane.
5. Optionally, type a Z factor to convert the z units to x,y units if the z units of the surface are not the same as the x,y units.

   You will get inaccurate results if the units are not the same and you do not include a Z factor.
6. Optionally, check the box to save the results to a text file.
7. Optionally, type a name for the text file where the results will be saved.
8. Click Calculate statistics.

   The 2D area, Surface area, and Volume will be reported in the area below this button and optionally written to a text file.

   You can change the parameters and click the button again to repeat the calculation.
9. Click Done.





Reclassifying data

What is reclassification?

Reclassifying your data simply means replacing input cell values with new output cell values. Reclassifying continuous data into categories is an important step in the process of transforming surface data into vector data for analysis.

The input data can be any raster format. If you add a multiband raster, the first band will be taken and used in the reclassification.

Why reclassify your data?

There are many reasons why you might want to reclassify your data. Some of the most common reasons are to:

- Replace values based on new information.
- Classify certain values together for display.
- Classify certain values together for conversion to vector format for analysis.
- Reclassify values to a common scale.
- Set specific values to NoData or to set NoData cells to a value.

Replacing values based on new information

Reclassification is useful when you want to replace the values in the input raster dataset with new values. This could be due to finding out that the value of a cell, or a number of cells, should actually be a different value. For example, this may happen if the land use in an area changed over time.

Grouping values together

You may want to simplify the information in a dataset. For instance, you may want to group together various types of forest into one forest class or group aspect values into general categories, such as north-facing and south-facing slopes.

You may also want to do overlay and selection on data from a surface—for example, finding and selecting areas that have low slope and southeast aspect. You can create the vector data for this kind of polygon overlay and selection using reclassified slope and aspect rasters.

Reclassifying values of a set of rasters to a common scale

Another reason to reclassify is to assign values of preference, sensitivity, priority, or some similar criteria to a raster. This may be done on a single raster (a raster of soil type may be assigned values of 1–10 that represent degree of susceptibility to erosion) or with several rasters to create a common scale of values.

Setting specific values to NoData or setting NoData cells to a value

Sometimes you want to remove specific values from your analysis. For example, a certain land cover type may have restrictions—such as wetlands restrictions—that mean you can't build there. In such cases you might want to change these values to NoData in order to remove them from further analysis.

In other cases, you may want to change a value of NoData to be a value. For example, you might acquire new information and want to update a value of NoData with the new value.

Reclassifying your data

The Reclassify dialog box enables you to modify the values in an input dataset and save the changes to a new output dataset.

There are many reasons why you may wish to do this, including replacing values based on new information, grouping entries, reclassifying values to a common scale—for example, for use in suitability analysis—or setting specific values to NoData or setting NoData cells to a value.

The Load button enables you to load a remap table that was previously created by pressing the Save button and applying it to the input raster dataset.

The Save button enables you to save a remap table for later use.

Tip

Replacing NoData values

NoData values can be turned into numeric values in the same way as you replace any value.

Tip

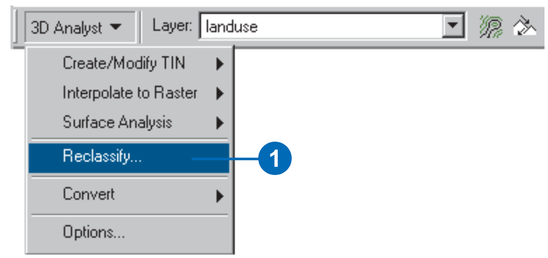
Changing the classes of your old values

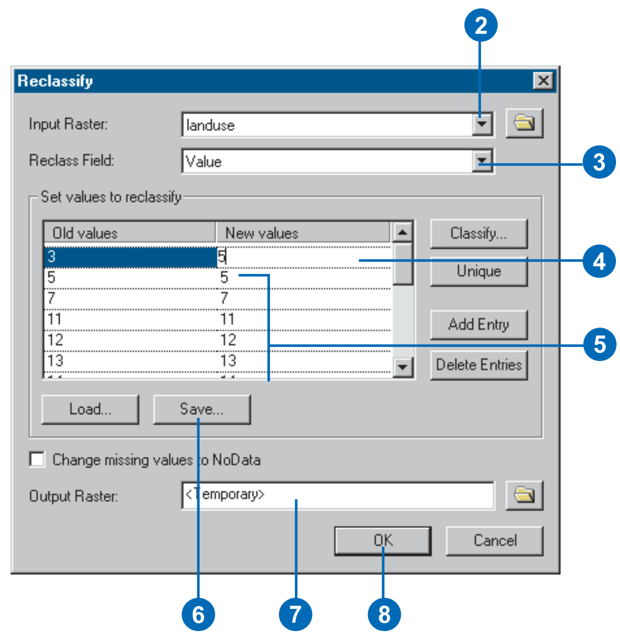
Click Classify to classify your old values differently.

Click Unique to separate classes of old values into unique values.

Replacing values based on new information

1. Click the 3D Analyst dropdown arrow and click Reclassify.
2. Click the Input Raster dropdown arrow and click the raster with a value you wish to change.
3. Click the Reclass Field dropdown arrow and click the field you wish to use.
4. Click the new value you wish to change and type a new value.
5. Type the Old values entry in each New values input box for all other values to keep them the same in the Output Raster.
6. Optionally, click Save to save the remap table.
7. Specify a name for the Output Raster or leave the default to create a temporary dataset in your working directory.
8. Click OK.





Tip

Ungrouping entries

To ungroup entries, right-click the group and click Ungroup Entry.

Tip

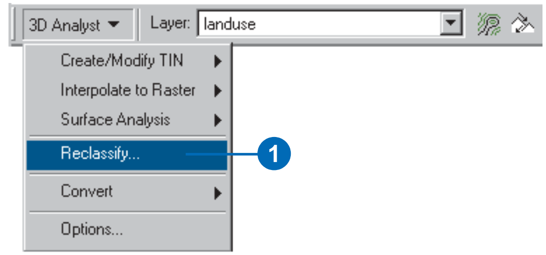
Changing the classes of your old values

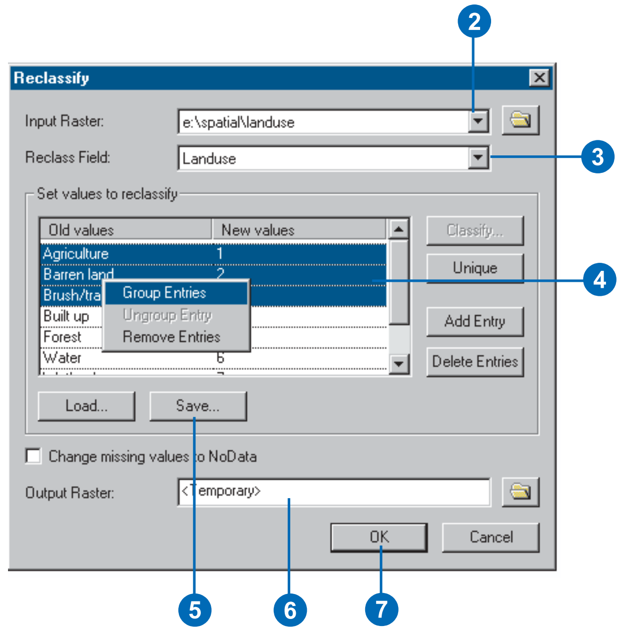
Click Classify to change the classification of your old values.

Click Unique to separate classes of old values into unique values.

Grouping entries

1. Click the 3D Analyst dropdown arrow and click Reclassify.
2. Click the Input Raster dropdown arrow and click the raster whose values you wish to group.
3. Click the Reclass Field dropdown arrow and click the field you wish to use.
4. Click the New values you wish to group—click one, then hold down the Shift key and click the next one—then right-click on Group Entries.
5. Optionally, click Save to save the remap table.
6. Specify a name for the Output Raster or leave the default to create a temporary dataset in your working directory.
7. Click OK.





Tip

Changing input ranges to unique values

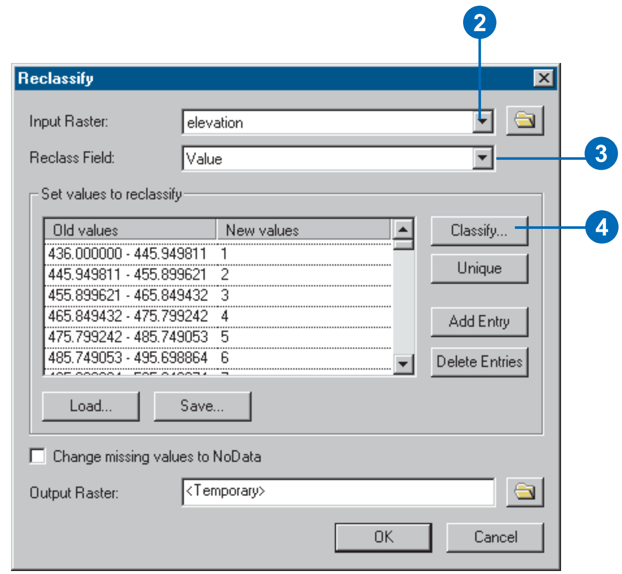
If your input values are split into ranges and you want them to be unique values, click Unique.

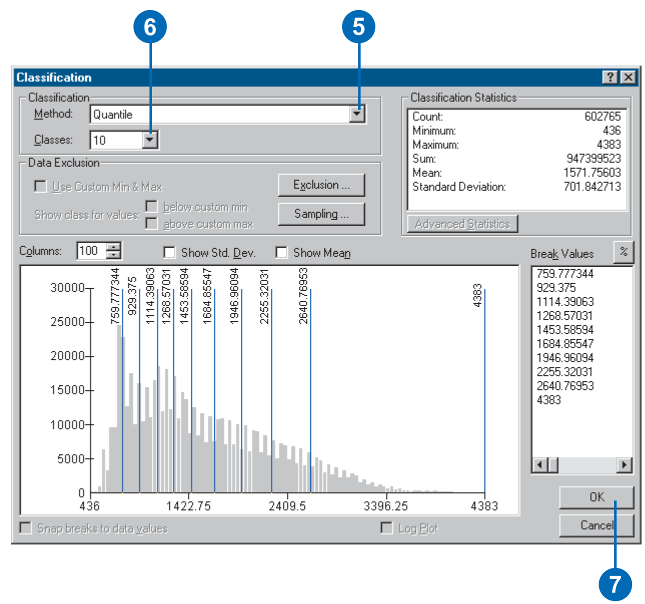
See Also

See 'Standard classification schemes' in Using ArcMap.

Changing the classification of input ranges

1. Click the 3D Analyst dropdown arrow and click Reclassify.
2. Click the Input Raster dropdown arrow and click the raster whose values you wish to reclassify.
3. Click the Reclass Field dropdown arrow and click the field you wish to use.
4. Click the Classify button.
5. Click the Method dropdown arrow and choose a classification method to use to reclassify your input data.
6. Click the Classes dropdown arrow and choose the number of classes into which to split your input data.
7. Click OK.
8. Modify the New values for your output grid if appropriate.
9. Specify a name for the Output Raster or leave the default to create a temporary dataset in your working directory.
10. Click OK in the Reclassify dialog box.





Tip

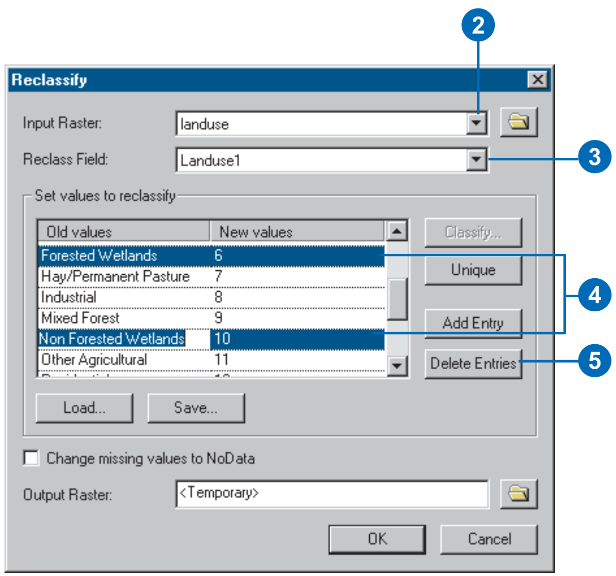
To change a value to NoData

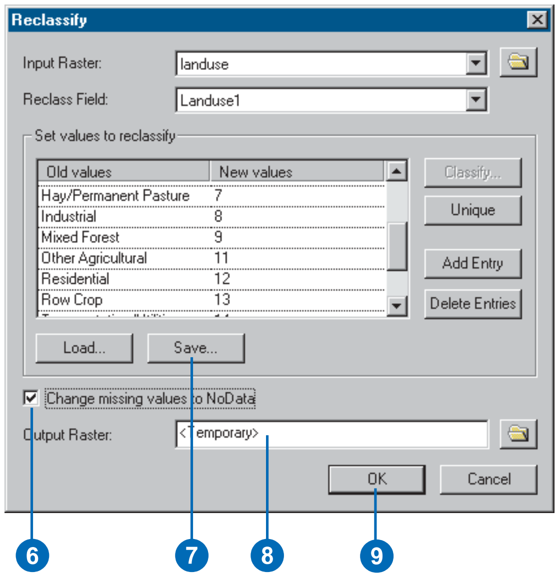
You can also type "NoData" in the input box for a new value to change an input value to NoData.

Setting specific values to NoData

1. Click the 3D Analyst dropdown arrow and click Reclassify.
2. Click the Input Raster dropdown arrow and click the raster with values you wish to set to NoData.
3. Click the Reclass Field dropdown arrow and click the field you wish to use.
4. Click the input boxes for the New values you wish to turn to NoData.
5. Click Delete Entries.
6. Check Change missing values to NoData.
7. Optionally, click Save to save the remap table.
8. Specify a name for the Output Raster or leave the default to create a temporary dataset in your working directory.
9. Click OK.

   The values you deleted will be turned to NoData in the output grid.





Converting rasters and TINs to vector data

Why convert from surfaces to vectors?

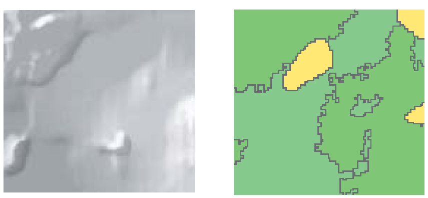
Raster and TIN surfaces may contain information that you would like to use in conventional, vector-based, GIS operations. Two such vector-based operations are overlay and selection by location.

Converting rasters to features

The general steps to convert rasters to features for analysis are:

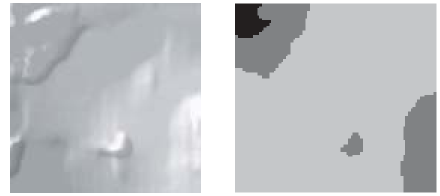
1. Convert the raster surface data to categorical data (elevation, slope, or aspect categories).
2. Convert the categories to polygons.
3. Use the polygons with other vector data and select the areas that meet some criteria.

For example, suppose you have a raster elevation model and a polygon feature class of vegetation.

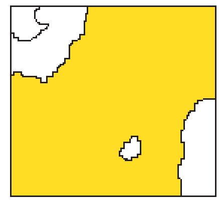


Raster elevation model and vector vegetation feature class.

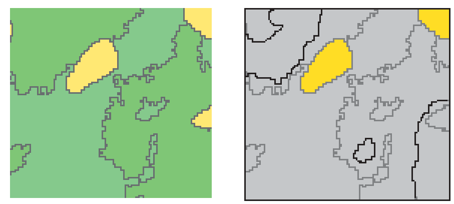
You might want to select parts of a study area that are below 1,000 meters in elevation and have a particular vegetation type. In order to do the vector overlay and selection with the elevation data, you need to convert the raster to polygons.



Raster elevation model with continuous values and raster converted to categorical values



Polygon feature class created from the raster categories



Polygon feature class of vegetation types and selected vegetation that falls within a selected elevation class polygon

You can also use this technique to extract linear features from rasters. For example, you could extract stream courses or roads from land cover rasters or remotely sensed images.

Converting TINs to features

Converting TINs to features involves fewer steps. You can extract slope and aspect polygons directly from TIN surfaces, or you can extract the elevation values of nodes in the TIN as a point feature class. You can use the slope and polygon features extracted from a TIN just as you would use such features extracted from a raster.

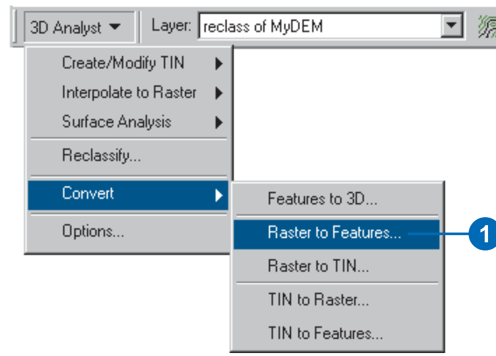
Converting surfaces to vector data

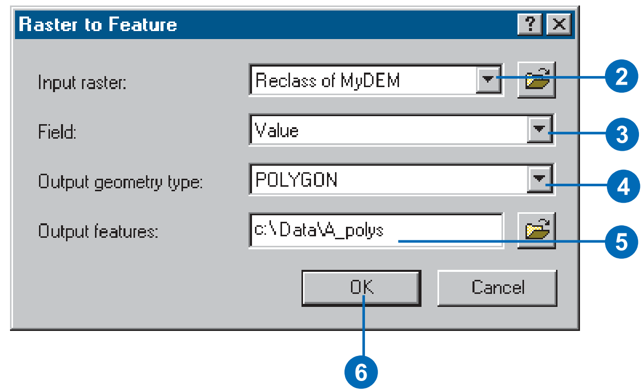
You can convert surface data to vectors—point, line, or polygon features—for use in selections or overlay or for editing.

Rasters must be reclassified from continuous data to categorical data—for example, from slope in degrees to slope categories or from elevation values to elevation classes.

Converting raster data to features

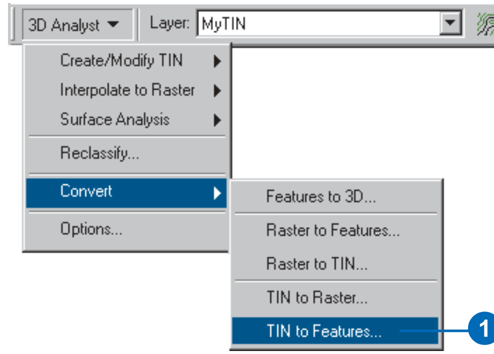
1. Click the 3D Analyst dropdown arrow, point to Convert, and click Raster to Features.
2. Click the Input raster dropdown arrow and click the raster dataset you want to convert to a feature.
3. Click the Field dropdown arrow and click the field you want to copy to the output features.
4. Click the Output geometry type dropdown arrow and click the type of feature you want to create from your raster data.
5. Specify a name for Output features or leave the default to create a temporary dataset in your working directory.
6. Click OK.

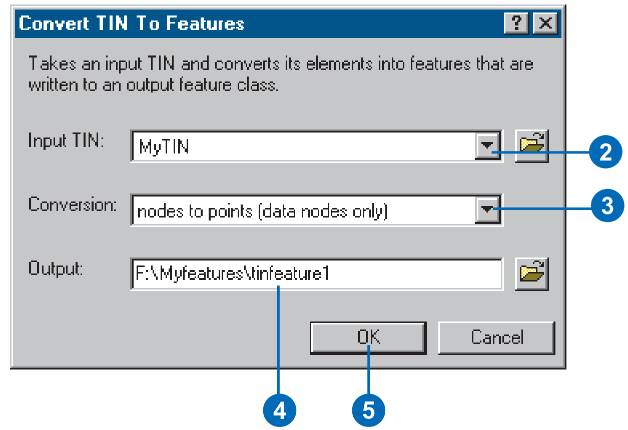




Converting TIN data to features

1. Click the 3D Analyst dropdown arrow, point to Convert, and click TIN to Features.
2. Click the Input TIN dropdown arrow and click the TIN you want to convert to features.
3. Click the Conversion dropdown arrow and click the type of conversion that you want to do.
4. Specify a name for the Output feature or leave the default to create a temporary dataset in your working directory.
5. Click OK.





Creating 3D features

It is often useful to have features with 3D geometry. Although you can display 2D features by draping them over a surface, 3D features are displayed more rapidly, and you can share them with others without having to send along the surface data.

You can convert existing 2D features to 3D in three ways. One method is to derive the height values of the features from a surface. The second is to derive the height value from an attribute of the features. The third is to derive the features' height from a constant value.

You can also digitize new features over a surface in ArcMap and interpolate the features' z-values from the surface during digitizing.

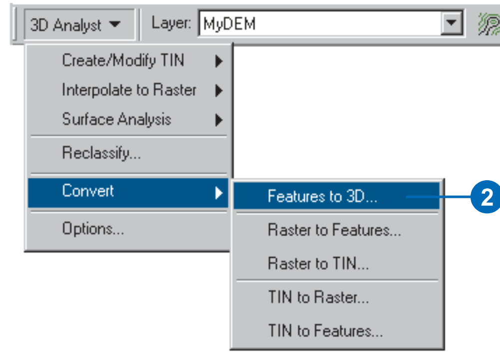
Tip

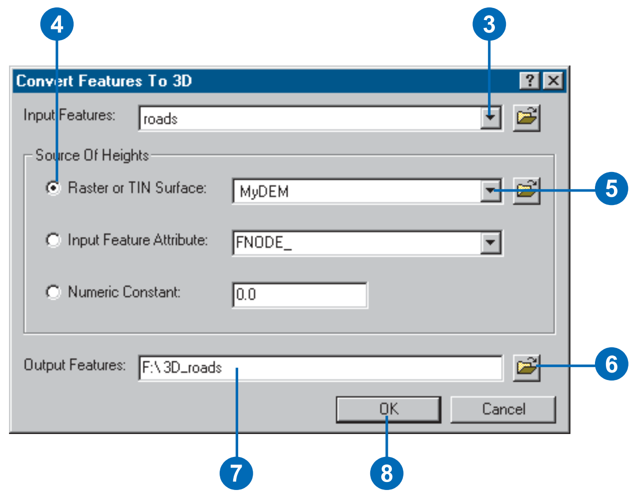
Creating 3D graphics

You use the same tools to digitize 3D graphics from a surface as you use to digitize 3D features. See 'Creating 3D graphics by digitizing over a surface'. You can copy and paste 3D graphics from ArcMap to ArcScene.

Deriving existing features' height from a surface

1. Add the 2D features and the surface to a map or scene.
2. Click 3D Analyst, point to Convert, and click Features to 3D.
3. Click the Input Features dropdown arrow and click the features that you want to convert to 3D.
4. Click the Raster or TIN Surface button to set the source for the features' heights.
5. Click the dropdown arrow and click the surface that you wish to use.
6. Optionally, browse to the location where you want to save the output feature class or shapefile.
7. Type the name of the output 3D feature class or shapefile.
8. Click OK.



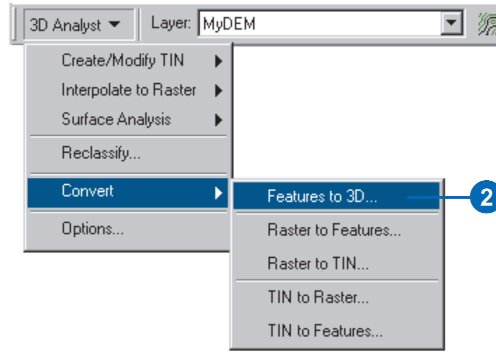


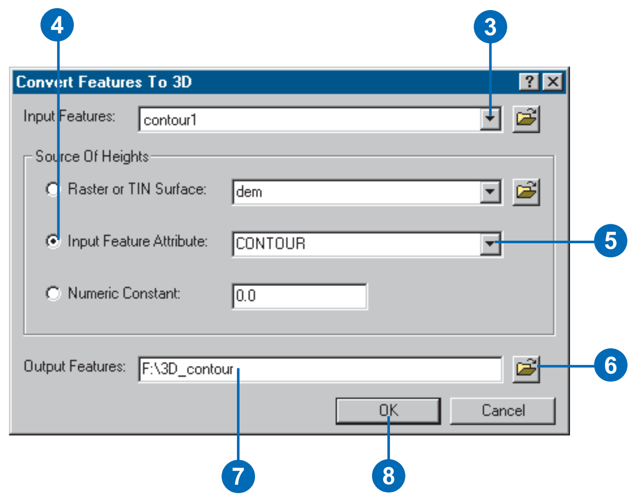
Giving all of the features the same z-value

You can click the numeric constant button if you want to give all of the output 3D features the same z-value.

Deriving existing features' height using an attribute

1. Add the 2D features to a map or scene.
2. Click 3D Analyst, point to Convert, and click Features to 3D.
3. Click the Input Features dropdown arrow and click the features that you want to convert to 3D.
4. Click the Input Feature Attribute button to set the source for the features' heights.
5. Click the attribute that you wish to use for the features' heights.
6. Optionally, browse to the location where you want to save the output feature class or shapefile.
7. Type the name of the output 3D feature class or shapefile.
8. Click OK.





Selecting the surface to supply z-values

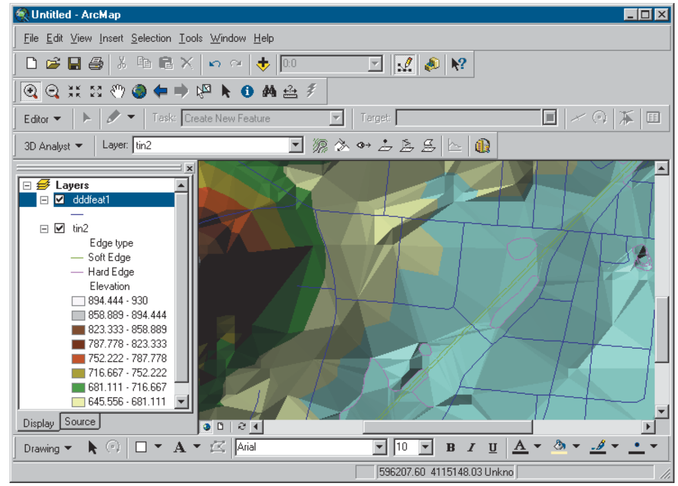
If you have more than one surface on the map, use the Layer dropdown list on the 3D Analyst toolbar to select the surface that will be the source of your features' z-values.

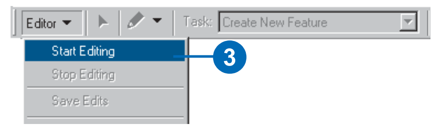
3D polygon feature limitation

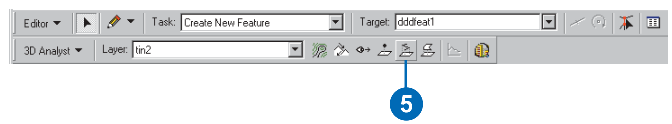
The perimeters of 3D polygons are where the z-values are stored. The interior elevations are interpolated based on these values. For relatively smooth surfaces, the interiors of 3D polygon features will reflect the actual surface reasonably well. If you need to accurately model the details of an area, use a TIN or raster surface instead of polygons.

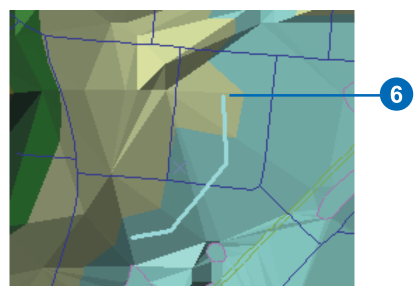
Creating 3D features by digitizing over a surface

1. Add the 3D feature class—an existing feature class with one of the following geometries: pointZ, polylineZ, polygonZ—to which you want to add features to the map.
2. Add the surface that you want to use as the source for the features' height to the map.
3. On the Editor toolbar, click Editor and click Start Editing.
4. If you have more than one feature class on the map, identify the workspace of the feature class in which you will be creating new 3D features. Click OK.
5. Click the Interpolate Point, Interpolate Line, or Interpolate Polygon button, depending on the geometry of the feature class you are creating.
6. Click on the surface and create the edit sketch for the feature just as you would for a 2D feature.
7. When you are finished digitizing, click Editor and click Save Edits.
8. Click Editor and click Stop Editing.
9. Click Yes to save your edits.









Tip

Selecting the surface to supply z-values

If you have more than one surface on the map, use the Layer dropdown list on the 3D Analyst toolbar to select the surface that will be the source of your graphics' z-values.

Tip

3D polygon graphics limitation

The perimeters of 3D polygons are where the z-values are stored. The interior elevations are interpolated based on these values. For relatively smooth surfaces, the interiors of 3D graphic polygons will reflect the actual surface reasonably well. If you need to accurately model the details of an area, use a TIN or raster surface instead of polygons.

Tip

Using 3D graphics in ArcScene

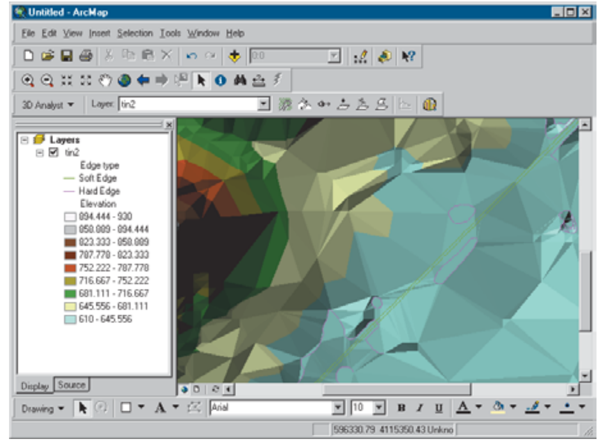
You can copy 3D graphics that you've created in ArcMap and paste them into ArcScene.

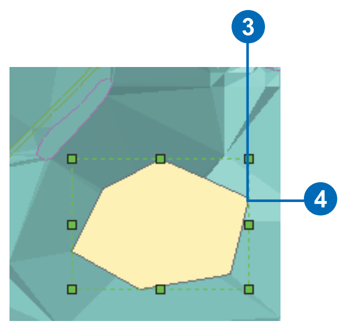
Creating 3D graphics by digitizing over a surface

1. Add the surface that you want to use as the source for the graphics' height to the map.
2. Click the Interpolate Point, Interpolate Line, or Interpolate Polygon button, depending on the geometry of the graphics you are creating.
3. Click the surface at the location where you want to start drawing.

   If you are using the Interpolate Point tool, a point will appear.

   If you are using the Interpolate Line or Interpolate Polygon tools, the first vertex will appear. Click the surface where you want to create the next vertex.
4. Double-click the surface to create the last vertex and finish drawing.





3D visualization

7

IN THIS CHAPTER

- **Creating a 3D scene**
- **Adding layers and graphics to a scene**
- **Defining 3D properties for layers**
- **Using the 3D Effects toolbar**
- **Navigating through a scene using the Fly tool**
- **Setting bookmarks**
- **Defining properties of a scene**
- **Identifying and selecting features**
- **Managing viewers**
- **View settings**
- **Exporting a scene**
- **Printing a scene**

Viewing data in three dimensions gives you new perspectives. 3D viewing can provide insights that would not be readily apparent from a planimetric map of the same data. For example, instead of inferring the presence of a valley from the configuration of contour lines, you can actually see the valley and perceive the difference in height between the valley floor and a ridge.

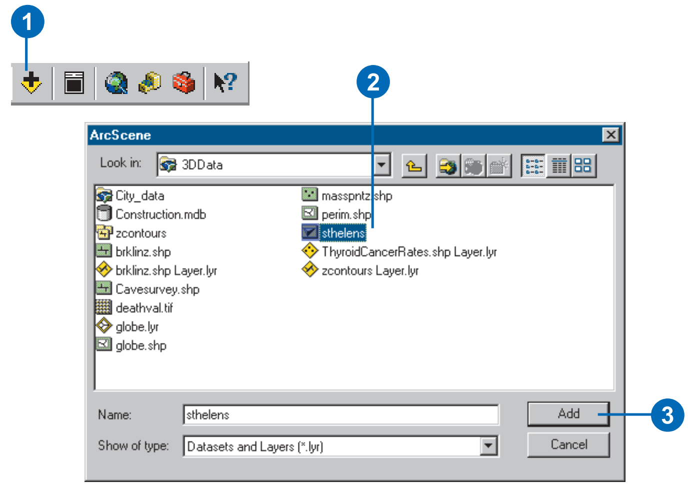
ArcScene lets you build multilayered scenes and control how each layer in a scene is symbolized, positioned in 3D space, and rendered. You can also control global properties for the scene, such as the illumination. You can select features in a scene by using their attributes or their position relative to other features or by clicking individual features in the scene. You can navigate around a scene or specify the coordinates of the observer and target for a viewer.

Creating a new scene

If you start ArcScene with an empty scene, you will probably want to add some data to it. You can add data surfaces, 2D or 3D feature classes, or layers that specify how the surface or feature data will be rendered in 3D. Once you add data to a scene, you can change how the layers of data in the scene are rendered by modifying their layer properties. You can also change general properties of the scene including the background color, the illumination of the scene, and the vertical exaggeration of the scene.

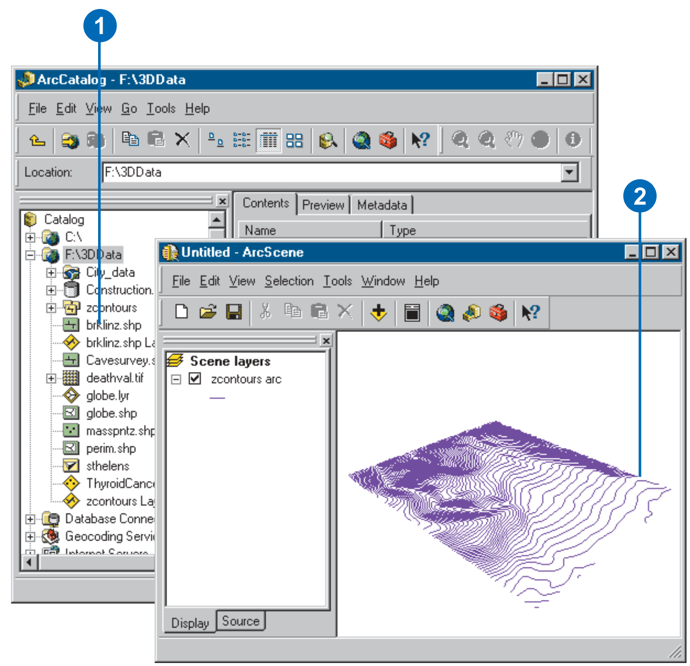
Adding data in ArcScene

1. Click the Add Data button on the ArcScene Standard toolbar.
2. Navigate to the surface or feature data.
3. Click Add.



Adding data from ArcCatalog

1. Navigate to the data in the Catalog tree.
2. Click and drag it onto the scene.



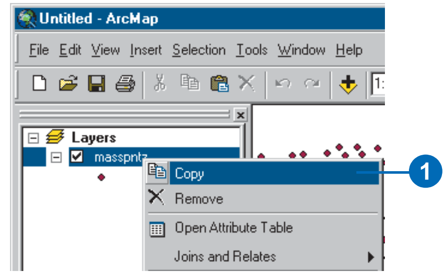
Tip

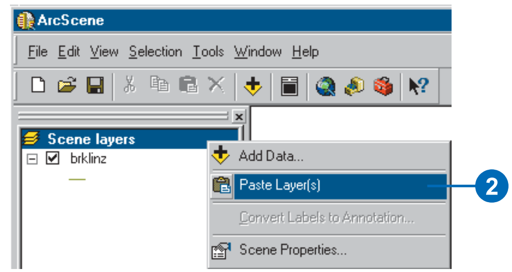
Why is the symbology of my layer different in ArcScene?

A layer file specifies how data should be rendered in ArcMap, ArcCatalog, or ArcScene. The 3D properties that you can set for a layer—those found on the Base Heights, Extrusion, and Rendering tabs—do not apply in ArcMap. Similarly, some symbology that you can set in ArcMap—multilayer symbols, dashed lines—will not show up in ArcScene. Features in ArcScene can be symbolized with simple symbols and using graduated symbols or colors.

Adding data from ArcMap

1. Right-click the layer in the ArcMap table of contents and click Copy.
2. Click the ArcScene window to make it active, right-click Scene layers, then click Paste Layer(s).



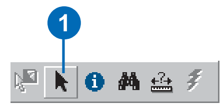


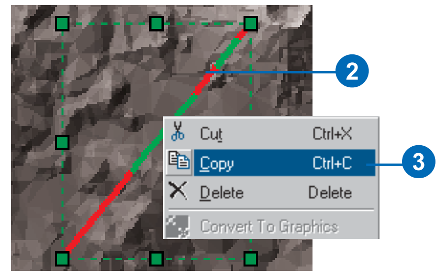
Adding 3D graphics to a scene

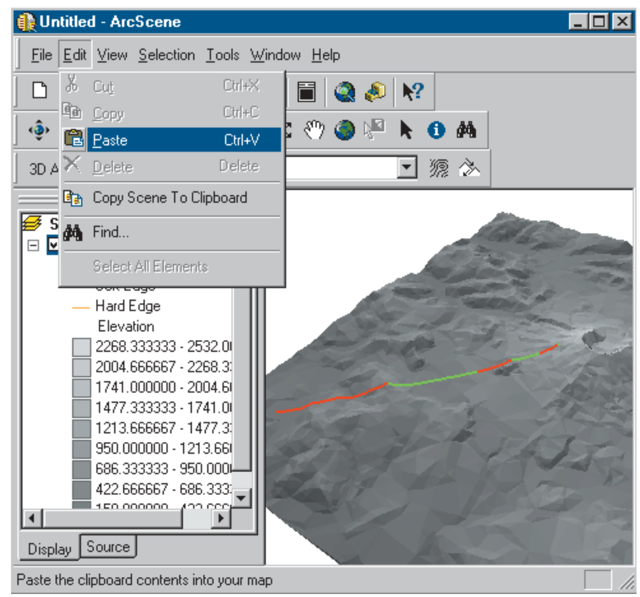
Once you've added data to a scene, you may find that you also want to add a 3D graphic. For example, you might have created a single contour line; a line of sight; a steepest path; or simple 3D point, line, or polygon graphics with one of the 3D Analyst tools in ArcMap. You can copy and paste these 3D graphics from ArcMap into ArcScene.

Copying and pasting 3D graphics from ArcMap

1. Click the Select Graphics button on the ArcMap Tools toolbar.
2. Click the graphic.
3. Right-click the graphic and click Copy.
4. In ArcScene, click Edit and click Paste.







Feature data and 3D

You may want to visualize feature data, as well as surfaces, in a perspective view.

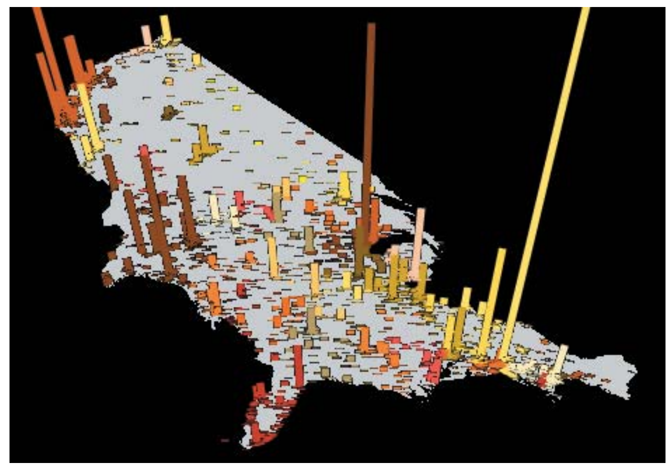
Feature data differs from surface data in representing discrete objects, rather than continuous phenomena. Features typically have a shape (geometry) and attributes.

Some typical feature geometries are point, line, or polygon. Point features might represent mountain peaks, telephone poles, or well locations. Lines might represent roads, streams, or ridgelines. Polygons might represent buildings, lakes, or administrative areas. The attributes of features can store values that refer to the elevation or height of the features. Some GIS features store elevation values with the feature geometry itself; for example, PointZ features are stored as a set of x,y,z coordinates. You can use z-values in the geometry or attributes of features to display the features in a 3D scene.

Building footprints extruded by building height

Sometimes features lack elevation or height values. You can still view these features in a 3D scene by draping or extruding them. If you have a surface model for the area, you can use the values in the model as z-values for the features. This is called *draping* the features. You can also use this technique to visualize image data in 3D. If you want to show building features in 3D, you can *extrude* them using an attribute such as building height or number of stories. You can also extrude features based on an arbitrary value.

Sometimes you'll want to view 2D features in a scene with z-values taken from some attribute other than a height value. For example, you might create a scene that shows city points extruded into columns based on their population.



U.S. cities, height extruded based on population in 1990

Defining the z-values for a layer

When you add a layer to a scene, it may not initially be rendered in 3D. TINs and features with 3D geometries—for example, pointZ, polylineZ, polygonZ, and multipatch shapes—are automatically drawn in 3D. Rasters (grids and images) and 2D features are drawn as though they were resting on a flat surface. In order to view rasters and 2D features in 3D, you need to define their z-values.

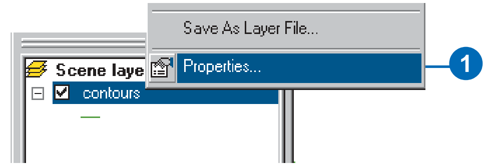
3D Analyst adds three tabs to the Properties dialog box that allow you to control how a feature layer is displayed in 3D. There are three ways to render 2D features in 3D. These are setting base heights using an attribute, setting base heights by draping features on a surface, and extruding features. There are some variations on these methods, and you can combine them—for example, you can set base heights from a surface, then extrude the features above or below the surface.

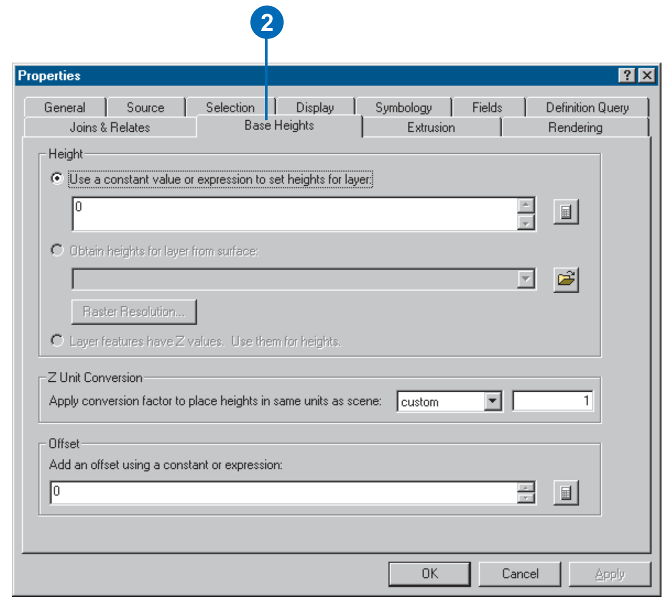
You can set the base heights of raster layers using a surface or a constant value.

Examining the base heights of a layer

1. Right-click the layer and click Properties in ArcScene or ArcCatalog.
2. Click the Base Heights tab.

   This layer does not have base heights.





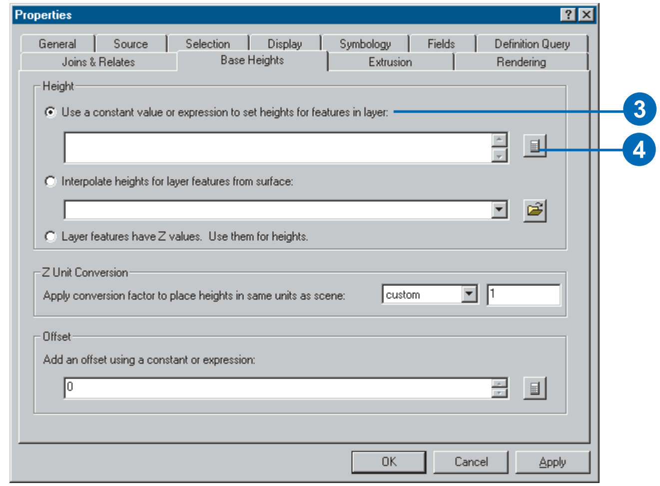
Tip

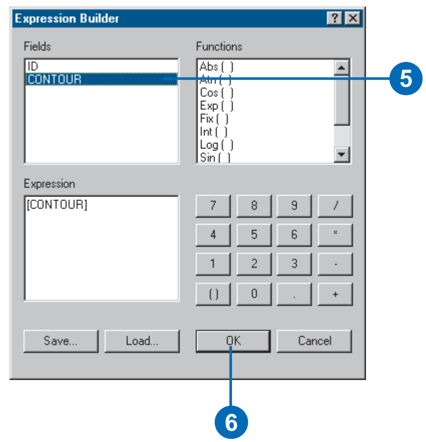
Using built-in z-values of a layer

Some feature data includes z-values in the feature geometry. If you examine the attribute tables of these feature layers, you will see that the shape field says PointZ, PolylineZ, or PolygonZ. Features with 3D geometry are automatically displayed in 3D in a scene using the z-values from the feature geometry. You can set the base height of these features using an attribute or a surface if you do not wish to use the built-in z-values. If you then want to switch back to using the z-values, click the Layer features have Z values. Use them for heights. option.

Setting the base height from an attribute

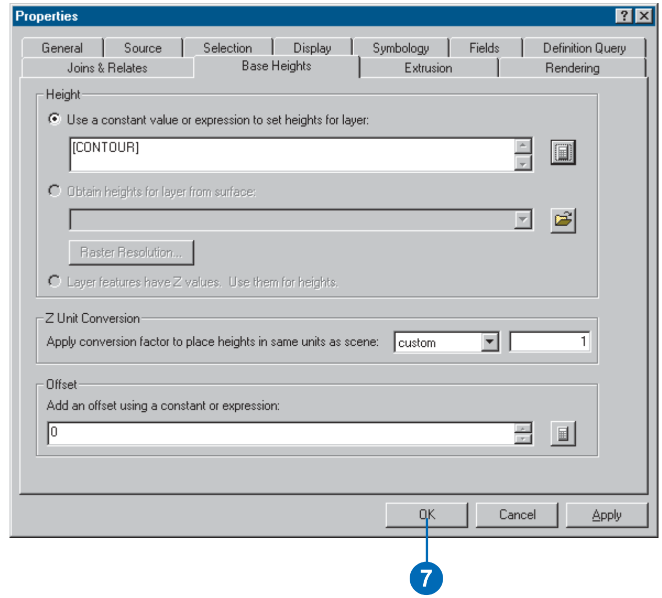
1. Right-click the layer and click Properties.
2. Click the Base Heights tab.
3. Click the option to Use a constant value or expression to set heights for features in layer.
4. Click the Calculate button.
5. Double-click the field that will provide the z-value for the features.
6. Click OK. ▸

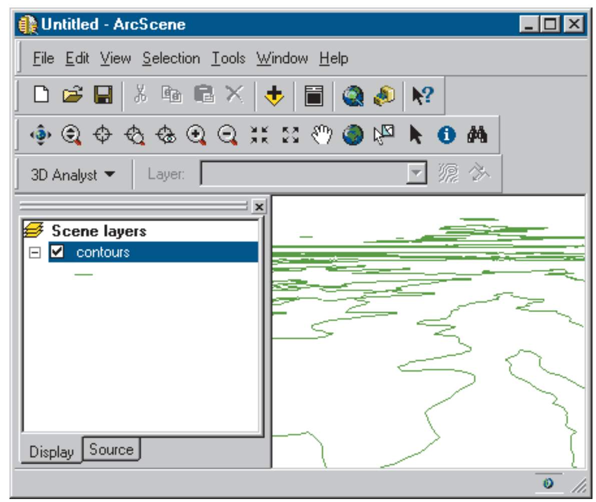




7. Click OK in the Properties dialog box.

The 2D features are now drawn in 3D, using the attribute that you selected as the z-value.



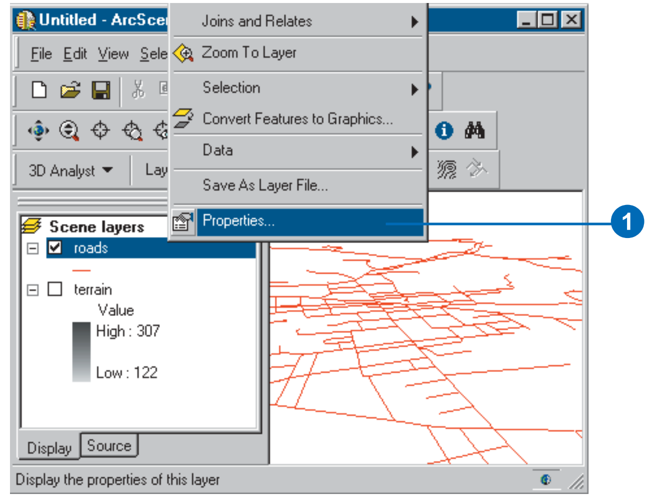


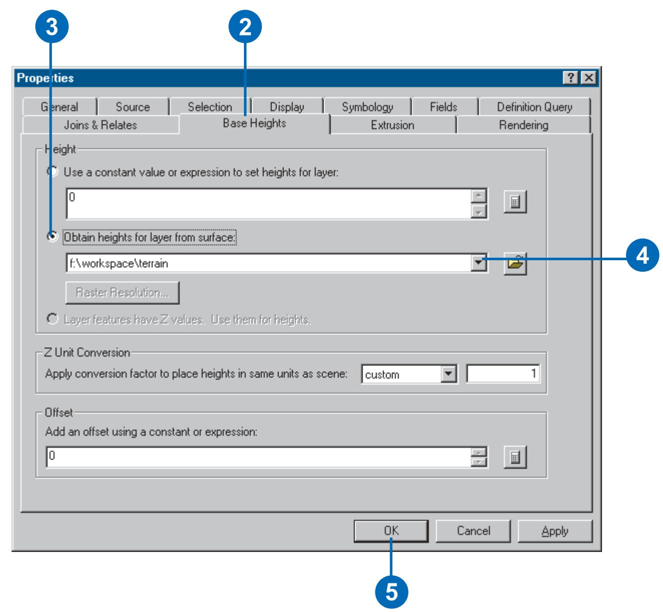
Perspective view of contours, base heights set using the contours' elevation attribute

Setting a feature layer's base heights using a surface

1. Right-click the layer and click Properties.
2. Click the Base Heights tab.
3. Click the option to Obtain heights for layer from surface.
4. Click the dropdown list and click the surface that you want to use for the base heights.
5. Click OK.

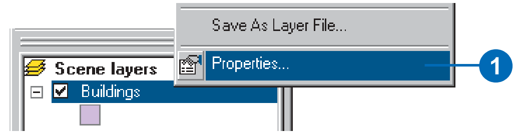
   The layer is now drawn in 3D, using the surface that you selected to provide the z-values.

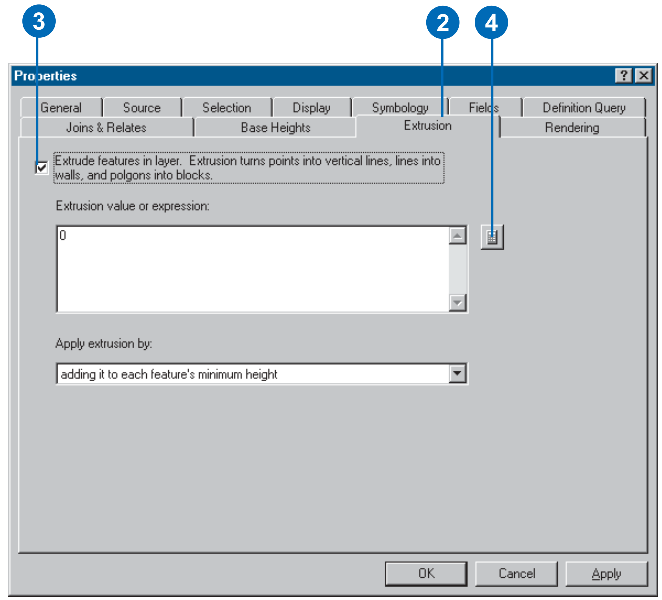




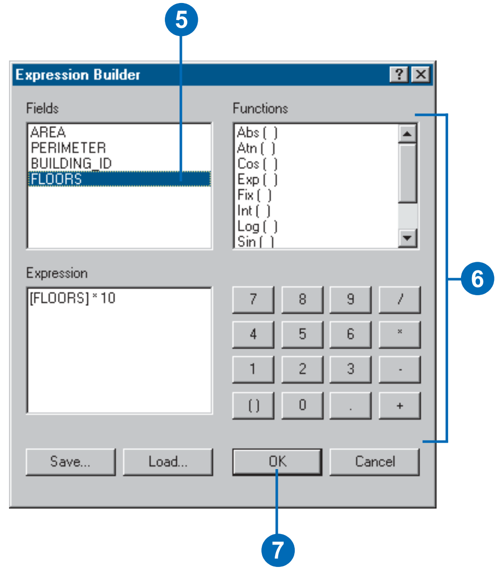
Extruding the features in a layer

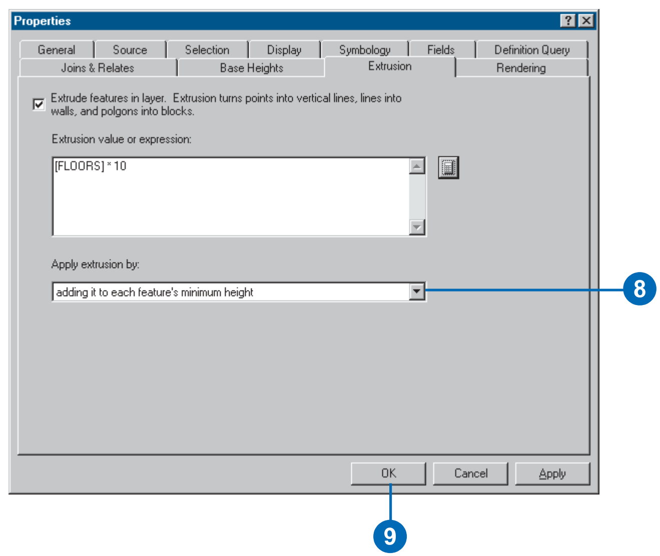
1. Right-click the layer and click Properties.
2. Click the Extrusion tab.
3. Check the check box to extrude the features in the layer.
4. Click the Calculate button to calculate an extrusion expression. ►



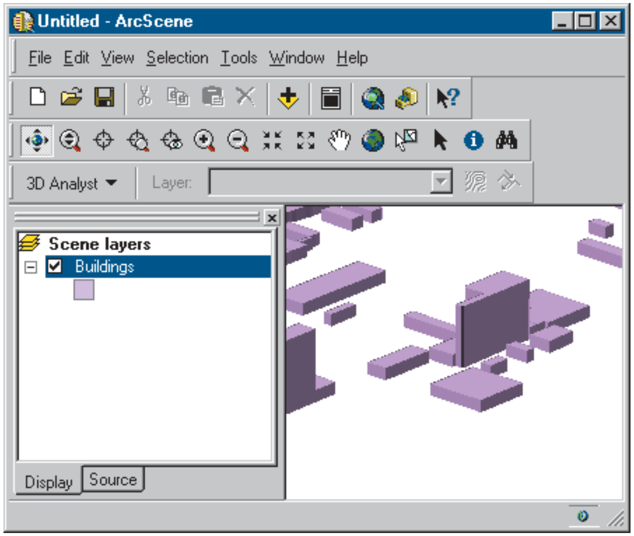


5. Double-click an attribute you want to include in the expression.
6. Optionally, build a calculation that includes the attribute.
7. Click OK.
8. Optionally, click the dropdown arrow to specify how the extrusion should be applied.
9. Click OK. ▸



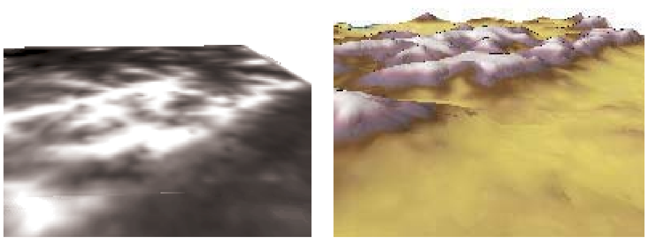


The features are extruded into 3D.



Raster data and 3D

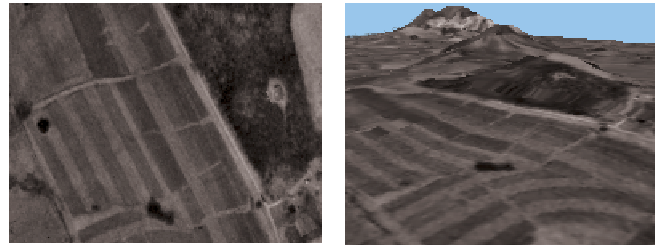
Rasters sometimes contain continuous surface data—when they do, rendering them in 3D is a simple matter of setting the raster layer's base height to itself. You can improve the appearance of the surface by setting the symbology, shading, base resolution, and—if the surface's z-units are not equal to its x,y units—z-unit conversion factor.



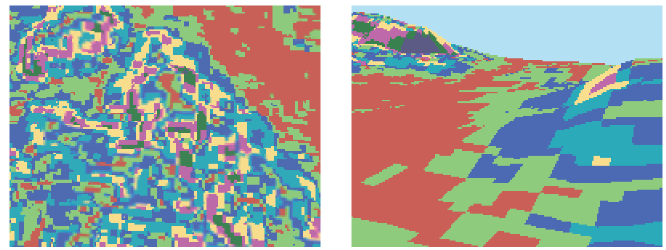
Terrain surface draped over a surface—itself

Rasters often contain other kinds of discontinuous (nonsurface) data about an area—for example, thematic land use data, remotely sensed image data, scanned maps, or the results of surface analyses—that can be mapped directly to a surface. You can display a nonsurface raster in 3D by draping it over a surface model of the area. This can be a very effective way of visualizing the relationship between the raster and the surface.

USGS quad map draped over a surface



Aerial photo draped over a surface



Categorical raster draped over a surface

Defining the 3D properties of a raster layer

When you add a raster layer to a scene, it is initially drawn as though it were resting on a flat surface. In order to view rasters in 3D, you need to define their z-values. 3D Analyst adds two tabs to the Properties dialog box that allow you to control how a raster layer is displayed in 3D. Rasters can be rendered in 3D by interpolating heights for the raster using values in the raster itself, by using values in another (raster or TIN) surface, or by giving the raster a constant base height.

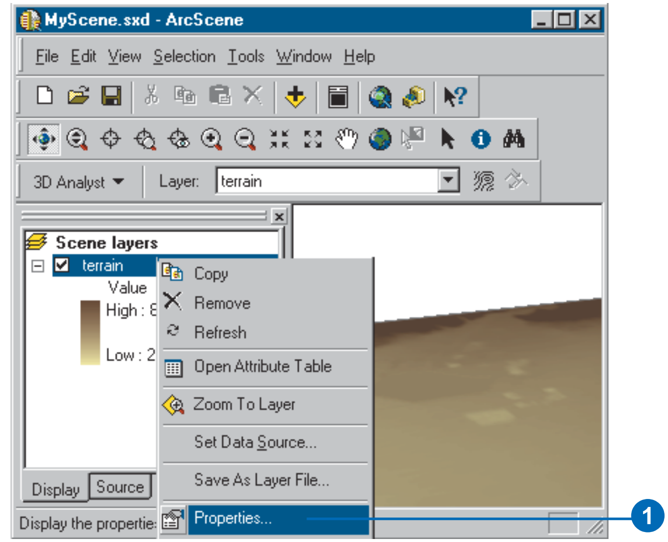
Tip

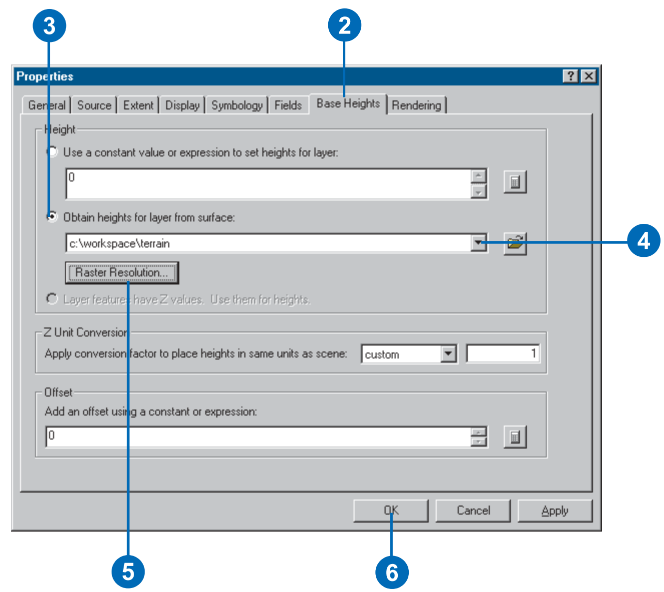
What is the Raster Resolution?

When you drape a raster over a raster surface, the base surface is resampled to 256 rows by 256 columns to improve performance. You can change the resolution of the base surface by clicking Raster Resolution and setting either the cell size or the number of rows and columns. A smaller number of rows and columns improves performance but reduces the resemblance of the base surface to the original.

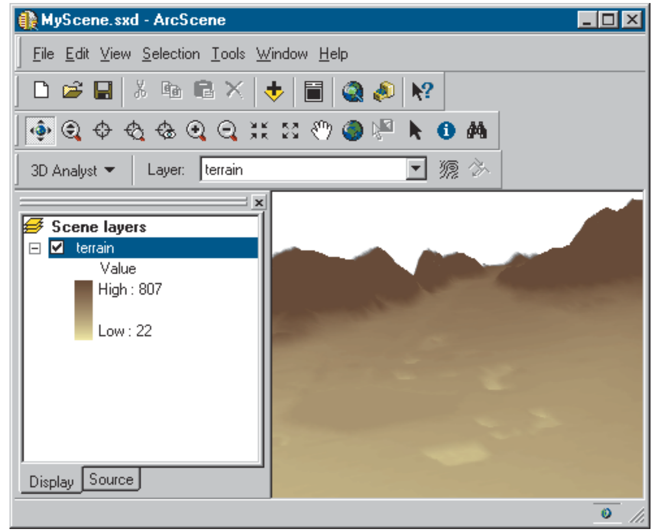
Setting a raster layer's base heights using a surface

1. Right-click the layer and click Properties.
2. Click the Base Heights tab.
3. Click the option to Obtain heights for layer from surface.
4. Click the dropdown arrow and click the surface that you want to use for the base heights.
5. Optionally, click Raster Resolution to set the resolution of the base surface.
6. Click OK. ▸





The raster layer is now drawn in 3D, using the surface that you selected to provide the z-values.



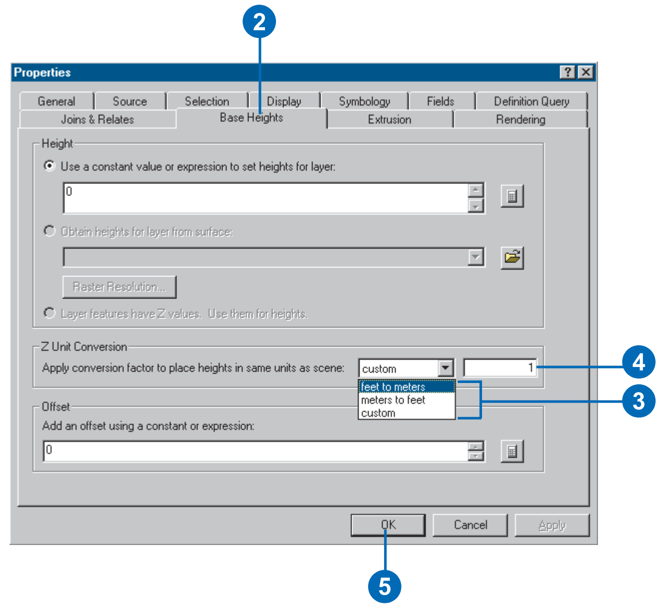
Converting z-units to x,y units

Geographic data is typically collected and represented in a coordinate system that has the same x and y units. However, when heights, depths, or elevations are recorded for features or surfaces, the z-units are not always the same as the x,y units of the coordinate system. For example, a set of well features might be stored in Universal Transverse Mercator (UTM) meters but have a well depth attribute in feet. In order to represent the wells correctly in 3D, the z-values must be converted to meters. Otherwise, when you extrude the wells in a scene, they will appear to be three times as deep as they really are.

Converting units

1. Right-click the layer and click Properties.
2. Click the Base Heights tab.
3. Click the Z Unit Conversion dropdown arrow and click one of the predefined types of conversion.

   If you need to apply a custom conversion, click custom.
4. Optionally, if you're using a custom conversion, type the conversion factor.
5. Click OK.

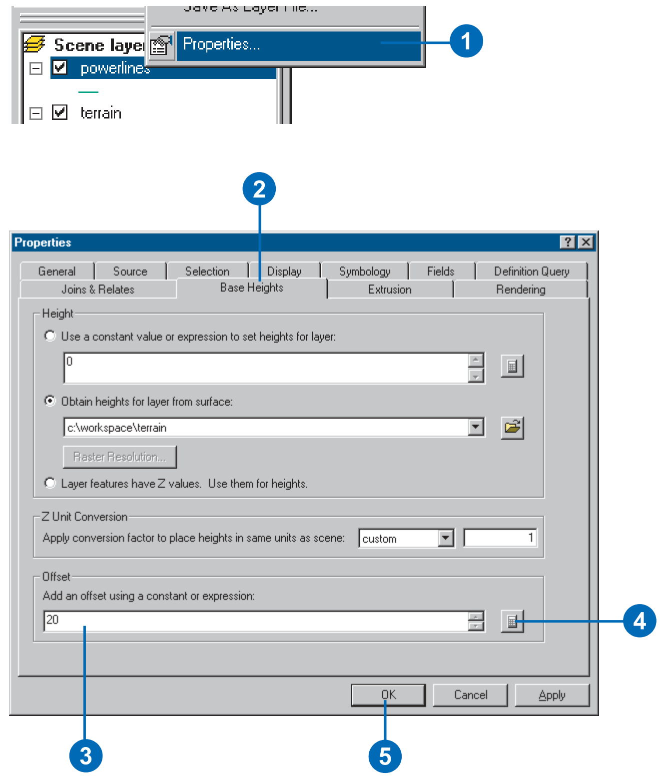


Offsetting the heights in a layer

You can add a constant amount or calculated value to the base height of a layer to raise—or lower—it relative to other layers in a scene. This can be useful when creating 3D visualizations to ensure that a layer is visible above a surface, allow comparison of the shapes of two surfaces, or provide z-values for power lines or pipes that have a given depth above or below a known surface.

The offset can be relative to base heights determined using a constant or expression, a surface, or z-values embedded in feature geometries.

1. Right-click the layer and click Properties.
2. Click the Base Heights tab.
3. Type a constant value for the offset.
4. Optionally, click the Calculate button to create an expression to define the offset.
5. Click OK.

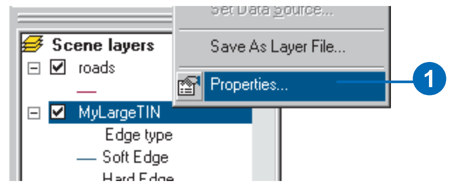


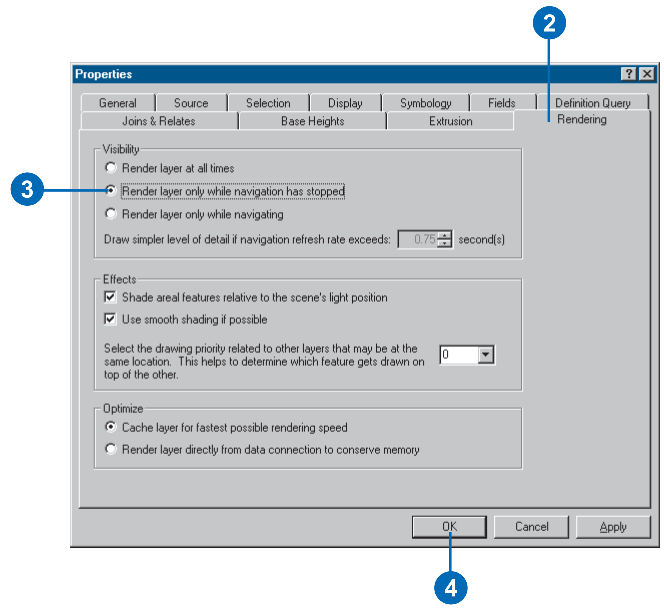
Controlling when a layer is rendered

Scenes can contain many layers, and rendering some layers can be quite demanding on your computer's resources. If you find that rendering a particular large or complex data layer causes navigation in a scene to be sluggish, you can set it to display only when navigation has ceased. Allowing a simpler layer to be rendered during navigation can provide landmarks to allow you to navigate accurately while the complex layer is not visible.

Rendering a layer only when navigation is stopped

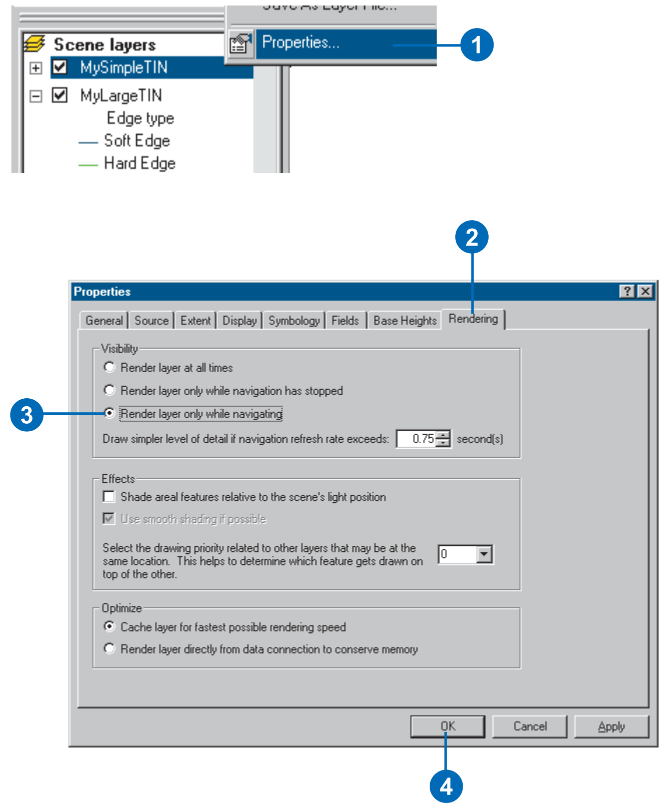
1. Right-click the layer and click Properties.
2. Click the Rendering tab.
3. Click the option to Render layer only while navigation has stopped.
4. Click OK.





Rendering a layer only during navigation

1. Right-click the layer and click Properties.
2. Click the Rendering tab.
3. Click the option to Render layer only while navigating.
4. Click OK.



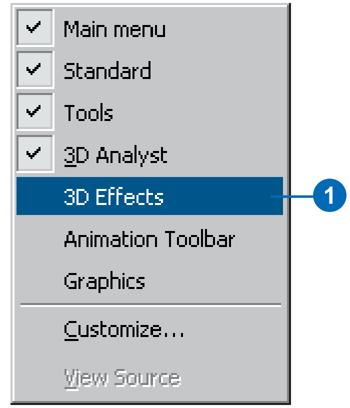
Using the 3D Effects toolbar

Using the 3D Effects toolbar, you can access a layer's display properties without opening its properties dialog box. Use the 3D Effects toolbar to adjust a layer's transparency, change its face culling, toggle its lighting, set its shading mode, or rank its depth priority.

Turning on the 3D Effects toolbar

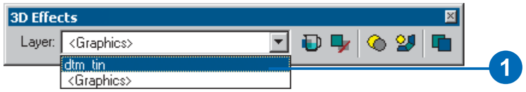
1. Right-click in the blank area near the menus and toolbars and click 3D Effects.

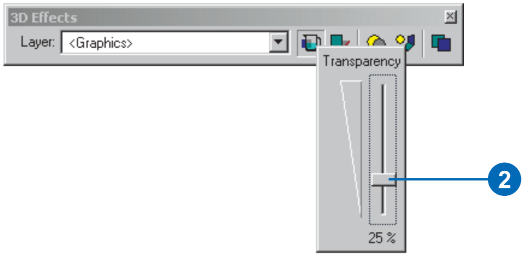
   The 3D Effects toolbar appears.



Using the 3D Effects toolbar to set a layer's transparency

1. Click the Layer dropdown arrow and choose the layer you want to change.
2. Click the Adjust Tranparency button and move the slider to a value.





Using face culling to control the way layers are drawn

Face culling is a property that is only accessible from the 3D Effects toolbar. Use culling to turn off the display of front or back faces of an areal feature or graphic. For example, if you have data that is surrounded by a multipatch shapefile used for a backdrop, use culling to turn off the display of one of the backdrop faces so that you can always see your data, even when zoomed outside of the backdrop extent.

Using the 3D Effects toolbar to adjust layer face culling

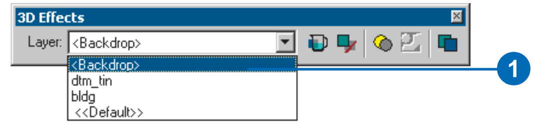
1. Click the Layer dropdown arrow and choose the layer you want to change.

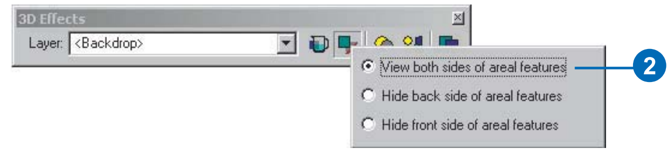
2. Click the Layer Face Culling button and click the option you want to use.

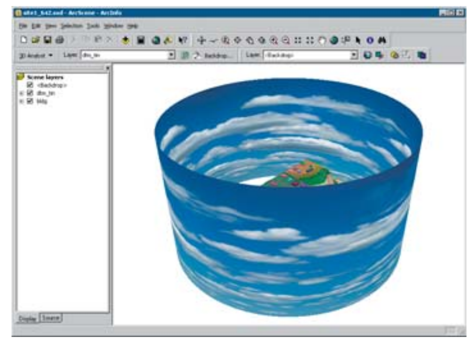
   The default setting is to view both sides.

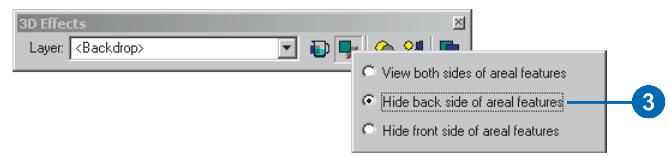
3. Optionally, click a different viewing option.

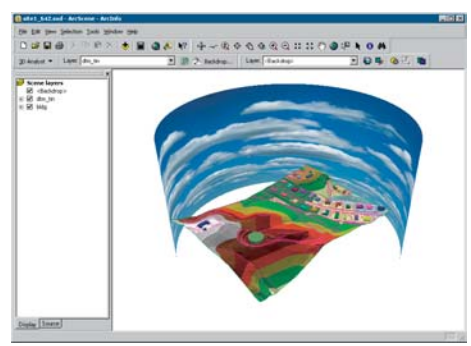
   View only one side of a layer to make the data inside it visible.









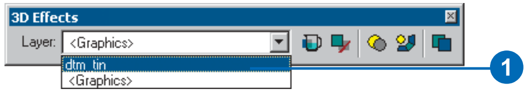


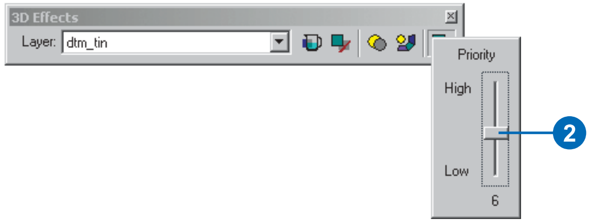
Changing a layer's drawing priority

If two layers occupy the same location in 3D space, a stitching effect may be seen when the layers are drawn in a scene. This is because they are competing with each other to be displayed. To reduce the effect, assign a priority to the layers so that you control the order in which they are drawn. For example, if your scene contains a feature layer draped on a surface layer, and you notice a conflict between the layers, you can lower the priority of the surface so that the features are always drawn first. Priority can only be changed for area-based features, such as polygons, rasters, and TINs.

Using the 3D Effects toolbar to change a layer's drawing priority

1. Click the Layer dropdown arrow and choose the layer you want to change.
2. Click on the Change Depth Priority button and move the slider to a value.

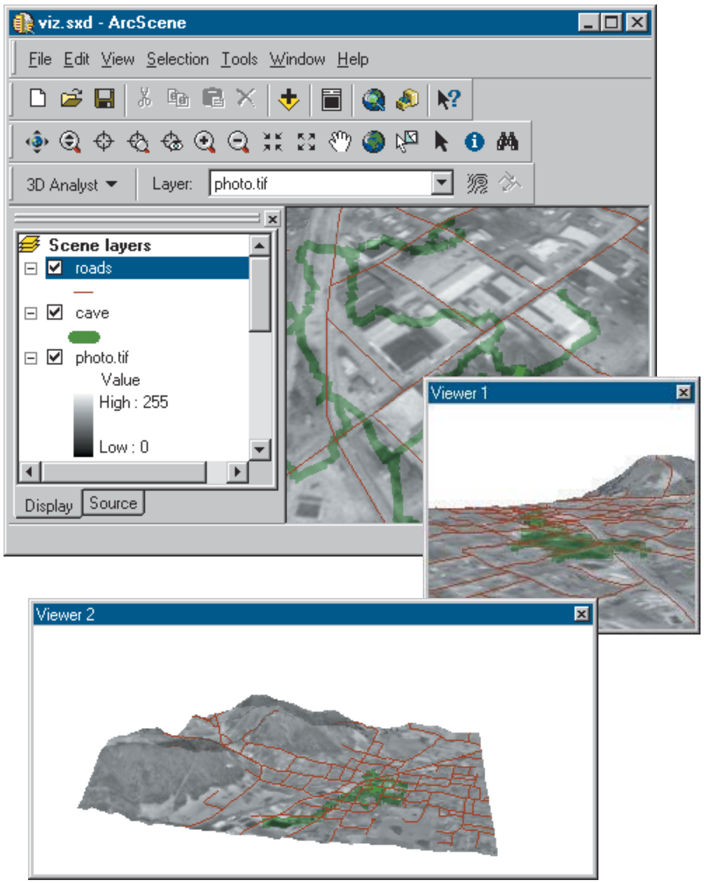




Viewing a scene from different angles

By default, ArcScene has a single window for viewing your scene, but you can create multiple viewer windows for a scene. Having additional viewers lets you focus on specific areas from the best angles while still seeing the whole.

You can navigate independently within each viewer window—the navigation tools control the view within the window they're used in. You can even make the scene rotate in one viewer and navigate in another.



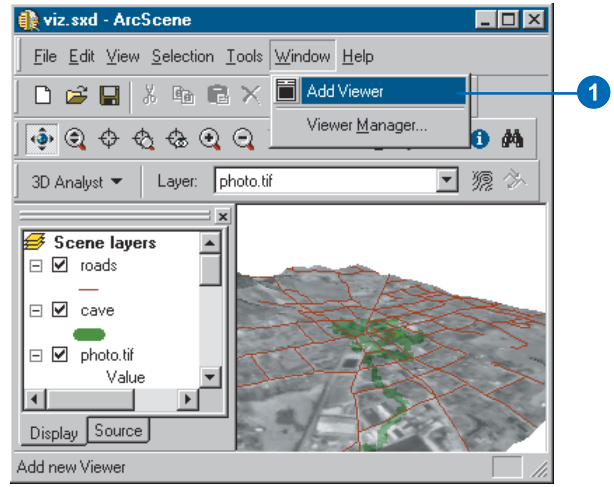
Managing scene viewers

Multiple viewers let you see a scene from different angles at the same time. Each viewer can be independently navigated. You can maximize viewers to fill the screen, minimize viewers to get them out of the way, restore viewers to their previous size, or close them altogether.

The properties of a scene apply to all of the viewers.

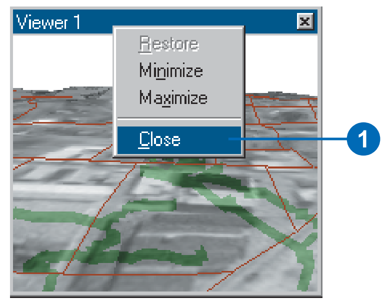
Adding a viewer

1. Click Window and click Add Viewer.



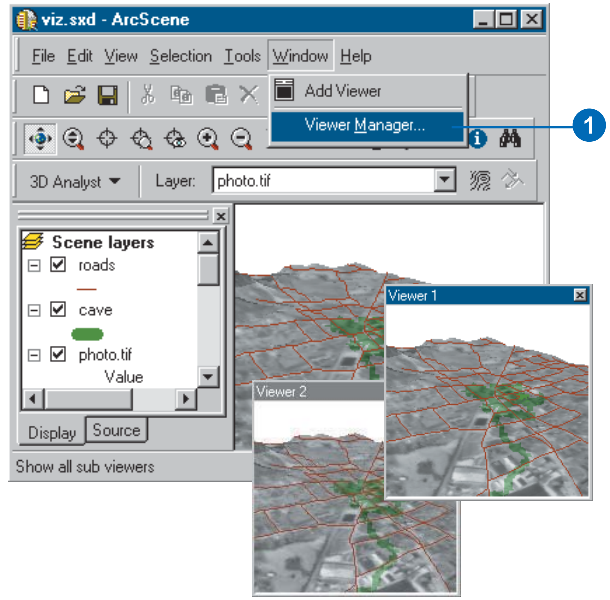
Closing a viewer

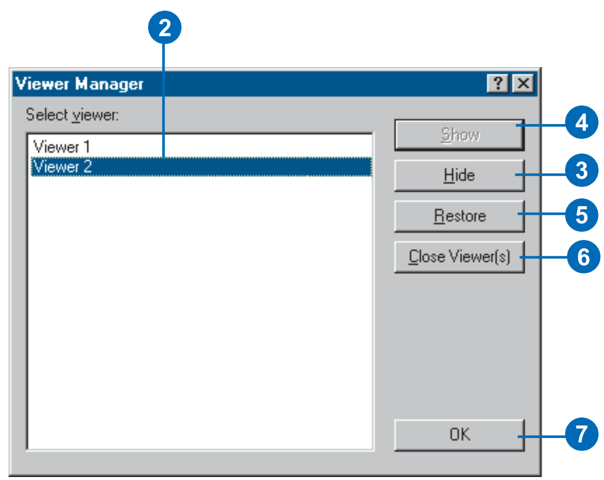
1. Right-click the title bar of the viewer and click Close.



Managing viewers

1. Click Window and click Viewer Manager.
2. Click a viewer in the Select viewer list.
3. Click Hide to hide the selected viewer.
4. Click Show to show the selected viewer.
5. If you have minimized a viewer, you can click Restore to restore it to its previous size.
6. Click Close Viewer(s) to permanently close the viewer.
7. Click OK to close the Viewer Manager.





Changing the viewer settings

You can also change the way you see a scene in each viewer in several ways.

You can switch a viewer from a perspective view to an orthographic view with no perspective distortion of scale. This lets you examine the data in the view as though it were on a 2D map.

You can change the roll angle and pitch of a viewer.

You can also change the position of a viewer and of its target point by typing x,y,z coordinates.

You can continue to navigate in a viewer as you view the viewer settings—the position, pitch, and roll values are updated in the View Settings dialog box when you stop navigating.

Tip

Extruded points limitation

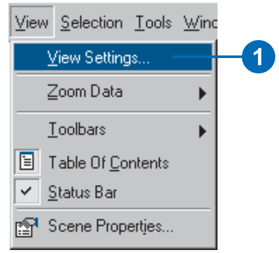
Extruded points are not visible in orthographic view.

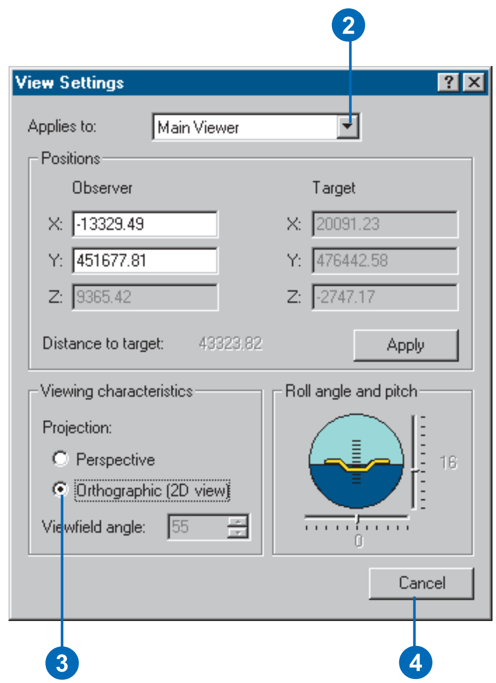
If you need to see extruded point data in an orthographic view, copy and paste the layer in the scene and turn off extrusion for the copy.

Setting a viewer to 2D

1. Click View and click View Settings.
2. If you have more than one viewer for the scene, click the dropdown arrow and select the viewer that you want to change.
3. Click Orthographic (2D view).

   You see an orthographic view of the scene with no perspective distortion of scale.
4. Click Cancel to close the View Settings dialog box.





Tip

Why can't I set the roll and pitch?

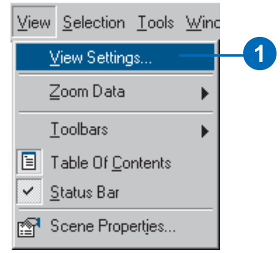
Roll and pitch are only applicable in perspective viewing. If you have the viewer set to Orthographic, the pitch and roll controls are disabled.

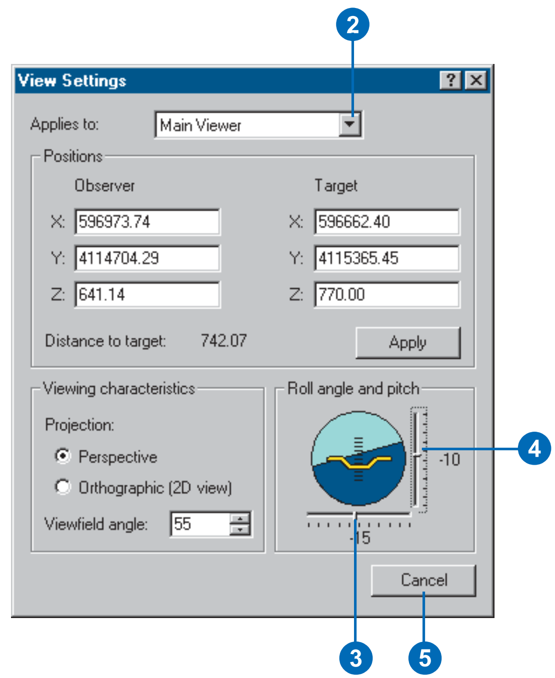
Changing the roll and pitch of a viewer

1. Click View and click View Settings.
2. If you have more than one viewer for the scene, click the dropdown arrow and select the viewer that you want to change.
3. Click the roll slider and drag it to change the roll.

   The artificial horizon and the viewer roll to the new roll angle.
4. Click the pitch slider and drag it to change the pitch.

   The artificial horizon and the viewer pitch to the new pitch angle.
5. Click Cancel to close the View Settings dialog box.





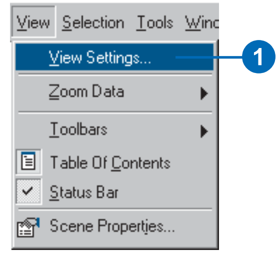
Tip

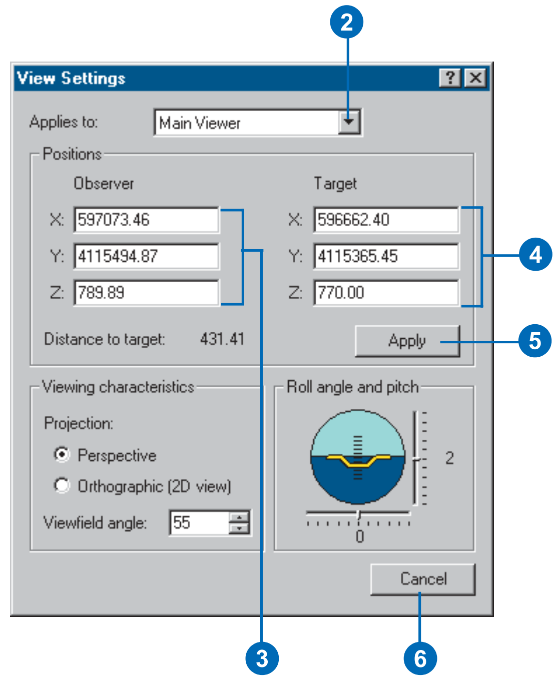
Finding a target's coordinates

With the View Settings dialog box open, click the Center on Target tool and click a location in the scene. The viewer centers on the target, and you can see its x,y,z coordinates.

Specifying the coordinates of the observer or target

1. Click View and click View Settings.
2. If you have more than one viewer for the scene, click the dropdown arrow and select the viewer that you want to change.
3. Type the x,y,z coordinates for the Observer.
4. Type the x,y,z coordinates for the Target.
5. Click Apply to set the viewer to the new observer and target coordinates.
6. Click Cancel to close the View Settings dialog box.





Navigating through a scene using the Fly tool

Use the Fly tool to investigate your scene by flying through it. You can fly in any direction and move forward or backward at different speeds.

Tip

Fine-tuning the fly speed

In between mouse clicks, press arrow up or down to increase or decrease speed, respectively.

Tip

Looking up or down while flying

Press Shift while flying to maintain a constant elevation. You can then point the mouse up or down to look in those directions without changing the direction of travel.

Using the Fly tool to navigate in a scene

1. Click the Fly button.

   The cursor changes to indicate fly mode is active.

2. Click once in the center of the scene.

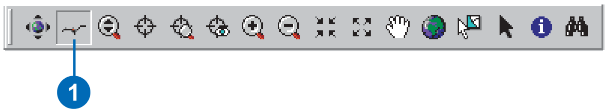
   The tool enters the suspended state. You can point the mouse to look in all directions, but there is no translational movement.

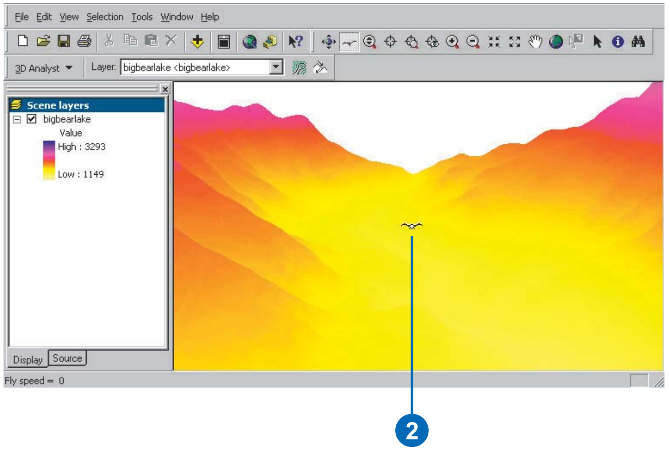
3. Click the mouse to move forward.

   Right-click to move in reverse. Successive clicks in either direction increase the speed. Speed is indicated in the status window.

4. Click the opposite button to slow down incrementally and then stop.

   Press Esc to immediately stop movement in either direction.





Setting bookmarks

A *3D bookmark* identifies a particular camera perspective that you want to save and refer to later. For example, you might want to create a 3D bookmark that identifies a certain view of a study area. That way, as you navigate through your scene, you can easily return to the study area view by accessing the bookmark. You can also use 3D bookmarks to highlight areas in your scene you want others to see.

The list of bookmarks applies to all viewers in a scene. If you create a bookmark in a viewer, it will be added to a central list and can be applied to any viewer.

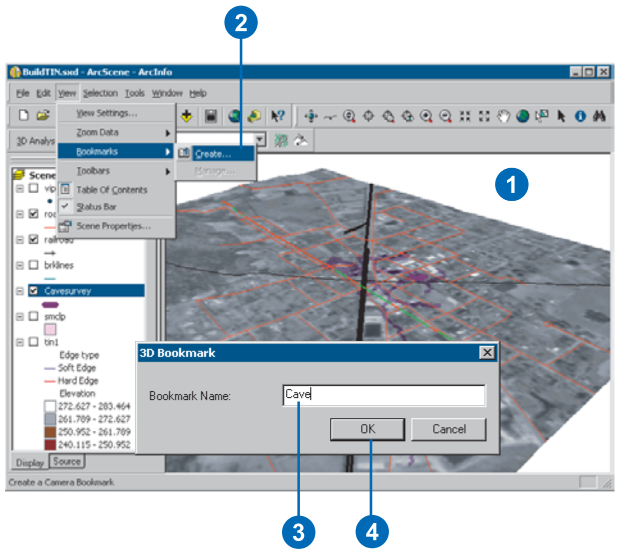
Tip

Using bookmarks in secondary viewers

To create or use a bookmark in a secondary viewer, right-click on its title bar and click Bookmarks.

Creating a perspective bookmark

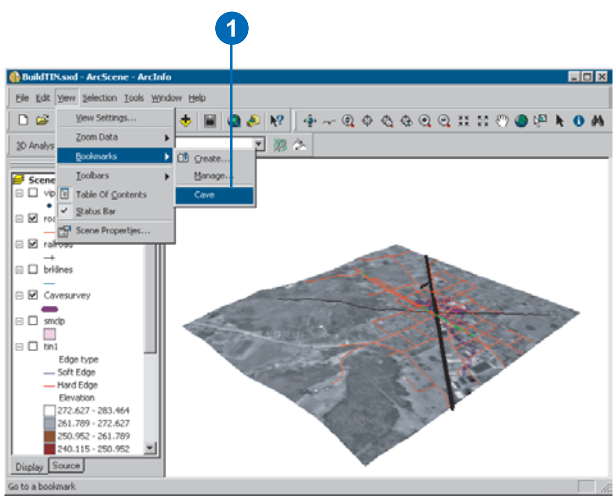
1. Navigate to the perspective for which you want to create a bookmark.
2. Click the View menu, point to Bookmarks, and click Create.
3. Type a name for the bookmark.
4. Click OK.



Using a perspective bookmark

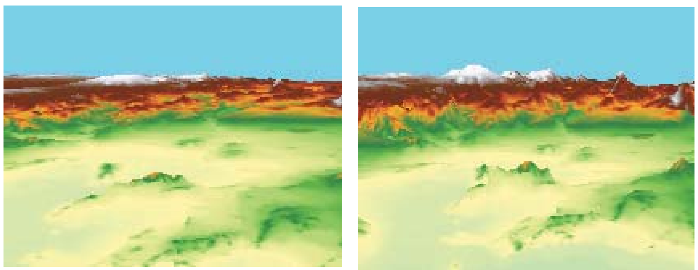
1. Click the View menu, point to Bookmarks, and click the name of the bookmark you want to use.

   The bookmarked view appears.

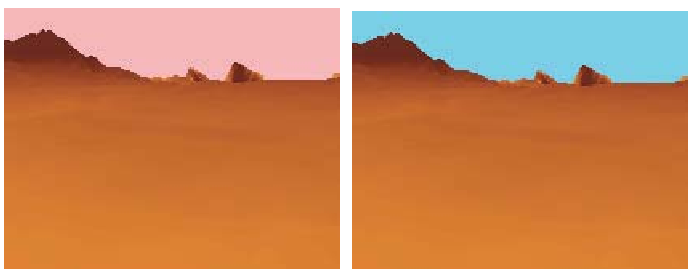


Setting the properties of a scene

In ArcScene, you can set certain properties—for example, vertical exaggeration, animated rotation, background color, extent, and illumination properties—that apply to the scene and all the layers within it. You can also add comments about the scene and set its coordinate system. If your scene has multiple viewers, these properties also apply to all of the viewers.

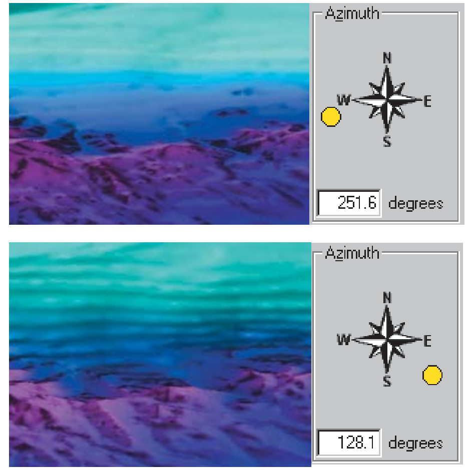


Vertical exaggeration emphasizes variation in a surface.

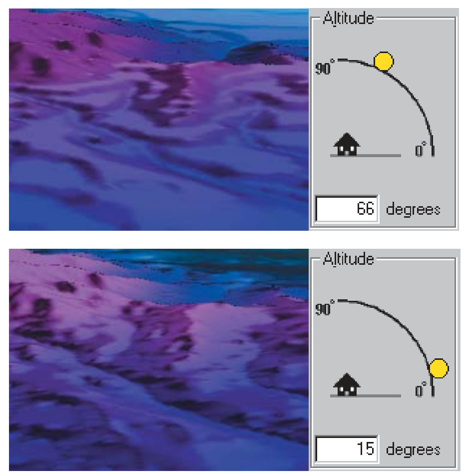


The background color can make visualizations more realistic.

The tabs of the Scene Properties dialog box let you set the various global scene properties.



Illuminating a scene from different angles emphasizes different parts of a surface.

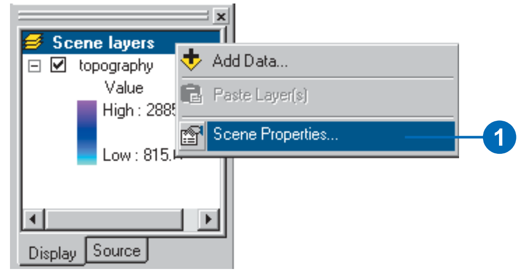


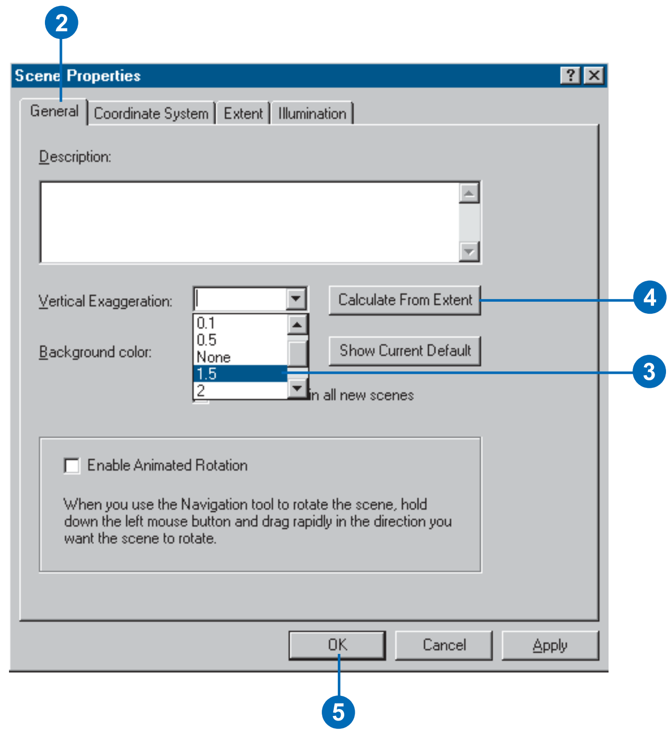
Changing the vertical exaggeration

Vertical exaggeration can be used to emphasize subtle changes in a surface. This can be useful in creating visualizations of terrain where the horizontal extent of the surface is significantly greater than the amount of vertical change in the surface. A fractional vertical exaggeration can be used to flatten surfaces or features that have extreme vertical variation.

The vertical exaggeration is applied to all layers in a scene. You can exaggerate a single layer by changing its z-unit conversion factor.

1. Right-click Scene layers and click Scene Properties.
2. Click the General tab.
3. Click the Vertical Exaggeration dropdown arrow and click a vertical exaggeration factor.
4. Optionally, click Calculate From Extent to automatically calculate an exaggeration factor based on the extent and z-variation in the scene.
5. Click OK.





Using animated rotation

Rotating a scene is a good way to get an overview of its contents. You can make a scene spin around the current center when animated rotation is enabled.

You can adjust the rotation speed and the observation angle and zoom in and out while the scene rotates.

Tip

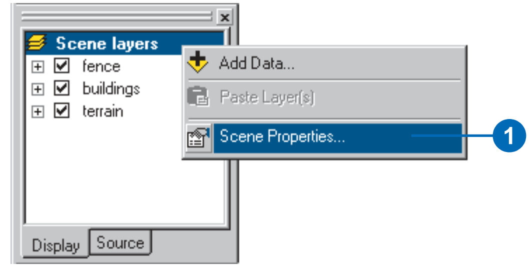
Toggling animated rotation using shortcut keys

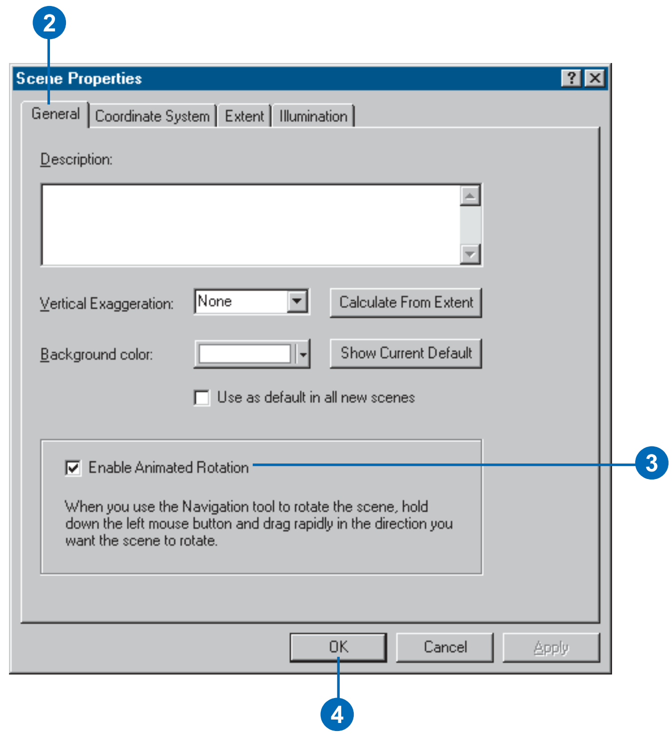
While in navigation mode, you can toggle on and off animated rotation in a specific viewer by pressing Ctrl and Shift while clicking in that viewer.

Enabling animated rotation

1. Right-click Scene layers and click Scene Properties.
2. Click the General tab.
3. Check the check box to enable animated rotation.
4. Click OK.

   The navigate cursor has a circle around it when animated rotation is enabled.





Starting animated rotation with complex data

To start complex data rotating, press Shift while dragging rapidly in the direction you want the scene to rotate.

Starting animated rotation

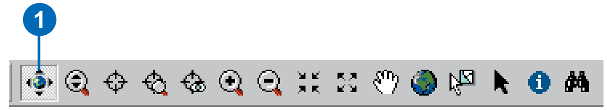
1. Click the Navigate button after enabling animated rotation.

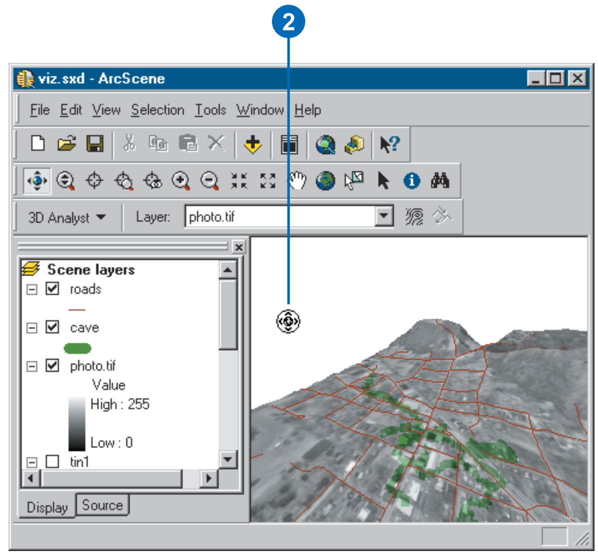
   The navigate cursor has a circle around it when animated rotation is enabled.

2. Click the scene, hold the mouse button down, drag to the left or right, and release the mouse button while the scene is in motion.

   The speed at which the scene rotates is proportional to the speed at which the mouse is moving when you release the button.

   Click the scene to stop the rotation.





Changing the rotation speed

1. While the scene is rotating, press the Page Up key to increase the rotation speed and press the Page Down key to decrease the rotation speed.

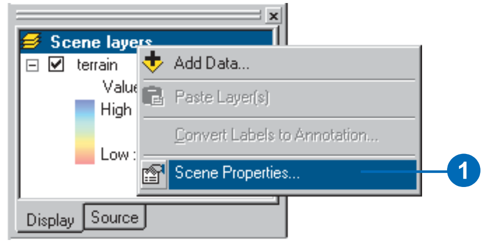
Changing the background color

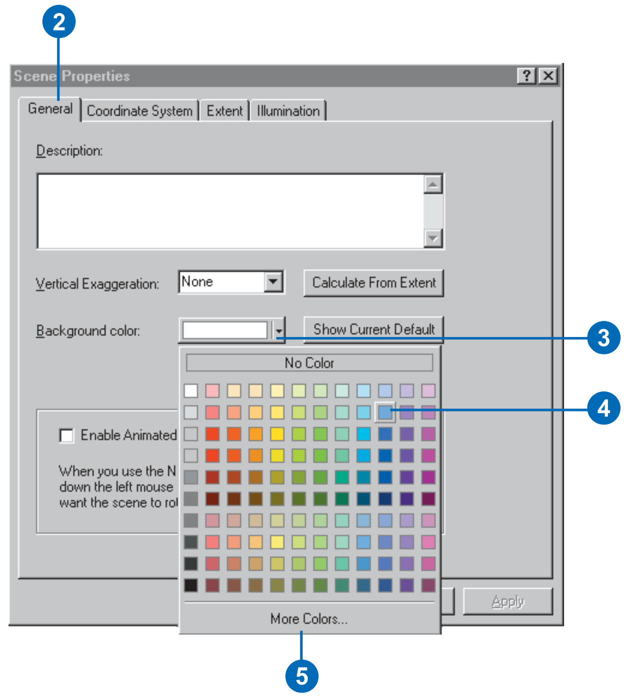
The background color for scenes is white by default. You can change the background color to suit your visualization needs. Various shades of blue can make the background appear to be a blue sky, while a black background can simulate night.

You can quickly set the backgound to one of the preset colors or mix your own color. You can also make the current background color the default for all new scenes.

Setting the background color

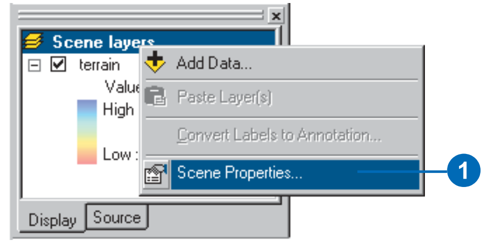
1. Right-click Scene layers and click Scene Properties.
2. Click the General tab.
3. Click the Background color dropdown arrow.
4. Click a color.
5. Optionally, click More Colors to mix your own background color.
6. Click OK.

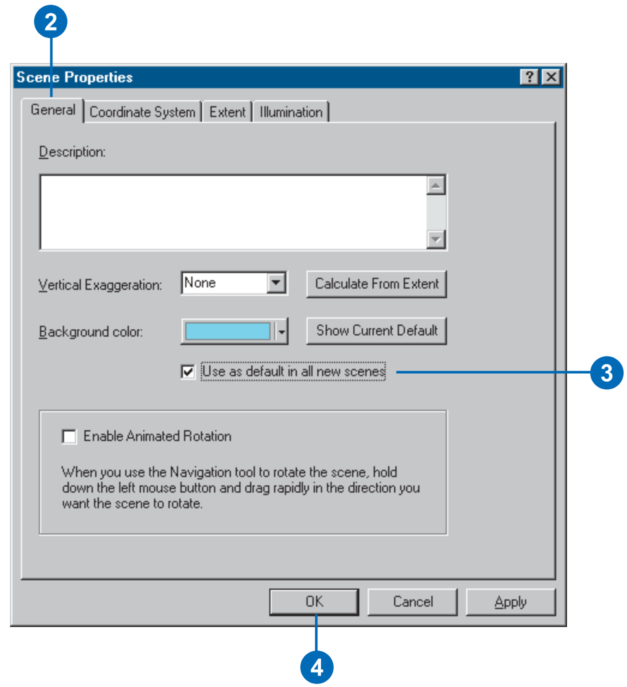




Setting the default background color for future scenes

1. Right-click Scene layers and click Scene Properties.
2. Click the General tab.
3. Check the check box to use the current color as the background in all new scenes.
4. Click OK.





Changing the scene illumination

You can set the azimuth and altitude of the light source, as well as the amount of contrast, used in rendering the illumination of the scene. The illumination properties for a scene apply to all of the areal features, including extruded polygon and line features, in a scene.

You can control whether individual layers are shaded by turning shading on or off on the Rendering tab of the layer's Properties dialog box.

Tip

What is the azimuth?

The azimuth is the compass direction from which the light source shines on the scene.

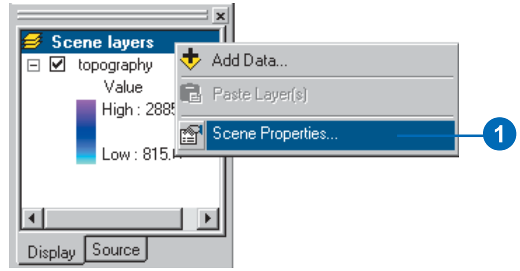
Tip

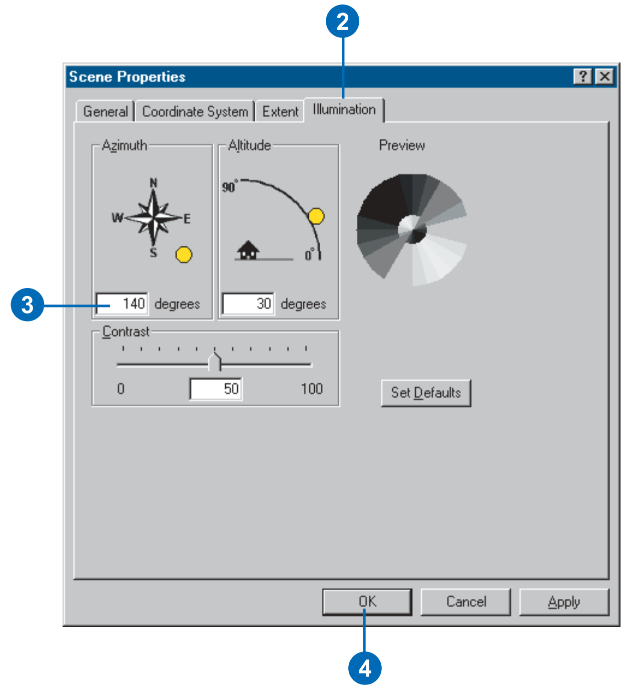
Changing illumination properties quickly

You can click on the sun graphic in the Azimuth and Altitude controls and drag it where you want it, instead of typing values into the text boxes. The illumination preview and the values in the text boxes will change to reflect the new position.

Setting the illumination azimuth

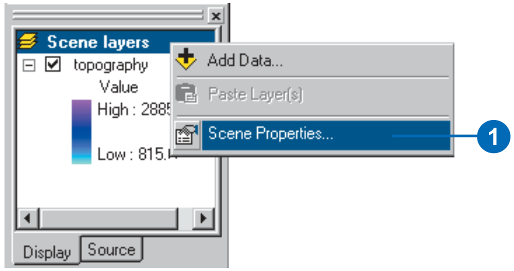
1. Right-click Scene layers and click Scene Properties.
2. Click the Illumination tab.
3. Type an Azimuth for the scene light source.
4. Click OK.

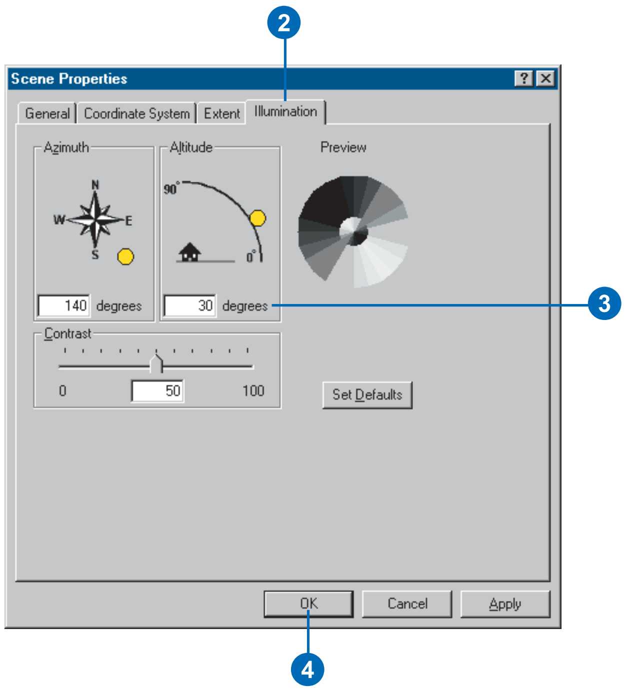




Setting the illumination altitude

1. Right-click Scene layers and click Scene Properties.
2. Click the Illumination tab.
3. Type an altitude for the scene light source.
4. Click OK.





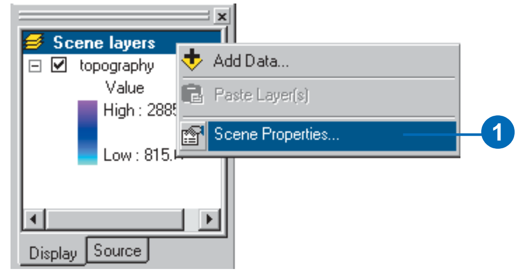
Tip

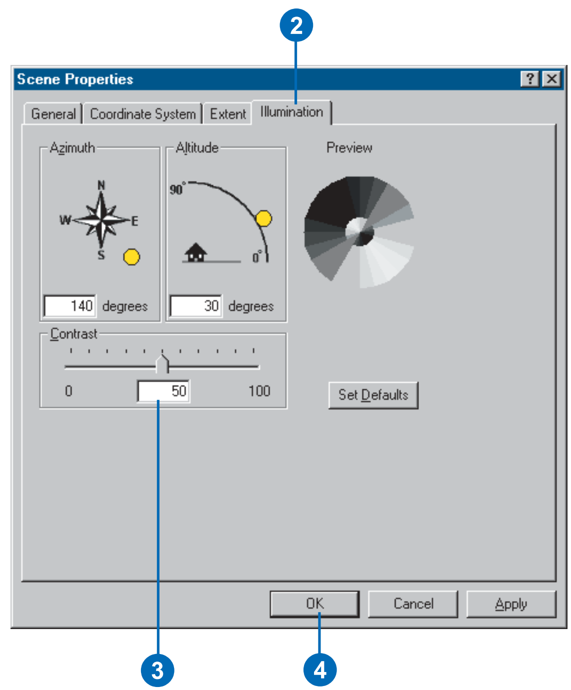
What is the contrast?

The contrast controls the amount of shading applied to a surface.

Setting the illumination contrast

1. Right-click Scene layers and click Scene Properties.
2. Click the Illumination tab.
3. Type a contrast value.
4. Click OK.





Changing the scene extent

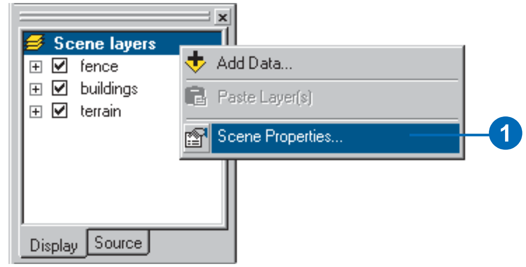
Reducing the extent of a scene can be a useful way to remove extraneous information and increase rendering performance.

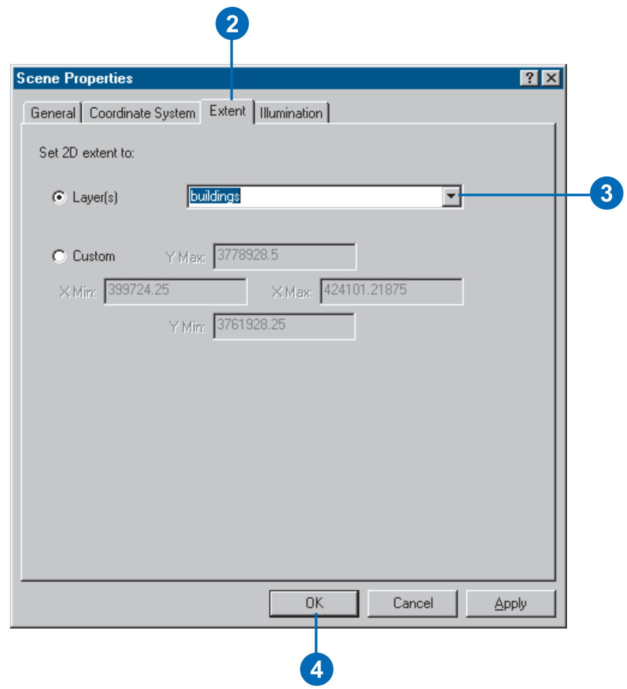
By default, the extent of a scene is the combined extent of all of the layers in the scene. You can change the extent of a scene to be the same as the extent of one of the layers or set it using specific minimum and maximum x- and y-coordinates.

Data that falls outside of the extent of the scene is not displayed.

Setting the extent to a layer

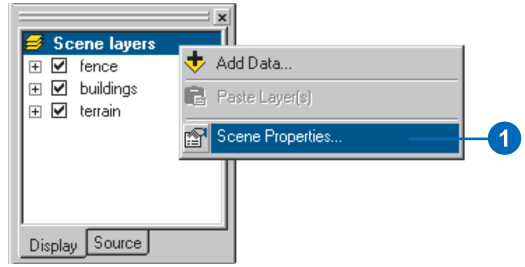
1. Right-click Scene layers and click Scene Properties.
2. Click the Extent tab.
3. Click Layer(s), click the Layer(s) dropdown arrow, then click the layer you want to define the scene extent.
4. Click OK.

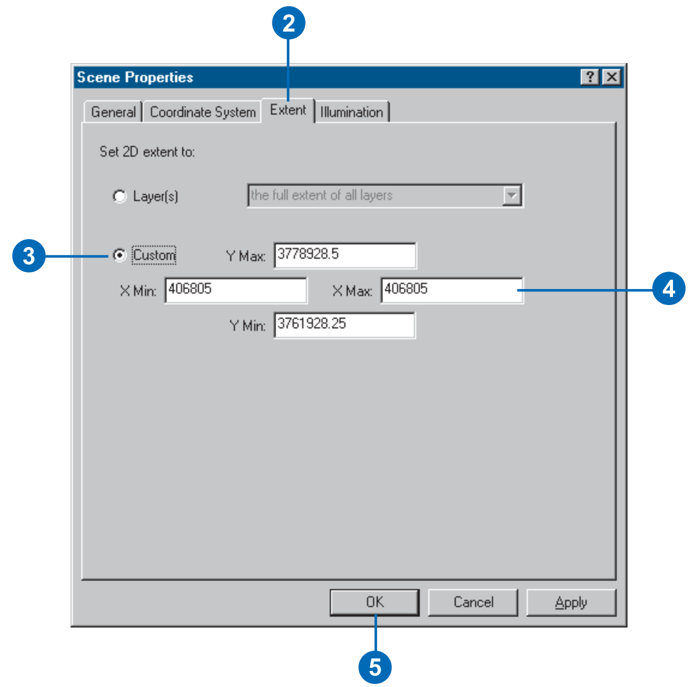




Setting the extent using coordinates

1. Right-click Scene layers and click Scene Properties.
2. Click the Extent tab.
3. Click Custom.
4. Type minimum and maximum x and y values to define the extent of the scene.
5. Click OK.





Changing the scene coordinate system

If all the data you want to display in a scene is stored in the same coordinate system—for example, you're using your organization's database—you can just add it to a scene and not consider whether the layers will overlay properly; they will. If, however, you've collected data from a variety of sources, you'll need to know what coordinate system each dataset uses to ensure ArcScene can display them together.

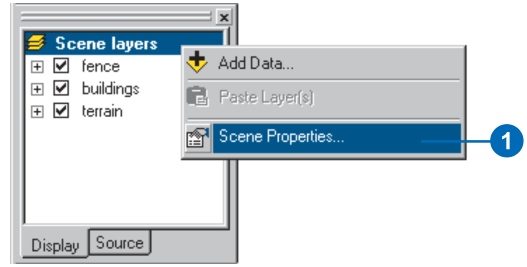
When you add a layer to an empty scene, that layer sets the coordinate system for the scene; you can change it later if necessary. As you add subsequent layers, they are automatically transformed to the scene's coordinate system as long as there's enough information associated with the layer's data source to determine its current coordinate system. If there isn't enough information, ArcScene will be unable to align the data and display it correctly. In this case, you'll have to supply the necessary coordinate system information yourself.

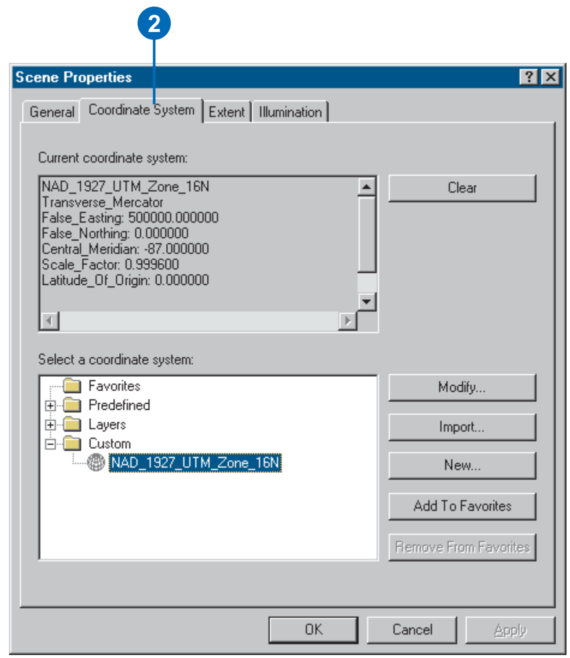
ArcScene expects coordinate system information to be stored with the data source. ►

Finding out what coordinate system your data is currently displayed in

1. Right-click Scene layers and click Scene Properties.
2. Click the Coordinate System tab.

   The details of the current coordinate system are displayed in the dialog box.





For a layer in a geodatabase, this information is part of the layer's metadata. For coverages, shapefiles, TINs, and rasters, it's stored on disk in a separate file named after the data source but with a .prj file extension (for example, streets.prj). These files are optional files; thus you may still need to define the coordinate system for one of these data sources. You can create a .prj file with ArcCatalog.

If no coordinate system information is associated with a data source, ArcScene will examine the coordinate values to see if they fall within the range: -180 to 180 degrees for x-values and -90 to 90 degrees for y-values. If they do, ArcScene assumes that these are geographic coordinates of latitude and longitude. If the values are not in this range, ArcScene simply treats the values as planar x,y coordinates.

Tip

Changing the coordinate system of a scene

Changing the coordinate system of a scene does not alter the coordinate system of the source data contained in it.

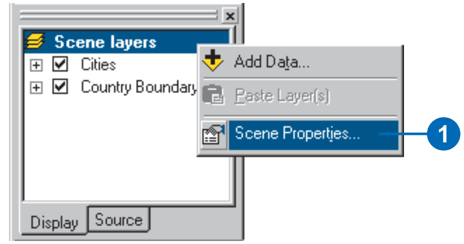
See Also

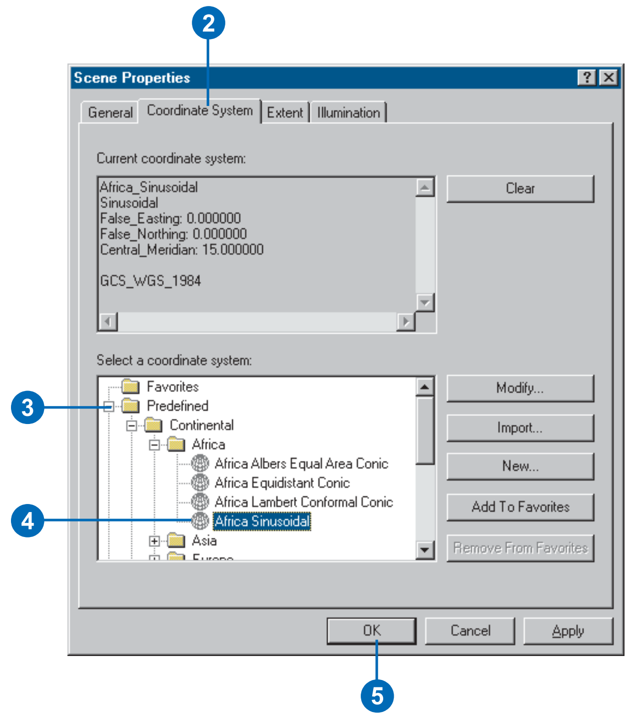
For more information on coordinate systems, see Understanding Map Projections.

Displaying data with a predefined coordinate system

1. Right-click Scene layers and click Scene Properties.
2. Click the Coordinate System tab.
3. Double-click Predefined.
4. Navigate through the folders until you find the coordinate system you want and click it.
5. Click OK.

   All layers in the scene will now be displayed with that coordinate system.





Selecting features in a scene

There are several ways to select features in a scene. The most direct way to select features is to click them in the scene with the Select Features tool or to click them in an attribute table. When a feature is selected it is highlighted.

You can also select features by their attributes as well as by their location with respect to other features. For example, you could select all of the polygons that you've classified as having moderately steep slope, then select all of the buildings that are within these polygons.

Tip

Learning more about selection

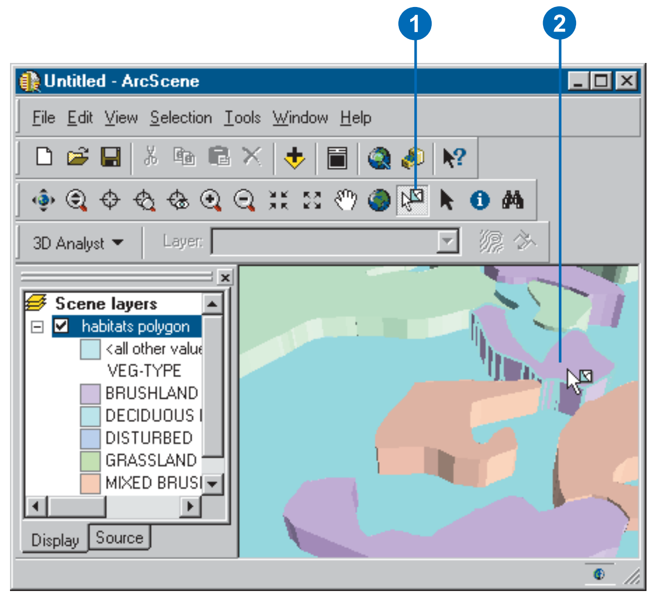
For more information about selecting features, see Using ArcMap.

Selecting features interactively by clicking in a scene

1. Click the Select Features tool.

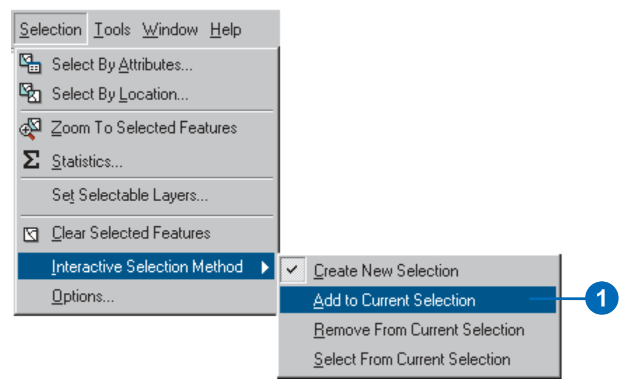
2. Click the feature you want to select.

   The selected feature is highlighted.



Changing the interactive selection method

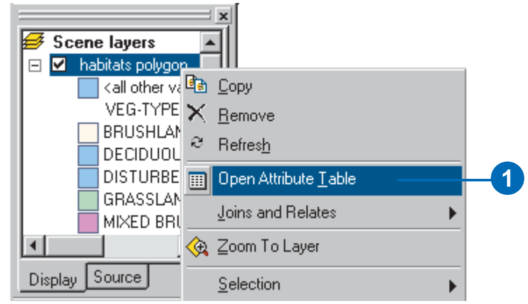
1. Click Selection, point to Interactive Selection Method, and click the selection method you want to use.

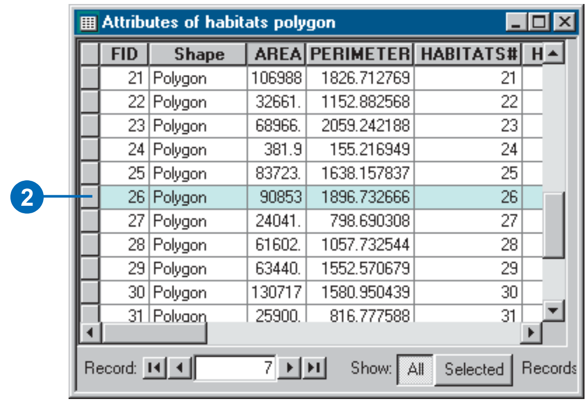


Selecting features interactively by clicking on a table

1. Right-click the feature layer and click Open Attribute Table.
2. Click the row belonging to the feature you want to select.

   The selected feature is highlighted.





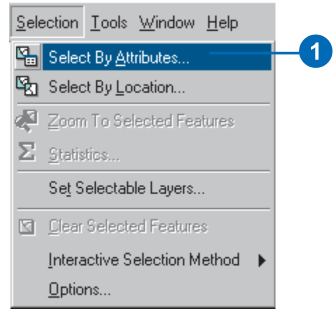
FID	Shape	AREA	PERIMETER	HABITATS#	H
21	Polygon	106988	1826.712769	21	
22	Polygon	32661.	1152.882568	22	
23	Polygon	68966.	2059.242188	23	
24	Polygon	381.9	155.216949	24	
25	Polygon	83723.	1638.157837	25	
26	Polygon	90853	1896.732666	26	
27	Polygon	24041.	798.690308	27	
28	Polygon	61602.	1057.732544	28	
29	Polygon	63440.	1552.570679	29	
30	Polygon	130717	1580.950439	30	
31	Polygon	25900.	816.777588	31	

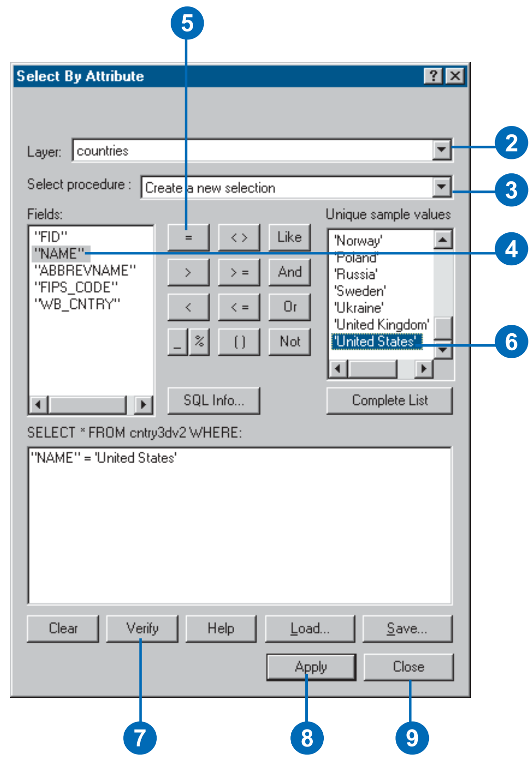
Selecting features by their attributes

1. Click Selection and click Select By Attributes.
2. Click the Layer dropdown arrow and click the layer from which you want to select features.
3. Click the dropdown arrow and click the selection procedure that you want to use.

   The default selection procedure is to create a new selection, but you can also add to, remove from, or select from the current selection.
4. Double-click the attribute field that you want to select from.
5. Click an operator, for example, the equals sign.
6. Double-click a value.
7. Optionally, click Verify to check your selection expression.
8. Click Apply.
9. Click Close.

   The features are selected.

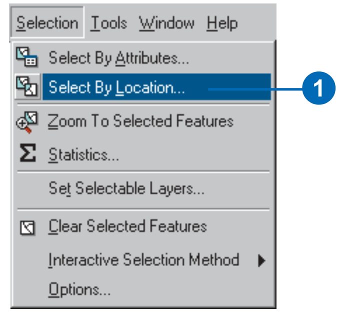


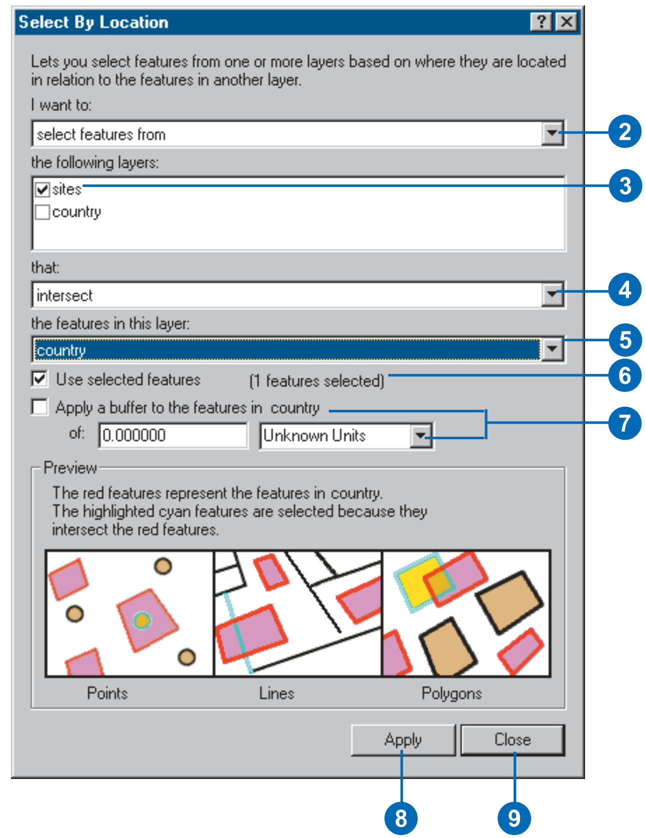


Selecting features by their location

1. Click Selection and click Select By Location.
2. Click the dropdown arrow and click a selection method.
3. Check the layers whose features you would like to select.
4. Click the dropdown arrow and click a selection method.
5. Click the dropdown arrow and click the layer you want to use to search for the features.
6. Optionally, check to use only the selected features.
7. Optionally, check Apply a buffer to the features in <layer> and set the distance within which to search for features.
8. Click Apply.
9. Click Close.

The features are selected.





Exporting a scene

You can export a 2D image of a scene to a graphics file, or you can export a 3D Virtual Reality Modeling Language (VRML) model. Images of scenes can be saved in several common file formats and placed in other documents—for example, in maps or reports.

Tip

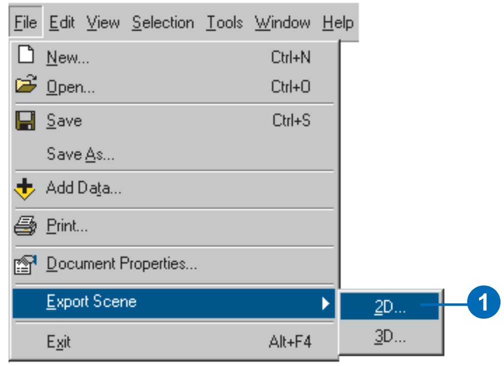
Taking a snapshot of a scene

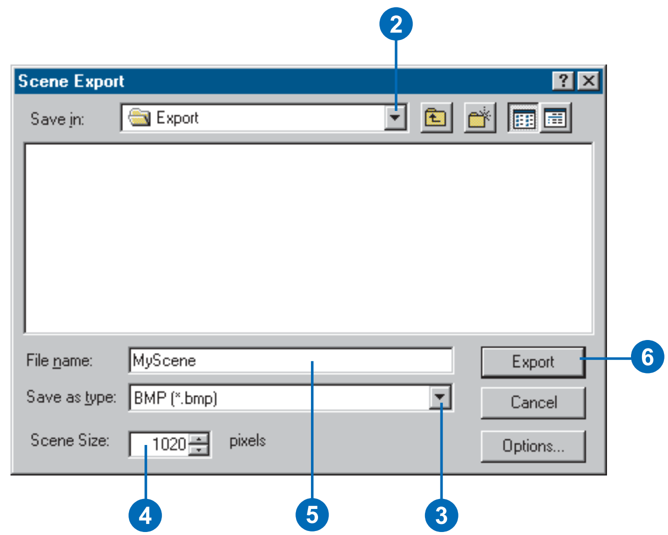
Sometimes you just need a quick snapshot of a scene. Click Edit and click Copy scene to clipboard. You can then paste the snapshot onto a map or other document.

Exporting a 2D graphic of a scene

1. Click File, point to Export Scene, and click 2D.
2. Navigate to the location where you want to save the image of the scene.
3. Click the dropdown arrow to choose the graphics file format to export.
4. Type the width of the exported graphic in pixels.
5. Type a name for the graphic.
6. Click Export.

   The scene is exported to a 2D image.

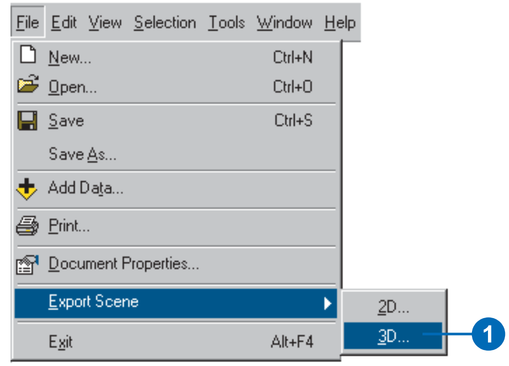


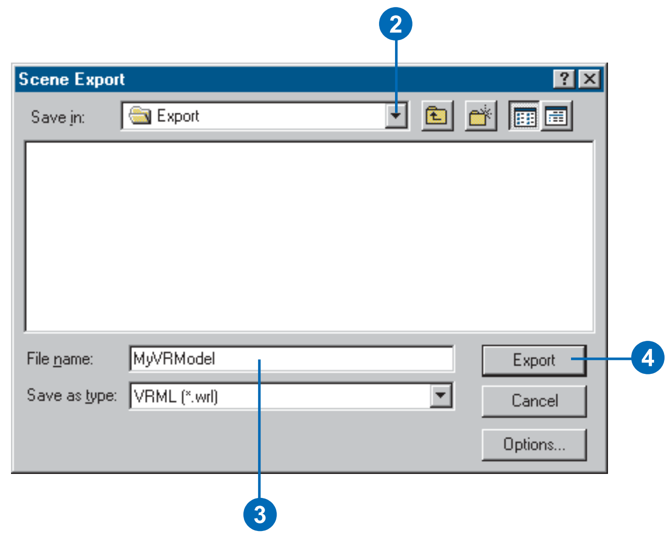


Exporting a 3D VRML model of a scene

1. Click File, point to Export Scene, and click 3D.
2. Navigate to the location where you want to save the VRML model.
3. Type a name for the VRML file.
4. Click Export.

   The scene is exported to a VRML file.

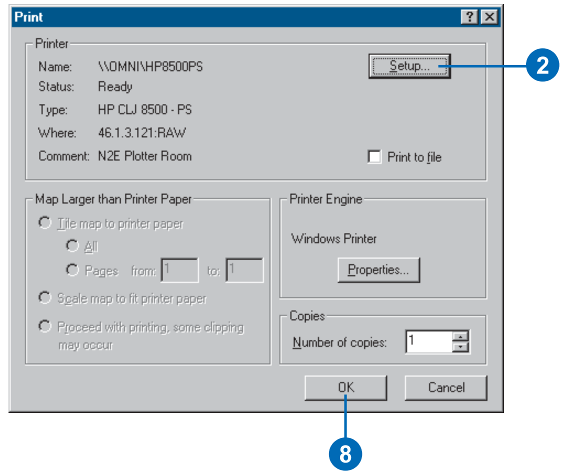


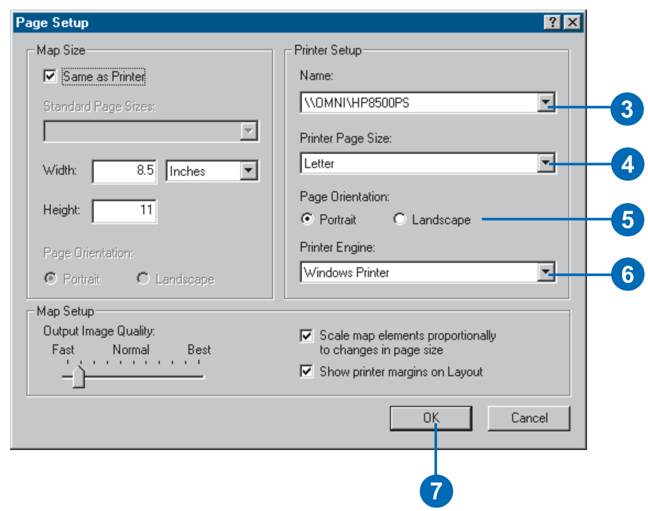


Printing a scene

When you want a hardcopy of a scene, you can print one. You can quickly print to your default printer, or you can change the page setup to select another printer and specify the details of how you want the scene to print.

1. Click the Print button.
2. Optionally, click Setup.
3. Optionally, click the dropdown arrow to select a printer.
4. Optionally, click the dropdown arrow to select a page size.
5. Optionally, click Portrait or Landscape to select the page orientation.
6. Optionally, click the dropdown arrow to select a printer engine.
7. Optionally, click OK.
8. Click OK on the Print dialog box.





8 Animation

IN THIS CHAPTER

- **ArcScene animation**
- **Creating animations**
- **Capturing perspective views in a scene**
- **Recording and playing back animation tracks**
- **Creating keyframes**
- **Making group animations**
- **Making animations from paths**
- **Using the Animation Manager**
- **Timing properties in the Animation Manager**
- **Saving an animation**
- **Sharing animations: Loading an ArcScene Animation file**

Animations make your scene come alive by storing actions so they can be replayed as you choose. They can help you visualize changes in perspective, changes in the scene's properties, geographical movements, and temporal changes.

You might create an animation that helps you visualize how moving satellites react with one another during their orbit. Additionally, you may model the earth's rotation and change in lighting at the same time.

You create animations using the Animation toolbar. You can make animations that manipulate data, perspective, and scenes.

An animation consists of one or more tracks. Tracks control dynamic changes of the properties of an object, such as a scene's background color, a layer's visibility, or a camera's location. Tracks are made up of a set of keyframes. A keyframe is a snapshot of a particular object's properties at a certain time. The object can be a scene, a layer, or a camera. For example, you can create a track with the scene object that animates scene property keyframes to show the background color changing from white to black.

Animation tracks are stored in the current scene document. They can also be saved into a separate file that may be shared by numerous scene documents. An animation may also be exported as an Audio Video Interleave (.avi) file that can be played by a third-party video player.

Animations provide a new and actuated aspect to your scenes. Use animations to automate the process you would undertake to demonstrate points that can only be made through visual dynamics.

ArcScene animation

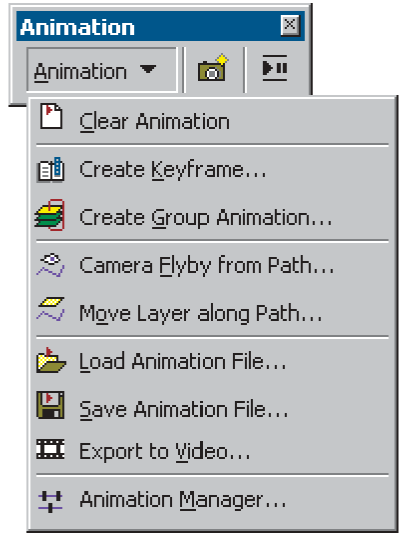
ArcScene lets you create, save, and share animations. You can create animations in different ways, composing the animation of multiple tracks that animate the scene properties, a layer, or the camera. An animation may be saved in a scene document, saved as an independent ArcScene Animation (.asa) file, or exported to a .avi file. You can share animations by exchanging scene documents, interchanging .asa files, or distributing .avi files.

The camera and the target

You view your scene through viewer cameras. A camera's location is defined by the observer property. The center of your view is called the target. Imagine what you see is what the camera sees. As you navigate through a scene, you are actually moving the observer in conjunction with the target. If you set a new target on the data, that point on the data is shifted to the center of your view. There is only one camera, observer, and target set in any viewer. Familiarizing yourself with the camera, observer, and target will help you gain a better understanding of animation.

Introducing the Animation toolbar

The Animation toolbar has all the tools you'll need to work with animations in ArcScene. Using these tools, you can record navigation, capture perspective views, save and export tracks, create video files, make group animations, create tracks from paths, and manage and preview your animations.



Creating an animation

You create an animation by composing the tracks it will contain. You can make tracks by creating a set of keyframes, recording actions, alternating the visibility of groups of layers, and importing paths that constrain movement. Use the Animation Manager to edit tracks and keyframes and organize how the tracks in an animation interact with each other.

Saving an animation

There are three ways to save an animation. A scene document will automatically store an animation that is present when the document is saved. In addition, you can save the animation tracks to an .asa file. Finally, you can create a standalone video by exporting the animation as a .avi file.

Sharing animations

You can share animations by loading a scene document that contains an animation, loading .asa files into ArcScene, and viewing .avi files that were created from animations. Use a shared scene document with an animation to demonstrate a particular point to colleagues. Independent .asa files can be used as templates for others to build on or as generic animations that can be utilized with various data. Share a .avi file for picture-perfect, highly detailed animations that can be played in real time to a wide and varied audience when you need to quickly demonstrate a problem that can only be shown dynamically.

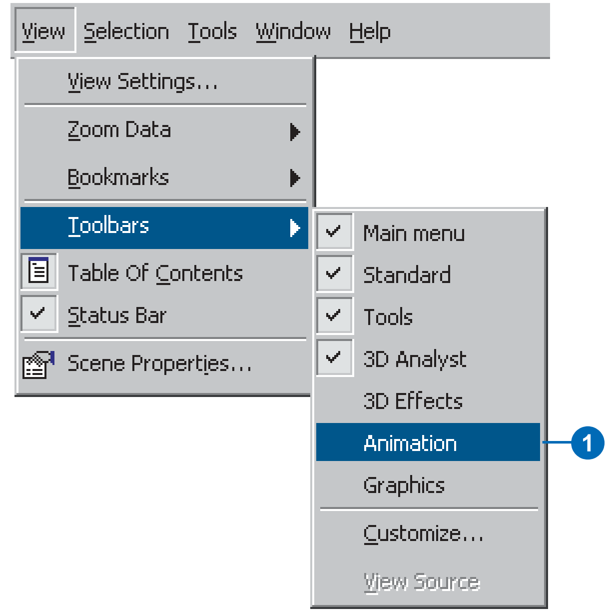
Creating animations

There are several ways to create animations. The simplest method, capturing perspective views, is quick and can be done with any data. A more complex method of creating animations is to create animations from paths defined by selected line features or graphics. This method requires specific types of data in the scene but can provide a more visually appealing animation.

Turning on the Animation toolbar

1. Click View, point to Toolbars, and click Animation.

   The Animation toolbar appears as an undocked toolbar.



Capturing perspective views in a scene

Use the Capture View command to save perspective views as keyframes in a camera track. The resulting track will be an interpolation between the keyframes, making a smooth animation. For example, you can create a track that rotates your scene, zooming in and out to points of interest along the way.

Tip

Is there a shortcut to capturing views?

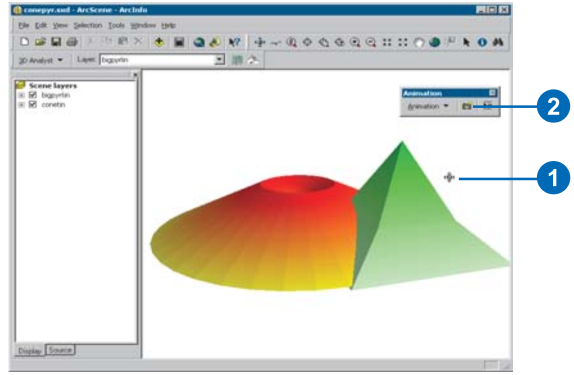
To capture a view to an animation, press Ctrl+A instead of clicking the Capture View button.

See Also

To learn how to play back an animation track, see 'Recording and playing back animation tracks' in this chapter.

Capturing views to make an animation

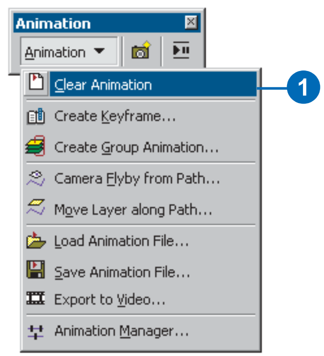
1. Navigate to the perspective you want to capture.
2. Click the Capture View button.
3. Repeat to capture more views as keyframes in a camera track.



Clearing an animation

1. Click Animation and click Clear Animation.

   All animation tracks are removed from the scene.



Recording and playing back animation tracks

Simple recording and playback are achieved using controls that resemble a VCR. Press the Record button to record your navigation and press the Play button to play it back.

Tip

Recording Fly tool navigation

You can make a flyby animation by recording as you fly through your scene with the Fly tool.

Tip

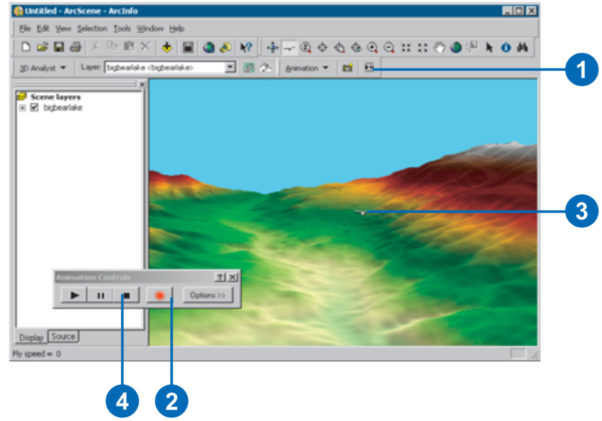
Controlling playback and recording options

Click Options on the Animation Controls toolbar to access animation duration, segments of animation to play back, looping mode, and overwriting options.

Recording navigation to create an animation

1. Click the Open Animation Controls button.
2. Click the Record button.
3. Navigate in the scene using any navigation tool.
4. Click the Stop button.

   A camera track is created, storing the navigation sequence.



Playing back an animation

1. Click the Open Animation Controls button.
2. Press the Play button.

   The animation is played back.

Creating keyframes

Keyframes are the most fundamental elements of an animation. A series of keyframes is assembled into a track. Create a keyframe to make a snapshot of an object's properties. Use keyframes to make snapshots of scene properties, camera properties, or layer properties, such as a scene's background color, a layer's transparency, or a camera's location.

Tip

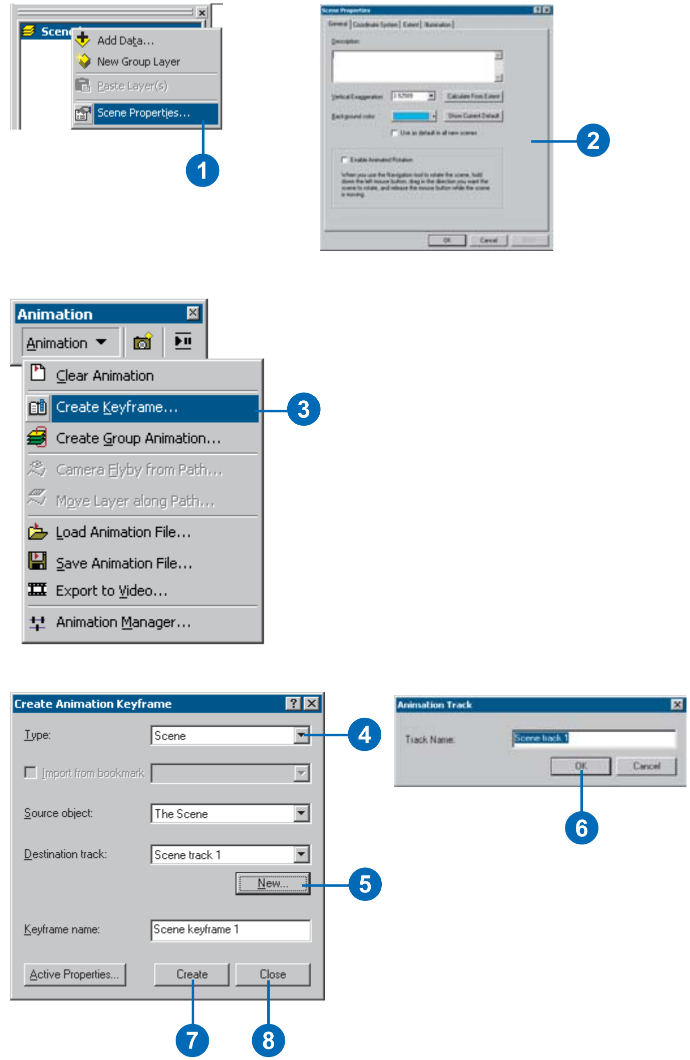
How can I edit keyframe properties?

You can edit the properties of a keyframe by using the Animation Manager.

Creating a keyframe of scene properties

1. Right-click Scene layers and click Scene Properties.
2. Set the scene property or properties you want to capture.
3. Turn on the Animation toolbar, click Animation, and click Create Keyframe.
4. Click the Type dropdown arrow and choose Scene.
5. Click New to create a new scene track.
6. Click OK.
7. Click Create.
8. Click Close.

   You can repeatedly change a property and create a new keyframe without closing the Create Animation Keyframe dialog box. Note that you need at least two keyframes to create a track that will show change.



Tip

How else can I create a keyframe?

You can create keyframes from the Keyframes page of the Animation Manager dialog box. Click Create to invoke the Create Animation Keyframe dialog box. For more information, see 'Using the Animation Manager'.

Tip

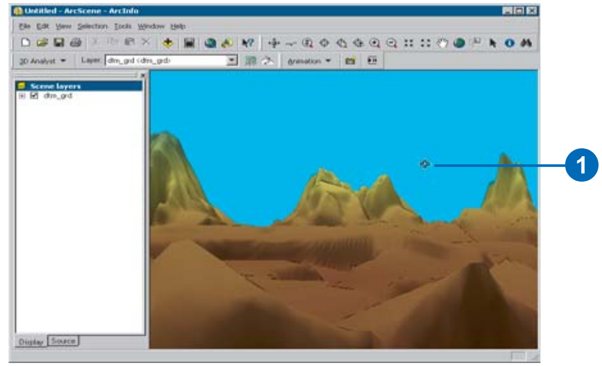
Using bookmarks as keyframes

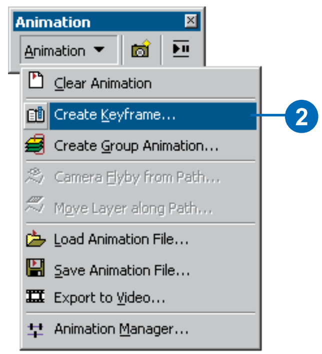
If your scene has bookmarks, you can import them to an animation as keyframes by checking Import from bookmarks and choosing a bookmark in the dropdown list.

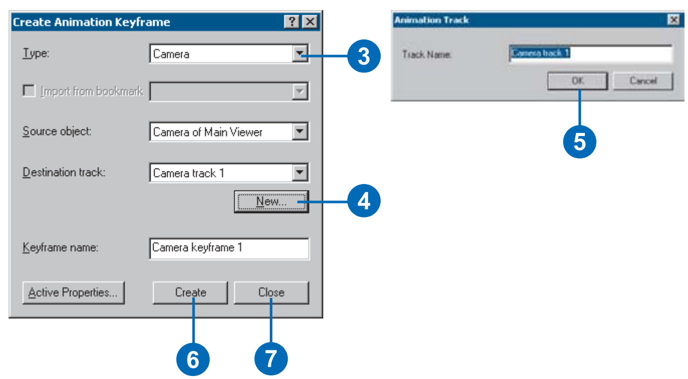
Creating a keyframe of camera properties

1. Navigate to the camera position you want to capture.
2. Click Animation and click Create Keyframe.
3. Click the Type dropdown arrow and choose Camera.
4. Click New.
5. Click OK.
6. Click Create.
7. Click Close.

   You can change a camera property by navigating to a new position and then create a new keyframe repeatedly without closing the Create Animation Keyframe dialog box. Note that you need at least two keyframes to create a track that will show change.







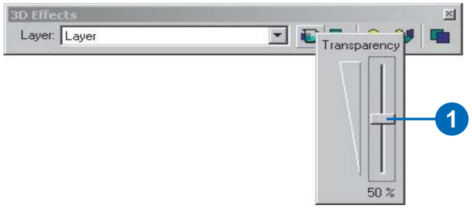
Creating a keyframe of layer properties

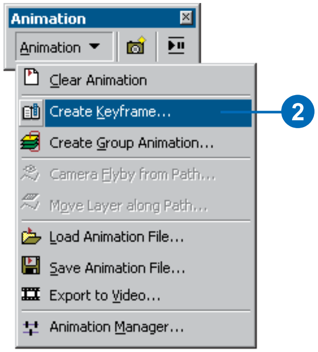
1. Change the layer property that you want to capture.

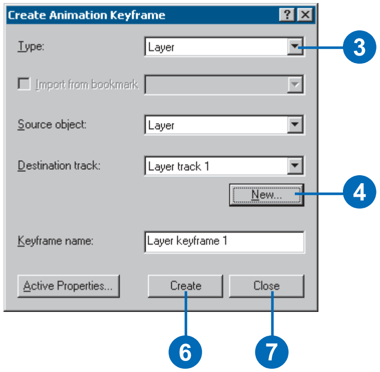
   For example, turn on the 3D Effects toolbar and set a transparency for a layer.

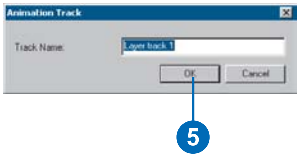
2. Click Animation and click Create Keyframe.
3. Click the Type dropdown arrow and choose Layer.
4. Click New.
5. Click OK.
6. Click Create.
7. Click Close.

   You can repeatedly change a property and create a new keyframe without closing the Create Animation Keyframe dialog box. Note that you need at least two keyframes to create a track that will show change.









Making group animations

You can create an animation from an existing group layer or individual layers within a scene. For example, you might have a group layer in which individual layers represent snapshots in time. If these layers are ordered sequentially in the TOC, you can create tracks that successively turn visibility on and off for each layer within the group. The animation will depend on the ordering in the TOC, so arrange layers in the order that you want them played. The tracks will show how the data changes through time.

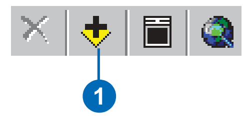
Making a group animation

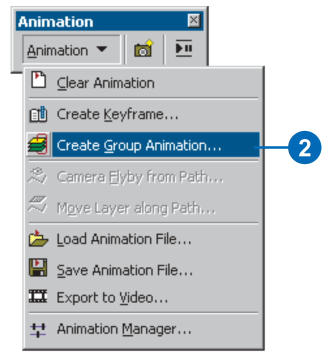
1. Click the Add Data button to add the layers or group layer that you want to animate.
2. Click Animation and click Create Group Animation.
3. Optionally, choose a Base name for tracks.

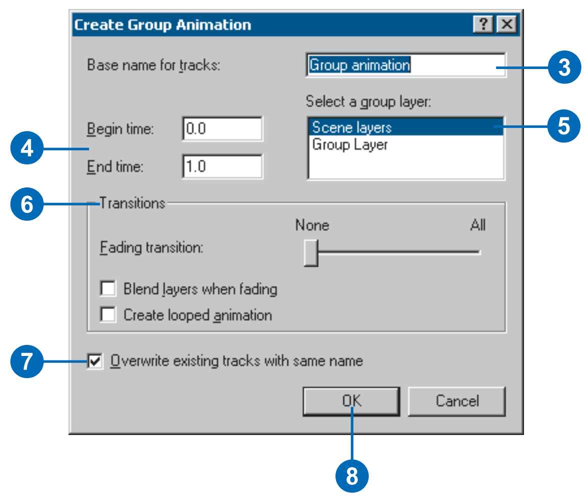
   You are provided with a default name for the tracks, but you can change it to something more meaningful.
4. Optionally, set the beginning and ending times.

   Setting these times allows you to determine when the group animation will play relative to other tracks that may exist.
5. Select a group layer. You can click a group layer or all the layers in your scene.
6. Optionally, change Transitions.

   These options will help you determine how layers in an animation change from one to another.
7. Optionally, uncheck Overwrite existing tracks with same name to add additional group animations.
8. Click OK.







Making animations from paths

You can create camera tracks and layer tracks from paths. A *path* is defined by a selected line feature or graphic, and its purpose is to constrain movement along the selection. A camera track is created by moving the camera along the selected path. A layer track is made by moving a layer along a path.

Use the Path destination options for a camera flyby path to modify the way the camera (observer) travels. There are three options for moving a camera along a path: move both the target and observer along the path for a flyby, move the observer along the path with the current target to point the camera at an area while it ▸

Tip

Modifying camera properties

When selecting flyby mode as the path destination, click Orientation Settings in the Camera Flyby from Path dialog box to modify the way camera azimuth, inclination, and roll properties are calculated from the path.

Making a camera flyby from a path

1. Click a selection tool and select the line feature or graphic you want to use as a path.
2. Click Animation and click Camera Flyby from Path.
3. Optionally, check Apply in reverse order.

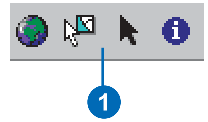
   The camera will begin at the other end of the path.
4. Optionally, type a value in the Vertical offset text box.

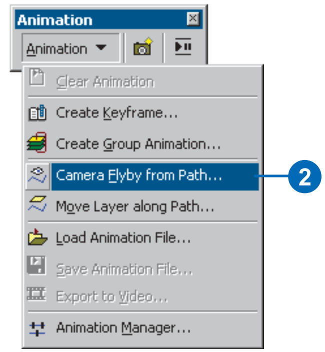
   An offset defines the height of the camera.
5. Optionally, slide the simplification factor.

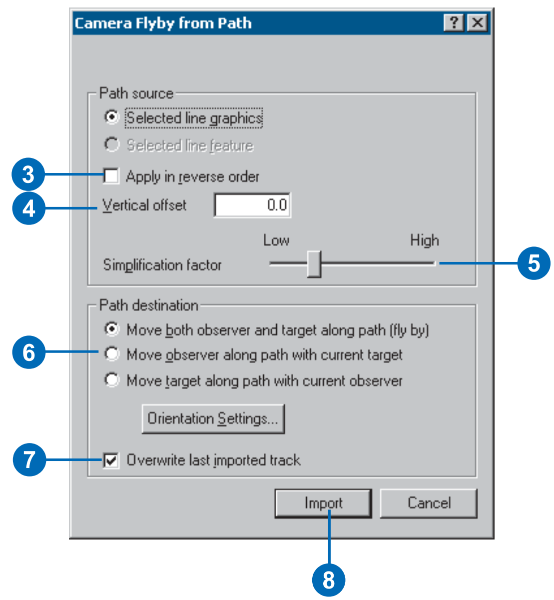
   The simplification factor determines how much the path will be generalized for the animation.
6. Optionally, choose a path destination.

   Change the path destination to determine how the observer and target are positioned during the animation.
7. Optionally, uncheck Overwrite last imported track.

   Disabling this option will allow you to add tracks to previous ones.
8. Click Import.







moves along the path, or move the target along the path with the current observer to let the camera follow a virtual point along the path in the track. You can then move a layer along this path to have the camera point toward the layer as it moves.

Tip

Using advanced layer orientation settings

Click Orientation Settings to modify the way a layer's azimuth, inclination, and roll properties are calculated from the path.

Moving a layer along a path

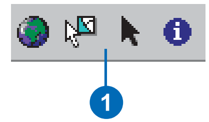
1. Click a selection tool and select the line feature or graphic you want to use as a path.
2. Click Animation and click Move Layer along Path.
3. Click the Layer dropdown arrow and choose the layer you want to move.
4. Optionally, check Apply in reverse order.

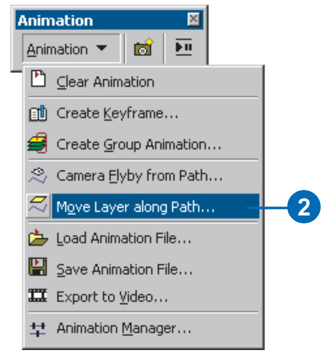
   This option will start the layer moving from the opposite end of the path.
5. Optionally, type a value in the Vertical offset text box.

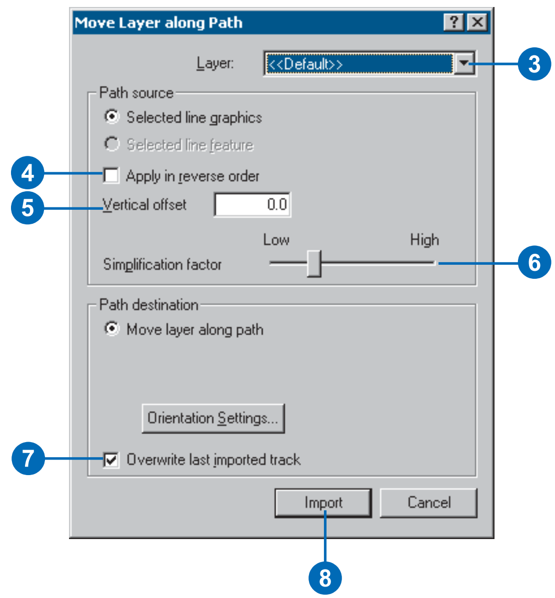
   The vertical offset determines the height of the layer.
6. Optionally, slide the simplification factor.

   The simplification factor indicates how much the path will be generalized when it is used for the animation.
7. Optionally, uncheck Overwrite last imported track.

   Disable this to allow additional layer tracks to be imported without overwriting existing ones.
8. Click Import.







Using the Animation Manager

The *Animation Manager* allows you to access properties of keyframes and tracks. In addition, you can access timing properties. You can manipulate these properties and then see the result using the Time View preview.

Tip

Why change a track or keyframe property?

Use the Animation Manager to modify the properties of a keyframe or track. For example, you can change the camera target's x-coordinate to move the target to a more pleasing position. Options to change properties are numerous, allowing you to make your animation look just as you intend.

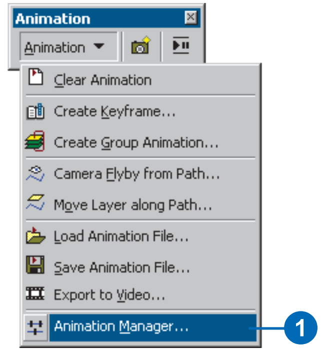
Tip

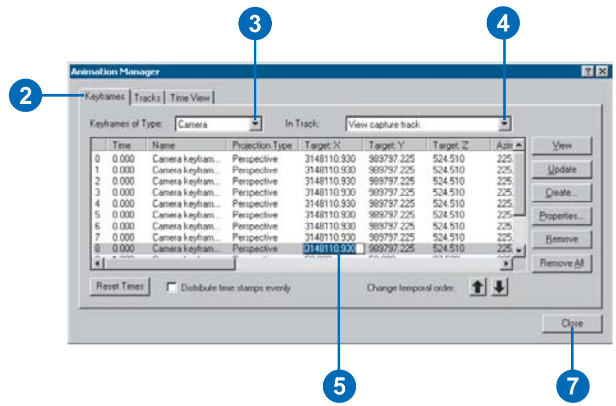
Updating a keyframe

To fine-tune a scene or camera keyframe in the Animation Manager, select the keyframe you want to change in the Keyframes page, modify the property in the Keyframes page, and click Update. The keyframe will be updated to reflect the change.

Using the Animation Manager to access keyframe properties

1. Click Animation and click Animation Manager.
2. Click the Keyframes tab.
3. Click the Keyframes of Type dropdown arrow and choose the type of keyframe you want to examine.
4. Click the In Track dropdown arrow and choose the specific track you want to access.
5. Click a Keyframe property to change it.
6. Press Enter.
7. Click Close.





Tip

Changing a track's priority

If multiple camera or scene tracks refer to the same object, they will be played back according to their priority. To change a track's priority, select the track in the Tracks page of the Animation Manager and use the arrow up or down buttons to move its ranking. If multiple layer tracks refer to the same layer, their transformations will be combined.

Tip

What is binding?

Tracks are bound to objects. For example, a camera track is bound to a particular viewer, and it will play in that viewer. To change the object that the track is bound to, click Binding.

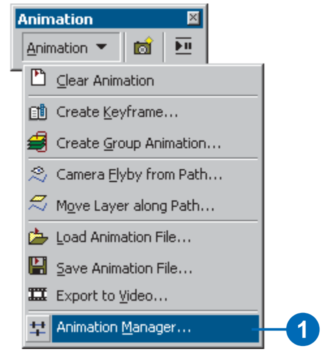
Tip

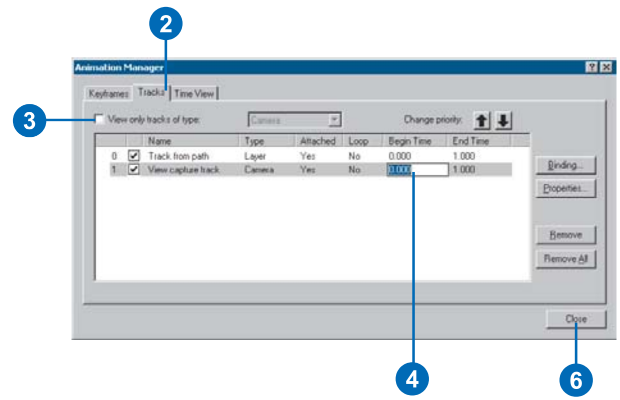
Can I interact with an animation track?

You can manually alter properties of a track to allow you, for example, to navigate during animation. Select a track and click Properties to see those that are available for that track. Uncheck the ones you want to control manually.

Using the Animation Manager to access track properties

1. Click Animation and click Animation Manager.
2. Click the Tracks tab.
3. Optionally, check View only tracks of type, click on the dropdown arrow, and select the type you want to examine.
4. Click a Track property to change it.
5. Press Enter.
6. Click Close.





Timing properties in the Animation Manager

Use the Time View page of the Animation Manager to modify when keyframes of one track are played relative to others or to organize how tracks are synchronized with one another.

When in the Time View page, the timing of all tracks is normalized to one time unit, and there is no indication of duration. Use the duration setting in the Animation Control options dialog box to determine how long your animation will play.

Tip

Previewing an animation

In the Time View page, you can examine your animation by clicking the display until a vertical red line appears and then dragging the line to preview the way your animation will play.

Tip

Why are my layer tracks colored differently?

In the Time View page, group layer tracks are color-coded to indicate their visibility. Light green means the layer is visible; red indicates that the layer is invisible; and pale yellow lets you know that the layer is semitransparent, or in transitional visibility.

Using the Animation Manager to modify timing properties of tracks

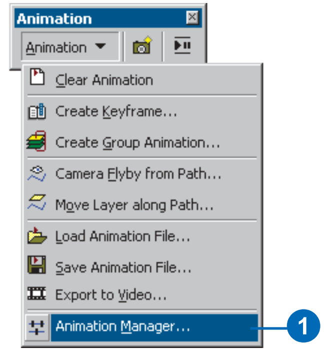
1. Click Animation and click Animation Manager.
2. Click the Time View tab.
3. Optionally, click the Plus or Minus buttons to change the time scale.

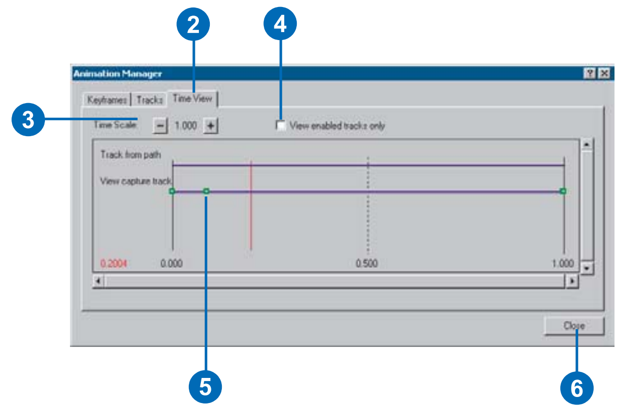
   Reducing the time scale gives you a more detailed view.
4. Optionally, check View enabled tracks only.

   Using this option shows only the tracks that are currently enabled. Disable a track in the Tracks page.
5. Click and drag a green keyframe box in a track to change the time when it will be played.

   Moving a keyframe to a later point in the track will make it be played later in the animation.

   Click and drag a keyframe at the start or end of a track to determine when that track will play relative to others.
6. Click Close.





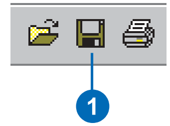
Saving an animation

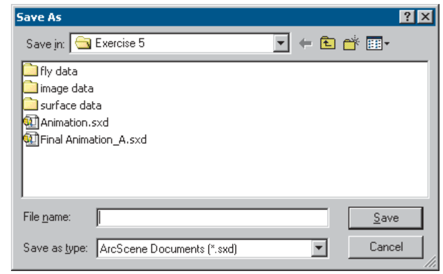
An animation that is created in a scene will be stored in the scene when it's saved. Animation tracks can also be saved into an independent .asa file. Another way to save an animation is to export it to a .avi file.

Saving an animation in a scene document

1. Click the Save button.

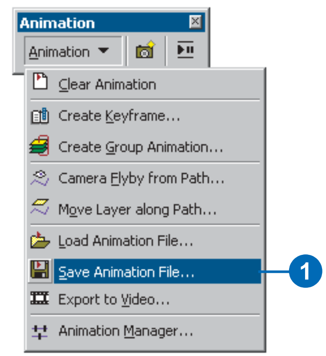
   An existing scene document will be appended with the animation, or if none exists, you will be prompted to provide a name for a new document that will contain the animation.

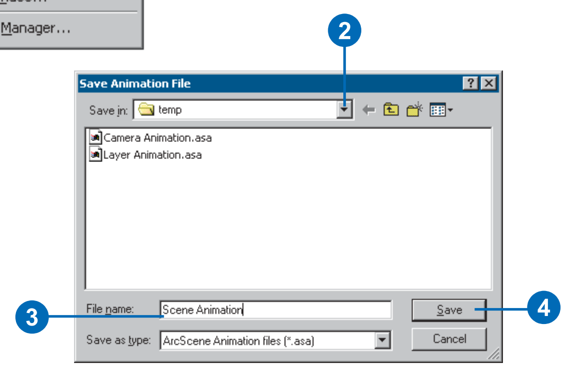




Saving an animation by exporting tracks to an ArcScene Track file

1. Click Animation and click Save Animation File.
2. Click the Save in dropdown arrow and choose a location.
3. In the File name text box, type the name you want to give the animation file.
4. Click Save.

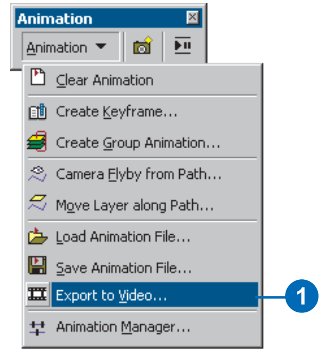


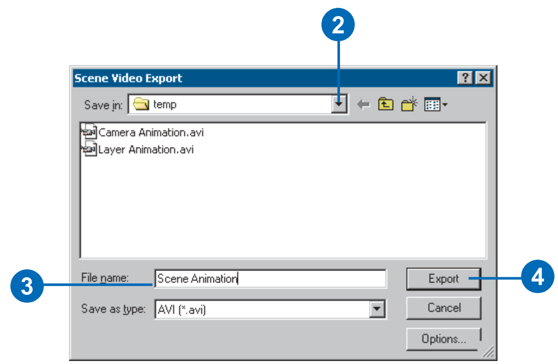


Exporting an animation to a video file

1. Click Animation and click Export to Video.
2. Click the Save in dropdown arrow and choose a location.
3. In the File name text box, type the name of the video file you want to create.
4. Click Export.

   A .avi file of the given name is created in the specified location.





Tip

Using advanced video options

To access frame rate and video quality properties, click Options on the Scene Video Export dialog box.

Tip

Preventing extraneous windows from appearing in video files

Make sure that when you export your animation to a video file you move other windows away so nothing obscures the viewer that is playing the animation. Otherwise, they may appear in the exported file.

Sharing animations: Loading an ArcScene Animation file

You can load an ArcScene Animation file into any scene. However, when attempting to load an animation into a new scene, care must be taken to ensure that the animation applies to the scene. If an animation track refers to an item that doesn't apply to the scene, the track will not play. For example, if you've created an animation containing a track that refers to a certain layer in a scene, that layer must be present in the new scene and must be in the same location in the TOC. Because a layer track refers to a layer by noting its ranking in the TOC, the track may animate an incorrect layer or may not be enabled to play if the ranking in the new scene's ►

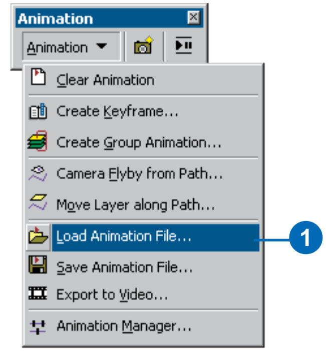
Tip

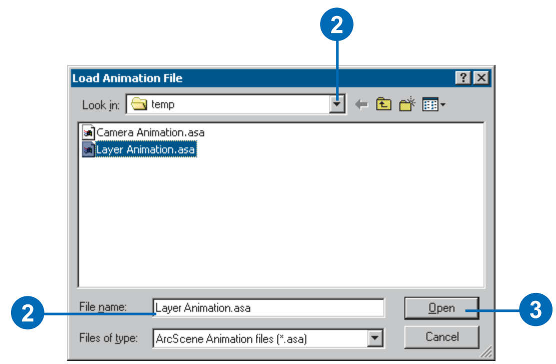
Identifying ArcScene Animation files

ArcScene Track files are appended with the .asa extension.

Loading an ArcScene Animation file

1. Click Animation and click Load Animation File.
2. Browse to the ArcScene Animation file you want to load.
3. Click Open.





TOC is different. Also, if a track refers to a viewer that doesn't exist in the new scene, that track will not play. In addition, if you've created a camera track in a scene's extent and want to play it in a new scene, make sure the extent of the new scene is the same, or else your track might move the camera in an area where there is no data. This is because a camera track is defined in part by a scene's extent.

Tip

Using animations stored in a scene document

Load a scene document with animation into ArcScene as you would any other scene document. Modify the name you give to animated scene documents to let others know that they contain animation.

Glossary

3D feature

A representation of a real-world object in a map or scene with z-values stored within the feature's geometry. Features have geometry and may also have attributes stored in a feature table. PointZ, PolylineZ, PolygonZ, and Multipatch feature classes and shapefiles may contain 3D features.

3D graphic

A graphic object in a map or scene with z-values stored in the graphic's geometry. Graphics do not have attributes.

altitude

1. The height, z-value, or vertical elevation of an object above, or below, a given reference datum—for example, sea level.
2. The height above the horizon—measured in degrees—from which a light source illuminates a surface; used when calculating a hillshade or for controlling the position of the light source in a 3D scene for on-the-fly shading.

animation

A collection of tracks in a scene that defines dynamic property changes related to associated objects.

Animation Manager

The interface that allows you to edit the properties of keyframes, tracks, and timing of an animation. It also allows you to preview an animation.

ArcScene

An ArcGIS application for combining and viewing features, surfaces, and graphics in 3D perspective.

area

1. The planimetric area of a polygon feature or surface.
2. The surface area of a 3D surface or of the portion of a surface above or below a reference plane. Surface is measured along the slope of a surface and is always greater than the 2D planimetric extent of the surface. When compared to the planimetric area, surface area gives you an idea of the surface roughness.

aspect

The direction a slope faces. The aspect for a TIN face is the steepest downslope direction of the face. The slope at a cell in a raster is the steepest downslope direction of a plane defined by the cell and its eight surrounding neighbors.

attribute

A piece of information describing a map feature. The attributes of a census tract, for example, might include its area, population, and average per capita income.

autocorrelation

The statistical relationship among the measured points, where the correlation depends on the distance and/or direction that separates the locations.

azimuth

A compass direction. In 3D Analyst and ArcGIS Spatial Analyst, the direction from which a light source illuminates a surface is called the azimuth.

background

1. You can set the color of the background of a scene to suggest sky, empty space, or any color that suits your visualization purpose. The default background color is white.
2. Some rasters, typically images, have border areas that are outside of the area for which image data was collected. This area is often assigned an arbitrary value—often black, or 255. You can control the display of these parts of a raster by setting the background color on the Symbology tab of the Layer Properties dialog box. See also nodata.

base height

The height at which a surface, raster, or feature is drawn in a scene. You can set the base height for features and rasters from a surface or by using a constant value or expression. Features with z-values stored in their geometry can have their base height set using the z-values. Setting the base heights from a surface is also called draping.

bin

A classification of lags, where all lags that have similar distance and direction are put into the same bin. Bins are commonly formed by dividing the sample area into grid cells or sectors.

camera

An object that defines the perspective of a scene's display.

Catalog tree

Contains a set of folder connections that provide access to geographic data stored in folders on local disks or shared on the network. It also includes folders that let you manage database connections and coordinate systems. The Catalog tree provides a hierarchical view of the geographic data in those folders.

categorical raster

A raster that represents the world with a set of values that have been aggregated into classes. For example, a satellite image that has been reclassified to extract a number of land cover types is a categorical raster. Categorical rasters represent an area, but the values do not form a continuous surface. See raster.

continuous raster

A raster that represents the world with a set of values that vary continuously to form a surface. For example, a raster digital elevation model and an interpolated chemical concentration surface are continuous rasters.

contour

A line that connects points of equal value on a terrain surface (an isoline).

drape

To set the base height for features or a surface using a surface. When you drape a layer over a raster surface, the resolution of the base surface is automatically downsampled to increase performance. You can change the resolution of the base surface if the default is unsatisfactory.

drawing priority

The order in which layers that occupy the same x,y,z positions are drawn in a scene. For example, if you have a road feature layer and an orthophoto draped over the same terrain model, the roads and raster may appear patchy or broken where they coincide. You can reduce the drawing priority for the raster so it will appear below the features. You can only change the drawing priority for areal features and surfaces.

edge

The linear segments between nodes in a TIN surface. Edges store information about the faces that they border. Linear features and the perimeters of polygon features used to generate a TIN become edges in the TIN.

extrusion

You can extrude 2D point, line, and area features in a scene into vertical lines, planes, and solids. Use extrusion to show the depth of well point features or the height of building footprint polygons.

face

Triangles form the faces on a TIN surface. Each face on a TIN surface is defined by three edges and three nodes and is adjacent to one to three other faces on the surface. TIN faces are used to calculate aspect and slope information and may have stored tag values.

grid

A geographic representation of the world as an array of equally sized square cells arranged in rows and columns. Each grid cell is referenced by its geographic x,y location. See raster.

hillshade

The hypothetical illumination of a surface. A hillshade raster can be calculated for a given surface, or hillshading can be applied on the fly to surfaces and areal features in a scene.

IDW

Inverse Distance Weighted. An interpolation method where cell values are estimated by averaging the values of sample data points in the vicinity of each cell. The closer a point is to the center of the cell being estimated, the more influence, or weight, it has in the averaging process.

image

Represents geographic features by dividing the world into discrete squares called cells. Examples include satellite and aerial photographs, scanned documents, and building photographs. See also raster.

interpolate

To predict values for a surface from a limited number of sample data points.

keyframe

In an animation, a snapshot of an object's properties at a certain time.

kriging

A geostatistical interpolation method based on statistical models that include autocorrelation—the statistical relationship among the measured points. Kriging weights the surrounding measured values to derive a prediction for an unmeasured location. Weights are based on the distance between the measured points, the prediction location, and the overall spatial arrangement among the measured points.

lag

The line, vector, that separates any two locations. A lag has length, distance, and direction, orientation.

line of sight

A graphic line between two points on a surface that shows whether or not the view along the line is obstructed.

natural neighbors

An interpolation method where cell values are estimated using weighted values of the closest surrounding input data points, determined by creating a triangulation of the input points.

navigate

To interactively change the perspective view of a scene by moving the mouse.

nodata

Some rasters, typically grids, have empty cells within the area for which data was collected. These cells are often assigned an arbitrary value—often -9999, or nodata. Rasters with some nodata cells are also created by some raster analysis tools. You can control the display of these parts of a raster by setting the nodata color on the Symbology tab of the Layer Properties dialog box. See also background.

node

A location on a TIN surface at the intersection of two or more edges. Nodes on a TIN surface store elevation (z) values and may have tag values. Point features used to generate a TIN become nodes in the TIN.

nugget

A parameter of a covariance or semivariogram model that represents independent error, measurement error, and/or microscale variation at spatial scales that are too fine to detect. The nugget effect is seen as a discontinuity at the origin of either the covariance or semivariogram model.

observer

The position of the camera in a scene.

observer offset

The height of the observer point above a surface used when calculating lines of sight and viewsheds.

offset

To change the z-value for a surface or features in a scene by a constant amount or by using an expression. Offsets may be applied to make features draw just above a surface.

orthographic view

Allows you to view the data in a scene as a 2D plane seen from above. There is no perspective foreshortening in orthographic view, so scale is constant across the entire display.

path

A single line feature or graphic that, when imported to an animation track, determines the movement of a camera or layer.

perspective view

Allows you to view the data in a scene in 3D from a perspective that you can control by navigating the scene or from a specified position.

profile graph

A graph of the height of a surface along a specified line.

range

A parameter of a variogram or semivariogram model that represents a distance beyond which there is little or no autocorrelation among variables.

raster

Represents any data source that uses a grid structure to store geographic information. See grid and image.

raster resolution

The size of the cells in a raster. Only features larger than or equal to the cell size can be discerned in a raster.

scene

A document used for interactive display and query of geographic data in 3D.

semivariogram

The variogram divided by two.

sill

A parameter of a variogram or semivariogram model that represents a value that the variogram tends to when distances get very large. At large distances, variables become uncorrelated, so the sill of the semivariogram is equal to the variance of the random variable.

slope

Slope is the incline, or steepness, of a surface. The slope of a TIN face is the steepest downhill slope of a plane defined by the face. The slope at a cell in a raster is the steepest downhill slope of a plane defined by the cell and its eight surrounding neighbors. Slope can be measured in degrees from horizontal (0–90) or percent slope, which is the rise divided by the run, times 100. A slope of 45 degrees equals 100 percent slope. As slope angle approaches vertical (90 degrees), the percent slope approaches infinity.

spline

An interpolation method in which cell values are estimated using a mathematical function that minimizes overall surface curvature, resulting in a smooth surface that passes exactly through the input points.

steepest path

A line that follows the steepest downhill direction on a surface. Paths terminate at the surface perimeter or in surface concavities or pits.

stretch

Applied to a raster to increase the visual contrast in brightness between its cells.

surface

A set of continuous data, such as elevation or air temperature, over an area or the boundary between two distinct materials or processes.

target

The center point in a scene’s view at which the camera is aimed.

target offset

The height of a target point above a surface used when calculating lines of sight and viewsheds.

TIN

Triangulated irregular network. A data structure that represents a continuous surface through a series of irregularly spaced points with values that describe the surface at that point—for example, an elevation. From these points, a network of linked triangles forms the surface.

track

An ordered collection of like keyframes that, when played as an animation, shows a dynamic transition between them.

triangle

Triangles form the faces on a TIN surface. Each triangle on a TIN surface is defined by three edges and three nodes and is adjacent to up to three other triangles on the surface. TIN triangles can be used to derive aspect and slope information and may be attributed with tag values.

variogram

A function of the distance and direction separating two locations that is used to quantify autocorrelation. The variogram is defined as the variance of the difference between two variables at two locations. The variogram generally increases with distance and is described by nugget, sill, and range parameters.

variography

The process of estimating the theoretical variogram. It begins with exploratory data analysis, then computing the empirical variogram, binning, fitting a variogram model, and using diagnostics to assess the fitted model.

vertical exaggeration

The amount by which the z-values in the scene are multiplied in order to enhance details in the shape of the surface. Scenes may appear flat when the range of x and y values is much larger than the z-values—setting vertical exaggeration can compensate for this by increasing relief.

viewer

An additional window that allows you to view the 3D data in a scene from another angle. You can have multiple viewers in a scene.

viewshed

The viewshed identifies the areas on a surface that can be seen from one or more observation points or lines. This is useful for applications in which the visibility of an object is important, such as finding well-exposed places for communication towers or hidden places for landfills or other facilities.

volume

The space—measured in cubic units—between a surface and a plane at a specified elevation. Volume may be calculated above or below the plane.

Z factor

The number of ground x,y units in one surface z unit. The input surface values are multiplied by the specified Z factor to adjust the output raster z units to another unit of measure.

z-value

A value represented on the z-axis in a three-dimensional x,y,z coordinate system. The values of a terrain or chemical concentration surface can be used as z-values when rendering the surface in 3D. Numeric attributes of features, for example, the number of floors in a building, can also be used as z-values.

Index

Symbols

A

B

C

E

F

M

N

O

P

R

S

T

U

V

W

Z