Geochemistry Data Classification and Integration

In this How-To Guide, you develop strategies for dealing with high volumes of geo-data and information. The guide walks you through the key steps required to classify your data and efficiently eliminate less prospective areas from investigation, so that follow-up is focused and cost-effective. This guide also demonstrates multi-disciplinary data integration.

Geosoft provides sample data for you to use when working through these How-To Guides. These data files can be found in your ".../Geosoft/resourcefiles/data/geochemistry" folder.

The guide includes the following sections:

1. Displaying a Geological Grid
2. Creating Location Point Plots
3. Using the Scatter Tool to Define and Post Masked Data
4. Classifying by Region
5. Subsetting a Database
6. Map Editing and CAD Tools
7. Creating and Displaying a TIN Grid

Displaying a Geological Grid

Using geological data to assist in identifying potential areas of interest is a fundamental process in geochemical exploration.

⚠️ *The procedure used for opening a grid is the same for all supported grid formats.*

Displaying a Grid

1. On the *GeochemMap* menu, select *Grids*, then select *Display Grid* and then **Single Grid**.
   The *Place a grid on a map* dialog appears.
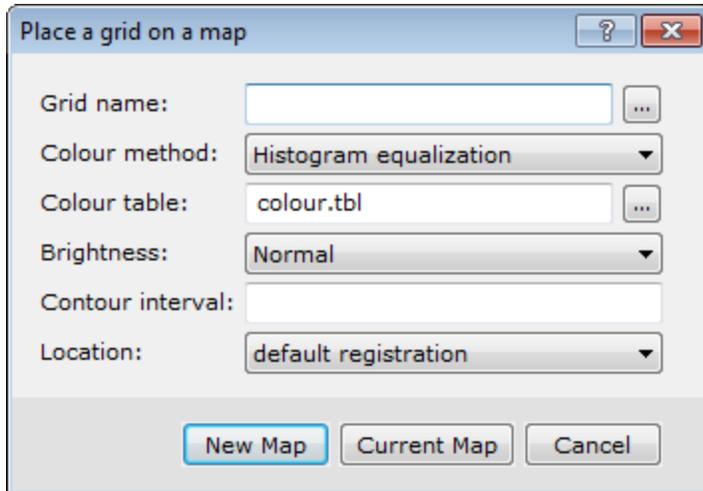
2. Use the *Browse* button to locate the *Grid name* from your project folder.

3. Define the other fields as required, or leave the intelligent default values.

4. Click **New Map** to display the grid on a new map or **Current Map** to display the grid on the current map. An example is shown below.
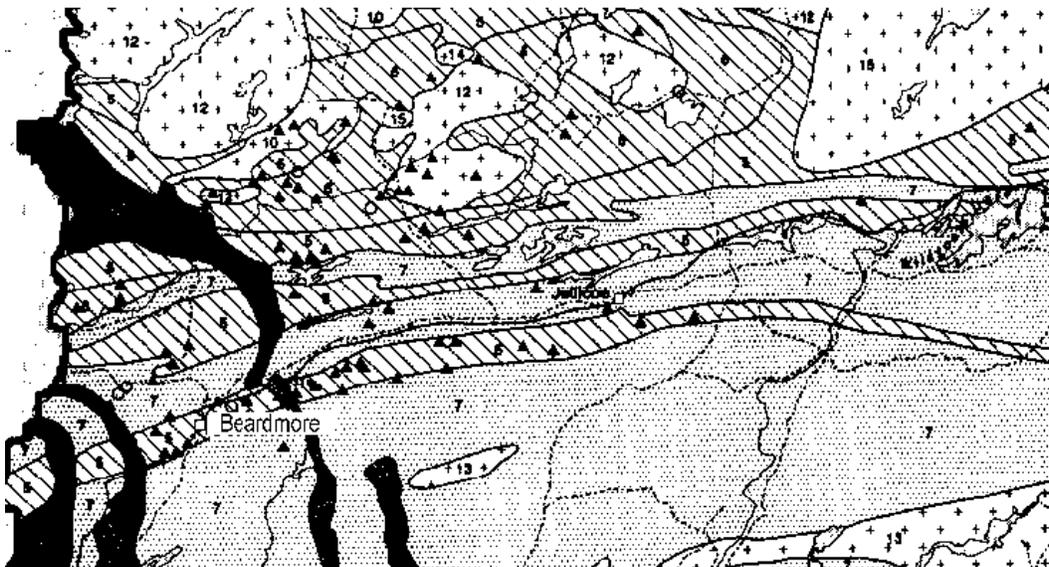

Location Point Plots

Location plots are used to represent the spatial distribution of sample locations on a map. When you use this functionality, the system searches the currently selected database for *X* and *Y* locations and plots a specified symbol on the currently selected map.
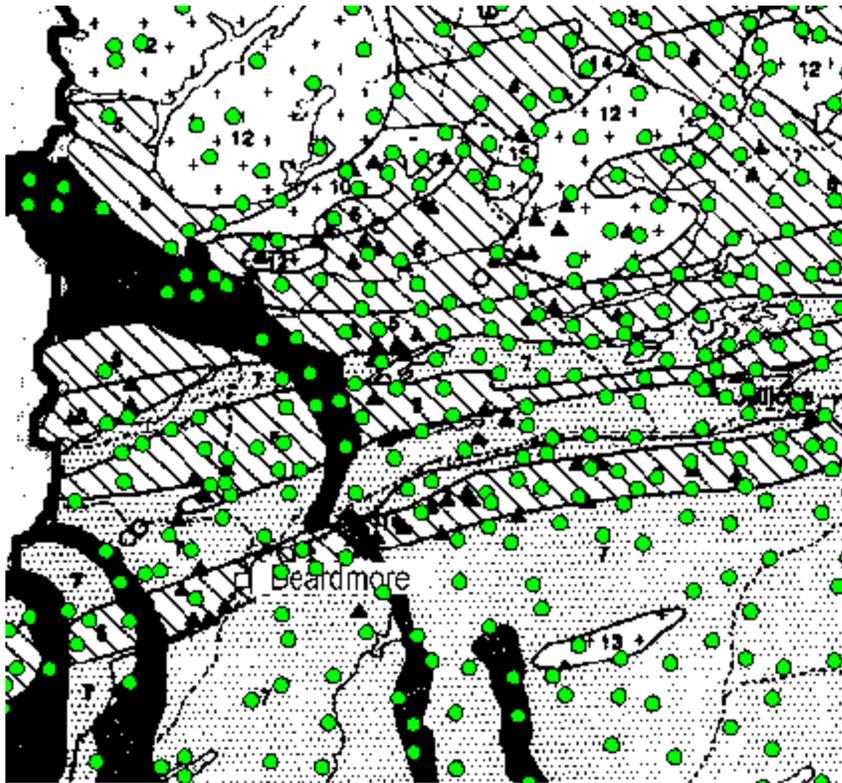
Plotting Location Point Plots

1. Make sure a map is open and selected in your project.

2. On the *GeochemMap* menu, click *Point Plots* and then select **Location plot**.
   The *Symbol plot* dialog appears.
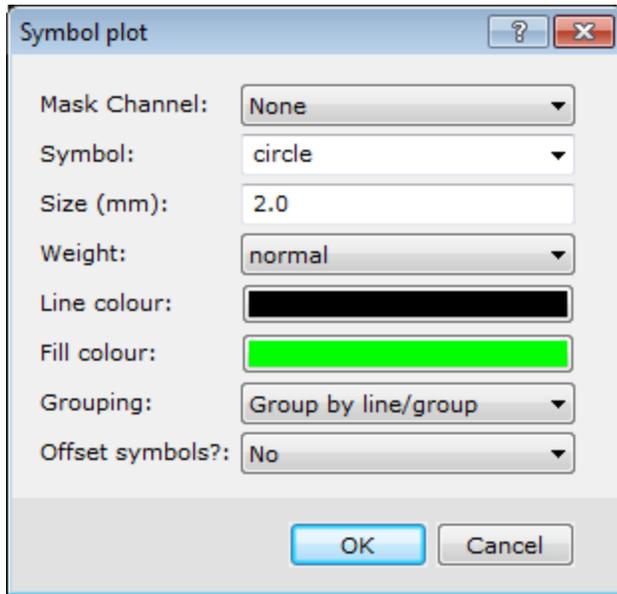
3. Specify the symbol plotting items that you want to apply and click **OK**. The system displays the symbols on your

map. An example is shown below.


Define Masking Limits with Scatter Tool

Defining mask limits and posting the mask values onto a map is one of the many features of the Geochemistry tools.

A 'mask' channel can be used for selecting a subset of data values from the database. When a mask channel is selected, it will be used and updated by the dialog as selections are plotted. A data value is highlighted in red if its corresponding mask value is not a "dummy" value.
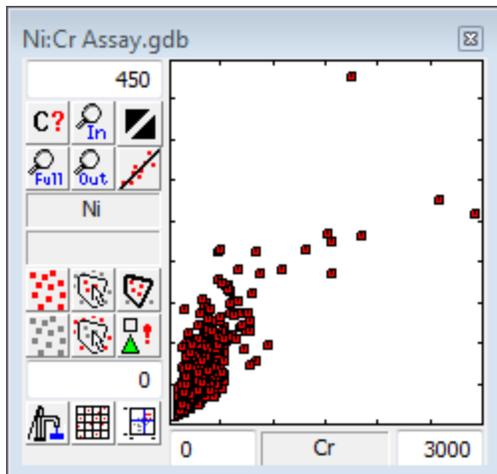
Using the Scatter Tool to Define Masking Limits

⚠️ *The Triplot tool also has the functionality to define and post mask limits on a map. The following procedure can be used with the Triplot tool, with a few modifications (i.e. you need to specify 3 channels instead of 2).*

1. Select the map with the mask channel.
2. On the *GeochemAnalysis* menu, select **Scatter Analysis**.
   The *Select scatter plot channels and symbols* dialog appears.
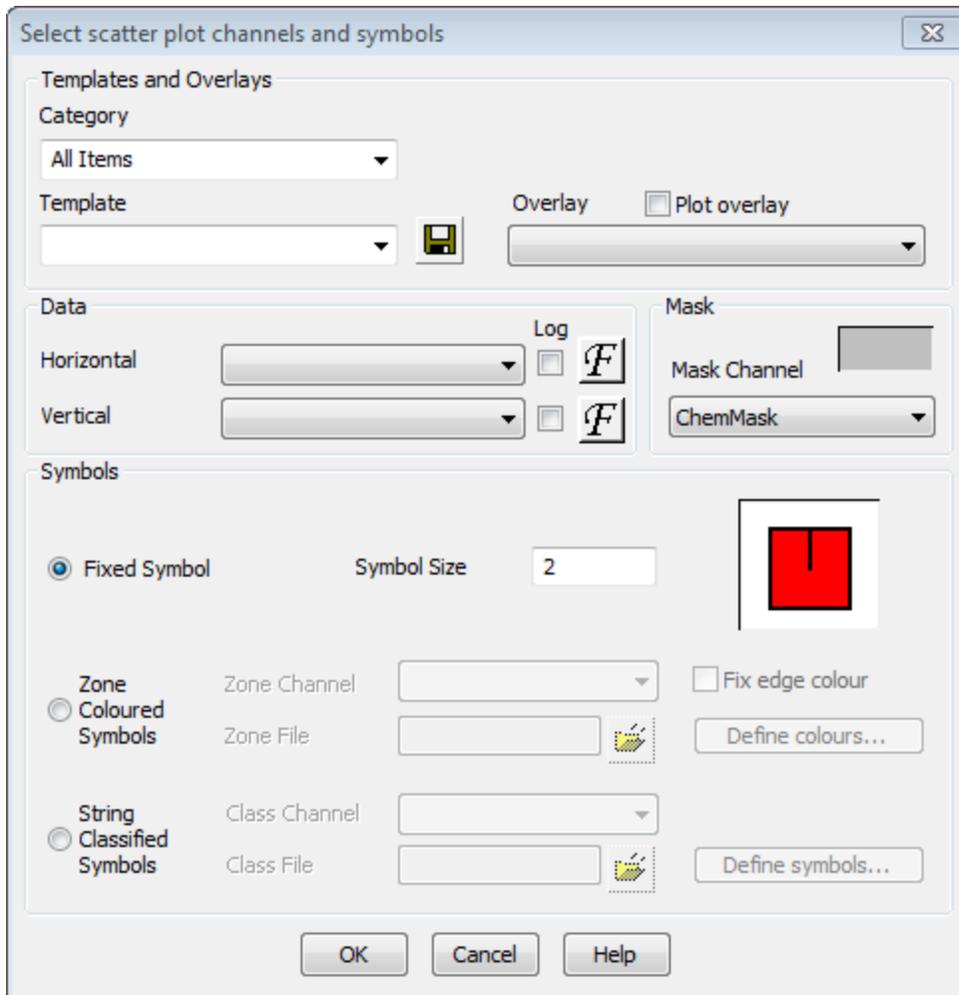
3. Select the *Template* and/or *Overlay* file to use (optional). The *Overlay*, *Horizontal* and *Vertical* parameters will be populated, based on the template settings.

4. Using the *Data* parameters, select the *Horizontal* and *Vertical* channels to display.

5. Check *Log* option for horizontal and vertical elements.

6. Select the *Mask* channel from the dropdown list.

7. Click **OK** to accept the parameters and display the *Scatter* tool.

⚠️ *If the data point values are displayed in grey, no values have been selected and no values will be plotted.*

⚠️ *Note that, the Horizontal and Vertical axis can be rescaled by typing the data range in the bottom (*

| 50	Cr	1000

) or left-side boxes.

8. To define a polygon mask, click the **Mask Using Polygon Tool** button ( ). The cursor changes from selection mode ( ) to digitizing mode ( ).

9. Select the area to be masked by drawing a polygon around data points. Once the polygon is closed, these samples will be highlighted in red.

To Plot Masked Data Points using the Scatter Tool

⚠️ *The Triplot Tool also has the functionality to define and post mask limits on a map. The following procedure can be used with the Triplot Tool, with a few modifications (i.e. you need to specify 3 channels instead of 2).*

1. Ensure that the Scatter tool is open.

2. To plot just the samples selected in the previous procedure on to the current map, click the **Plot Selected Points on the Map** ( ) button. The *Symbol Plot* dialog appears.
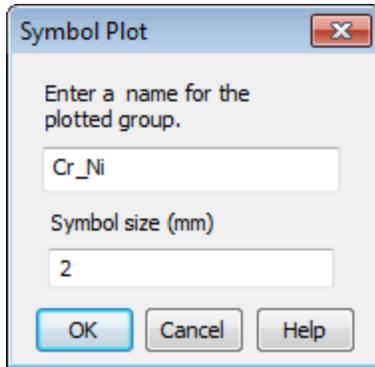
3. Use this dialog to specify the new group name. Note that the group name is automatically generated from the axis displayed, but we suggest that you change this name so that groups are not overwritten.

4. Select the *Symbol size (mm)* or you can leave it to the default.

5. Click **OK** to plot the selected data points on the current map.

Classify by Region

Creating polygonal regions is a quick process consisting of a digitizing step and a display step (in which you confirm that your polygon region is accurate). The procedures include creating a polygon mask and displaying the polygon mask outline on a map.

Creating a Polygon Mask

1. Make sure a map is open and selected in your project.

2. On the *GeochemAnalysis* menu, select *Classify by Region* and then select **Create Polygon Mask**. The *Create a polygon file* dialog appears.
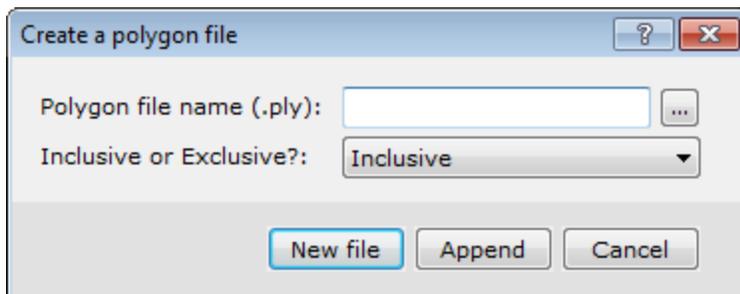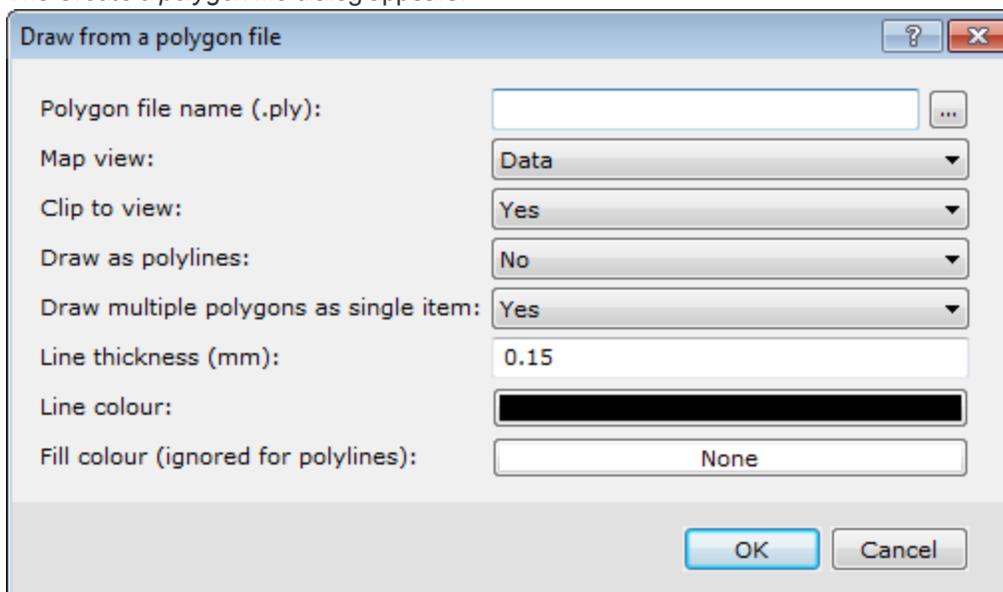

3. Specify the *Polygon file name*.

4. Select *Inclusive* or *Exclusive*).

5. Click *New file*. The system displays the *Define polygon* dialog box.
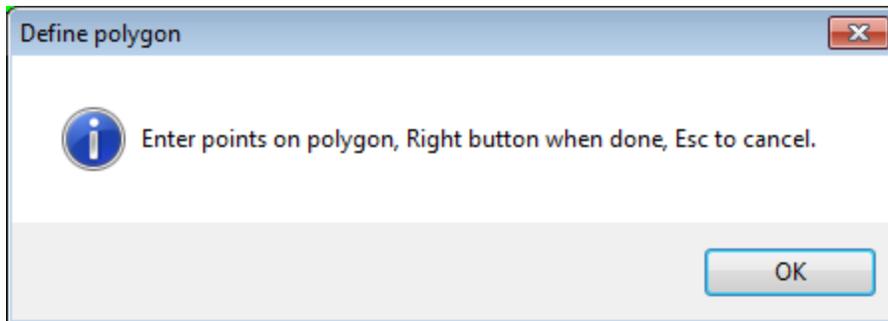
6. Click **OK** and then digitize the area of interest.

7. *Right-click* when finished and select **Done** from the pop-up menu.

8. Repeat steps 2 to 5 above, but click **Append** on the *Create a polygon file* dialog. This will include separate polygons in the same polygon file. Repeat until you have digitized the areas of interest.

Displaying Mask Outline on Map

1. On the *GeochemAnalysis* menu, select *Classify by Region* and then select **Draw Mask Outline**.
   The *Create a polygon file* dialog appears.

2. Use the *Browse* button to locate the *Polygon file name (*ply)*.

3. Specify the remaining parameters as required, or you can leave the intelligent defaults.

4. Click the **OK** button and the Mask polygon outline will be displayed on your map.

Sub-set a Database

Sub-setting a database based on a regional polygon file consists of two simple steps, applying a polygon mask to the database and sub-setting the database based on the polygon mask file.

Applying a Polygon Mask to a Database

1. Make sure a database is selected in your project.

2. On the *GeochemAnalysis* menu, click *Classify by Region* and then select **Apply Mask to Data**.
   The *Create a mask channel from a polygon file* dialog is displayed.
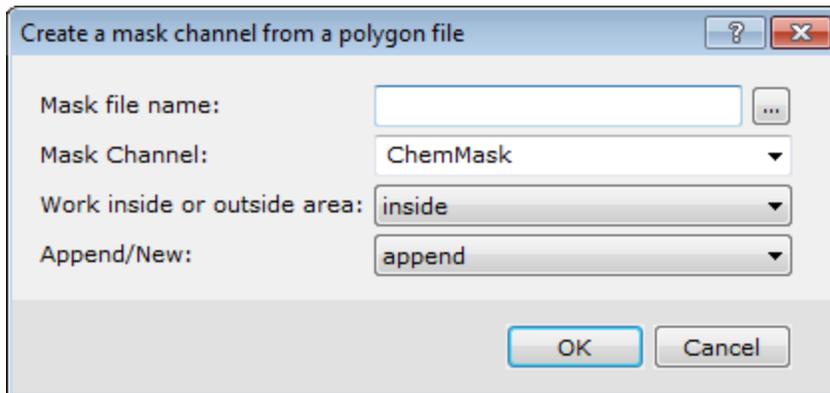
3. Use the *Browse* button to select the *Mask file name* (*.ply).

4. Select the *Mask Channel* from the dropdown list.

5. Use the dropdown lists to select *Work inside or outside area* and to *Append/New*.

6. Click **OK**. The system writes fiducial values to the selected channel.

Map Editing and CAD Tools

The map editing, masking, clipboard, and CAD tools in Oasis montaj are described in the Oasis montaj Map Editing How-To Guide. To learn how to edit maps using these tools, we recommend that you run through this How-To Guide.

The Oasis montaj Map Editing How-To Guide is designed to introduce you to the basic map editing and CAD functions available in Oasis montaj. The procedures are designed to show you how you can use these tools to draw an interpretation from geophysical data. The guide also shows how you can use the clipboard to copy, import, and export map groups and images.

Creating and Displaying a TIN Grid

The Geochemistry system enables you to create a TIN (Triangular Irregular Network), and to use this TIN for data gridding. Three data interpolation methods are available:

> Linear – Triangles in the TIN are interpolated using a plane defined by the triangle vertices
> Natural Neighbour – Uses the Natural Neighbour algorithm described by Sambride et al. (1995)
> Nearest Neighbour – Uses the values of the nodes closest to the given locations

The TIN is created from a set of spatial data using the public domain Sweepline algorithm implemented by Steven Fortune of Bell Laboratories (Fortune, S 1987). The TINDB GX applies the Sweepline algorithm to the X, Y (Z-optional) data values in a Geosoft database (*.gdb) to create a binary TIN (*.tin) file.

Creating a TIN Grid Using the Natural Neighbour Method

When Z values are included in the (*.tin) file, a TIN grid can be created using the Natural Neighbour algorithm (Sambridge, Brown & McQueen 1995) on these Z values.

1. Select the database you want to grid.

2. On the *GeochemMap* menu, click *Grids/Tinning* and then select **Create TIN File from Database**.
   The *Create a TIN from X, Y positions* dialog is displayed.
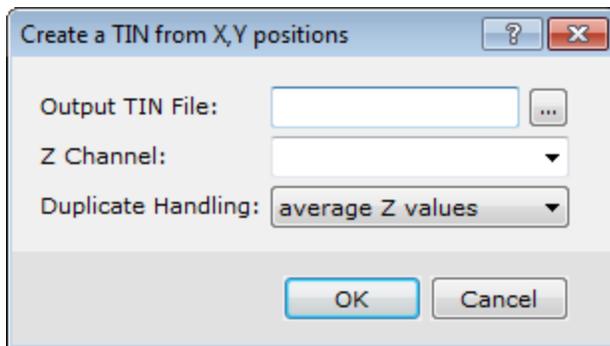
3. In the *Output TIN File* field, enter the file name (for example "Cu.tin").

4. Use the *Z Channel* dropdown list to select the channel which holds the Z values.

5. Use the *Duplicate Handling* dropdown list to select *average Z values* or *sum z values*. The TIN procedure cannot handle more than a single data point at any (X, Y) location. This parameter determines what to do with the "Z" values for duplicated locations.

6. Click **OK** to create the TIN file.

7. On the *GeochemMap* menu, click *Grids/Tinning* and then select **Natural-Neighbour TIN Gridding**.
   The *Grid a TIN* dialog is displayed.
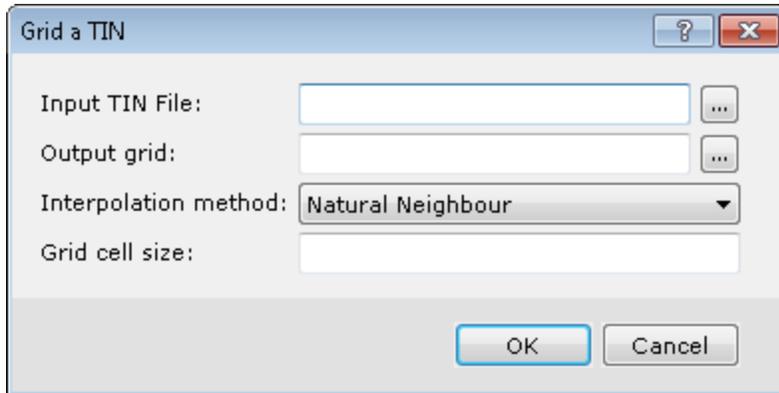
8. In the *Input TIN File* field, enter the file you created above.

9. In the *Output grid* field, enter the name of the grid file.

10. On the *Interpolation method* dropdown list, select **Natural Neighbour**.

11. Set the *Grid cell size*. If this value is not specified, a default is chosen, based on the average density of points in the TIN. (The cell size is chosen to give a density about twice this value).

12. Click **OK**.The grid is created and displayed in your project workspace.

⚠️ *In Geochemistry, grids and images are displayed on a map in the Map window. Grids are listed in the Project Explorer pane. For quick viewing, newly created grids are automatically displayed in a temporary map, using default display options. When you open a grid, it opens as a view of the grid in its own internal "map". This grid view is like a real map, and can include the same graphics as a map. It can also be saved as a map, in which case the map will appear under the Map node in the Project Explorer pane.*
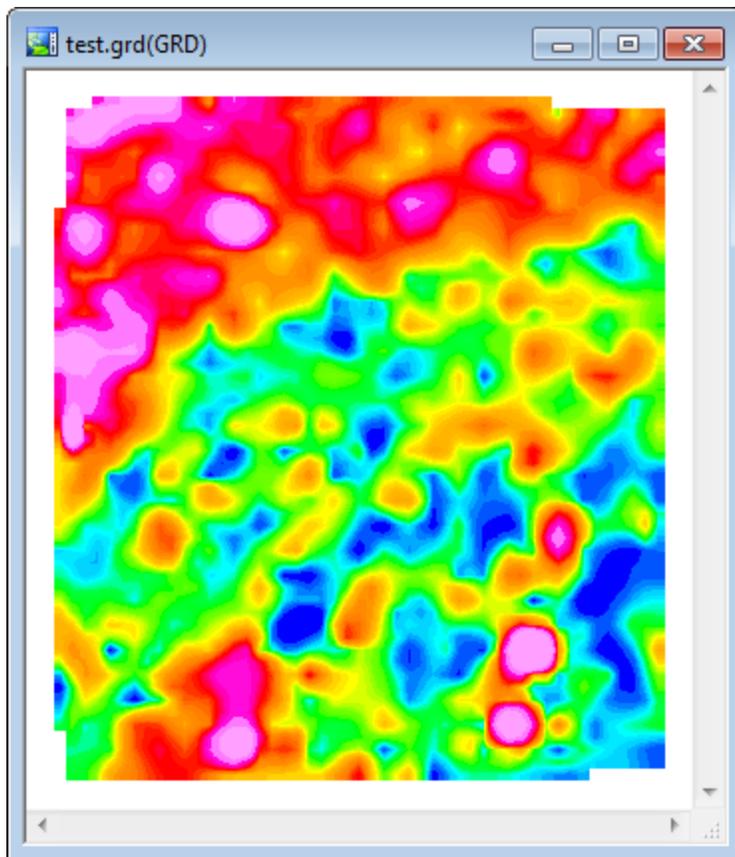
Displaying a TIN Grid on a Map

1. Select the required map.

2. On the *GeochemMap* menu, click *Grids / Display Grid* and then select **Colour-shaded grid**. The *Color-shaded grid image* dialog appears.
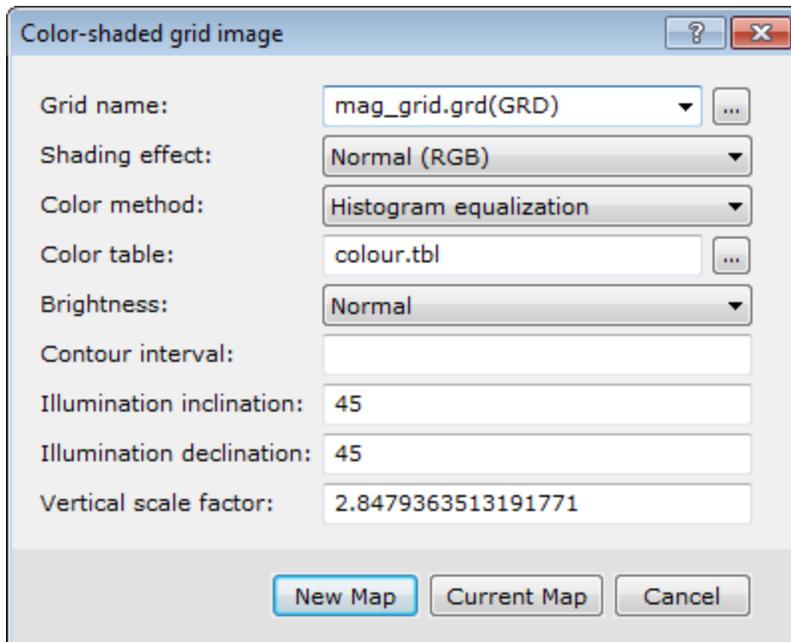
3. Use the *Browse* button to locate the *Grid name* from your project folder.

4. Using the dropdown lists, select the *Shading effect* and *Colour method*.

5. Select the *Colour table* from the "*../Geosoft/Oasis montaj/resource files/tbl/*folder.

6. You can leave the remaining parameters to the default values, or fine-tune the way the grid will look.

7. Click **New Map** to display the grid on a new map or **Current Map** to display the grid on the current map. An example is shown below.
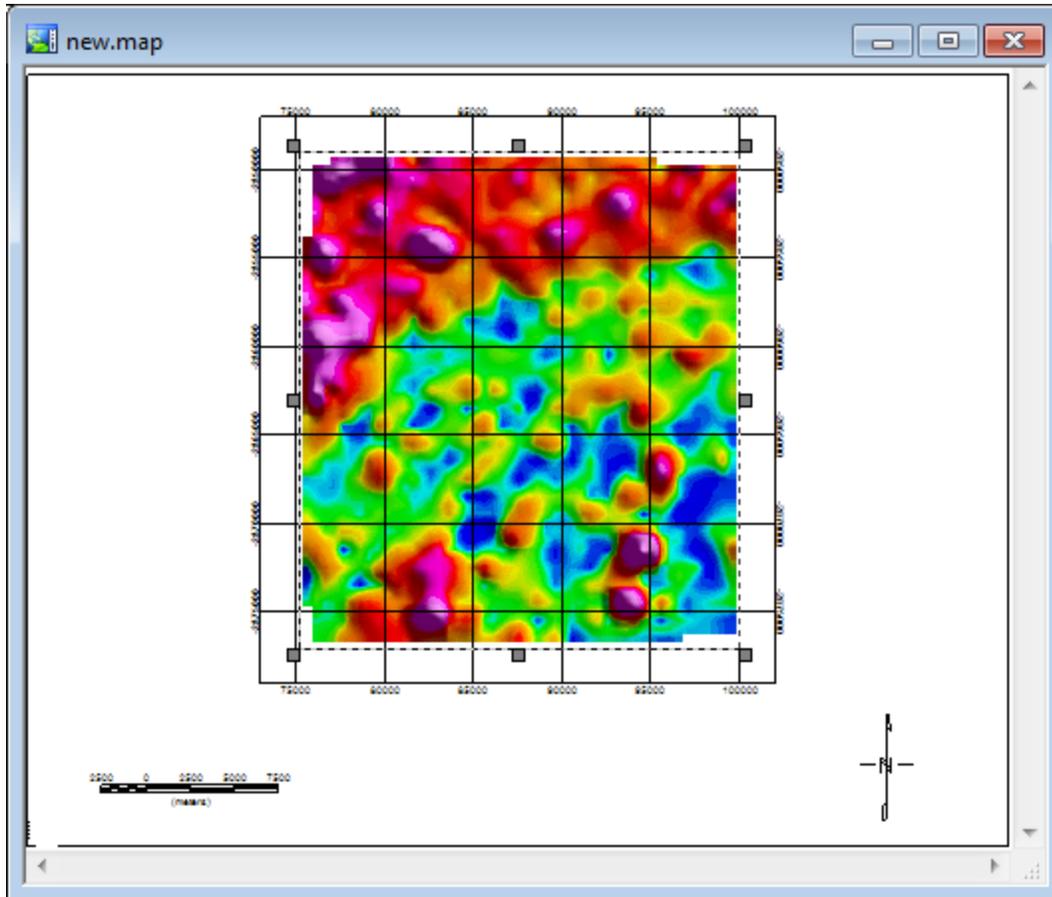
⚠ *Geosoft Tinning provides a number of ways for visualizing the TIN, including the ability to plot TIN Nodes, TIN Mesh (or Delaunay triangulation), Convex Hull, and the Voronoi cells of your data. For more information on these topics, download the technical note, Tinning - Triangular Irregular Network Gridding for Oasis montaj from www.geosoft.com/resources/technotes/.*

Publication Date: December-16-15