SURFACES

Tutorial

Surface Modeling

with

TNTmips®

Before Getting Started

This booklet introduces the powerful Surface Modeling process in TNTmips®. Surface modeling creates approximations of functional surfaces from the 3D information you provide. A functional surface combines spatial location with the value for some variable (such as elevation, chemical concentration, or population density) at that location. Functional surfaces are commonly represented as Digital Elevation Models (raster), isolines (vector contours), or Triangular Irregular Networks (TIN object). You can create any of these surface forms and convert between them in the Surface Modeling process. You can also create serial profiles of functional surfaces.

Prerequisite Skills This booklet assumes that you have completed the exercises in the tutorial booklets *Displaying Geospatial Data* and *TNT Product Concepts*. Those exercises introduce essential skills and basic techniques that are not covered again here. Please consult those booklets and the TNTmips reference manual for any review you need.

Sample Data The exercises presented in this booklet use sample data that is distributed with the TNT products. If you do not have access to a TNT products DVD, you can download the data from MicroImages' web site. In particular, this booklet uses sample files in the SURFMODL data collection.

More Documentation This booklet is intended only as an introduction to Surface Modeling. Details of the processes described here can be found in a variety of tutorial booklets, color plates, and Quick Guides, which are all available from MicroImages' web site.

TNTmips Pro, Basic, and Free TNTmips comes in three versions: TNTmips Pro (which requires a software license key), low-cost TNTmips Basic, and TNTmips Free. This booklet refers to all versions as "TNTmips." TNTmips Basic and TNTmips Free provide all the capabilities of TNTmips Pro but limit the size of the geospatial objects and attribute tables that can be used in your project.

Surface Modeling is not available in TNTview, TNTedit, or TNTatlas. All the exercises can be completed in TNTmips Free using the sample geodata provided.

Randall B. Smith, Ph.D., 14 August 2013

You can print or read this booklet in color from MicroImages' web site. The web site is also your source for the newest tutorial booklets on other topics. You can download an installation guide, sample data, and the latest version of TNTmips.

http://www.microimages.com

Welcome to Surface Modeling

The Surface Modeling process in TNTmips includes a set of operations that allow you to transform spatial data representing a three-dimensional surface from one form to another. The most familiar example of such data is probably the variation in elevation of the Earth's surface. However, any variable can be visualized and analyzed as a three-dimensional surface as long as it varies relatively smoothly at the chosen map scale and has only a single value at each location. Examples include crop yield data, population density data, the concentration of dissolved chemicals at the ocean surface or groundwater table, geophysical measurements such as gravity, and many others.

A three-dimensional surface can be approximated in a number of forms, including irregularly-spaced point observations, a regular grid of values, or contour lines of equal value (isolines). In TNTmips, irregularly-spaced point data can be stored as points in a vector or shape object, as nodes in a TIN (Triangulated Irregular Network), or in a database object containing X and Y coordinates in addition to the value to be mapped. Gridded measurements are stored as a raster object, and contours as a vector or shape object. Each of these data types can be used as input for one or more of the Surface Modeling operations.

Each Surface Modeling operation produces a specific type of object. The Surface Fitting operation produces a raster grid, Contouring produces vector contour lines, and Triangulation produces a TIN. The Profiling operation creates a series of parallel vertical profiles of a surface raster. Most operations provide a choice of several different methods for producing the desired surface. Your choice of method may depend on the type of input data as well as the intended use for the output surface.

STEPS
- ☑ launch TNTmips
- ☑ select Terrain / Surface Modeling from the TNTmips menu

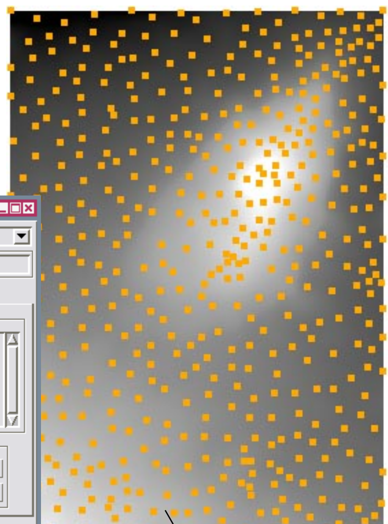
The exercises on pages 4-15 of this booklet show you how to create surface rasters with the **Surface Fitting** operation. Techniques for producing vector contours with the **Contouring** operation are introduced on pages 16-19. Pages 20-24 lead you through the creation of TIN objects with the **Triangulation** operation. Pages 25-26 show you how to create stacked vertical profiles of a surface raster with the **Profiling** operation. Page 27 presents graphical and tabular summaries of all Surface Modeling operations and methods.

Begin Surface Fitting

STEPS

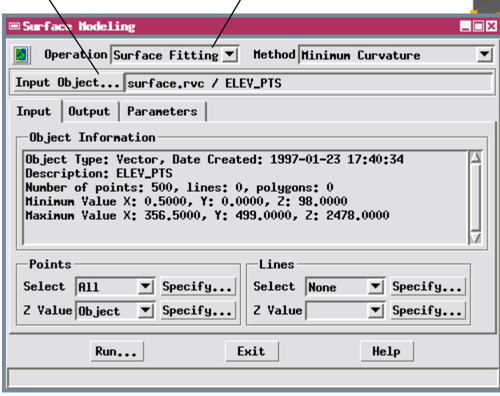
- ☑ select Surface Fitting from the Operation option menu
- ☑ press the Input Object button
- ☑ select the ELEV_PTS vector object from the SURFACE Project File in the SURFMODL data collection
- ☑ press the Run button and name a new Project File SURFOUT
- ☑ accept the default name provided for the output surface raster

Let's start by running a sample Surface Fitting operation. *Surface Fitting* interpolates a regular grid of values from data in the input object and outputs the grid as a raster object. The input data can be in the form of points stored in a vector object or in a database that has X and Y coordinate fields for each record. You can also use vector contour lines or the nodes and/or edges in a TIN object as input. The input object used in this exercise is a 3D vector object containing 500 irregularly-spaced sample elevation points from a topographic surface. The elevation is stored as a Z-value for each point.



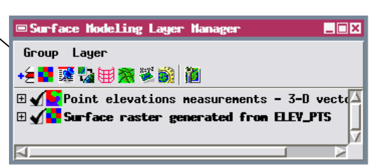
Select a Surface Modeling operation

Click on the Input Object button to choose the input object.



Surface Modeling uses a standard View window to automatically display input and output objects.

Use the standard Layer Manager window to change display settings for any input or output objects shown in the Surface Modeling View. If the Layer Manager is not open, press the Layer Controls icon button on the View to open it.



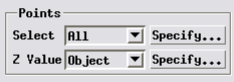
Keep the Surface Modeling window open with the current settings for the next exercise.

Set Input and Output Parameters

The Input and Output tabbed panels of the Surface Modeling window let you control the selection of data from the input object and the size and spatial resolution of the output surface raster. In this exercise you examine these controls and set a cell size for the next surface raster to be generated from the ELEV_PTS vector object.

The controls on the Input tabbed panel vary depending on the operation and input object type you have selected. Since the current input vector object contains points, the Points subpanel is active. These controls determine which points are used to generate the surface raster values and where to find the "elevation" value. In this case all of the points in the object are valid elevation measurements, so the default selection of All that appears on the Select option button is appropriate. The default selection of Object on the Z Value: option menu is also appropriate, as we are surface fitting the elevation value stored as the Z-value in the 3D (XYZ) vector object. A By Query option is also provided on each of these menus, allowing you to use a database query to select a subset of the points as input for the process and to use values stored in any database field as the Z value.

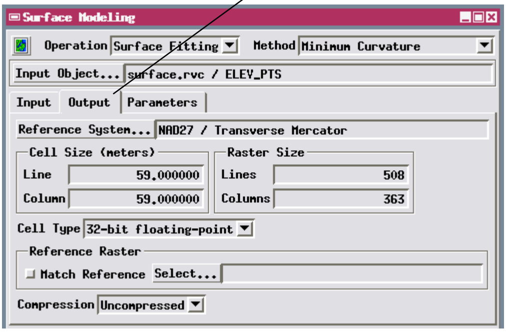
The Cell Size subpanel on the Output tabbed panel is used to set the size of the output raster cells in meters. The previous Surface Fitting operation calculated a cell size of 59 meters from the geographic extents of the input object and the default output raster size. When you enter new cell size values in the Line and Column numeric fields, the size of the output raster is recalculated and the Raster Size parameter fields are automatically updated.



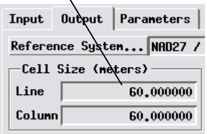
STEPS

- ☑ examine the Points controls on the Input tabbed panel
- ☑ click the Output tab to reveal the Output tabbed panel
- ☑ in the Cell Size subpanel, enter 60.0 in the Line and Column text fields
- ☑ choose 16-bit signed from the Cell Type menu

Click a tab to reveal its attached panel.



To change a parameter value, highlight the field with the mouse cursor and type in the desired value.



Keep the Surface Modeling window open with the current settings.

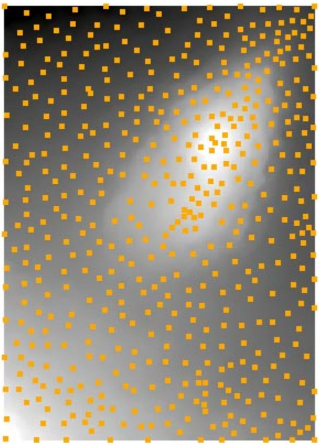
Surface Fitting by Inverse Distance

STEPS

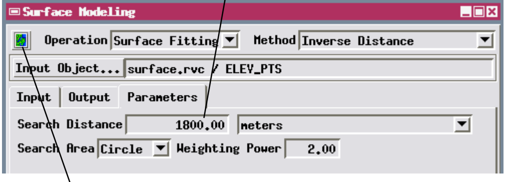
- ☑ select Inverse Distance from the Method option menu
- ☑ click on the Parameters tab and choose Circle from the Search Area option menu
- ☑ set the Search Distance value to 1800 and choose meters from the units menu
- ☑ set the Weighting Power to 2.0
- ☑ press the Run button and direct the output raster to the SURFOUT Project File
- ☑ accept the default name provided for the output surface raster

The Inverse Distance method interpolates a value for each cell in the output raster using a set of nearby input elements. The Z-values of these elements are weighted so that nearby elements contribute more to the result than those farther away. This method can be used with vector or shape objects containing points or contours, or with database or TIN objects.

The Search Area parameter determines the shape of the selection area, while the Search Distance value determines its size. You can set the Search Distance in raster cells or in map distance units. The settings used here create a circular selection area with an 1800-meter radius (30 cells with the current output cell size of 60 meters). The spacing of adjacent points in this input object varies from 200 to about 2000 meters, so these settings should provide an adequate set of points for each raster cell location. The Weighting Power parameter determines the exponent used in the distance function that determines the weighting factors applied to the input Z-values. With the default setting of 2.00, the weighting factors decrease in value by the square of the distance.

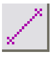


The Search Distance parameter controls the size of the area used to select input data values for interpolation.



Press the Open 2D View icon button to re-open the View window if you have closed it or to bring the open View to the foreground.

To measure the distances between points in an input object, press the GeoToolbox icon button on the View window and use the Ruler tool. For more information see the tutorial booklet entitled *Sketching and Measuring*.



Keep the Surface Modeling window open with the current settings for the next exercise.

Polynomial Trend Analysis

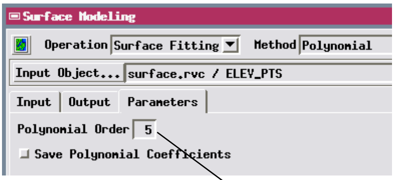
The Polynomial surface fitting method finds a best-fit surface defined by a polynomial equation that treats the mapped value as a mathematical function of geographic position. You can use vector point, TIN, and database objects as input with this method.

The polynomial method finds the best-fit surface by minimizing the sum of the squared deviations between the input values and the calculated surface. Because this is a best-fit for the *entire set* of input points, typically the output surface does not match the original value at each input point. This method is most useful for portraying generalized spatial trends for a "noisy" mapped value.

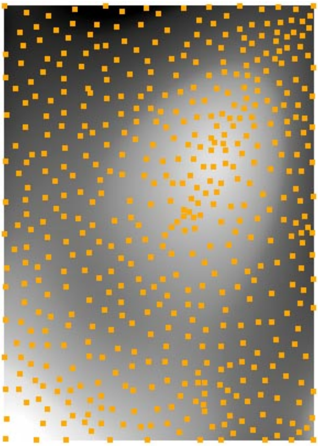
The Polynomial Order parameter controls the form of the polynomial equation, which in turn defines the complexity of the computed surface. A second-order polynomial equation defines a parabolic curved surface with only one sense of curvature (concave or convex). A third-order (cubic) equation allows one change in sense of curvature in any cross-section. Higher-order equations allow for increasing complexity and more local detail. The fifth-order polynomial surface you generate here depicts the generalized trends in elevation in the input point object, but does not convey the detail present in the surface raster produced in the previous exercise by the Inverse Distance method.

STEPS

- ☑ select Polynomial from the Method option menu
- ☑ on the Parameters tab set the Polynomial Order parameter value to 5
- ☑ press the Run button and direct the output raster to the SURFOUT Project File
- ☑ accept the default name provided for the output surface raster



The Polynomial Order parameter controls the complexity of the computed surface.



Keep the Surface Modeling window open with the current settings for the next exercise.

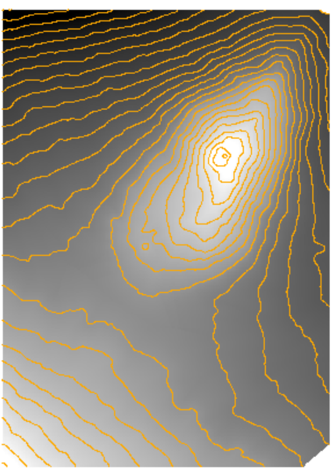
For each operation Surface Modeling offers a variety of methods that can be used with particular types of input objects. The Method option menu shows only those methods that can be used with the current input object type.

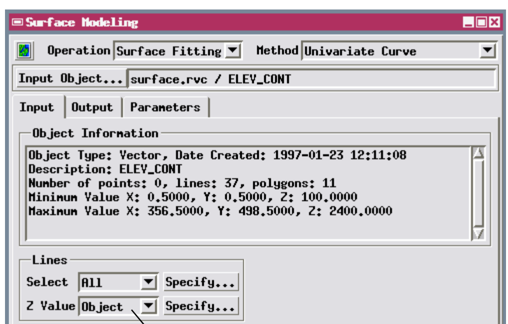
Surface Fitting by Univariate Curve

STEPS
- ☑ press the Input Object button and select the ELEV_CONT vector object from the SURFACE Project File
- ☑ press [Yes] on the Question dialog to remove previous result layers
- ☑ select Univariate Curve from the Method option menu
- ☑ on the Output panel set the Line and Column cell size to 60.0
- ☑ on the Parameters panel choose Cubic from the Interpolation menu and Linear from the Slope menu
- ☑ press the Run button and direct the output raster to the SURFOUT Project File

The Univariate Curve surface fitting method is specially tailored to create a surface raster from contour lines. Its strategy is to assign each output cell value by interpolating between the enclosing pair of contours along an approximation of the steepest path of descent through that cell. To approximate that path, the closest points on the uphill and downhill contour lines are found and those distances and contour elevations control the interpolation.

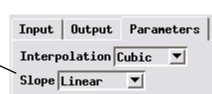
Three interpolation methods are provided. The Linear method creates a smooth surface between each contour pair but leaves an abrupt change in slope at each contour. The Cubic and Hermite methods are curve-fitting procedures that include the local slope values at each contour line as inputs to the interpolation. These methods interpolate curving surfaces that meet more smoothly at the contours. Along ridgelines and valley bottoms, however, opposing slope surfaces may meet at sharp angles.





When the input vector object contains elevation contours, the Lines subpanel is shown. For 3-D input objects, use the default selection of Object on the Z Value option menu.

NOTE: in subsequent exercises in which you are asked to choose a new input object, always choose Yes on the Question dialog to remove previous result layers.



Slope values at the contour lines are calculated using either a Linear method (neighboring 4 cells) or Quadratic method (neighboring 8 cells).

Evaluate the Surface Raster Result

The display interface in the Surface Modeling process includes many tools that you can use to evaluate the quality of any computed surface raster. DataTips allow you to compare the elevation value of an element in the input object with the computed elevation at the same location in the surface raster. Relief shading the surface raster provides a more natural and revealing view of the surface than a simple grayscale display, highlighting the shape of the surface at all scales, including minor details. The Profile View in the GeoToolbox lets you construct and view vertical elevation profiles along any transect to help assess the shape of the surface. More information about these tools is available in the tutorial booklets *Sketching and Measuring* and *Analyzing Terrain and Surfaces*.

STEPS

☑ in the Layer Manager window, left-click on the icon for surface raster layer entry to open the Layer Controls

☑ click on the Shading tab in the Raster Layer Controls window

☑ turn on the Relief Shading toggle

☑ drag the radius line in the circle graphic to the northwest quadrant

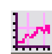
☑ enter 3.0 in the Z Scaling field

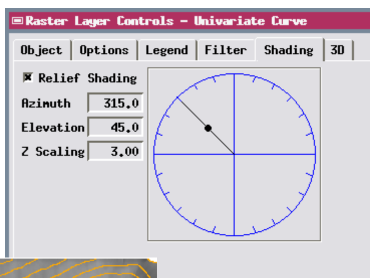
☑ click [OK] on the Raster Layer Controls window

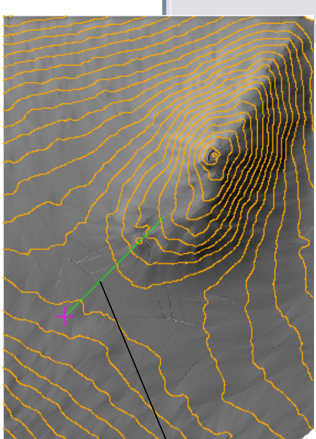
☑ press the GeoToolbox icon button on the View 

☑ press the Ruler icon button in the GeoToolbox 

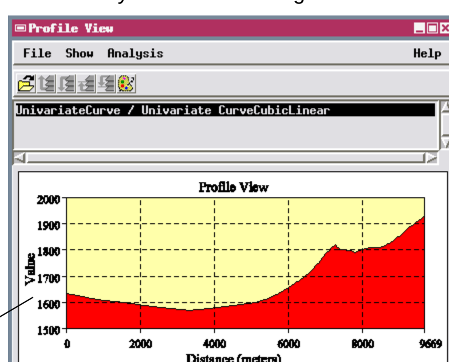
☑ left-click and drag with the mouse in the View to draw a ruler line

☑ press the Open Profile View icon button in the GeoToolbox 





Relief shading provides a more understandable image of the output surface and can reveal artifacts created by the surface fitting method.



Profile line set by Ruler tool and resulting Profile View.

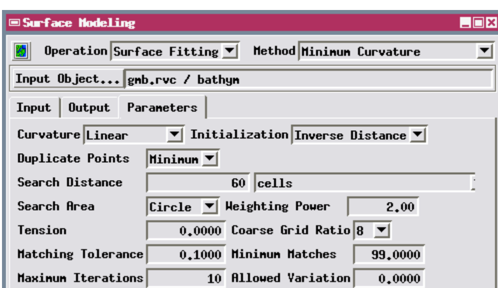
Surface Fitting by Minimum Curvature

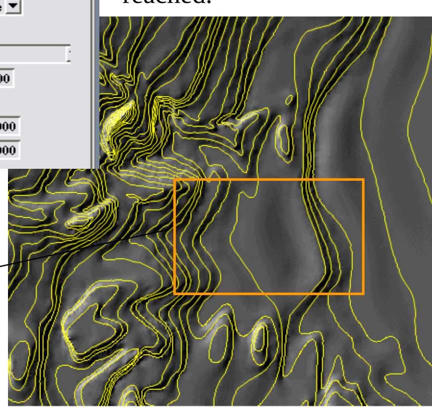
STEPS

- ☑ select Minimum Curvature from the Method option menu
- ☑ press the Input Object icon button and select the BATHYM vector object from the GMB Project File
- ☑ on the Parameters panel set the Search Distance value to 60 cells
- ☑ set the Tension value to 0.0
- ☑ set the Coarse Grid Ratio menu to 8
- ☑ set the other parameters to the values shown in the interface illustration
- ☑ press the Run button
- ☑ since you have already created a raster named Minimum Curvature, modify the default object name and direct the output raster to the SURFOUT Project File
- ☑ turn on Relief Shading for the output raster and set the Z Scaling to 12.0

The Minimum Curvature method fits a two-dimensional cubic spline surface to the input points, TIN nodes, or contour line vertices. This surface is constructed to have a minimum overall curvature, producing a smoothly varying surface. The *Allowed Variation* parameter lets you determine whether the surface must pass exactly through the data points (value 0) or can deviate from them by the specified amount.

This method uses an interative approach beginning with an initial coarse-resolution grid of output cells. The Coarse Grid Ratio parameter sets the spacing (in number of output cells) between cells in this initial grid. Initial cell values for the coarse grid are determined from the input object using either the Inverse Distance or Profiles Method. A cubic surface is fit to the coarse grid values, which are iteratively adjusted to minimize the surface curvature. Iterations cease for each grid value when the change falls below the Matching Tolerance value or the Maximum Iterations value is reached. A finer-resolution grid is interpolated from the adjusted coarse grid and its values are iteratively adjusted, with the process repeating until the final raster dimensions are reached.





The contours used in this exercise depict seafloor bathymetry, and so their Z values are negative. The area outlined by the box is shown in more detail on the following page.

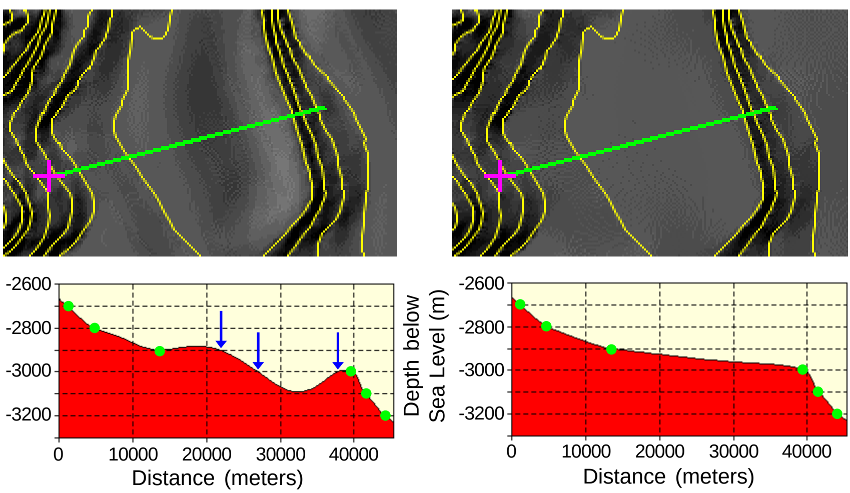
Minimum Curvature with Tension

The Minimum Curvature method can produce surfaces with large oscillations and unnecessary inflections along the the boundary and in areas where input values are widely spaced. Examples of such oscillations occurs in the boxed area in the surface illustration on the previous page (shown in more detail below) and between the other widely-spaced contours on the right side of the sample area.

STEPS
☑ set the Tension parameter for the Minimum Curvature method to 0.75
☑ set the Coarse Grid Ratio option menu to 16

To reduce these spurious oscillations, use the largest Coarse Grid Ratio setting (16) to produce an initial smooth, low resolution surface for further iterative refinement. You can also increase the Tension setting, which varies between 0 and 1.0. The effect is similar to that of increasing the tension on a physical elastic surface stretched to fit the data values. Increasing the tension value simplifies the surface shape between input data points (reduces its curvature) and increases curvature at the data point locations.

☑ press [Run], rename the output raster and direct it to the SURFOUT Project File
☑ turn on Relief Shading for the output raster and set the Z Scaling to 12.0

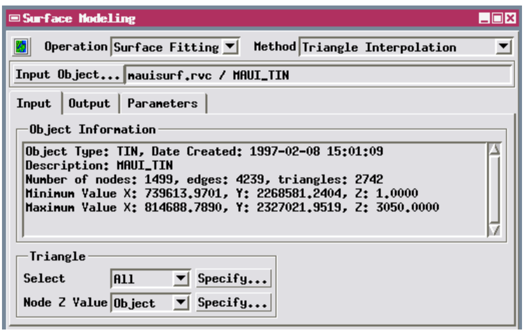


Detail of computed surface from previous page (left) and this page (right) with profiles along the green line. Green dots on profiles mark points where profile crosses input contour lines; arrows on left profile mark crossings of nonexistent contours due to unconstrained oscillations of the minimum curvature surface between widely-spaced contours. Increasing the Coarse Grid Ratio and Tension values helps damp these extraneous oscillations.

Surface Fitting by Triangle Interpolation

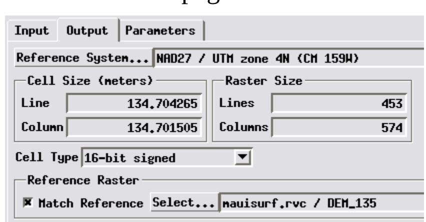
STEPS
- ☑ press the Input Object button and select the MAUI_TIN object from the MAUISURF Project File
- ☑ select Triangle Interpolation from the Method option menu
- ☑ on the Output panel turn on the Match Reference toggle button
- ☑ choose the DEM_135 object from the MAUISURF Project File as the reference

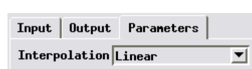
The Triangle Interpolation surface fitting method is designed for TIN objects. A vector or shape object can also be used as input, in which case the method builds a temporary TIN object from the input elements. The method uses the elevation values at the nodes of the TIN triangles to compute a surface that fits each triangular area. “Holes” in the TIN are not filled, and separate TIN hulls generate separate elevation surfaces (as illustrated by the island surfaces created in this exercise). Raster cells in areas outside the TIN hulls are marked as null and are automatically displayed transparent in the View.

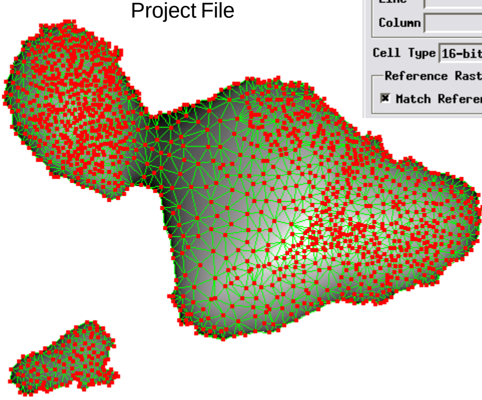


The Match Reference option on the Output tabbed panel enables you to match the dimensions, orientation, and cell size of your output surface raster to an existing georeferenced raster object that covers the same area as your input data. This option works with any Surface Fitting method except the Bidirectional method, which is discussed on a later page.

- ☑ on the Parameters panel choose Linear from the Interpolation menu
- ☑ press [Run], name the output raster Linear and direct it to the SURFOUT Project File



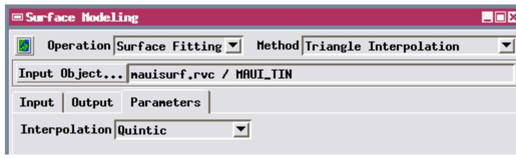




When the input TIN object has more than one hull, Triangle Interpolation produces a separate elevation surface for each hull, with intervening cells marked as null.

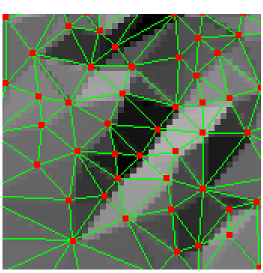
Triangle Interpolation Options

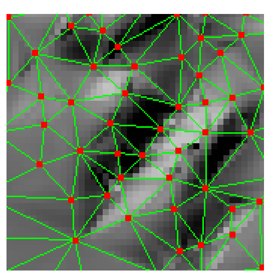
You can choose from several interpolation procedures on the Parameters tabbed panel. The Linear option used in the previous exercise fits a simple planar surface to each triangle in the input TIN object. The planar surfaces of adjacent triangles may meet at distinct angles along their shared edges. This option essentially reproduces the triangular-faceted TIN surface in raster form.

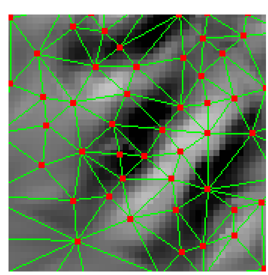


The Quintic and Nonic options fit a curving polynomial surface to each triangle. The Quintic option uses a 5th-order polynomial, while the nonic option uses a 9th-order polynomial. These polynomial expressions are derived using not only the nodes of the current triangle, but those of the surrounding triangles as well. The slope and change in slope in different directions around each triangle node are computed and used to constrain the shape of the current triangle surface to ensure that it will join relatively smoothly with the surfaces derived from the adjacent triangles. The Quintic option produces a smoother, less angular surface than the Linear option but may not eliminate all triangle edge artifacts. The Nonic option provides more degrees of freedom to produce a more complexly curved local surface with fewer triangle edge artifacts.

STEPS
- ☑ on the Parameters panel choose Quintic from the Interpolation menu
- ☑ press [Run], name the output raster Quintic and direct it to the SURFOUT Project File
- ☑ on the Parameters panel choose Nonic from the Interpolation menu
- ☑ press [Run], name the output raster Nonic and direct it to the SURFOUT Project File
- ☑ turn on Relief Shading for the surface rasters created by the Linear, Quintic, and Nonic options to compare their shapes


Linear Interpolation

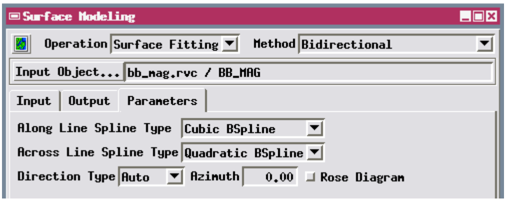

Quintic Interpolation


Nonic Interpolation

Bidirectional Surface Fitting

STEPS

- ☑ press the Input Object button and select the BB_MAG object from the BB_MAG Project File
- ☑ select Bidirectional from the Method option menu



- ☑ on the Output panel set the Line and Column cell size to 31.0
- ☑ choose 32-bit floating-point from the Cell Type option menu
- ☑ on the Parameters panel choose Cubic BSpline from the Along Line Spline Type menu
- ☑ select Quadratic BSpline from the Across Line Spline Type menu
- ☑ press [Run] and direct the output raster to the SURFOUT Project File

The Bidirectional surface fitting method is designed for use with aeromagnetic and other geophysical data that are collected along groups of nearly parallel transect lines. Input data for the Bidirectional method must be in the form of 3D vector lines, with one line for each transect and line vertices representing measurement locations. In most cases the distance between measurements along each transect is much less than the spacing between adjacent transects, so there is an inherent directional bias in the distribution of the observation points.

The Bidirectional method interpolates raster values in two steps: first along each transect line, then perpendicular to the dominant transect direction. With the Direction Type option set to Auto, the process automatically determines the predominant transect direction. Alternatively, you can set this option to Manual and enter an azimuth value to be used as the predominant direction.

You can choose different methods for interpolating along and across the transect lines. Three interpolation methods are provided for each direction. The methods are Linear, Quadratic BSpline, and Cubic BSpline. The latter two methods produce surfaces with smoother, more gradual changes in curvature than the Linear splining method.

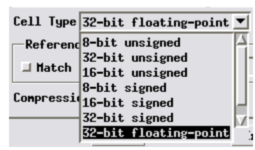
These transect lines are from an aeromagnetic survey of a single 7.5-minute map quadrangle, with magnetic intensity values in nanoteslas.

Other Surface Fitting Methods

The Surface Modeling process in TNTmips also includes several additional surface fitting methods that are not used in the exercises in this booklet. Brief summaries of these methods are provided below.

Kriging Kriging is a statistical approach to interpolation that assumes that the input data values sample a continuous surface, so that nearby data values should be similar in value (dependent) and widely separated values should be nearly independent. Kriging interpolates a value for each output raster cell by calculating a weighted average of the values at nearby points. The statistical variation in values over different distances and in different directions (depicted graphically as a variogram) is analyzed to determine the shape and size of the point selection area and the set of weighting factors that will produce the minimum error in the elevation estimate. The variation in input values can be assumed to consist of local variations on a regional trend (the drift). Ordinary kriging assumes no drift, or the drift can be modeled as a linear or nonlinear function. Known directions of anisotropy can also be factored into the calculations. Kriging can be used with input vector points, TINs, and database objects.

Profiles This method uses a multi-directional linear interpolation procedure to create a surface raster from contour lines. The process searches for pairs of input elevations on opposite sides of each output cell. After edge cells are interpolated, the process searches in eight different directions (up, down, left, right, and diagonally) and uses the closest pair of values (including edge cells) to assign an output elevation value. A Search Distance parameter determines the radius of the search.



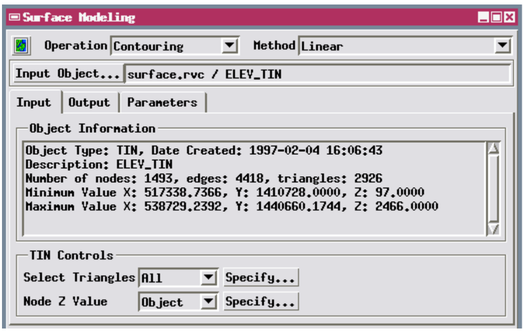
Use the Cell Type option menu on the Output tabbed panel to choose the appropriate data type for the computed surface raster. When the input represents elevations of the Earth's surface in meters or feet, a16-bit signed integer cell type is usually appropriate, since its range of -32,768 to +32,767 covers the entire elevation range. For sub-meter or sub-foot vertical accuracy in elevation, or for surface fitting other numeric values with a smaller range, you can choose 32-bit floating point output, but the stored size of the raster will be significantly larger. In addition, floating-point rasters cannot be compressed.

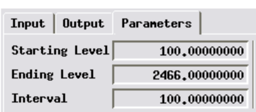
Contouring a TIN Object

STEPS
- ☑ choose Contouring from the Operation option menu
- ☑ press [Yes] on the Question dialog to remove previous result layers
- ☑ press [Input Object] and select the ELEV_TIN object from the SURFACE Project File

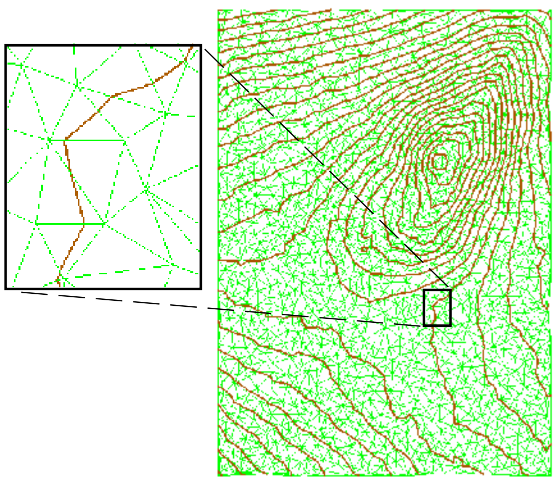
The next series of exercises explore the **Contouring** operation, which creates a 3D vector object with lines of equal value (contours or isolines) at a specified interval. TIN and raster objects can serve as input for contouring.

The Linear method is the only one available for contouring TIN objects. It treats each TIN triangle as a planar surface. When a contour is found to pass between two TIN nodes, the location of its intersection with the triangle edge is determined by linear interpolation from the node Z-values (or values you specify by a query). Each output contour line is made up of straight-line segments (one segment per triangle crossed), with direction changes occurring at the triangle edges.





- ☑ on the Parameters panel set the Starting Level parameter value to 100
- ☑ set the Interval parameter value to 100
- ☑ press [Run] and direct the output vector object to the SURFOUT Project File



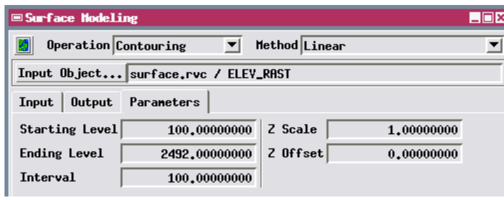
IMPORTANT: Choosing a new Surface Modeling operation clears the previous Input Object selection. You are also asked to choose whether or not to remove previous result layers from the View window before beginning the new Surface Modeling operation.

Contouring a Raster: Linear Method

The Linear method is also available for contouring raster objects. It locates contours from the raster values by linear interpolation in the line and column directions.

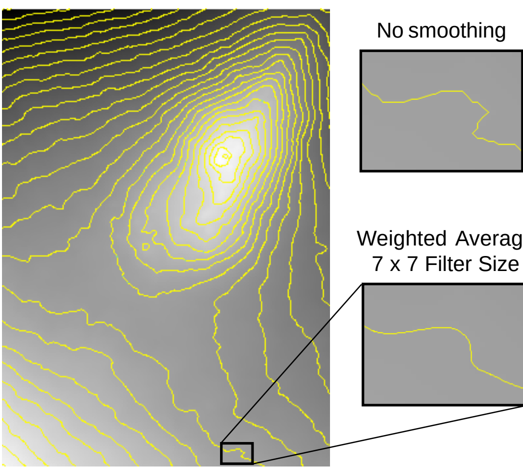
You have the option of smoothing the input raster values to reduce local detail prior to finding contours. The Smoothing Method options are found on the Input tabbed panel. With no smoothing, contour lines may appear jagged. The smoothing methods produce increasing smoothing as you increase the Filter Window Size. The smoothing filters include Weighted Average, Gaussian, Quadratic, Cubic, and Quartic.

STEPS
☑ press [Input Object] and select the ELEV_RAST object from the SURFACE Project File
☑ on the Parameters panel set the Starting Level parameter to 100 and the Interval parameter to 100



☑ on the Input panel choose None from the Smoothing Method option menu
☑ press [Run] and direct the output vector object to the SURFOUT Project File

☑ on the Input panel choose Weighted Average from the Smoothing Method menu
☑ select 7 x 7 from the Filter Window Size menu
☑ choose None from the Resampling Method menu
☑ press [Run] and direct the output vector object to the SURFOUT Project File



Choose raster smoothing options from the Smoothing Method option menu on the Input tabbed panel.

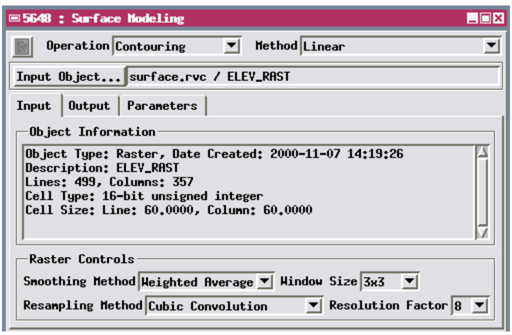
Increase the Filter Window Size to produce smoother, more curving contours.

Contouring with Resampling

STEPS

☑ on the Input panel choose 3x3 from the Window Size menu; keep the Weighted Average smoothing method

☑ choose Cubic Convolution from the Resampling Method menu

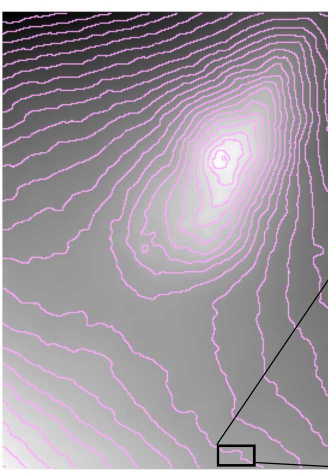


☑ choose 8 from the Resolution Factor menu

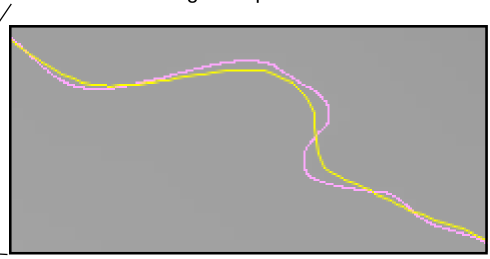
☑ press [Run] and direct the output vector object to the SURFOUT Project File

The Linear contouring method can also produce smoother contours by resampling the input raster to a smaller cell size prior to computing contour lines. This procedure uses the values of the surrounding cells in the input raster to interpolate values for the new smaller cells.

Use the Resampling Method menu to choose either the Bilinear Interpolation or Cubic Convolution method to interpolate the new cell values. The Resolution Factor sets how fine a subdivision of the original cell grid is created by the resampling. A resolution factor of 2 subdivides each cell into a 2 by 2 grid of smaller cells (4 cells), a factor of 4 divides each cell into a 4 by 4 grid (16 cells), and so on. Resampling to higher resolution can be used alone or in combination with raster smoothing. In this exercise minimal smoothing is applied in combination with resampling by a factor of 8.



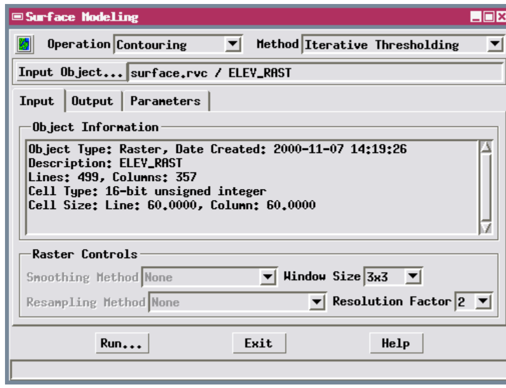
The illustration below compares results of smoothing with resampling the input raster to higher resolution. The yellow contour line was produced in the previous exercise using Weighted Averaging smoothing with a 7 x 7 filter window. The pink contour line produced in the current exercise using resampling captures more local detail while still showing smooth curves at the scale of the original input raster.



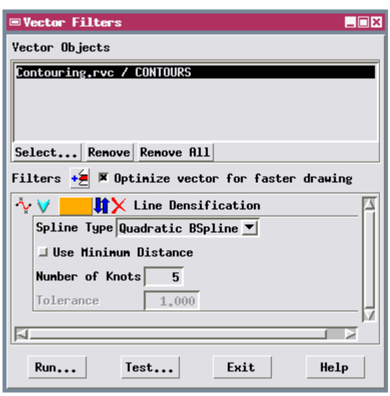
Contouring by Iterative Thresholding

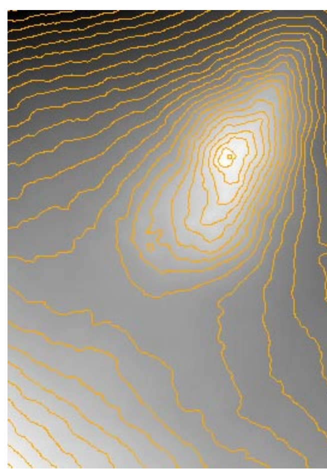
STEPS
- ☑ choose Iterative Thresholding from the Method option button
- ☑ press [Run] and direct the output vector object to the SURFOUT Project File

The Iterative Thresholding contour method for raster objects takes an image segmentation approach to locating contours. Each contour value is used as a threshold to segment the elevation raster into areas equal to or below the threshold and those with raster values above the threshold. The contour line is generated along the boundary of these regions in such a way that the line passes through the centers of all cells with a value equal to the contour value (unless the cell is a local maximum or minimum) and always passes between cells whose values bracket the contour value.



Iterative thresholding produces a mathematically accurate result, but one that may produce noisy, angular contours. This method is also several times slower than the other contouring methods.





You can smooth the contour lines produced by this and other contouring methods using the Vector Filters process (Geometric Filter from the TNTmips menu). The Line Densification filter adds vertices to lines to make them better approximate a smooth curve. You can find more information about Line Densification in the *Advanced Vector Editing* and *Digitizing Soil Maps* tutorials.

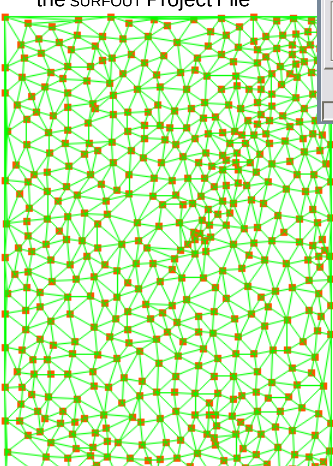
Triangulation from Point Data

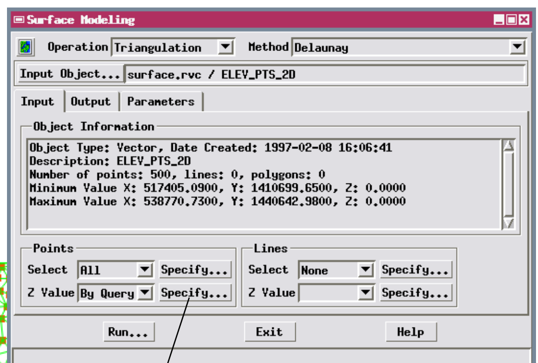
STEPS

- ☑ choose Triangulation from the Operation option menu
- ☑ press [Yes] on the Question dialog to remove previous result layers
- ☑ press [Input Object] and select the ELEV_PTS_2D object from the SURFACE Project File
- ☑ in the Points section of the Input panel, press [Specify...] next to the Z Value option button
- ☑ in the Script Editor window, press the Insert Field icon button 
- ☑ in the Insert Field window, choose ELEV_PTS from the Table menu and Z_VAL from the Field menu; press [Insert]
- ☑ press [OK] in the Script Editor window
- ☑ press [Run] and direct the output TIN object to the SURFOUT Project File

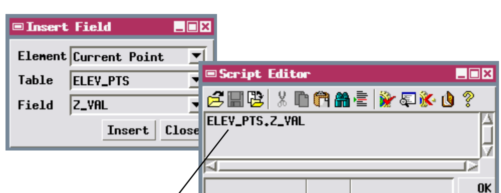
You have worked with TINs as input objects in previous exercises. To create a TIN object, use the Triangulation operation. **Triangulation** computes a TIN from points in a vector or database object, from vector contours, or from a raster object.

When the input object contains point data or contours, the Delaunay triangulation method is used. This method uses the input points (or contour line vertices) to create a triangular network meeting the Delaunay criterion (described in the exercise on Surface Fitting by Triangulation). For an input database object or 2D vector object, By Query is the active selection on the Z Value option button on the Input tabbed panel. You must use a query to specify a database table and field containing the values to assign as Z-values for the output TIN nodes.





Press [Specify...] to open the Script Editor window and create a query that specifies the table and field containing the desired Z values.



A value query has the simple form TABLE.FIELD, specifying the field in the attached database table that contains the desired values.

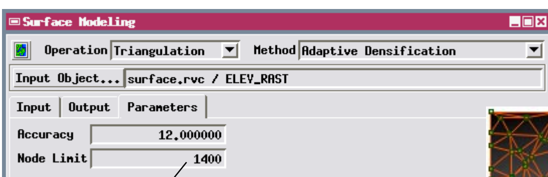
Triangulation from a Raster

The Adaptive Densification method of Triangulation is used to create a TIN from a raster object. After placing initial TIN nodes at the corners of the input raster to form two large triangles, this method subdivides triangles in a number of iterations to create a denser TIN structure. A triangle is subdivided by placing a new node at the raster cell location with the highest deviation from the planar surface defined by the triangle.

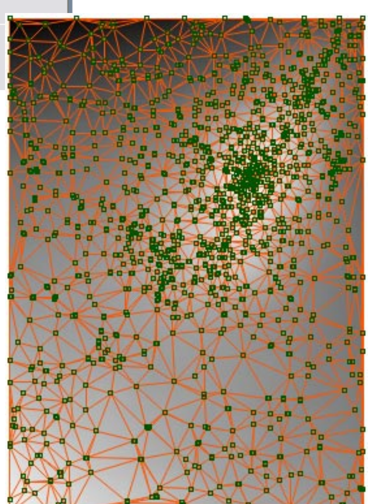
You can control the complexity and fidelity of the output TIN using the Accuracy and Node Limit parameters. The Accuracy parameter value sets the maximum Z-value deviation between a triangle and the raster surface it represents. If a triangle's deviation is less than this value, the triangle is not subdivided further. The Node Limit parameter sets a rough upper limit on the number of nodes in the final TIN object.

STEPS
- ☑ press [Input Object] and select the ELEV_RAST object from the SURFACE Project File
- ☑ click the Parameters tab
- ☑ on the Parameters tabbed panel, set the Accuracy parameter value to 12 and the Node Limit value to 1400
- ☑ press [Run] and direct the output TIN object to the SURFOUT Project File

The Adaptive Densification method is automatically selected when you choose a raster object as input for the Triangulation operation.



The number of nodes in the final TIN object may be less than the Node Limit if all triangles satisfy the current Accuracy parameter setting before the Node Limit is reached. By contrast, if the node limit is reached partway through a processing iteration, subdivision continues until all current triangles have been processed. In this case the final number of nodes exceeds the Node Limit value by a small amount, and some triangles may not satisfy the Accuracy parameter setting. For example, the TIN produced using the settings in this exercise contains 1417 nodes.



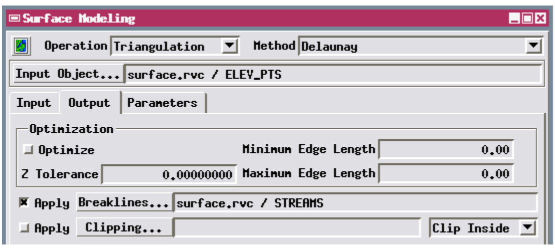
Triangulation with Breaklines

STEPS

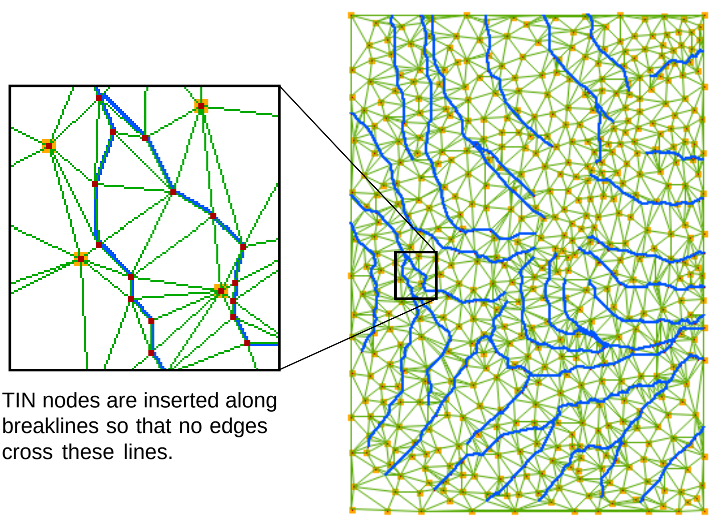
- ☑ press [Input Object] and select the ELEV_PTS object from the SURFACE Project File
- ☑ on the Output panel turn on the Apply Breaklines toggle button

The Triangulation operation includes several processing options that are available for any input object type, including the use of breaklines. **Breaklines** are 3D vector lines or polygons that modify or refine the structure of the resulting TIN object.

The Apply Breaklines option uses vector lines to guide the creation of the TIN structure. TIN nodes are inserted along the breaklines so that each breakline is reproduced in the TIN as a series of interconnected triangle edges, and no TIN edge crosses a breakline. The breaklines used in this exercise represent a drainage network. In addition to providing supplemental elevation control, they also mark the change in slope direction at the bottom of valleys and as such are significant topographic features. Using breaklines representing stream lines and ridge crests creates a TIN that better represents the landscape shape.



- ☑ select object STREAMS from the SURFACE Project File
- ☑ press [Run] and direct the output TIN object to the SURFOUT Project File



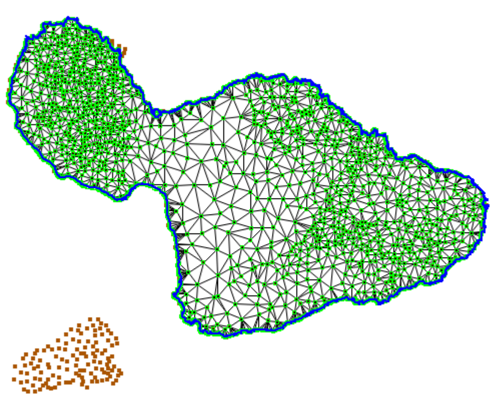
TIN nodes are inserted along breaklines so that no edges cross these lines.

Triangulation Using Breaklines to Clip

The Apply Clipping option allows you to use one or more polygons in a vector object to limit the extent of the TIN produced by the Triangulation operation.

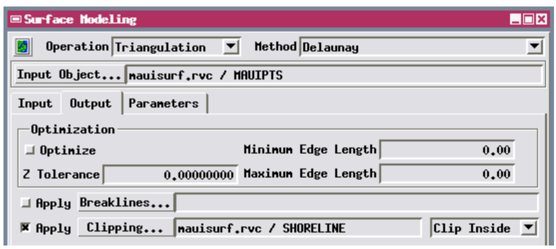
The Clip Inside option is appropriate when the polygon represents the outer boundary of the area of interest. The TIN structure inside the breakline polygon is retained, while edges and nodes outside the polygon are eliminated. In the example in this exercise, the input elevation point database represents the island of Maui and a smaller outlying island, while the breakline polygon outlines the shoreline of the main island. Only the main island is covered by the resulting TIN object.

The Clip Outside option creates "holes" within the TIN: the TIN structure outside the polygon is preserved, and edges and nodes inside the polygon are eliminated. In a topographic example, this option might be used when the polygon represents the complex shoreline of a large, irregularly shaped lake. Without clipping, the lake surface would be represented by a large number of horizontal triangles.



STEPS

- ☑ press [Input Object], navigate to the MAUISURF Project File, and double click on the MAUI_PTS_DB object
- ☑ select the MAUIPTS table and press [OK]
- ☑ in the Pins section of the Input panel choose By Query from the Z Value menu and press [Specify...]

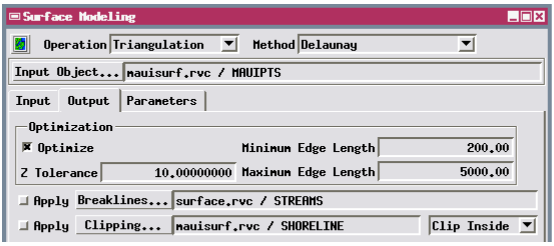


- ☑ in the Script Editor window press the Insert Field icon button 
- ☑ in the Insert Field window, MAUIPTS should be automatically selected in the Table List; choose ELEV in the Field list and press [Insert]
- ☑ press [OK] in the Script Editor window
- ☑ on the Output tabbed panel, turn off the Apply Breaklines toggle button
- ☑ turn on the Apply Clipping toggle button and choose object SHORELINE in the MAUISURF Project File
- ☑ choose Clip Inside from the clipping menu
- ☑ press [Run] and direct the output TIN object to the SURFOUT Project File

Triangulation with Optimization

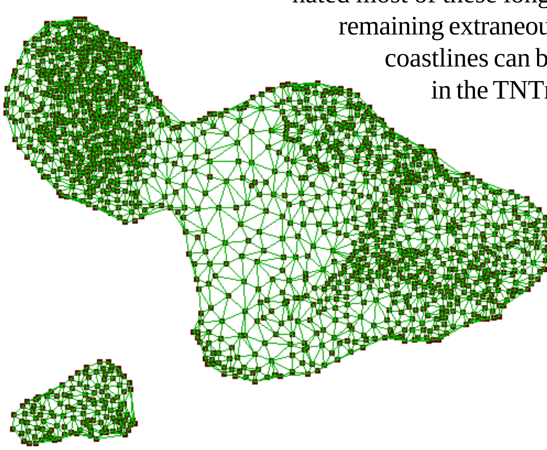
STEPS
- ☑ turn off the Apply Clipping toggle button on the Output tabbed panel
- ☑ turn on the Optimize toggle button in the Optimization box on the Output tabbed panel

TIN Optimization provides several functions to filter out redundant TIN nodes and control the geometry of the resulting TIN object. If a TIN node is too close to another node with a similar elevation, it is identified as redundant and eliminated. The Z Tolerance parameter defines the minimum elevation difference that is allowed for nearby nodes in the final TIN object. The Minimum Edge Length parameter value quantifies “nearby”: it sets the minimum triangle edge length allowed in the output TIN.



- ☑ set the Z Tolerance value to 10.0
- ☑ set the Min. Edge Length value to 200
- ☑ set the Max. Edge Length value to 5000
- ☑ press [Run] and direct the output TIN object to the SURFOUT Project File
- ☑ in the Layer Manager, turn off the Show/Hide checkbox for the SHORELINE vector used in the previous exercise

The Maximum Edge Length parameter value sets an upper limit on the length of triangle edges in the TIN. This parameter can be useful when the set of input points has an irregular margin or includes two distinct clusters of points, and you don’t have a suitable breakline polygon available for clipping. For the point database used in this exercise, triangulation without optimization would create a number of long edges that would span indentations in the coastlines and connect the two islands. The selected value of the Maximum Edge Length parameter eliminated most of these long edges. The few remaining extraneous edges along the coastlines can be easily removed in the TNTmips Editor.



Profiling a Surface Raster

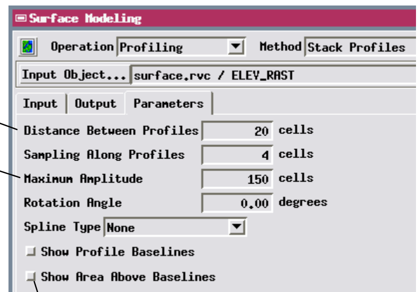
The **Profiling** operation creates a series of parallel ("stacked") vertical profiles of a surface raster. Stacked profiles provide a quick alternative means of visualizing a three-dimensional surface from different directions and with differing vertical scales. The profiles are stored as a CAD object.

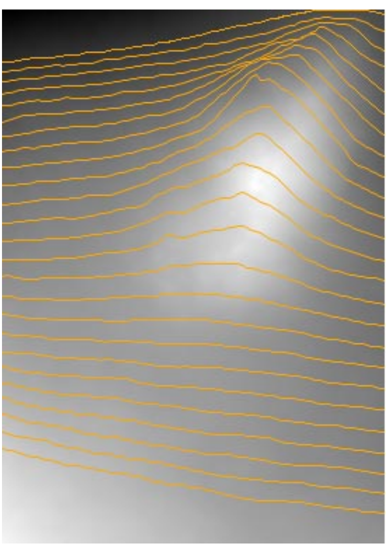
You can use the settings on the Parameters tabbed panel to control the spacing between profile lines, vertical scaling, the profile line direction, and optional profile line smoothing.

STEPS
- ☑ select Profiling from the Operation option menu
- ☑ press [Input Object] and select the ELEV_RAST object from the SURFACE Project File
- ☑ on the Parameters panel, set the Distance Between Profiles parameter value to 20 and the Maximum Amplitude to 150

The Distance Between Profiles parameter controls the profile spacing.

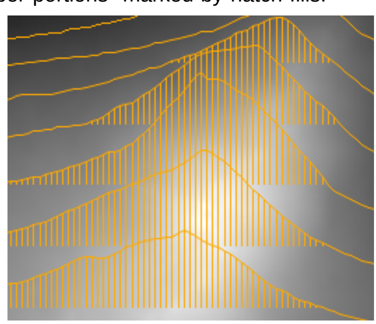
Set the maximum vertical dimension of a profile (in raster cells) with the Maximum Amplitude parameter value. You should adjust the profile spacing and amplitude in tandem to ensure that the profiles show sufficient detail without confusing overlap.





- ☑ press [Run] and direct the output CAD object to the SURFOUT Project File

Turn on the Show Area Above Baselines toggle button to create profiles with the upper portions marked by hatch fills.

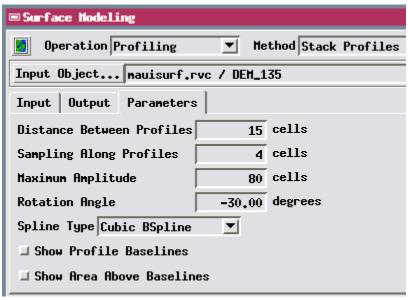


Creating Rotated Profiles

STEPS

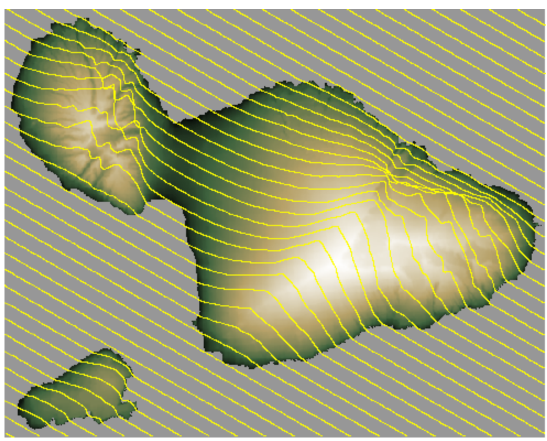
- ☑ press [Input Object] and select the DEM_135 object from the MAUISURF Project File

The default profiling direction is horizontal. To create profiles in other orientations, specify a rotation angle (positive degrees counterclockwise from horizontal or negative degrees clockwise from horizontal).



If profiles appear "noisy" (too detailed), you can either increase the value of the Sampling Along Profile parameter or smooth the profile. The profile is smoothed by splining, with a choice of either Cubic (third order) B Spline, or Quadratic (second order) B Spline methods.

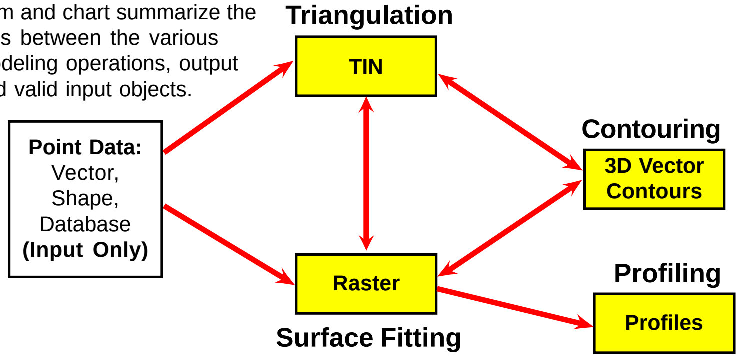
- ☑ on the Parameters panel, set the Distance Between Profiles parameter value to 15 and the Maximum Amplitude to 80
- ☑ set the Rotation Angle to -30.00
- ☑ choose Cubic BSpline from the Spline Type option menu
- ☑ press [Run] and direct the output CAD object to the SURFOUT Project File



You can easily set up DataTips to show Z-values in the output objects you create in the Surface Modeling process. Press the Layer Controls icon button in the Layer Manager for the relevant layer to open its Layer Controls window. Open the Object tabbed panel (for a raster) or the panel for the desired element type in a vector or TIN object. Choose a predefined DataTip source from the Show menu or choose Select Attribute to open the Select Table/Field window and choose the appropriate database table and field (for example, NODE.Z for TIN nodes). For further information, see the tutorial booklets *Displaying Geospatial Data* and *TNT Product Concepts*.

Surface Modeling Summary

This diagram and chart summarize the relationships between the various Surface Modeling operations, output objects, and valid input objects.



Operation	Method	Input Objects						Output Object
		Raster	TIN	Vector Lines	Vector Points	Shape	Data-base	
Surface Fitting	Minimum Curvature		Yes	Contours	Yes	Yes	Yes	Raster
	Univariate Curve			Contours				
	Inverse Distance		Yes	Contours	Yes	Yes	Yes	
	Profiles		Yes	Contours				
	Polynomial		Yes		Yes	Yes	Yes	
	Triangle Interpolation (Linear, Quintic, Nonic)		Yes		Yes	Yes		
	Kriging		Yes		Yes	Yes	Yes	
	Bidirectional			Transects				
Contouring	Linear	Yes	Yes					Vector Contours
	Iterative Thresholding	Yes						
Triangulation	Delaunay			Contours	Yes	Yes	Yes	TIN
	Adaptive Densification	Yes						
Profiling	Stack Profiles	Yes						CAD

Advanced Software for Geospatial Analysis

SURFACES

Index