Creating and Analyzing Surfaces Using ArcGIS® Spatial Analyst

Exercises

Juan Salcedo Carbajal
2011

TELEMATICA
Tecnología Integrada GIS
www.telematica.com.pe

esri Official Distributor

Creating and Analyzing Surfaces Using ArcGIS® Spatial Analyst

Course version 2.1. Version release date August 2010.

Printed in the United States of America.

C O N T E N T S

1

Getting started with surface analysis

There are no exercises in this lesson.

2

Interpolating surfaces

Exercise 2A: Interpolate surfaces from point data
Estimated time: 30 minutes

Exercise 2B: (Optional) Interpolate a surface and specify a barrier
Estimated time: 15 minutes

Exercise 2A: Interpolate surfaces from point data

Estimated time: 30 minutes

This exercise introduces you to some of the different surface interpolation techniques available in the ArcGIS Spatial Analyst extension.

You are provided with a set of fictitious sample points for snow depth measurements at specific locations throughout your study area, which includes a portion of southwest Lake Tahoe, California. You will use several of the interpolation methods available in ArcGIS Spatial Analyst to create terrain surfaces from the sample point layer and visually compare the results. The sample points are regularly spaced throughout the area and, therefore, any of the interpolation methods may be suitable for creating the surface.

After completing this exercise, you will be able to:

- Create surfaces using a variety of interpolation techniques.
- Evaluate interpolation results.

Step 1: Open a map document and turn on the ArcGIS Spatial Analyst extension

First, you will make a connection to your CASA folder that contains the student data.

☐ Start ArcMap.

You will see the ArcMap – Getting Started dialog box. This dialog box gives you the ability to open a template or assign a default geodatabase for this map document. Each map document has a default geodatabase, which is the home location for the spatial content of your map. There are a few different ways to set this default geodatabase. One is to use this ArcMap – Getting Started dialog box. Later on, you will explore another option by right-clicking any geodatabase in the Catalog window, and setting it as the default geodatabase.

☐ At the bottom of the ArcMap – Getting Started dialog box, click Do not show this dialog in the future in the lower left corner and click OK.

> *Note:* If you would like to see this dialog box when you open every map document, you can go to the Customize menu, ArcMap options, and check the option to Show startup dialog.

By default, the Catalog window is set to auto hide on the right side of the ArcMap interface. You can use the Auto Hide pushpin to toggle auto hide on and off.

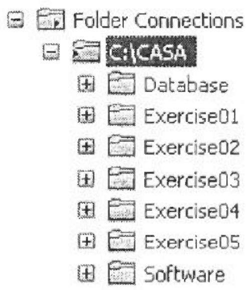
☐ Click the Catalog window tab to open it.

Note: If you close the Catalog window, you can reopen it by clicking the Catalog window button on the Standard toolbar.

☐ In the Catalog window, click the Connect To Folder icon.

☐ In the Connect To Folder dialog box, navigate to ..\CASA and click OK.

The CASA folder displays in the Catalog window.



☐ Expand the Exercise02 folder and double-click Interpolate.mxd.

You see a land/water raster, called arlandmask, for the study area. The red points represent sample points where the depth of snow has been recorded in inches.

Now you will turn on the ArcGIS Spatial Analyst extension.

☐ From the Customize menu, choose Extensions.

☐ In the Extensions dialog box, check the box next to Spatial Analyst, and then click Close.

Step 2: Set the analysis environment

Before you use any tools, you should set the appropriate geoprocessing environment.

☐ From the Geoprocessing menu, choose Environments.

☐ In the Environment Settings dialog box, click the Workspace heading to expand it.

☐ For Current Workspace, click the Browse button and browse to the Database folder. Click Interpolate.gdb and click Add to set it as the current workspace.

☐ Do the same for Scratch Workspace.

☐ Scroll down and expand Raster Analysis.

☐ For Cell Size, choose Same as layer arlandmask.

☐ For Mask, choose arlandmask from the drop-down list.

☐ Click OK to close the Environment Settings dialog box.

Step 3: Interpolate using the IDW method

IDW interpolates by giving greater weight to the sample points closest to the cell whose value is being interpolated. As the distance between input point and cell increases, the significance of the sample decreases.

IDW will not interpolate values beyond the vertical extremes of the provided samples (i.e., it cannot predict ridges or valleys if they are not represented in the sample set). The best results are obtained with dense sampling.

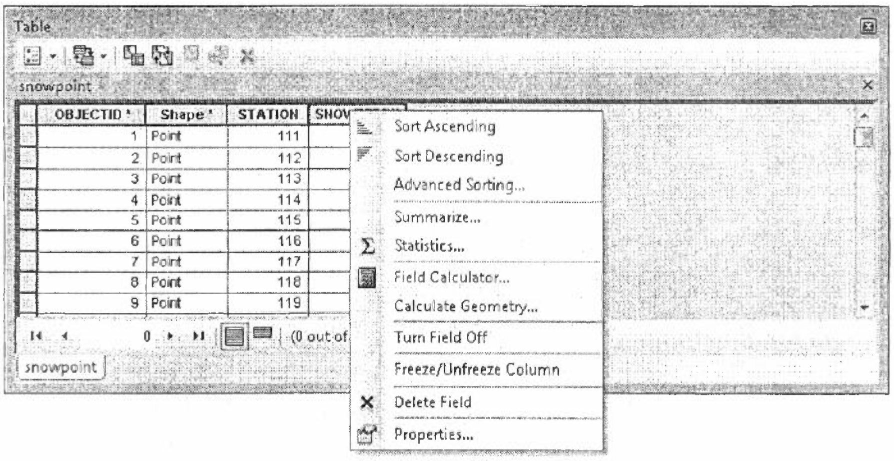
> *Note:* Before using the interpolation tools, you should make sure that your input sample points dataset does not have more than one point in coincidence on any x,y coordinate location. You can use the Delete Identical tool in the Data Management Toolbox > General toolset, if you have an ArcInfo license. As an alternative, you can use the Collect Events tool (in the Spatial Statistics Tools toolbox > Utilities toolset) or the Collect Events with Rendering tool (in the Spatial Statistics Tools toolbox > Rendering toolset) to discover this. Select and delete any duplicate points before using the interpolator. For this exercise, your snowpoint layer is suitable to use.

☐ In the table of contents, right-click the snowpoint layer and choose Open Attribute Table.

The Attribute window behaves like the Catalog window. You can set it to auto hide and you can dock the window to the ArcMap interface.

The SNOWDEPTH field contains measurements of average snow depth for each of the 271 sample points. You will use this field for your interpolation to create a surface of snow depth across the study area.

☐ Right-click the SNOWDEPTH field heading and choose Statistics.



Notice that the snow depth ranges from 0 to 28 inches, and the mean snow depth is about 12 inches.

☐ Close the Statistics of snowpoint window and the Table window.

IDW works best when the sample points capture the surface extremes and the surface is relatively smooth. If the sampling of input points is sparse or very uneven, the results may not represent the actual surface very well.

☐ In the Catalog window, expand Toolboxes, System Toolboxes, and then expand the Spatial Analyst Tools toolbox.

☐ Expand the Interpolation toolset.

☐ Double-click the IDW tool to open its dialog box.

☐ Set its parameters as follows:

- Input point features: snowpoint
- Z value field: SNOWDEPTH
- Output raster: ...\CASA\Database\Interpolate.gdb**idw1**
- Output cell size: 30
- Power: 2
- Search radius: Variable
- Search Radius Settings:
  - Number of points: 12
  - Maximum distance: [Leave blank]
- Input barrier polyline features: [Leave blank]

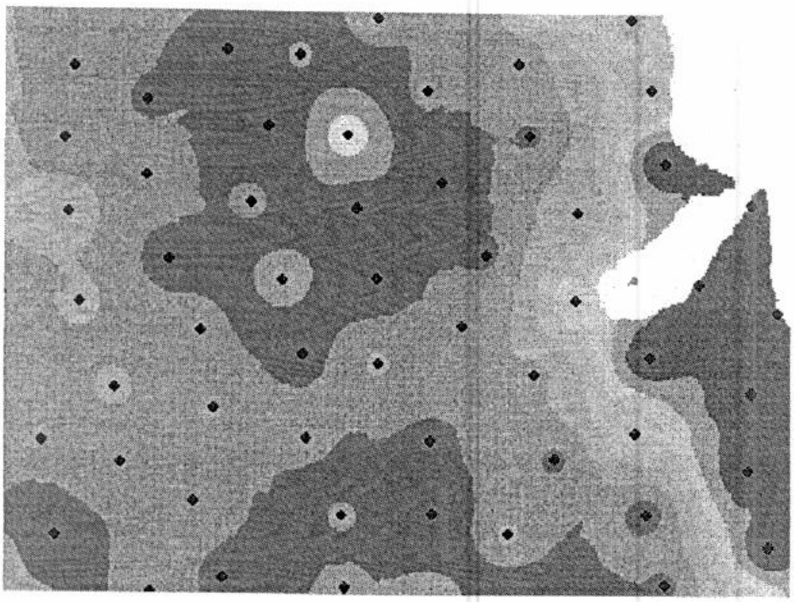
If you would like to find more information about these parameters, feel free to use the Tool help.

☐ Click OK to run the tool.

It may take up to one minute for the tool to run. You can tell that it is running by observing the blue "IDW" line at the bottom of the ArcMap window.

The output raster layer, idw1, is added to ArcMap.

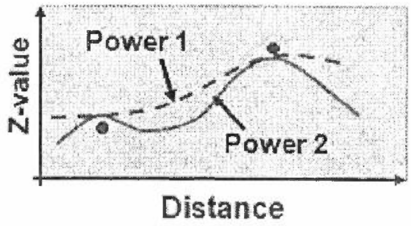
☐ Zoom in on the display to get a closer look at the output.



☐ In the table of contents, examine the legend for idw1.

Note that IDW honors the mask. The maximum and minimum values in the interpolated idw1 surface only occur at the sample points because the interpolator averages the samples, and the surface does not necessarily pass through them. Each point has a dominating influence on the surface in its immediate vicinity.

IDW lets you control the significance of the samples based on their distance from the cell being interpolated. When a higher value for the Power parameter is specified, more emphasis can be put on the nearest points. Specifying a lower value for Power provides more influence to samples that are farther away.



You can set either a variable or a fixed search radius.

Using a variable search radius, you can specify the minimum number of samples to use when calculating the value of the interpolated cell. This makes the search radius variable for each interpolated cell, depending on how far it has to look to find the minimum number of samples.

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 raster. By specifying a minimum number of samples, you can ensure that at least that many samples will be used in the interpolation of each cell.

☐ Open the Layer properties for idw1, click the Source tab, and scroll down to Statistics.

Question 1: What is the significance of the minimum and maximum values?

____

____

____

☐ Close the Layer Properties dialog box.

☐ Collapse the idw1 and arlandmask layers.

☐ Click the Full Extent button.

Step 4: Interpolate using the spline method

The Spline tool uses a minimum-curvature spline interpolation method. Unlike IDW, the spline method can predict hills and valleys, and the surface passes through the samples.

First you will run the Spline tool using the Regularized method. In this method, higher values used for the Weight parameter produce smoother surfaces. The Weight values are typically set between 0 and 0.5. The default is 0.1.

☐ In the Catalog window, double-click the Spline tool (in the Interpolation toolset).

☐ Set its parameters as follows:

- Input point features: snowpoint
- Z value field: SNOWDEPTH
- Output raster: **spline_r**
- Output cell size: 30
- Spline type: REGULARIZED
- Weight: 0.1
- Number of points: 12

☐ Click OK.

☐ Confirm that the snowpoint layer is at the top of the table of contents. If it is not, drag it to the top.

You will compare the spline and IDW results using tools on the Effects toolbar.

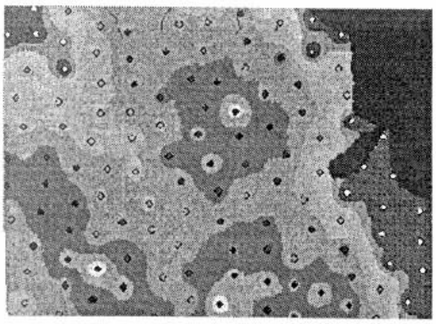
☐ From the Customize menu, choose Toolbars, and click the Effects toolbar to turn it on.

☐ On the Effects toolbar, select the spline_r layer from the Layer drop-down list.

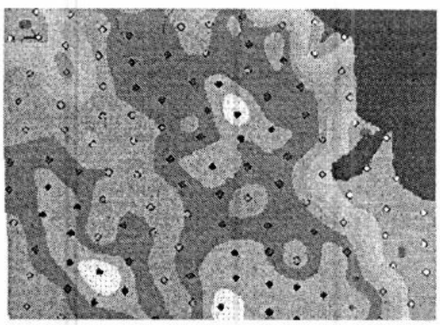
☐ Click the Swipe Layer tool .

☐ Click inside the map and hold the mouse button down. Drag the mouse pointer from top to bottom or from side to side.

The spline_r layer slides back, allowing you to compare it with the idw1 layer underneath.



idw1



spline_r

In the legend for the spline_r layer, notice that the surface values exceed the sample range of 0 to 28 inches; the spline method has predicted hills and valleys. Spline also honors the mask.

☐ Turn off the idw1 layer.

Next, you will use the Tension method. The Tension method tunes the stiffness of the surface according to the character of the modeled phenomenon. It creates a coarser surface with values more closely constrained by the sample data range.

A value of 0 for the Weight parameter results in the smoothest surface, whereas higher Weight values result in a coarser surface. Typical Weight values are 0, 1, 5, and 10.

☐ Double-click the Spline tool and set its parameters as follows:

- Input point features: snowpoint
- Z value field: SNOWDEPTH
- Output raster: **spline_t**
- Output cell size: 30
- Spline type: TENSION
- Weight: 0.1
- Number of points: 12

☐ Click OK.

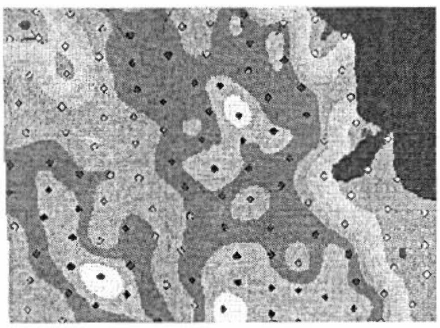
☐ Confirm that the snowpoint layer is at the top of the table of contents.

☐ On the Effects toolbar, set the Layer to spline_t.

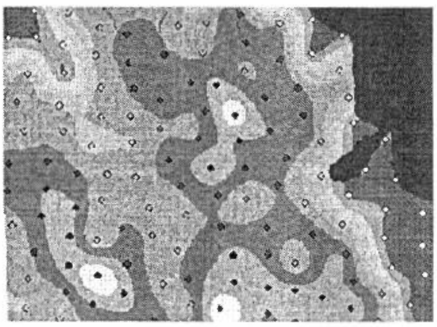
☐ Click the Flicker Layer button.

The spline_t layer flickers on and off, allowing you to compare it with the spline_r layer underneath. In the box next to the Flicker button, the flicker rate is shown in milliseconds.

☐ If you like, increase or decrease the flicker rate.



spline_r



spline_t

☐ Click the Flicker button again to stop the flickering.

Note in the legend that spline_t has a smaller range of values than spline_r. The Tension method created a tighter surface with shorter hills and shallower valleys.

☐ Turn off spline_r and spline_t, and collapse all layers.

Step 5: Interpolate using the natural neighbors method

The natural neighbors interpolation method is similar to IDW. A major difference is how the sample points are selected and weighted as each cell is interpolated. It uses Thiessen polygons to find the natural neighbors for a cell and uses an area-weighted (not distance-weighted) method to weight the samples. Like IDW, the output surface will not exceed the range of the input samples.

This interpolator has an advantage over the others. It can handle an extremely large number of samples that might cause other interpolators to fail.

☐ Double-click the Natural Neighbor tool (in the Interpolation toolset) and set its parameters as follows:

- Input point features: snowpoint
- Z value field: SNOWDEPTH
- Output raster: **natural1**
- Output cell size: 30

☐ Click OK.

☐ Examine the output raster.

The natural neighbors method only interpolates values for the cells that fall within the convex hull of the sample points (that is, the polygon made up by connecting all the outermost points). Also, it does not honor the mask. You will need to clip the output surface to the mask and manually assign symbology to the clipped surface.

☐ Expand the Extraction toolset.

☐ Double-click the Extract by Mask tool and set its parameters as follows:

- Input raster: **natural1**
- Input raster or feature mask data: arlandmask
- Output raster: **natural2**

☐ Click OK.

☐ Open the layer properties for the natural2 layer.

☐ On the Symbology tab under Show, set the renderer to Classified.

☐ In the upper right corner of the dialog box, click Import.

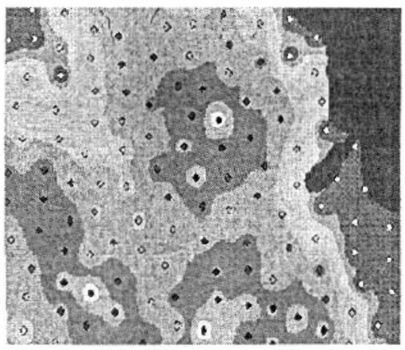
☐ Choose natural1 in the Layer drop-down list and click OK.

☐ Click OK to close the Layer Properties dialog box and apply the new symbology.

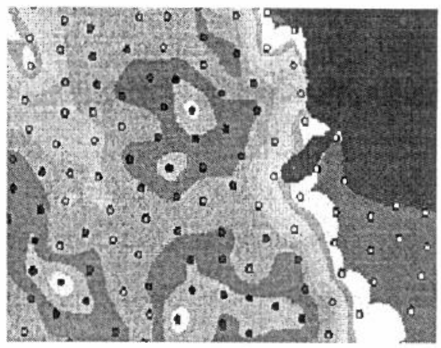
☐ Turn off all layers except snowpoint, natural2, and idw1.

☐ On the Effects toolbar, set the Layer to natural2.

☐ Use either the Swipe tool or Flicker button to compare natural2 with idw1.



idw1



natural2

All the interpolators you have used in this exercise are best applied to surfaces like rainfall, noise levels, and so on, rather than topographic surfaces. The Topo to Raster tool is designed to return high-quality topographic and hydrologically correct surfaces.

☐ Turn off and collapse all layers except snowpoint and arlandmask.

Step 6: Evaluate interpolation results

In this step, you will test the accuracy of your interpolation results.

It is difficult to evaluate the validity of an interpolated surface because there is no correct surface to which you can compare your results. The only information available to you is the input sample set.

One method for validating an interpolated surface is to remove some of the sample points, interpolate the surface using the remaining sample points, and then check to see whether the interpolated surface predicted the value of the sample points you removed.

Organizations that create critically important surfaces, like those performing global warming studies, follow this method exhaustively. That is, if they have 1 million samples, they might run the interpolation 1 million times, excluding each sample in turn, and test the results—which is why these organizations use supercomputers.

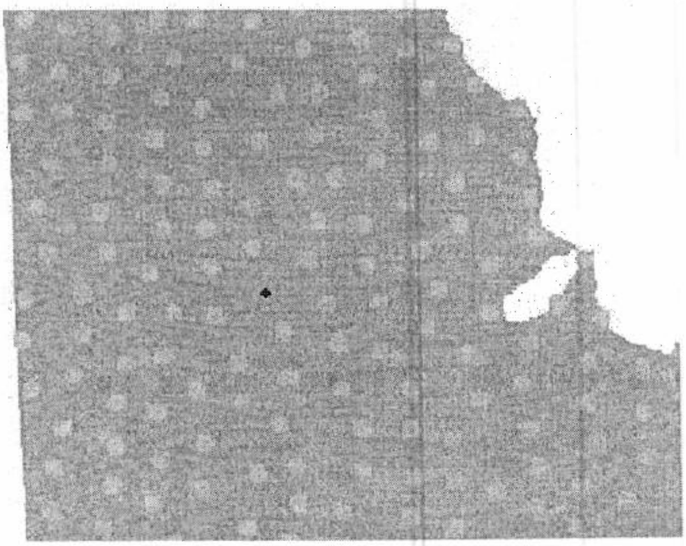
In this step, you will use a simplified version of this method. You will remove one sample, run different interpolators against the remaining samples, and check to see which interpolator came closest to predicting the value of the missing sample.

☐ If necessary, expand the snowpoint layer.

☐ Select all the points in the snowpoint layer by right-clicking the layer and choosing Selection > Select All.

☐ On the Tools toolbar, click the Select Features by Rectangle tool.

☐ Hold the Shift key on your keyboard and click a random point in the middle of the map to deselect it.



Only the selected points will be used by the IDW interpolation tool.

☐ Double-click the IDW tool and set its parameters as follows:

- Input point features: snowpoint
- Z value field: SNOWDEPTH
- Output raster: **idw2**
- Output cell size: 30
- Power: 2
- Search radius: Variable
- Search Radius Settings:
  - Number of points: 12
  - Maximum distance: [Leave blank]
- Input barrier polyline features: [Leave blank]

☐ Click OK.

☐ Zoom in to the deselected sample point.

☐ On the Tools toolbar, click the Identify tool.

☐ Click the deselected point in the map.

☐ In the Identify window, click the Identify from drop-down arrow and choose snowpoint, if it is not already selected.

☐ Note its SNOWDEPTH value, as shown in the Identify window.

Now you will identify the snow depth value of the same location in the idw2 layer.

☐ In the Identify window, choose idw2 from the Identify from drop-down list.

☐ Click the same deselected point in the map and note its Pixel value, as shown in the Identify window.

☐ Repeat the preceding process to compare these values with the results of the spline interpolation.

☐ When you are finished, close the Identify window.

☐ Exit ArcMap without saving changes to your map.

Conclusion

Using default parameters, all the interpolation methods create useful and basically similar surfaces. The differences between them are in the details.

Your choice of an interpolation method should be influenced by your knowledge of the surface you are modeling. You can use your knowledge of reality to evaluate how well the interpolators are doing.

Your choice of method might also be influenced by your understanding of the method. For example, an advanced interpolation method might be more sophisticated than IDW (and create a more accurate surface), but IDW might be preferable if you can understand the results better.

Answers to Exercise 2A Questions

Question 1: What is the significance of the minimum and maximum values?

Answer: The surface values range from 0 to 28 inches—the same range as the snow depth samples. These are the highest and lowest z-values returned by the IDW interpolator. IDW cannot create values greater than or less than those of the samples. That is, IDW cannot predict hills or valleys.

Exercise 2B: (Optional) Interpolate a surface and specify a barrier

Estimated time: 15 minutes

A barrier is a line feature class that limits the interpolator's search for input sample points. A line may represent a cliff, an earthquake fault, or some other interruption in the behavior of a surface. Only those sample points on the same side of the barrier as the current processing cell are used in the interpolation process.

Barriers are time-consuming to process with IDW. You can improve performance somewhat by generalizing the barrier lines to remove extraneous vertices.

You will use terrain ridge lines as barriers in interpolating snow depth.

After completing this exercise, you will be able to:

- Create surfaces using a barrier to restrict calculations used by IDW.

Step 1: Open a map document

☐ If necessary, start ArcMap.

☐ In the Catalog window, navigate to your ..\CASA\Exercise02 folder.

☐ Open the Barriers.mxd.

You see the land/water mask for the study area and the snow depth sample points. A layer showing terrain ridge lines is present but turned off.

☐ Set the environment settings for this exercise as follow:

- Current Workspace: ..\CASA\Database\Interpolate.gdb
- Scratch Workspace: ..\CASA\Database\Interpolate.gdb
- Output Coordinates: Same as Layer Land Mask
- Processing Extent: Same as Layer Land Mask
- Cell Size: Same as Layer Land Mask
- Mask: Land Mask

Step 2: Use IDW without barriers

In this step, you will use the Search window to find and open tools. It is an easy way to find tools if you are not sure in which toolbox or toolset they are located.

First you will run IDW without any barriers.

☐ If you don't have the Search window open, click the Search window button on the Standard toolbar.

Remember, the Search window functions like the Catalog window.

☐ In the Search window, type IDW and click the Search button.

☐ Click IDW (Spatial Analyst) to launch the tool and set its parameters as follows:

- Input point features: Snow Depth
- Z value field: SNOWDEPTH
- Output raster: **idw_nobar**
- Output cell size: 30
- Power: 2
- Search radius: Variable
- Search Radius Settings:
  - Number of points: 12
  - Maximum distance: [Leave blank]
- Input barrier polyline features: [Leave blank]

☐ Click OK.

Step 3: Use IDW with barriers

In this step, you will create another surface representing snow depth, but this time you will include the ridge lines in the analysis.

The ridge lines will serve as barriers that restrict calculations used by IDW. When the ridge lines are used as barriers, they restrict the IDW function from using the depth values from sample points on the other side of the ridge. In a sense, they compartmentalize the IDW process.

☐ Click the IDW (Spatial Analyst) tool again and set its parameters as follows:

- Input point features: Snow Depth
- Z value field: SNOWDEPTH
- Output raster: **idw_bar**
- Output cell size: 30
- Power: 2
- Search radius: Variable
- Search Radius Settings:
  - Number of points: 12
  - Maximum distance: [Leave blank]
- Input barrier polyline features: Ridges

☐ Click OK.

Barriers represent the locations of linear features known to interrupt surface continuity in an interpolated surface. Barrier features do not have z-values, but act as limits on the selected set of the input sample points used to interpolate output z-values. Separation by a barrier is determined by line-of-sight analysis between each pair of points. The barrier ensures that only those samples on the same side of the barrier are used when interpolating a cell value. Cliffs, faults, or embankments are typical examples of barriers incorporated when interpolating.

Step 4: Compare interpolated surfaces

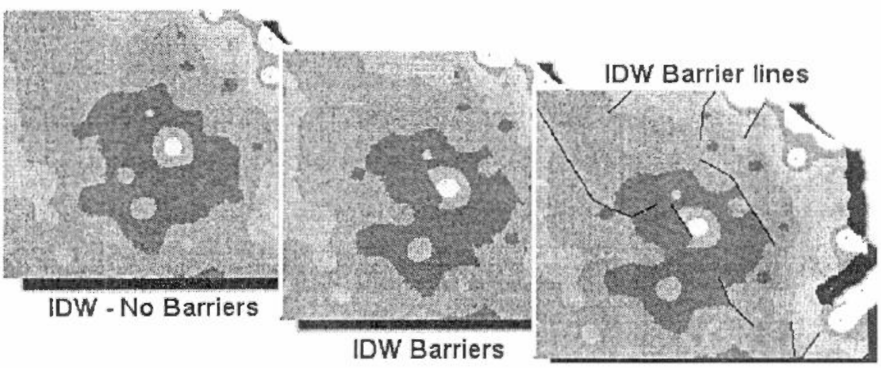
☐ Turn off the Snow Depth layer.

☐ On the Effects toolbar, set the Layer to idw_bar.

☐ Use the Flicker Layer tool on the Effects toolbar to compare the idw_bar layer with idw_nobar.

☐ Turn on the Ridges layer to see the barriers.

☐ Use the Identify tool to query several locations along the barriers to observe how the interpolation is affected by a barrier. Be sure to choose <All Layers> in the drop-down list.



Because a barrier was used, the surface was interpolated from the edge of the extent up to the barriers and then stopped. The same is true of the opposite side of the barriers; interpolation occurred from the edge up to the barriers and then stopped.

☐ Exit ArcMap without saving changes.

Conclusion

You may be able to improve the accuracy of an IDW surface by using line layers as barriers. On elevation surfaces, barriers can represent abrupt changes in elevation, such as cliffs.

3

Introduction to kriging

There are no exercises in this lesson.

4

Calculating density

Exercise 4: Calculate density
Estimated time: 20 minutes

Exercise 4: Calculate density

Estimated time: 20 minutes

Density surfaces are useful for showing concentrations of phenomena, like population. When you calculate population density, you spread the population over the landscape and return a value such as number of people per square mile. ArcGIS Spatial Analyst has three density tools: Line Density, Point Density, and Kernel Density. You will use each of these.

After completing this exercise, you will be able to:

- Calculate the density of line features.
- Calculate the density of point features.
- Calculate the density of point features using a kernel function.

Step 1: Open a map document

☐ If necessary, start ArcMap.

☐ In the Catalog window, navigate to your ..\CASA\Exercise04 folder.

☐ Open Density.mxd.

You see the Tahoe study area with a layer of roads and a layer from the USGS Geographic Names Information System (GNIS).

☐ From the Geoprocessing menu, choose Environments and set the environment settings for this exercise as follows:

- Current Workspace: ..\CASA\Database\Tahoe.gdb
- Scratch Workspace: ..\CASA\Database\Tahoe.gdb
- Output Coordinates: Same as Layer Land
- Processing Extent: Same as Layer Land
- Cell Size: Same as Layer Land
- Mask: Land

Step 2: Calculate line density

First you will calculate line density.

The Line Density tool calculates the density of linear features in the neighborhood of each output cell, like miles of road per square mile. It works by creating a circle of a specified radius around each cell. The length of the portion of each line inside the circle is multiplied by its Population field value. The values are summed and the total is divided by the circle's area to calculate the density.

Note: For the remaining exercises, either use the Search window or navigate to specific tools within the Catalog window to open tools. Remember, you need to double-click a tool in the Catalog window to open its dialog box.

☐ Open the Line Density (Spatial Analyst) tool and set its parameters as follows:

- Input polyline features: Roads
- Population field: NONE
- Output raster: **road_den**
- Output cell size: 30 (*Hint*: Raster Analysis settings)
- Search radius: **800**
- Area units: SQUARE_KILOMETERS

Note: By default, the search radius is equal to the shortest width or height of the extent of the input (Roads) features in the output (road_den) spatial reference divided by 30. The software calculated this to be 752, but you are rounding up to 800.

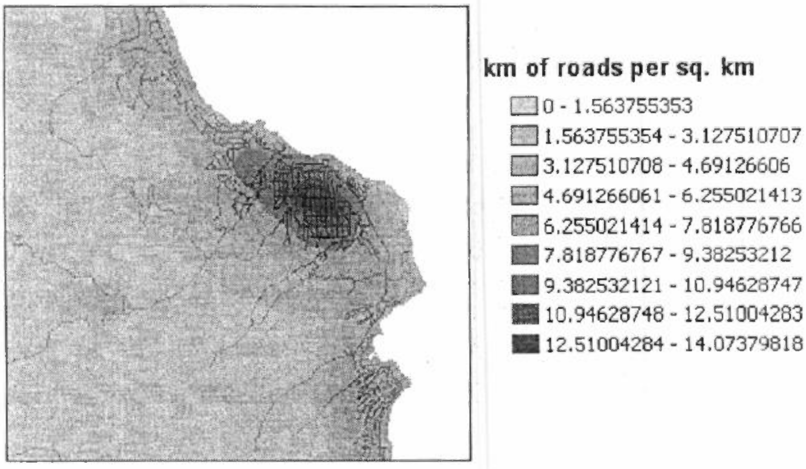
☐ Click OK.

☐ In the table of contents, click the Roads layer and drag it to just below road_den.

☐ Turn off all layers except road_den and Roads.

☐ On the Effects toolbar, set the transparency to 20 percent. (*Hint*: Click the Adjust Transparency button and use the slider.)

☐ Zoom in on the area shown in the following graphic (near the top of the map).



The values in the legend represent kilometers of road per square kilometer. Notice that the values of road_den increase in areas with more roads. Uses for the Line Density tool might include helping to find undisturbed areas for a wildlife habitat model or, conversely, helping to design new roads to reach remote areas. Road density can also be a proxy measurement of population density.

☐ Zoom to the full extent of the map.

☐ Turn off and collapse all layers.

☐ Turn on the USGS GNIS layer.

Step 3: Calculate point density

In this step, you will select populated places from the USGS Geographic Names Information System (GNIS) layer and then create a surface to show the population density in your Lake Tahoe study area. The selection will be honored by the Point Density tool.

☐ From the Selection menu, choose Select By Attributes.

☐ In the Select By Attributes dialog box, create the expression **"GNIS_TYPE" = 'populated place'** by doing the following:

- Double-click "GNIS_TYPE" in the list of attributes.
- Click the equals operator = .
- Click Get Unique Values, then double-click 'populated place' in the list of values.

☐ Click OK.

The status bar on the bottom left of the ArcMap interface indicates that 22 points are selected.

Point density is calculated by finding the number of points inside the search radius for a cell and then dividing that number by the area of the circle. If a population field is specified, the attribute values inside the circle are summed instead of the number of points. The result is the value density for each cell, like people per square mile.

☐ Open the Point Density tool in the Density toolset and set its parameters as follows:

- Input point features: USGS GNIS
- Population field: Capacity
- Output raster: **gnis_den**
- Output cell size: 30
- Neighborhood: Circle
- Neighborhood Settings:
  - Radius: **1000**
  - Units: Map
- Area units: SQUARE_KILOMETERS

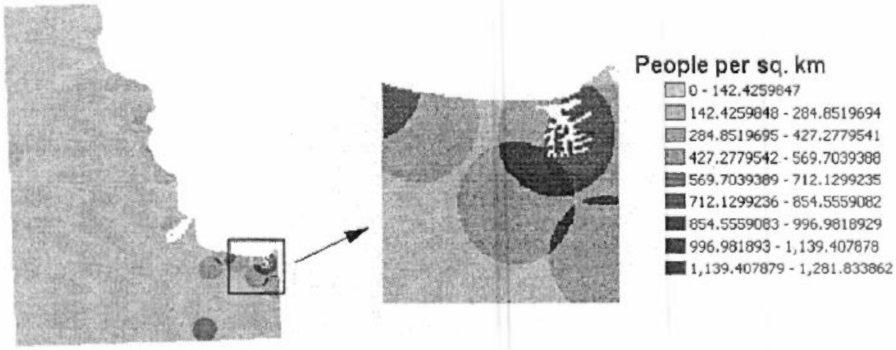
☐ Click OK.

☐ Open the gnis_den layer properties dialog box.

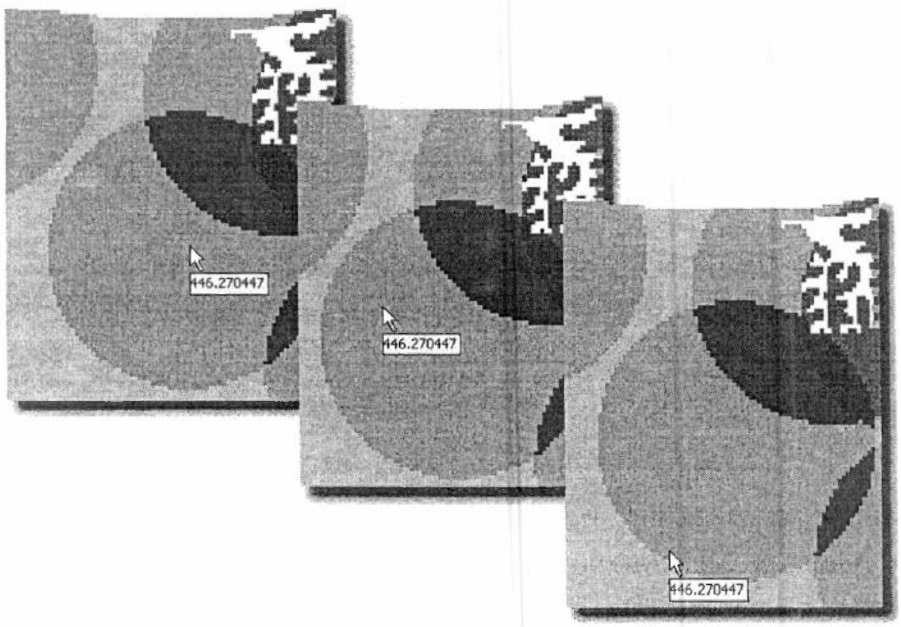
☐ On the Display tab, check Show Map Tips.

☐ Click OK to close the layer properties.

☐ Turn off the USGS GNIS layer and zoom in on the area of the map shown in the following graphic.



☐ Use the Select Elements tool and move your mouse pointer inside the density areas and note the cell values on the Map Tip.



Notice that within each of the overlapping shaded areas, all the cell values are exactly the same. This is an important characteristic of both the Point Density and the Line Density tools: they assign the same density values to all the cells in the intersecting areas defined by the overlapping circles that are constructed around the features by the search radius.

In the next step, you will use the Kernel Density tool, which is similar to the other two density tools but which uses a smoothing technique to distribute values.

Step 4: Calculate kernel density

The Kernel Density tool is similar to Point Density and Line Density; it accepts both point and line inputs. Unlike them, its density values decrease as distance from the features increases.

For points, the Kernel Density tool fits a smoothly curved surface over each input point whose shape is the circle defined by the search radius (like a buffer). The surface is built so that the volume under it is equal to the Population field value for the point (or 1 if no population field is set). The value of the surface is highest at the point location and diminishes with distance to zero at the limit of the search radius. The density of an output cell is calculated by adding the values of all the kernel surfaces where they overlay the cell center.

For lines, the Kernel Density tool fits the same smooth surface over each input line, but it is defined so that its volume is equal to the line length multiplied by the line's Population field value. Again, the overlapping kernel surfaces are added to calculate the densities of the output cells.

Now, you will run the Kernel Density tool with the GNIS data and compare the results with those created using the Point Density tool.

☐ Open the Kernel Density tool (in the Density toolset) and set its parameters as follows:

- Input point or polyline features: USGS GNIS
- Population field: Capacity
- Output raster: **gnis_kern**
- Output cell size: 30
- Search radius: **1000**
- Area units: SQUARE_KILOMETERS

☐ Click OK.

☐ Enable Map Tips for the gnis_kern layer.

☐ If necessary, zoom in on the same area of the map that you examined after running the Point Density tool.

☐ On the Effects toolbar, set the Layer to gnis_kern.

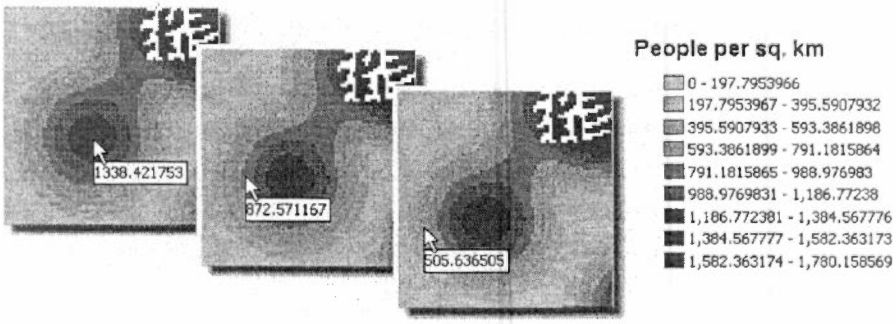
☐ Use either the Swipe Layer tool or the Flicker Layer button to compare it with gnis_den.

You should see a distinct difference between the gnis_den and gnis_kern layers, where gnis_den looks like a set of overlapping circles and gnis_kern looks more like contours.

☐ Click the Flicker Layer button again to turn it off.

☐ Turn off the gnis_den layer.

☐ Move your mouse pointer over the gnis_kern layer and note the values shown in the Map Tips.



In the gnis_kern surface, the density values decrease with distance from the points.

☐ Exit ArcMap without saving changes.

Conclusion

Density surfaces reveal patterns in the data that might not otherwise be evident. Even though a larger search radius tends to generalize the data, you might need to increase the size of the search neighborhood in order to find meaningful patterns. The kernel method of density analysis creates a smoother-looking density surface than the point method.

5

Analyzing surfaces

Exercise 5A: Create hillshades
Estimated time: 20 minutes

Exercise 5B: Calculate slope and aspect
Estimated time: 15 minutes

Exercise 5C: (Optional) Create contours
Estimated time: 15 minutes

Exercise 5D: (Optional) Calculate visibility and viewshed
Estimated time: 20 minutes

Exercise 5A: Create hillshades

Estimated time: 20 minutes

In this exercise, you will use the ArcGIS Spatial Analyst surface analysis tools to explore cartographic uses of hillshades.

After completing this exercise, you will be able to:

- Create a hillshade.
- Create shaded relief.
- Create a soft hillshade.

Step 1: Open a map document

☐ If necessary, start ArcMap.

☐ In the Catalog window, navigate to your ..\CASA\Exercise05 folder.

☐ Open Hillshade.mxd.

You see a raster elevation surface. Another raster of land and water areas is present in the table of contents but is currently turned off.

Confirm that the environment settings for this exercise have been set as follows:

- Current Workspace: ..\CASA\Database\Exercise5.gdb
- Scratch Workspace: ..\CASA\Database\Exercise5.gdb
- Output Coordinate System: Same as Layer Elevation
- Processing Extent: Same as Layer Elevation
- Cell Size: Same as Layer Elevation
- Mask: None

> *Note:* If you examine the environment settings, you will see the actual values for the extent, coordinate system, and cell size, rather than the layer on which they are based.

Step 2: Create a standard hillshade

In this step, you will create a standard hillshade for comparison with the other hillshades that you will create. You will create two hillshades using the Hillshade tool—one with the sun in the northwest and the other with the sun in the southeast—to see the effect the shadows have on your perception of the surface.

☐ In the table of contents, expand the Elevation layer.

The map of the raw elevation surface does not provide much topographical information. The gray values indicate high areas (white) and low areas (black), but most of the surface details are hidden. A hillshade can reveal the hidden details.

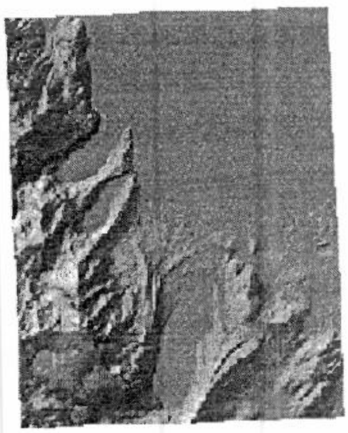
First, you will create a standard hillshade using the default values for azimuth and altitude.

☐ Open the Hillshade tool (Spatial Analyst) and set its parameters as follows:

- Input raster: Elevation
- Output raster: **hill_std**
- Azimuth: 315
- Altitude: 45
- Model shadows: [Leave unchecked]
- Z factor: [Accept the default value]

☐ Click OK.

Elevation



hill_std

As you see, it is much easier to identify mountains and valleys from a hillshade than it is from a raw elevation surface.

The Hillshade tool works by comparing the orientation of a cell (its slope and aspect) with the location of the light source, which is set with the azimuth and altitude. Cells that directly face the light source are fully illuminated and are given a grayscale value of 255 (white). Cells that face directly away from the light are given a grayscale value of 0 (black). Other cells are given intermediate shades of gray.

The azimuth (compass heading) and altitude (degrees above the horizon) control the location of the sun. Changing the azimuth exposes different cells, whereas changing the altitude makes shadows longer or shorter.

For cartographic applications, you should always position the sun in the north so that the shadows in the hillshade are cast at the bottom of the mountains. If the sun is in the south, where it normally belongs, the shadows will be cast at the top of the mountains, and most people will perceive the mountains as being depressions.

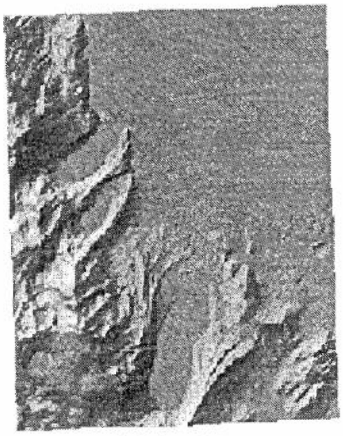
Now you will run the Hillshade tool again with the sun in the southeast so you can see this effect.

☐ Open the Hillshade tool again and set its parameters as follows:

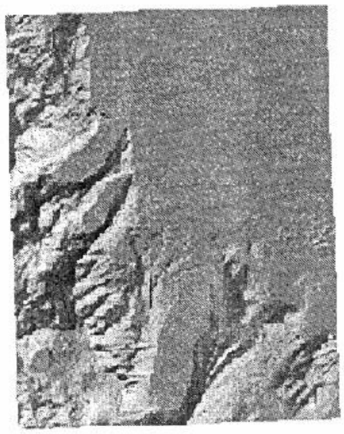
- Input raster: Elevation
- Output raster: **hill_depr**
- Azimuth: **135**
- Altitude: 45
- Model shadows: [Leave unchecked]
- Z factor: **1**

☐ Click OK.

☐ Turn hill_depr on and off to see the effect.



hill_std



hill_depr

In hill_depr, most people will see the lakes as plateaus and the mountains as valleys.

☐ Turn off and collapse all layers.

Step 3: Create a shaded relief map with a hillshade

Combining a hillshade with a transparent layer of colored thematic data, like vegetation, produces a very attractive cartographic product, and it is easy to do. The hillshade provides a sense of dimension to the map and allows you to visualize the terrain under the data. In this step, you will create a shaded relief map.

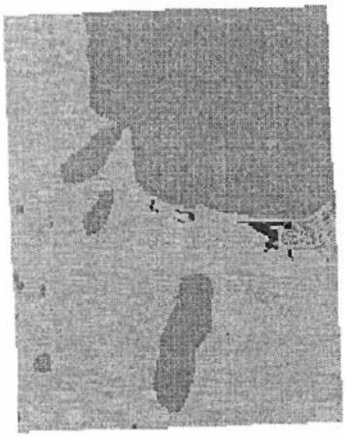
☐ Move LandWater to the top of the table of contents.

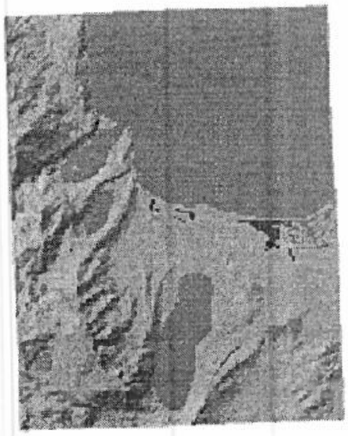
☐ Turn on the LandWater and hill_std layers.

The LandWater layer is symbolized simply, with water in blue, land in brown, and marshes in green. By itself it is not very impressive.

☐ Open the Image Analysis window and click the LandWater layer.

☐ In the Reset Transparency box, type **50** and press Enter.


LandWater


LandWater and hill_std

The hillshade is now partially visible through the LandWater layer and gives the map a sense of dimension.

You can draw any transparent thematic layer over a hillshade, like vegetation, land use, roads and streams, forest fires, and so on. You can also use multiple layers with different transparencies.

☐ Close the Image Analysis window and turn off all layers.

Step 4: Create a softer hillshade

For some cartographic applications, you want the hillshade to be softer so it does not intrude on the thematic layers you draw over it. The Swiss style of hillshading does just that; it washes out detail in the surface, and it simulates aerial perspective by making the higher elevations lighter and the lower elevations darker.

With the Swiss style method, you simply create a standard hillshade, smooth it using a neighborhood function, weight the standard hillshade with the elevation, and combine the smoothed and weighted hillshades with transparency.

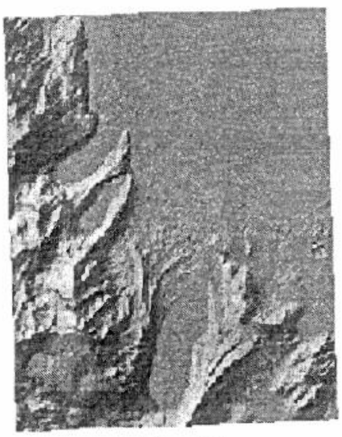
First, you will smooth your standard hillshade using the Focal Statistics tool and the median statistic type.

☐ Open the Focal Statistics (Spatial Analyst) tool and set its parameters as follows:

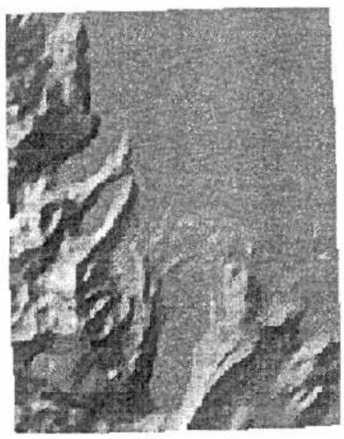
- Input raster: hill_std
- Output raster: **hill_med**
- Neighborhood: Circle
- Neighborhood Settings:
  - Radius: 4
  - Units: Cell
- Statistics type: MEDIAN
- Ignore NoData in calculations: Checked

☐ Click OK.

☐ Use either the Swipe tool or the Flicker Layer button to compare hill_med with hill_std.



hill_std



hill_med

The smoothed hillshade appears blurry because the Focal Statistics tool eliminated much of its detail.

☐ Turn off the hill_med layer.

The next step is to weight the standard hillshade with the elevation raster in order to emphasize the higher elevations. You will do this by dividing the elevation by five and adding the result to the standard hillshade.

First you will divide the values in the standard hillshade by five.

☐ Open the Divide tool (Spatial Analyst) and set its parameters as follows:

- Input raster or constant value 1: **Elevation**
- Input raster or constant value 2: **5**
- Output raster: **Elev_div**

☐ Click OK.

Now you will add the result to the standard hillshade.

☐ Open the Plus tool (Spatial Analyst) and set its parameters as follows:

- Input raster or constant value 1: **Elev_div**
- Input raster or constant value 2: **hill_med**
- Output raster: **hill_wt**

☐ Click OK.

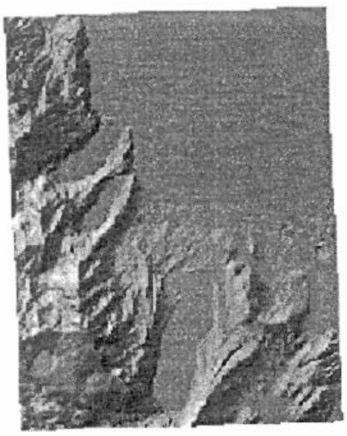
☐ Import the raster symbology from hill_std to hill_wt.

Hint: Open the hill_wt layer properties.

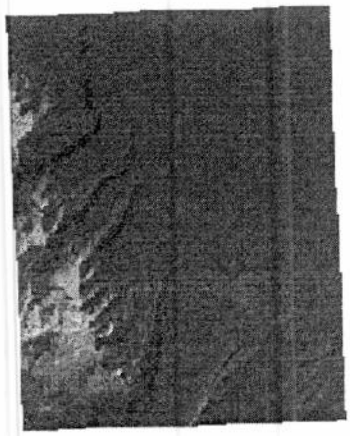
☐ Turn off and collapse the Elev_div layer.

☐ In the Image Analysis window, click hill_wt.

☐ Use either the Swipe tool or the Flicker Layer button to compare hill_wt with hill_std.



hill_std



hill_wt

In hill_wt, notice that the higher elevations are brighter than the lower elevations. This will provide the aerial perspective effect in the final map product.

To get the full effect of the Swiss style, you will combine the hill_med and hill_wt layers with a thematic layer using transparency. The drawing order is important.

☐ Turn off all layers except hill_wt, hill_med, and LandWater.

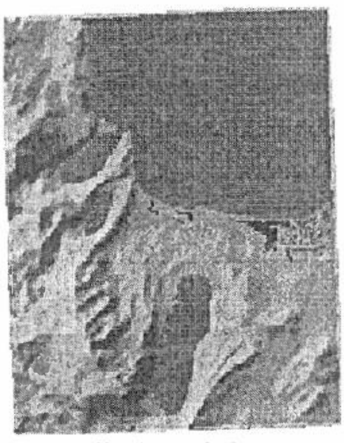
☐ Reorder the layers in the table of contents and set their transparencies as follows:

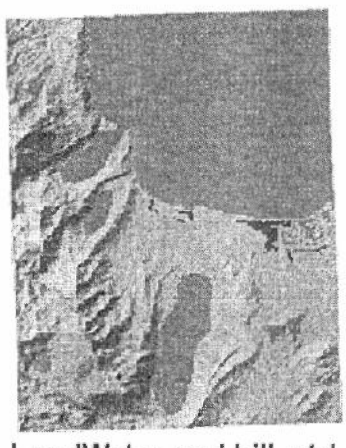
Layer/Order	Transparency
1. LandWater	55%
2. hill_med	35%
3. hill_wt	0%

The map now shows the full effect of the Swiss hillshading style. You can compare it with the standard hillshade.

☐ Move hill_std to just below LandWater in the table of contents and turn it on.

☐ Use either the Swipe tool or the Flicker Layer button to compare hill_std with the Swiss style of hillshading.


Swiss style


LandWater and hill_std

In the Swiss style, the terrain is less detailed and the higher elevations are brighter. It is possible to adjust the appearance by modifying the transparency, brightness, and contrast of the three layers.

Note: You can change the layer transparency, brightness, and contrast properties by using the respective tools in the Image Analysis window.

You could smooth the surface more by running the Focal Statistics tool several times on the original hillshade (or on the elevation surface before you create the original hillshade), using the output of each as the input to the next.

The benefits of this cartographic hillshading technique are best realized when printed on a high-quality printer at their intended scale, which for this data is 1:63,360 (the scale normally used for a 15-minute USGS quadrangle).

☐ Exit ArcMap without saving changes.

Conclusion

This exercise showed how you can create hillshades from an elevation raster to achieve a 3D effect. The simulated shadow effect is created by assigning an illumination value from 0 to 255 to each cell according to a specified azimuth and altitude for the sun. Azimuth is the angular direction of the sun (its compass direction). Valid values range from 0 to 360. The default is 315 degrees (northwest). Altitude is the angle of the sun above the horizon. Valid values range from 0 to 90. The default is 45 degrees. You can create shaded relief maps by displaying the original elevation raster transparently with the hillshade raster.

Exercise 5B: Calculate slope and aspect

Estimated time: 15 minutes

Slope and aspect are both measures of the shape, or morphology, of a surface. Slope is the change in z over distance (or steepness) through a cell, and aspect is the direction of the steepest path through the cell relative to north (or the orientation of the cell). In this exercise, you will compute both slope and aspect.

After completing this exercise, you will be able to:

- Calculate slope.
- Calculate aspect.
- Reclassify slope into zones.

Step 1: Open a map document

☐ If necessary, start ArcMap.

☐ Navigate to your ..\CASA\Exercise05 folder and open Surface.mxd.

Confirm that the environment settings for this exercise have been set as follows:

- Current Workspace: ..\CASA\Database\Exercise5.gdb
- Scratch Workspace: ..\CASA\Database\Exercise5.gdb
- Output Coordinate System: Same as Layer Elevation
- Processing Extent: Same as Layer Elevation
- Cell Size: Same as Layer Elevation
- Mask: None

> *Note:* If you examine the environment settings, you will see the actual values for the extent, coordinate system, and cell size, rather than the layer on which they are based.

Step 2: Calculate slope as a percent

Slope is the steepness of a cell. It is measured as the change in z-value over distance for the nine-cell neighborhood around each cell (a plane is fitted to the neighborhood). Slope is typically used with elevation data, but may be useful in analyzing other types of surfaces as well. For example, with a surface of rainfall, the slope reflects the rate of change in the rainfall (that is, steeper slopes are changing faster).

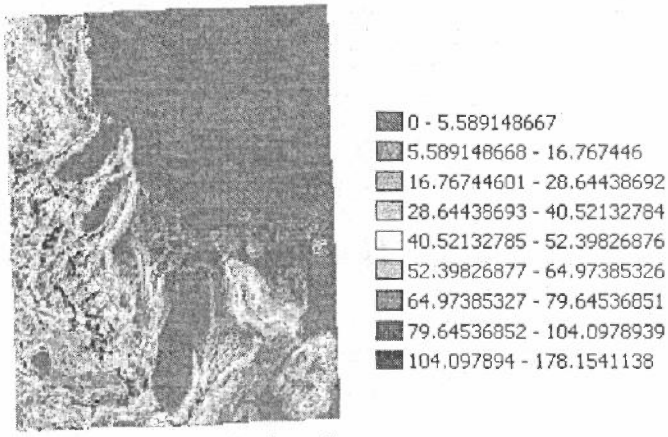
Slope may be expressed either as degrees or as a percentage of rise over run. Degrees are commonly used in scientific applications, whereas percentages are commonly used in transportation studies (for example, a sign that reads "Caution: 6% grade ahead").

In this step, you will run the Slope tool to return percent slope.

☐ Open the Slope tool (Spatial Analyst) and set its parameters as follows:

- Input raster: Elevation
- Output raster: **slp_per**
- Output measurement: PERCENT_RISE
- Z factor: **1**

☐ Click OK.



slp_per (percent slope)

In the legend for slp_per, notice that the values range from 0 to about 178 percent. Percent slope is computed as the rise divided by the run, so if there is a large rise over a very short run, the values can approach infinity.

The Slope tool automatically assigns an appropriate renderer to the output layer. It uses a Classified renderer with nine classes and the Natural Breaks (Jenks) method and assigns the special green-to-red Slope color ramp to the classes.

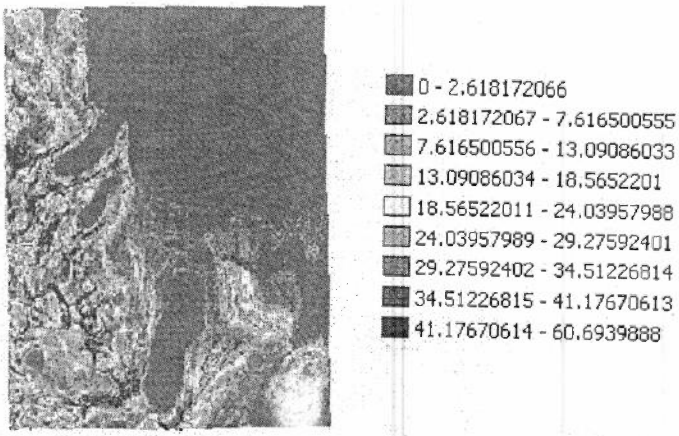
Step 3: Calculate slope in degrees

Now you will run the Slope tool to return degrees slope.

☐ Open the Slope tool and set its parameters as follows:

- Input raster: Elevation
- Output raster: **slp_deg**
- Output measurement: DEGREE
- Z factor: **1**

☐ Click OK.



slp_deg (degrees slope)

Notice in the legend for slp_deg that the values range from 0 to about 61 degrees. The highest possible slope is 90 degrees: a vertical cliff.

In a ski resort model scenario, you could assume that the best ski runs are those with slopes between 15 and 45 degrees. Less than 15 degrees is too flat, while those over 45 degrees are too dangerous. You will reclassify the slope values into ranges that reflect their ski slope suitability when you build the ski resort model.

Step 4: Reclassify slope into zones

You have probably heard people talk about conquering a mountain or beating a steep slope. But just how steep is steep? In this step, you will rate the terrain around Tahoe as to how difficult it is for hiking or skiing. The ratings you will use are summarized in the following table.

Slope	Description
< 15 degrees	Flat terrain
15–45 degrees	Prime terrain
45+ degrees	Dangerous terrain

You will use the Reclassify tool to reclassify the slp_deg raster into these three classes.

☐ Open the Reclassify (Spatial Analyst) tool and set its parameters as follows:

- Input raster: slp_deg
- Reclass field: Value
- Output raster: **Rating**

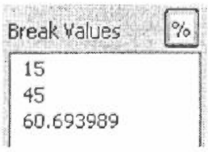
☐ Under Reclassification, click Classify to display the Classification dialog box.

☐ For the classification Method, make sure Natural Breaks (Jenks) is chosen. This is the default.

☐ For Classes, choose 3.

☐ In the Break Values box, click the first value to make it editable, and then type **15**.

☐ Change the second break value to **45**. Do not change the last break value.



You can see how many elements are in each class by clicking each of the break values. The number of elements is reported below the histogram on the Classification dialog box.

Question 1: Which class is the largest?

☐ Click OK to close the Classification dialog box.

☐ Click OK to run the Reclassify tool.

☐ Open the Rating layer properties and click the Symbology tab.

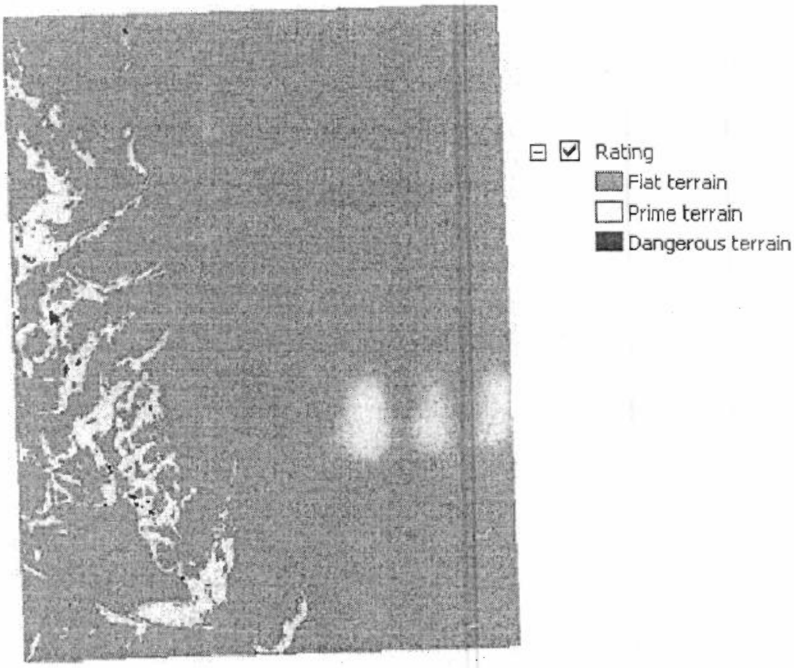
☐ Under Show, click Classified.

☐ Right-click the Color Ramp field and click Graphic View to uncheck it.

☐ Choose the Slope color scheme from the drop-down list.

☐ In the Label column, label the symbols using the descriptions given in the table at the beginning of this step.

☐ Click OK on the Layer Properties dialog box.



Step 5: Calculate aspect

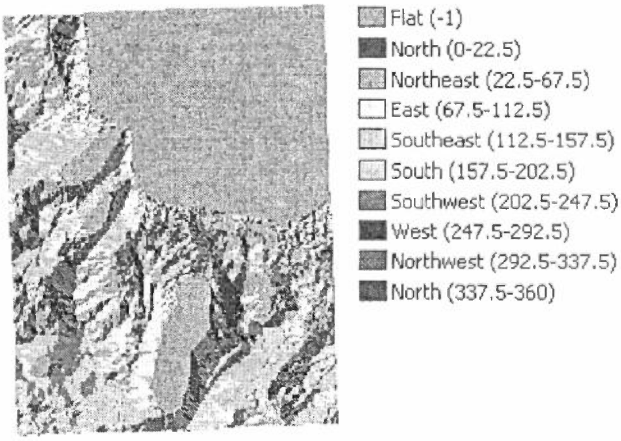
Aspect is the orientation of a cell relative to north. It works by finding the downslope direction of the maximum rate of change in z-value from a cell to its neighbors. In other words, aspect is the slope direction. Internally, the same algorithm is used by both the Slope and the Aspect tools—they merely return different measures of the gradient. Aspect is output as compass degrees from 0 to 360, with -1 reserved for flat cells.

Now you will run the Aspect tool.

☐ Open the Aspect (Spatial Analyst) tool and set its parameters as follows:

- Input raster: Elevation
- Output raster: **asp**

☐ Click OK.



asp

Like the Slope tool, Aspect assigns appropriate symbology to the output layer.

In the context of a ski model, you could have used the Aspect tool to find slopes with low sun exposure by assuming that north-facing slopes get less sun than south slopes, but the hillshade method you used returns a more reliable measure. Regardless, the Aspect tool is useful in many terrain analysis applications.

Using the Aspect tool in some types of analysis is a challenge because of the nature of the angular measurement; an aspect of 359 degrees is very close to an aspect of 1 degree (they are only 3 degrees apart), but mathematically they are 358 degrees apart.

☐ Exit ArcMap without saving changes.

Conclusion

You have just learned how to calculate slope and aspect using the Slope tool and the Aspect tool. Slope measures the incline, or steepness, of a surface and can be expressed in either degrees or percent slope. Aspect is the compass direction in which a topographic slope faces, usually expressed in terms of degrees from the north.

Answers to Exercise 5B Questions

Question 1: Which class is the largest?

Answer: The first class (less than 15% slope)

Exercise 5C: (Optional) Create contours

Estimated time: 15 minutes

Contours are lines that connect points of equal elevation or, more generally, points of equal z-values for data like rainfall, air pollution, or noise. Contours are best used in maps as indicators of underlying topography, and very little analysis can be performed with them. In this exercise, you will create contours from the Elevation layer.

After completing this exercise, you will be able to:

- Create intermediate and index contours.
- Symbolize contours.
- Create contours for specific values.

Step 1: Open a map document

☐ If necessary, start ArcMap.

☐ Navigate to your ..\CASA\Exercise05 folder and open Contours.mxd.

Confirm that the environment settings for this exercise have been set as follows:

- Current Workspace: ..\CASA\Database\Exercise5.gdb
- Scratch Workspace: ..\CASA\Database\Exercise5.gdb
- Output Coordinate System: Same as Layer Elevation
- Extent: Same as Layer Elevation
- Cell Size: Same as Layer Elevation
- Mask: None

Note: If you examine the environment settings, you will see the actual values for the extent, coordinate system, and cell size, rather than the layer on which they are based.

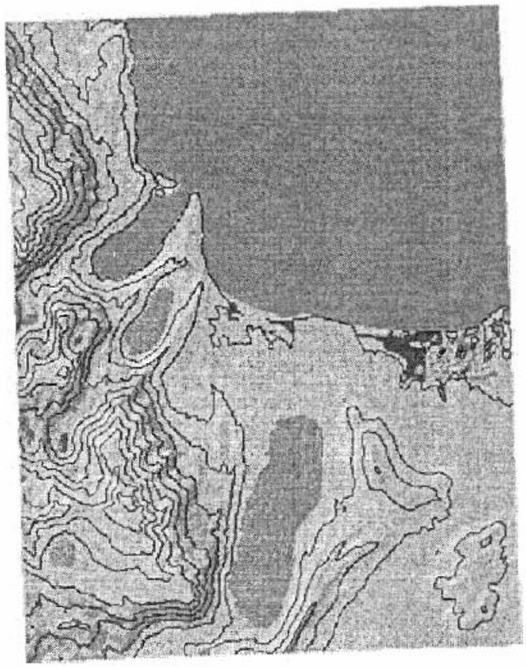
Step 2: Create intermediate contours

Now you will run the Contour tool. It returns the contours as lines in a feature class.

☐ Open the Contour (Spatial Analyst) tool and set its parameters as follows:

- Input raster: Elevation
- Output polyline features: **Conlines100**
- Contour interval: **100**
- Base contour: 0
- Z factor: 1

☐ Click OK.



The Contour tool produces engineering-quality contours. The lines can occasionally cross, appear to intersect, or form unclosed branching lines. These are not errors, just an exact interpretation of the raster surface model. The lines can be edited for cartographic or aesthetic purposes.

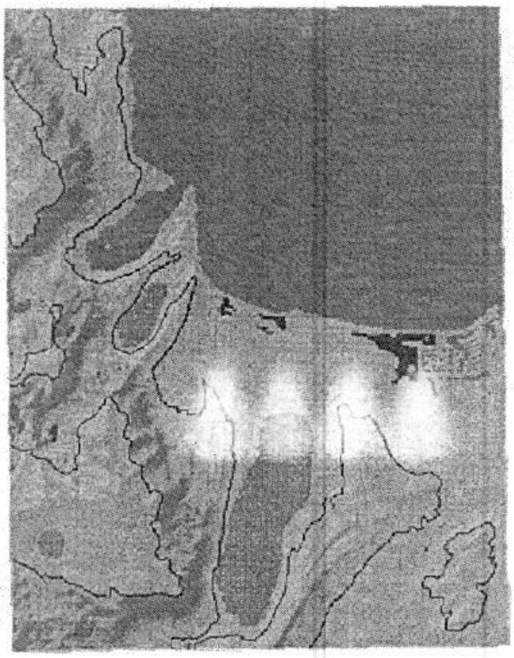
Contours can sometimes be created that appear to follow the raster cell boundaries. This can occur when the raster z-values are integers and they fall exactly on a contour. This is not a bug, merely an exact contouring of the data. There are three ways to overcome this condition if you would like smoother contours: smooth the surface before contouring, adjust the base contour, or both.

Step 3: Create index contours

It is common cartographic practice to make every fourth or fifth line noticeably thicker than the other contours. These thick contours are called *index contours* and should be labeled with the elevations they represent. The three or four contours that fall between the index contours are called *intermediate contours*.

☐ On your own, use the Contour tool to create additional contours, but this time specify a contour interval of **500**. Save the file as **Conlines500**.

☐ When the process completes, turn off the Conlines100 layer to better see the new contours.



Step 4: Symbolize the contours

The output line features have a CONTOUR attribute that may be used to label the contour lines. Standard ArcMap label properties may be set so that the labels follow the lines and have masks to make them more visible.

☐ Turn the Conlines100 layer back on.

☐ Open the Conlines100 layer properties.

☐ On the General tab, change the layer name to **Intermediate Contours**.

☐ On the Symbology tab, click the symbol in the Symbol area to display the Symbol Selector.

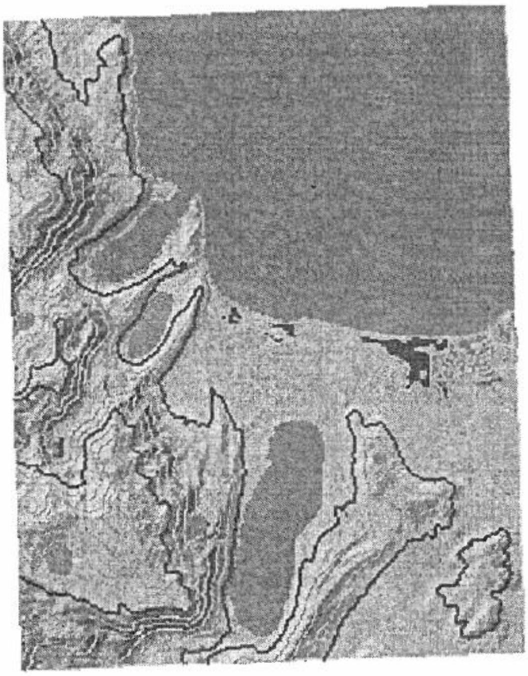
☐ In the Symbol Selector, type **contour intermediate** and click the Search button.

☐ In the category ESRI, select the Contour, Topographic, Intermediate line symbol, which is the first one.

☐ Click OK on the Symbol Selector and on the Layer Properties dialog box.

☐ Repeat this process to change the name of the Conlines500 layer to **Index Contours** and symbolize the contours using the Contour, Topographic, Index line symbol.

Hint: Search for "contour index."



Step 5: Create contours using a list

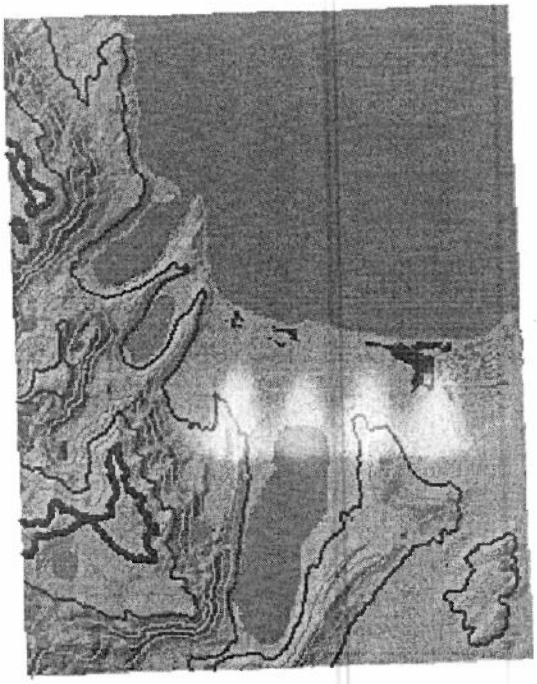
Sometimes you may be interested in specific isolines that represent a constant value (for example, sea level or a particular precipitation value). Suppose that you want to see where Tahoe's approximate tree line (2,743 meters) is located. You will use the Contour List tool to find out.

☐ Open the Contour List (Spatial Analyst) tool and set its parameters as follows:

- Input raster: Elevation
- Output polyline features: **TreeLine**
- Contour values: Type **2743** and click the Add button ✚ to add the value to the contour list

☐ Click OK to run the Contour List tool.

☐ Symbolize the TreeLine layer using a dark green line with a width of 3.



You see several contours representing the approximate level of the tree line.

☐ Exit ArcMap without saving changes.

Conclusion

This exercise showed you how to create isolines from a surface. Isolines are lines connecting points of equal value. In this case, you created contours, a type of isoline connecting points of equal elevation. You create contours by specifying a contour interval and a base contour.

The contour interval is the change in vertical measurement for the contour lines. The base contour is the starting location from which all contours will be generated. You should choose an interval for the contour lines that is small enough to give some definition to the surface, but not so small that the lines become too close together and the map becomes difficult to read.

You can label contour lines with their values to make it easier to see the values, as well as patterns. Using a bold line for every fourth or fifth interval makes the values easier to read. For example, if the interval is 10 meters, you could use bold lines for values of 50, 100, and 150 meters.

Exercise 5D: (Optional) Calculate visibility and viewshed

Estimated time: 20 minutes

Viewshed analysis examines every cell to determine if it has a clear line of sight to one or more observation points, based on whether there are high intervening cells that block the line of sight between the cell and the observer. It returns a simple count of the number of observers that can see the cell. There are many potential applications for viewshed analysis in GIS, like hiding unsightly oil storage tank farms, determining lines of fire for a military fortification, or siting cellular phone antennas.

After completing this exercise, you will be able to:

- Determine the raster surface locations visible to a set of observer features.
- Determine the visual exposure of multiple observer locations.

Step 1: Open a map document

☐ If necessary, start ArcMap.

☐ Navigate to your ..\CASA\Exercise05 folder and open Viewshed.mxd.

You see the following layers:

- ParkFacilities: The observation points
- LandMask: A mask; land has a value, and Lake Tahoe is NoData
- TopoMap: A scanned topographic map for background
- erhill: A hillshade
- Elevation: A raster elevation surface

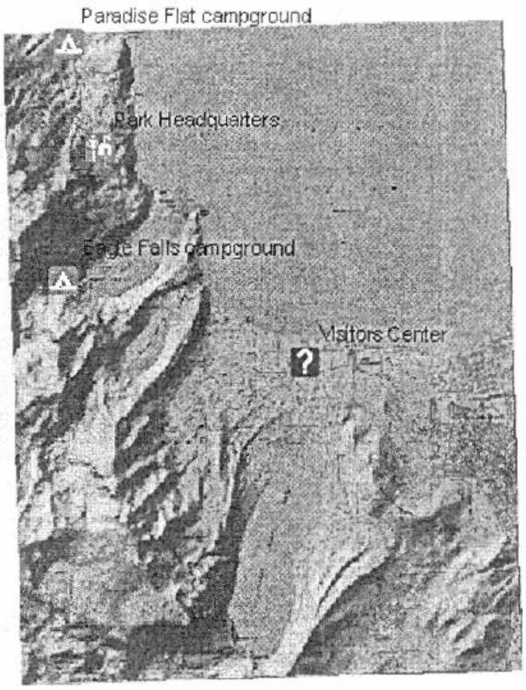
Confirm that the environment settings for this exercise have been set as follows:

- Current Workspace: ..\CASA\Database\Exercise5.gdb
- Scratch Workspace: ..\CASA\Database\Exercise5.gdb
- Output Coordinate System: Same as Layer Elevation
- Extent: Same as Layer Elevation
- Cell Size: Same as Layer Elevation
- Mask: None

> *Note:* If you examine the environment settings, you will see the actual values for the extent, coordinate system, and cell size, rather than the layer on which they are based.

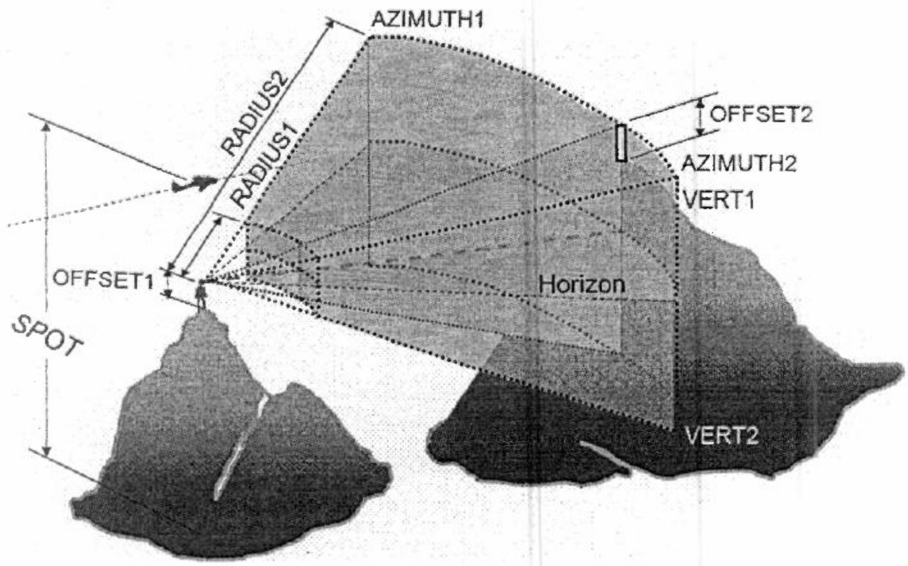
Step 2: Examine an attribute table

In this scenario, the National Park Service has problems with radio communications among its various facilities and it needs to build a repeater tower. The tower site must provide a view of all the facilities.



Each facility has a radio mast. The headquarters mast is 100 feet tall, and all the others are 50 feet tall. The repeater tower will be 100 feet tall.

The Viewshed tool is easy to use and has few parameters to set. This is because most of the parameters are carried as attributes of the input observers, which may be either points or lines in a feature class. All the parameter attributes are optional (they have defaults), but if used, they must store numbers and the fields must have special names.



☐ Open the attribute table for the ParkFacilities layer.

Shape *	Id	Ranger_Des	OffsetA	OffsetB	OBJECTID *
Point	0	Park Headquarters	100	100	1
Point	0	Visitors Center	50	100	2
Point	0	Paradise Flat campground	50	100	3
Point	0	Eagle Falls campground	50	100	4

The ParkFacilities layer has OffsetA and OffsetB attributes. OffsetA is the height of each facilities radio mast and OffsetB is 100 feet (the height of the proposed repeater tower) for every point.

☐ Close the attribute table.

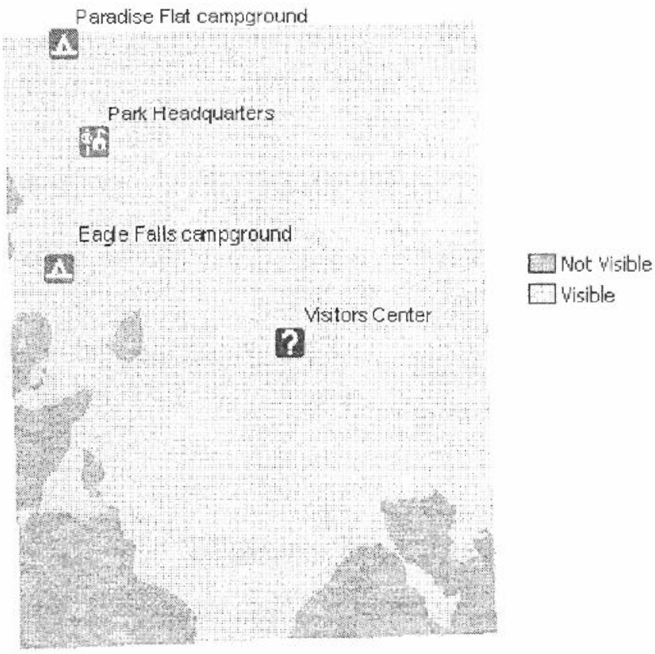
Step 3: Calculate viewshed

Now you are ready to run the Viewshed tool.

☐ Open the Viewshed (Spatial Analyst) tool and set its parameters as follows:

- Input raster: Elevation
- Input point or polyline observer features: ParkFacilities
- Output raster: **View1**
- Z factor: **1**
- Use earth curvature corrections: [Leave unchecked]
- Refractivity coefficient: [Leave blank]

☐ Click OK.



The tool classified the cells as visible (green) or not visible (pink). The actual cell values are a count of the number of observers that can see each cell. In the next step, you will resymbolize this layer to show how many facilities can see each cell.

Step 4: Make a map of potential relay tower sites

The Viewshed tool does not honor an analysis mask (which you have not set), and it calculated visibility for the cells in the lake. These are not very attractive, so you will turn them into NoData.

Remember, masks are set in the Environment Settings dialog box under Raster Analysis Settings.

☐ Change the Environment Settings so that Mask is set to LandMask.

Hint: Refer to exercise 2, step 2.

If you were just to run the Extract by Mask tool, there would be no need to set the Mask in the Environment Settings—the tool does that for you. However, you are setting it in the Environment Settings so that it persists for all the subsequent tools you will be using.

☐ Open the Extract by Mask (Spatial Analyst) tool and set its parameters as follows:

- Input raster: **View1**
- Input raster or feature mask data: LandMask
- Output raster: **View2**

☐ Click OK.

Now you will modify the colors of the View2 symbols and make the final map.

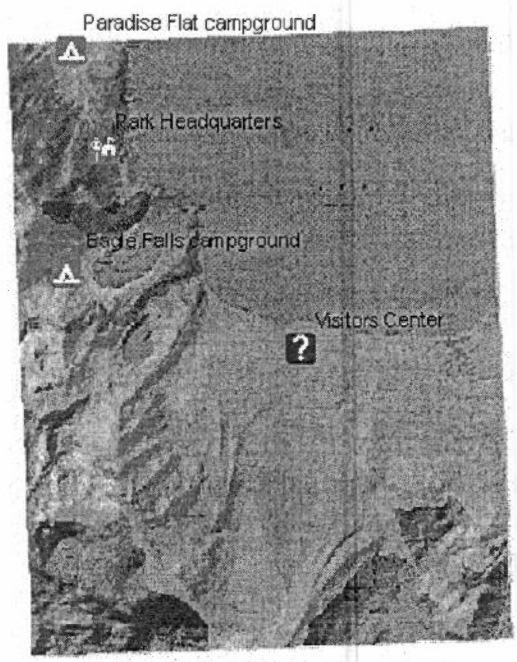
☐ Right-click each symbol in the View2 legend and assign the following colors (specific recommendations are given in parentheses):

- Value 0: No Color
- Value 1: Light green (Tzavorite Green)
- Value 2: Medium green (Peridot Green)
- Value 3: Dark green (Tarragon Green)
- Value 4: Bright red (Mars Red)

☐ Turn off all layers except ParkFacilities, View2, TopoMap, and erhill.

☐ In the Image Analysis window, set the View2 transparency to 50%.

The bright red cells are those that are visible from all four park facilities. The new relay tower could be located in any of these locations.



Potential radio tower sites

☐ Turn off the View2 layer.

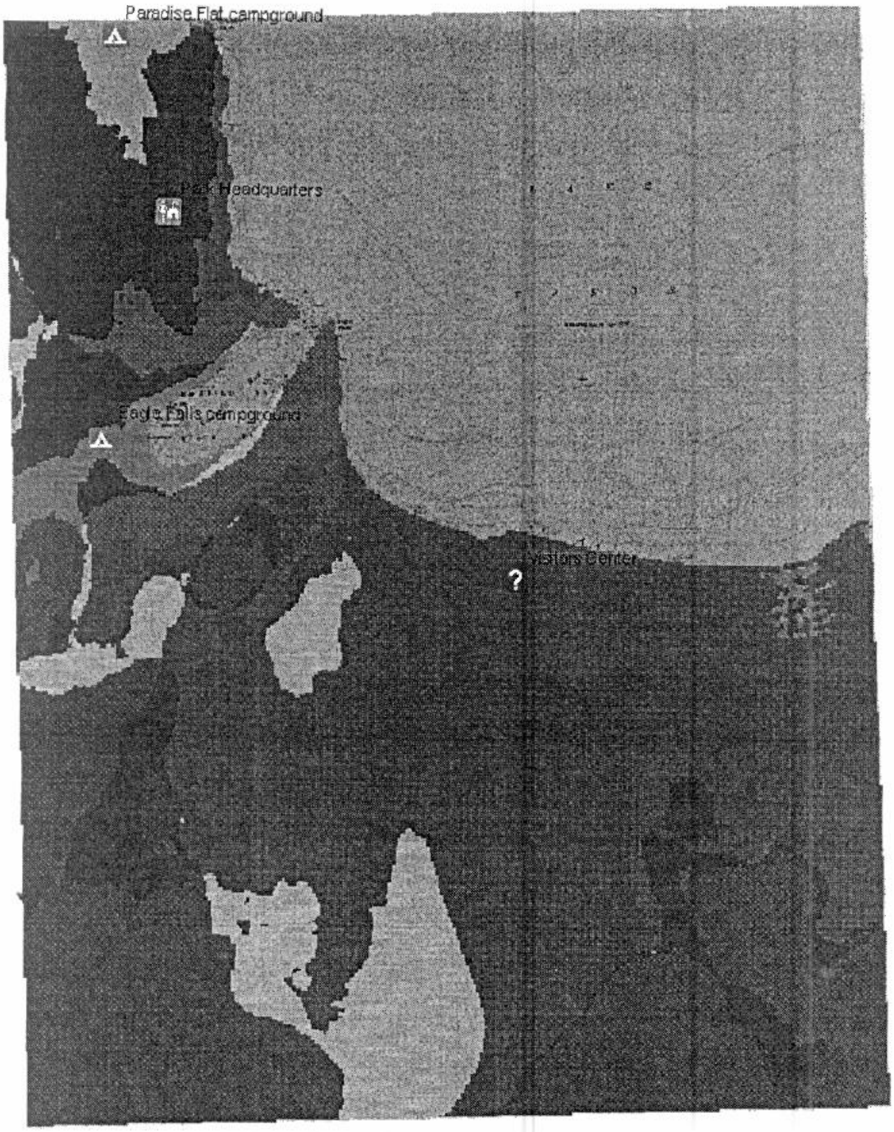
Step 5: Determine the visual exposure of park facilities

The Observer Points tool is similar to the Viewshed tool, except that it only accepts point features for the input observers and is limited to 16 points. The IDs of the observers that can see each cell are recorded in the output raster's attribute table.

☐ Open the Observer Points (Spatial Analyst) tool and set its parameters as follows:

- Input raster: Elevation
- Input point observer features: ParkFacilities
- Output raster: **ObsPoints**
- Z factor: **1**
- Use earth curvature corrections: [Leave unchecked]
- Refractivity coefficient: [Leave blank]

☐ Click OK.


Paradise Flat campground
Park Headquarters
Eagle Falls campground
Visitors Center

☐ Open the attribute table for the ObsPoints layer.

OBJECTID *	Value	Count	OBS1	OBS2	OBS3	OBS4
1	0	25708	0	0	0	0
2	1	6160	1	0	0	0
3	2	9926	0	1	0	0
4	3	48427	1	1	0	0
5	4	1	0	0	1	0
6	5	1584	1	0	1	0
7	6	41	0	1	1	0
8	7	8474	1	1	1	0
9	8	1840	0	0	0	1
10	9	210	1	0	0	1
11	10	2547	0	1	0	1
12	11	4591	1	1	0	1
13	14	33	0	1	1	1
14	15	315	1	1	1	1

In addition to the standard VALUE and COUNT attributes, there are attributes named OBS1 through OBS4. These attributes are used to record the visibility of each cell from every input point.

For example, every raster cell that can be seen by Observer 1 will contain a value of 1 in the OBS1 field. Cells that cannot be seen from the observation point are assigned a value of 0.

☐ Close the attribute table.

Step 6: Extract raster surface locations visible from a single observation point only

You can use the OBS fields to identify those raster cells that can be seen from a specific observation point. For example, suppose you want to display all the regions of the raster that can be seen only at park headquarters (OBS1). You could select the row where the Observer 1 (OBS1) value is 1 and the values for all other observers (OBS2, OBS3, and OBS4) are 0. The regions of the raster that can be seen only by Observer 1 will be highlighted in the map.

You can also use the Extract by Attributes tool to create a permanent surface.

☐ Open the Extract by Attributes (Spatial Analyst) tool and set its parameters as follows.

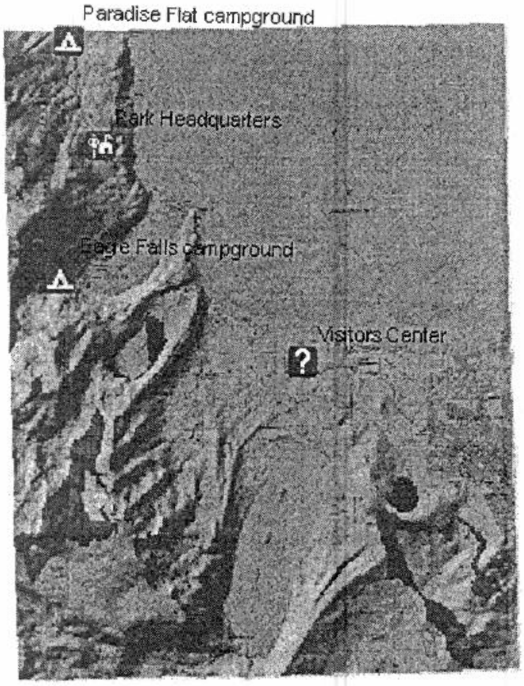
Hint: For Where clause, you can click the SQL button and create the expression on the Query Builder, or you can type the expression directly into the field.

- Input raster: ObsPoints
- Where clause: **"OBS1" = 1 AND "OBS2" = 0 AND "OBS3" = 0 AND "OBS4" = 0**
- Output raster: **HQViz**

☐ Click OK.

☐ Symbolize the HQViz layer with a red color and set its transparency to 50%.

☐ Turn off all layers except ParkFacilities, HQViz, TopoMap, and erhill.



The red cells represent those areas that can be seen only from the park headquarters.

☐ Exit ArcMap without saving changes.

Conclusion

In this exercise you calculated a viewshed for Tahoe using four observer points (park facilities). To calculate viewshed, you need an elevation layer and an observation layer of point or line features. The value of each cell in the viewshed raster represents the number of observer points from which the cell is visible.

A

Suggested reading

Reading list

1. *ArcGIS Geostatistical Analyst product page*
   Links to brochures, white papers, and other resources on the ArcGIS Geostatistical Analyst extension.
   www.esri.com/software/arcgis/extensions/geostatistical/about/literature.html
2. *Using ArcGIS Geostatistical Analyst*
   ISBN: 1-58948-106-2, 300 pages, ESRI 2004
   http://store.esri.com/esri/showdetl.cfm?SID=2&Product_ID=1138&Category_ID=121