



Target for ArcGIS

Course Manual



The software described in this manual is furnished under license and may only be used or copied in accordance with the terms of the license.

Manual release date: Thursday, June 21, 2018.

Please send comments or questions to info@geosoft.com

© 2017 Geosoft Inc. All rights reserved. Geosoft is a registered trademark and Oasis montaj is a registered trademark of Geosoft Inc. Other brand and product names mentioned herein are properties of their respective trademark owners. No part of this publication may be reproduced, stored in a retrieval system or transmitted, in any form, or by any means, electronic, mechanical, photocopying, reading, or otherwise, without prior consent from Geosoft Inc.

The software described in this manual is furnished under license and may only be used or copied in accordance with the terms of the license. TFAI.cm.2013.02TFAll.cm.2013.09

Windows™, and Windows NT are either registered trademarks or trademarks of Microsoft Corporation.

Geosoft Incorporated
Queen's Quay Terminal
207 Queen's Quay West
Suite 810, PO Box 131
Toronto, Ontario
M5J 1A7
Canada
Tel: (416) 369-0111
Fax: (416) 369-9599

Website: www.geosoft.com

E-mail: info@geosoft.com

Table of Contents

Table of Contents	i
Introduction: Target for ArcGIS	1
Module 1: Mapping Surface Data	3
Lesson 1.1 Creating an MXD and Adding Data	4
Lesson 1.2 Displaying Data	10
Lesson 1.3 Creating Gridded Data	18
Lesson 1.4 Creating Geosoft Contours	30
Module 2: Managing Drillhole Data	41
Lesson 2.1 Creating a Drillhole Project and Importing Data	42
Applying Filters When Importing Data	50
Lesson 2.2 Working with Geosoft Databases and Profiles	54
Lesson 2.3 Using QA/QC Tools	65
Lesson 2.4 Creating Composite Databases	69
Lesson 2.5 Using Drillhole Data Management Tools	76
Module 3: Mapping Drillhole Data	85
Lesson 3.1 Selecting Drillholes	86
Lesson 3.2 Creating a Plan Map	89
Lesson 3.3 Creating a Section Map	106
Lesson 3.4 Creating a Stacked Section Map	123
Lesson 3.5 Creating a Fence Diagram	129
Lesson 3.6 Creating a Strip Log	134
Lesson 3.7 Creating a 3D Map	147
Module 4: Implicit Modelling	157
Lesson 4.1 Creating Voxels	158
Lesson 4.2 Creating Isosurfaces	164
To create and display the isosurface:	165
Lesson 4.3 Creating a Lithology Voxel and Contact Surface from Geology	170
Lesson 4.4 Extracting Slices from a Voxel	175
Module 5: Explicit Modelling	177
Lesson 5.1 Creating an fGDB and Adding Feature Classes	178
Geological Interpretation and Wireframing	178
Lesson 5.2 Digitizing Interpretations on Section Maps	182
Lesson 5.3 Editing Feature Classes	186
Lesson 5.4 Wireframing Interpretations	190

Introduction: Target for ArcGIS

Target for ArcGIS

Target for ArcGIS™ is a surface and drillhole mapping extension to Esri's ArcGIS™ software that simplifies the visualisation, mapping and analysis of subsurface drillhole and borehole geology data within your Esri GIS environment. Built on the same technology as Geosoft's industry-leading **Oasis montaj®** software, **Target for ArcGIS** adds powerful spatial data processing capabilities and 3D viewing to your GIS.

About this Training Manual

The **Target for ArcGIS** course manual is for use in a classroom setting with a qualified Geosoft instructor. Your instructor is available throughout the course to answer any questions you may have.

Each module in this book contains a series of hands-on lessons that let you work with the software and a supplied dataset. Lessons have brief introductions followed by tasks and procedures in numbered steps.

The following modules are included in this course:

- › Module 1 - Mapping Surface Data
- › Module 2 - Managing Drillhole Data
- › Module 3 - Mapping Drillhole Data
- › Module 4 - Implicit Modelling
- › Module 5 - Explicit Modelling

Course Data

The data used in this course is contained in a folder called **Geosoft Training Data**. Please copy the whole folder of data to your C:\ drive or other hard drive where you have at least 200 MB of available space.

Module 1: Mapping Surface Data

Module 1 has Four Lessons:

Lesson 1.1	Creating an MXD and Adding Data	4
Lesson 1.2	Displaying Data	10
Lesson 1.3	Creating Gridded Data	18
Lesson 1.4	Creating Geosoft Contours	30

Lesson 1.1 Creating an MXD and Adding Data

In this lesson you will:

- Create a new map document
- Add data layers
- Add x,y tabular data and assign a coordinate system
- Save the map document

Creating an MXD

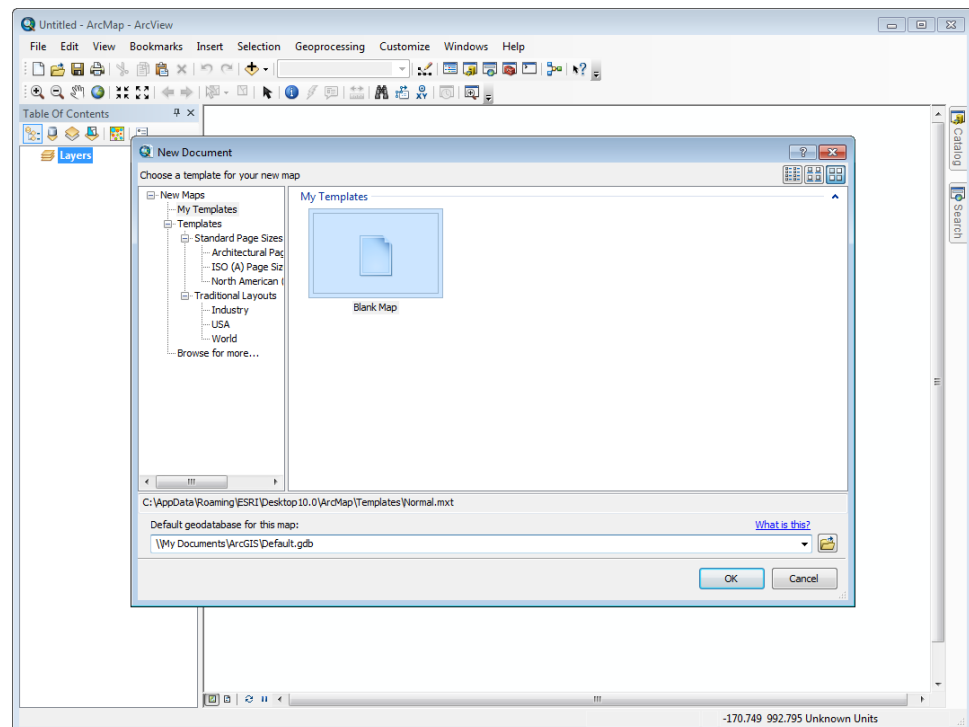
An .MXD is the native project file format for Esri ArcGIS. It contains the project map and any data contained within, as well as information about how those layers are displayed. You will begin by creating a new ArcGIS map document.

To create and save an MXD:

1. Start **ArcMap**.

The ArcMap starting dialog opens.

Figure 1.1 ArcMap starting dialog



2. Ensure **Blank Map** is selected and click **OK**.
3. From the **File** menu, select **Save**.
4. Locate the **Geosoft Training Data** folder.

This is the location where all your data output will be saved.

5. For the **File name**, enter **Surface Mapping.mxd** and **Drillhole Mapping.mxd** and click **Save**.

Now that you have started ArcMap and created an .MXD file, you will display the **Target for ArcGIS** toolbars.

Displaying Target for ArcGIS Toolbars

Once you have installed **Target for ArcGIS** and opened ArcMap, you need to display the **Geosoft Target for ArcGIS** toolbars. The toolbars can then be docked in various locations.

To display the Target for ArcGIS toolbars:


1. From the **Customize** menu, select **Toolbars** and then select **Geosoft Database**.
The Geosoft Extensions Setup and DAP toolbar opens.
2. Repeat step 1 to display the **Geosoft Extensions Setup and Seeker**, **Target Drillhole**, and **Target Surface** and **Target Wireframing** toolbars.

You will now add data to your map document.

Adding Data

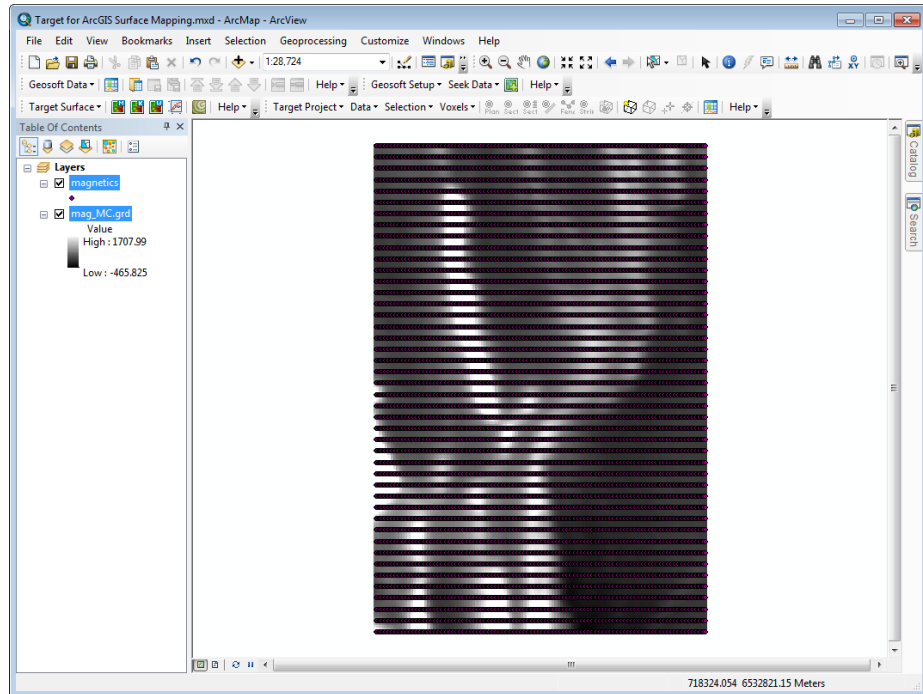
Any data that you add to your map is represented as a layer in the MXD Table of Contents. You can add data from folders (such as shapefiles and Geosoft grids) and from databases.

To add data to a map:

1. On the Standard Toolbar, click the **Add Data** button .
2. Locate the **Geophysics** folder.
3. Hold down the **CTRL** key, select **mag_MC.grd** and **magnetics.shp** and click **Add**.

The mag_MC.grd and magnetics.shp layers are listed in the Table Of Contents and displayed in the map display area. By default, the mag_MC grid is displayed as greyscale.

Figure 1.2 Data added to map



Adding XY Data and Defining the Coordinate System

In **Target for ArcGIS** you can create Geosoft grids, by interpolating data that is stored within shapefiles and event layers. However, in order for the resulting grid to have a coordinate system, you must grid either a shapefile or an event layer that already has a defined coordinate system. When you do this, the output grid will inherit the coordinate system of the source data.

In this example, you will add ASCII data into the MXD as a layer and then define its coordinate system. Any grid created from this x,y coordinate event layer will inherit its coordinate system



When adding x,y data, the coordinate system defaults to that of the current data frame. You must ensure the coordinate system is appropriate for the data you are adding.

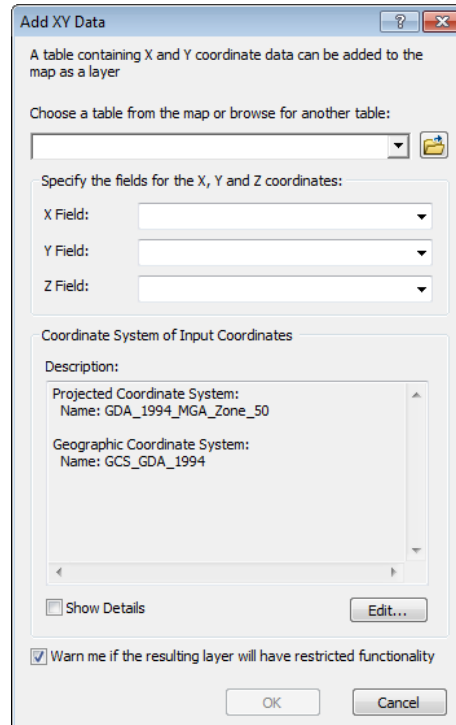


To add x,y data, your coordinates must be in projected units or decimal degrees.

To add x,y data and define the coordinate system:

1. From the **File** menu, select **Add Data** then **Add XY Data**.

The Add XY Data dialog opens. The coordinates of the current data frame are listed by default.

Figure 1.3 Add XY Data dialog

2. Click the **Browse** button .

The Add dialog opens.

3. From the **Geochemistry** folder, select **geochemistry_mga.csv** and click **Add**.
4. From the **X Field** list, select **East**.
5. From the **Y Field** list, select **North**.

The data that we first imported into the project has the "GDA_1994_MGA_Zone_50" projected coordinate system assigned to it. Because of this, any new data that is imported will by default be assigned to that system. You can change the coordinate system by clicking the "Edit button". In our case though, the displayed coordinate system is the one we want to assign to our new data.

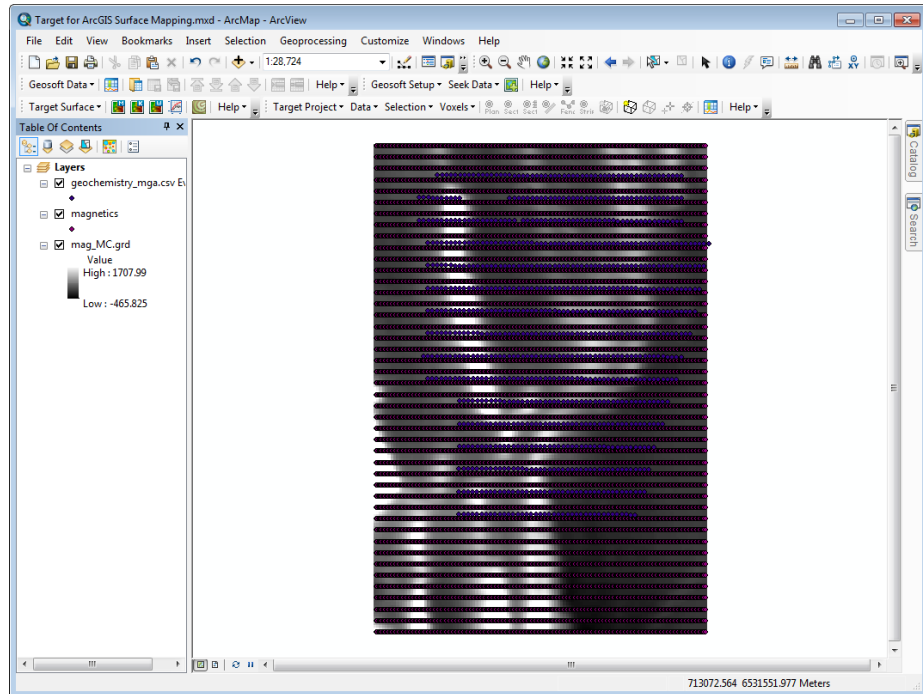
6. Click **OK**.

A message window opens indicating the table does not have an Object-ID field.

7. Click **OK**.

The geochemistry_mga.csv Events layer is listed in the Table Of Contents and displayed in the map display area.

Figure 1.4 Tabular data added as X,Y Event layer



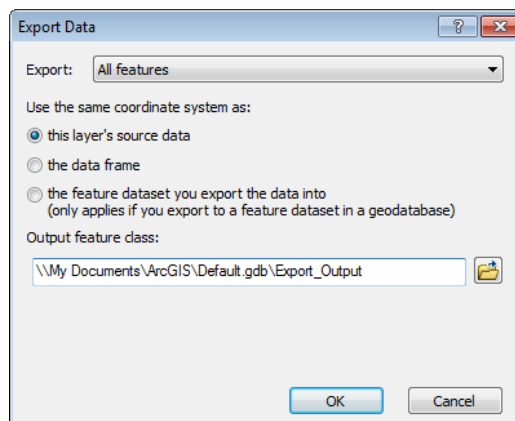
The XY Event data (geochemistry_mga.csv Events) only exists in the current map document. To save the layer as a separate file, you must export it as a shapefile or feature class.


To save the layer as a shapefile:

1. In the Table Of Contents, right-click the **geochemistry_mga.csv Events** layer and select **Data** then **Export Data**.

The Export Data dialog opens.

Figure 1.5 Export Data dialog



2. For **Output feature class**, click the **Browse** button .
3. Locate the **Geosoft Training Data** folder.
4. For **Name**, enter **geochemistry**.
5. From the **Save as type list**, select **Shapefile** and click **Save**.

6. Click **OK**.

An ArcMap message window opens asking if you would like to add the exported data to the map as a layer.

7. Click **Yes**.

The geochemistry.shp file is saved in the specified folder and is displayed as a layer on your current map.

Because you have created a shapefile of the surface geochemistry data, you can remove the Events layer.

To remove the Events layer:

- › In the Table Of Contents, right-click the **geochemistry_mga.csv Events** layer and select **Remove**.

Saving a Map Document

When you save a map document, the data itself is not saved, but rather it saves references to the data sources and their display properties.

To save your map document:

- › From the **File** menu, select **Save**
- OR -
- › Click the **Save** button on the Standard toolbar.



If you close your map document without saving it, you will be prompted to save changes to your .MXD file.

In this lesson you:

- › Created a new map document
- › Added data layers
- › Added ASCII data and assigned a coordinate system
- › Converted an Events Layer to a Shapefile
- › Saved the map document

Lesson 1.2 Displaying Data

In this lesson you will:

- > Change the grid display
- > Apply a colour ramp to your grid
- > Symbolise point data using colour and symbol sizes
- > Manually classify symbol ranges
- > Save symbolised points as a layer file

Displaying Geosoft Grids

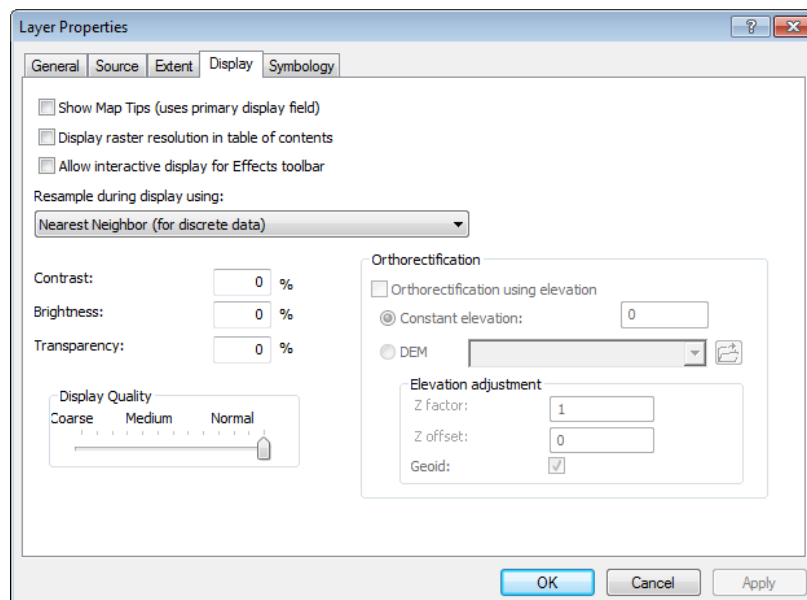
You can modify the display of Geosoft grids in many ways in ArcMap, including assigning different colour ranges, applying a shaded effect, and applying transparency.

You will begin by changing the resampling method used to display the grid layer. Then, you will select a colour ramp and choose how that colour is mapped to the data.

To change the grid display:

1. In the Table Of Contents, right-click **mag_MC.grd** and select **Properties**.
The Layer Properties dialog opens.
2. Click the **Display** tab.

Figure 1.6 Layer Properties Display tab



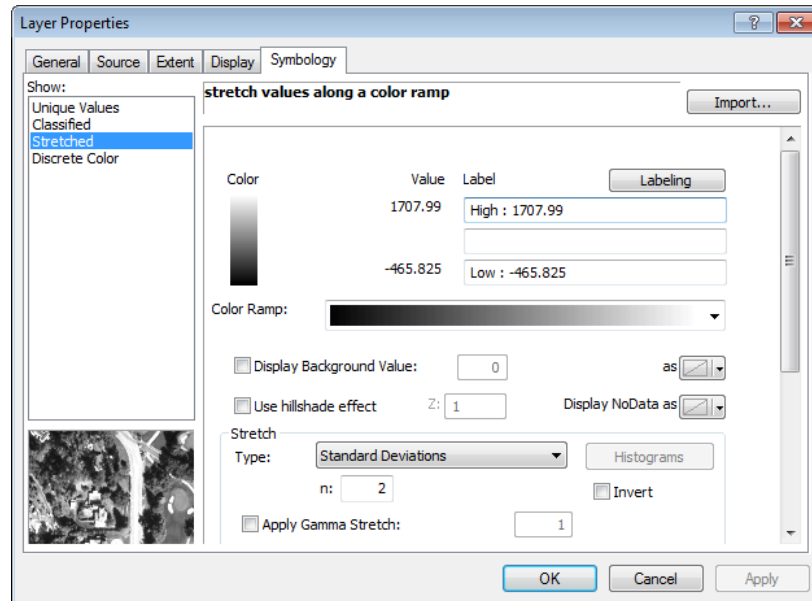
3. From the **Resample during display using** list, select **Bilinear Interpolation (for continuous data)**.
This will give a smooth appearance to your raster data.
4. Click **Apply**.

Now, you will select a colour ramp and how you want to map that colour to the data - the colour "stretch".

To apply a colour ramp:

1. Click the **Symbology** tab.

Figure 1.7 Layer Properties Symbology tab



2. Right-click in the **Color Ramp** list and click to clear the **Graphic View** option. The Color Ramp view switches from a graphic view to text.
3. From the **Color Ramp** list, select **Geosoft Default**.
4. Click **Apply**.

The Geosoft Default colour ramp is applied to the data; low values are shown in blue and as the values increase, the colours change from green to yellow, orange, red, and magenta.

Next, you will modify the colour display by changing the colour stretch.

To modify the colour stretch:

Note that the default color stretch in ArcMap is the "Standard Deviations" type.

1. From the **Type** list in the Stretch section of the tab, select **Histogram Equalize**.

The Compute Histogram message window opens asking you how you would like to estimate the histogram.

2. Click **Yes**.
3. Click **OK**.

The grid is now displayed with a histogram equalize stretch. This stretch is designed to enhance mid-range features in data, by assigning an equal number of data points to each colour in the colour ramp.

Examine the colour display of the grid layer. You may wish to turn off the display of the magnetics and geochemistry point layers or change the display order of the layers in your data frame.

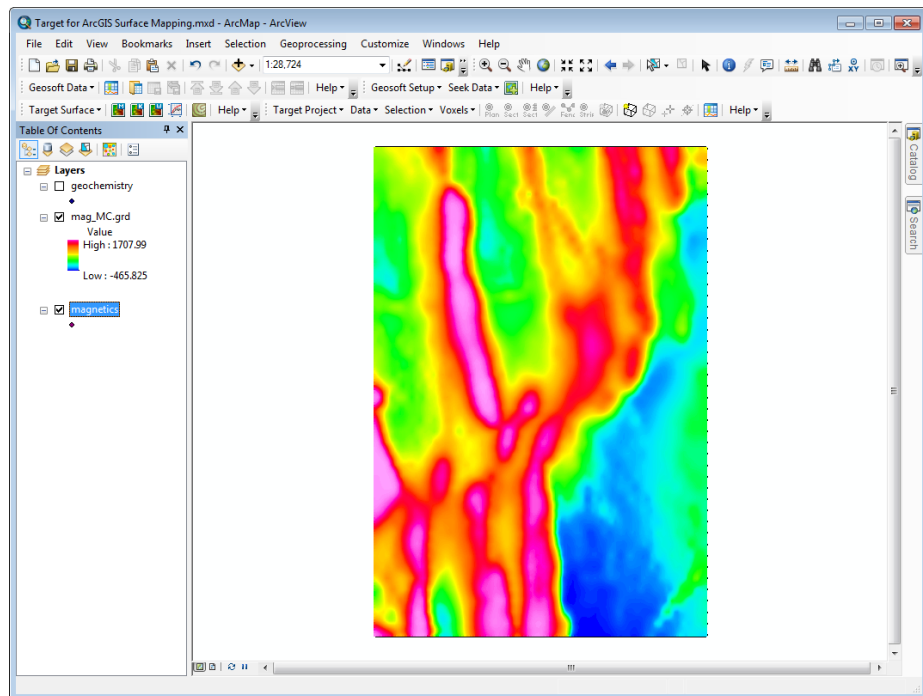
To turn off the display of a layer:

- In the Table Of Contents, clear the check box beside the **geochemistry** layer.
This layer is no longer displayed.

To change the display order:

- In the Table Of Contents, select the **magnetics** layer and drag it below the **mag_MC.grd** layer.
The mag_MC.grd layer is now displayed on top of the magnetics layer.

Figure 1.8 Grid with coloured display



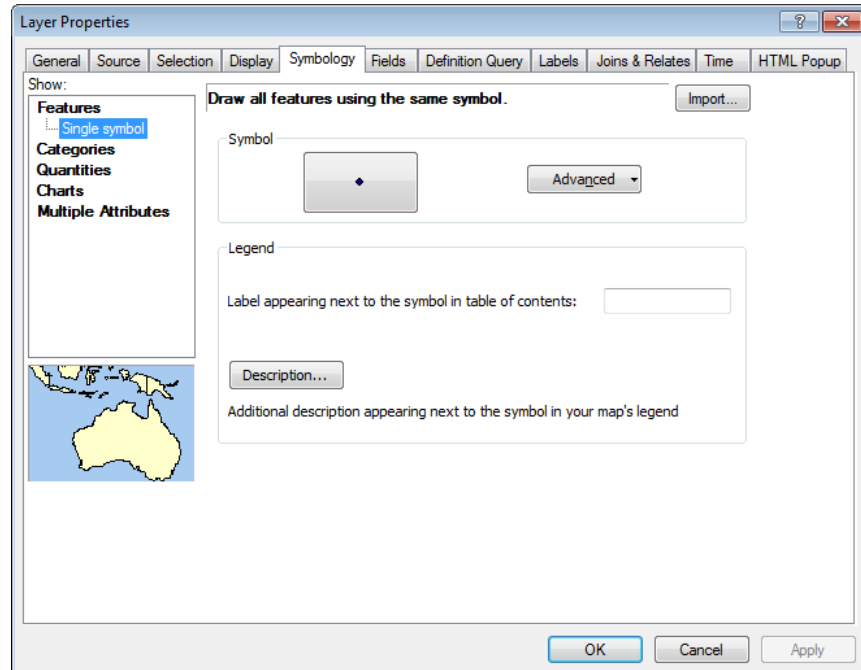
Symbolising Point Data

Now you will explore displaying data as symbols. You will modify the display of your soil geochemistry layer in your project to show symbols that change in colour and size, according to gold concentration.

To symbolise point data based on quantity:

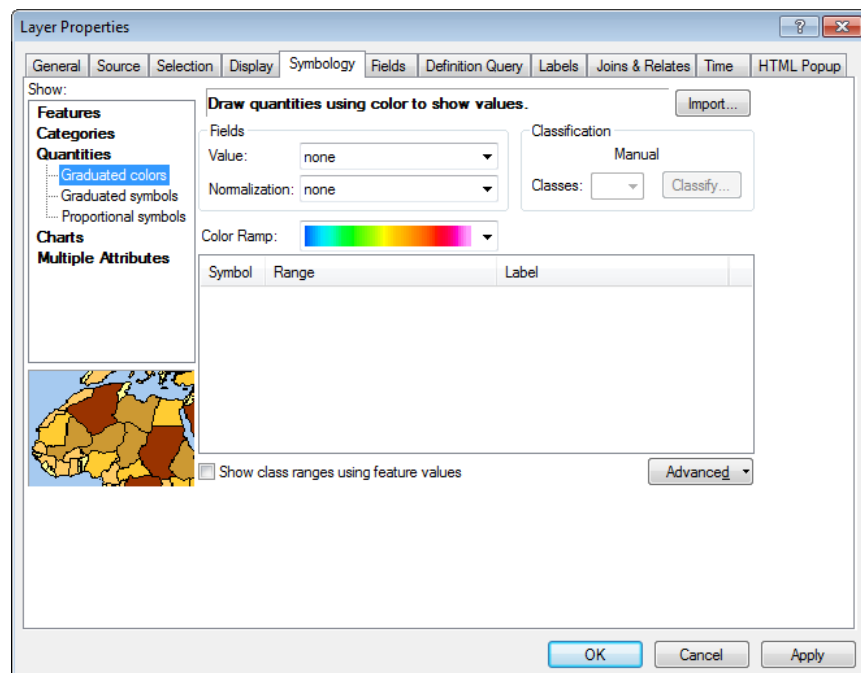
1. In the Table Of Contents, click the check box beside the **geochemistry** layer.
This layer is now visible again.
2. Right-click the **geochemistry** layer and select **Properties**.
The Layer Properties dialog opens.
3. Click the **Symbology** tab.

Figure 1.9 Layer Properties Symbology tab for point data



- In the Show area on the left, click **Quantities**.
ArcMap automatically selects Graduated colors.

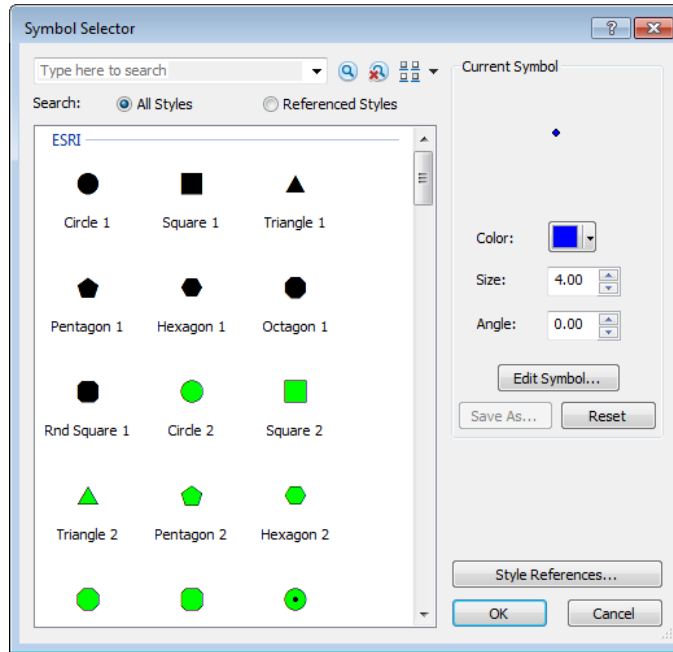
Figure 1.10 Layer Properties Symbology tab with Graduated colors selected



- From the **Value** list, select **Au**.
The Symbol, Range and Label columns update to show the data classified based on the natural statistical groupings of data values.
- For the **Color Ramp**, select **Geosoft Default**.

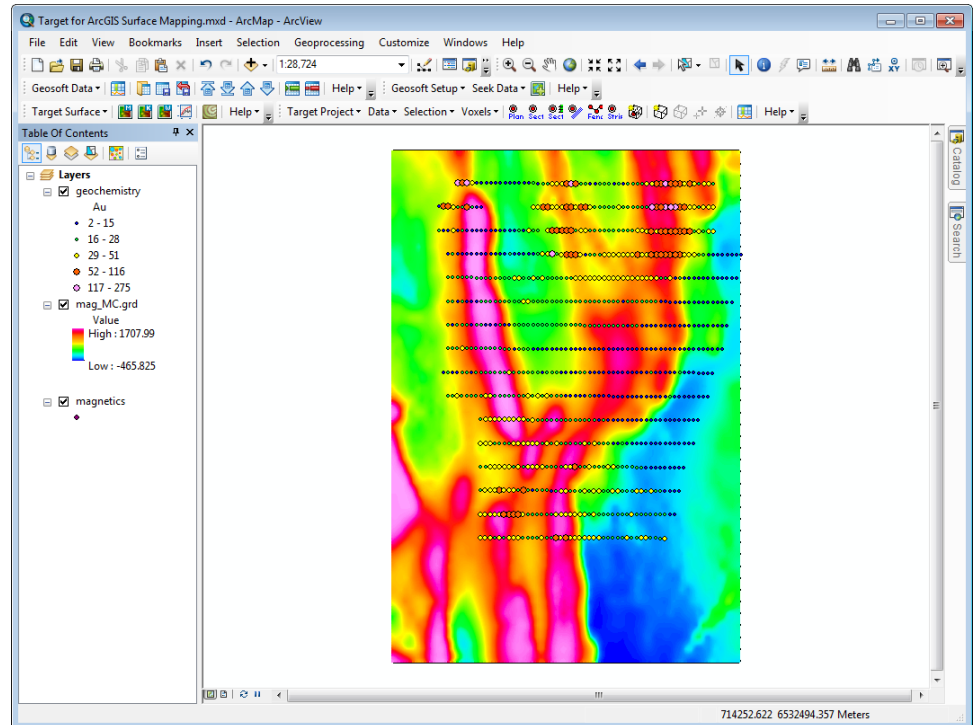
7. Double-click the first class in the Symbol column.
The Symbol Selector dialog opens.

Figure 1.11 Symbol Selector dialog



8. Change the **Size** to **2** and click **OK**.
9. Double-click the symbol for each remaining class and change the size so they range from **3** to **6**.
10. Click **Apply**.
The points are now displayed using colour and size to represent the amount of Au at each sample location.

Figure 1.12 Coloured point symbols



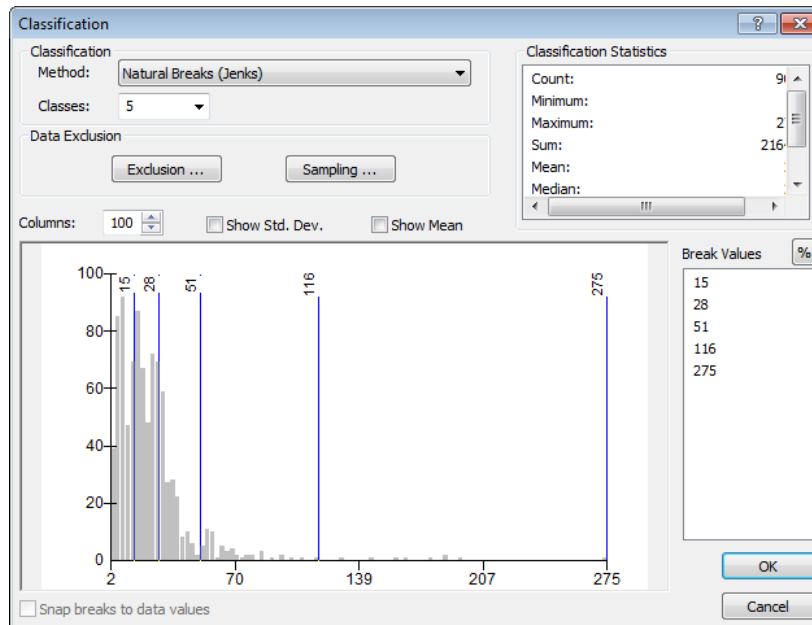
The symbol properties for this point layer are shown in the Table Of Contents; you can see the colour, size and classes that are used to display the data.

Next, you will manually edit the class ranges based on the data distribution. You are able to manually define significant or useful breaks in the data so you can compare it with other maps or identify populations in the data distribution.

To classify the point symbols manually:

1. In the Symbology tab of the Layer Properties dialog, click **Classify**.
The Classification dialog opens showing the default Natural Breaks classification method.

Figure 1.13 Classification dialog



The data in this example is log distributed; there is a lot of data in the low end and very little data in the high end. Notice the Classification Statistics area in the Classification dialog. The Mean for this data is 24 and the Standard Deviation is 23. You will use this information to determine the classification Break Values.

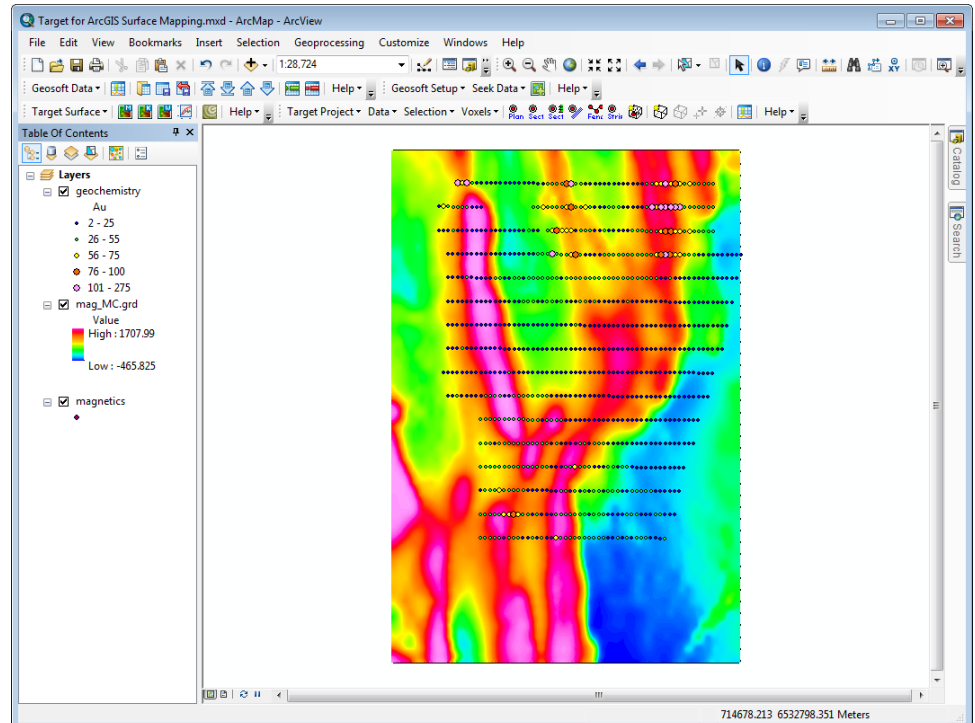
- In the **Break Values** area, click the default values and enter the following values:

Break Value	Explanation
25	Approximate Mean value
50	Mean value plus approximately 1 standard deviation
75	Mean value plus approximately 2 standard deviations
100	Mean value plus approximately 3 standard deviations
275	Maximum value

- Click **OK**.
- In the Layer Properties dialog, click **OK**.

The display of the symbols updates based on the redefined classes.

Figure 1.14 Manually classified point symbols



Saving as a Layer File

A .LYR file stores any symbology attributes used to display the data. It is a stand alone file that can be shared with others and applied to other datasets. For example, you might be working with others that are analysing geochemical data from the same region as yours and you want to display your data in the exact same way; a shared LYR file allows you to do this.

To save the shapefile as a layer file:

1. In the Table Of Contents, right-click the **geochemistry** layer and select **Save As Layer File**.
The Save Layer dialog opens.
2. If necessary, select the **Geosoft Training Data** folder from the **Look in** list.
3. Ensure the layer **Name** is **geochemistry.lyr** and click **Save**.
The layer file is saved on disk.

In this lesson you:

- Changed the grid display
- Modified the colour ramp and colour stretch your grid
- Symbolised point data using colour and symbol sizes
- Manually classified symbol ranges
- Saved symbolised points as a layer file

Lesson 1.3 Creating Gridded Data

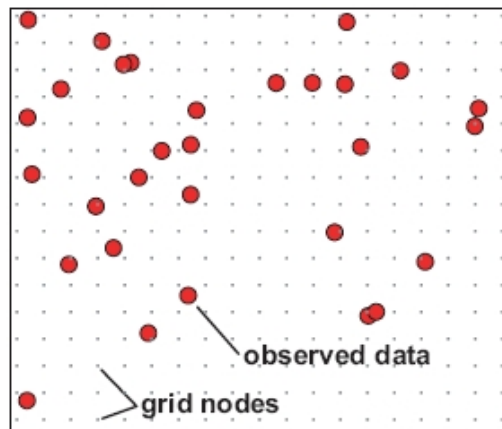
In this lesson you will:

- › Create a grid using the minimum curvature method
- › Create a grid using the minimum curvature advanced options
- › Create a grid using the kriging method

Gridding Data

Gridding data is the process of spatial interpolation. The process of gridding takes point data and interpolates to determine the values at the nodes of a grid in between the data points. The resulting interpolated dataset is known as a grid.

Figure 1.15 Gridding process

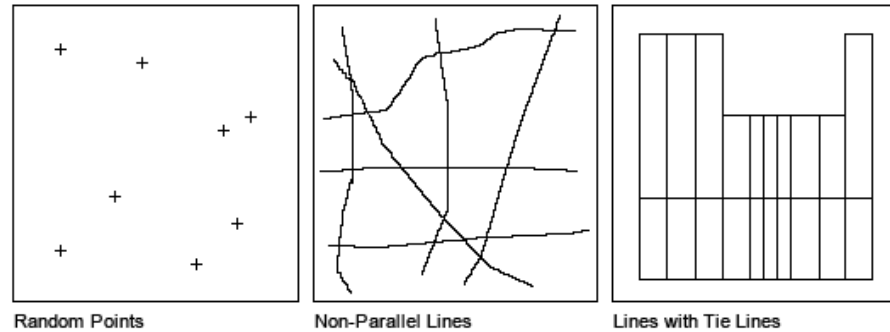


Target for ArcGIS provides two methods for interpolating raw XYZ data to create a grid:

- › **Minimum Curvature:** Interpolates the data by fitting a minimum curvature surface to the raw XYZ data points. This method is ideal when data is sparsely sampled and a relatively smooth variation is expected between data points.
- › **Kriging:** Interpolates data using a geostatistical gridding method to determine a value at each grid node. It first calculates a variogram of the data, which shows the correlation of the data as a function of distance. This method is ideal when data is clustered and randomly distributed.

Minimum Curvature Gridding

Minimum curvature gridding is a gridding method that fits a smoothest possible surface to the data points. It is best used when data is randomly distributed, when data is sampled along arbitrary lines or if you want to include tie lines. The following figure illustrates these types of data:

Figure 1.16 Data distribution suitable for minimum curvature gridding

If the data is relatively smooth between sample points or survey lines, minimum curvature gridding should be used. If the data may be variable between sample locations, or is known to be statistical in nature (such as geochemical data), is poorly sampled or clustered, use the kriging method.

Minimum curvature gridding has the following strengths:

- > works with any spatial distribution of data
- > does not impose a directional bias


Determining Cell Size

If your data is distributed as random points, the cell size should be 1/4 to 1/2 of the nominal sample interval. If the cell size is not specified, the data points are assumed to be evenly distributed and the default cell size will be calculated as:

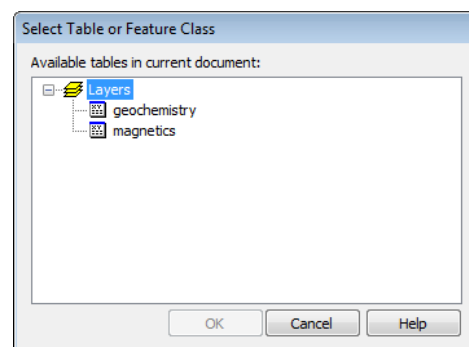
$$1/4 * (\text{sqrt}(\text{grid area} / \# \text{ data points}))$$

Because our geochemical data is randomly distributed, it is difficult to manually measure the nominal sample interval. In this lesson, you will grid the geochemistry data using the minimum curvature method and have the algorithm determine the cell size for you. You will then run it again after adjusting some of the advanced parameters to optimise the output grid.

To grid data using the minimum curvature method:

1. On the Target Surface toolbar, click the **Minimum curvature gridding from table in current document** button .

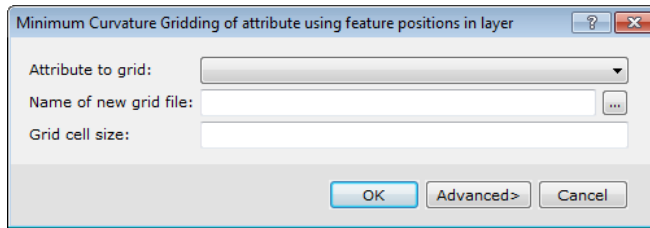
The Select Table or Feature Class dialog box opens.

Figure 1.17 Select Table or Feature Class dialog box

2. Select **geochemistry** and click **OK**.

The Minimum Curvature Gridding of attribute using feature positions in layer dialog box opens.

Figure 1.18 Minimum Curvature Gridding of attribute using feature positions in layer dialog box



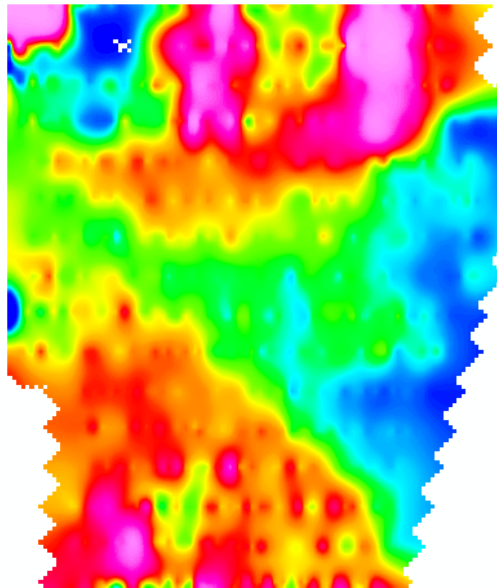
3. From the **Attribute to grid** list, select **Au**.
4. For the **Name of new grid file**, enter **Au_MC.grd**.

The Grid cell size parameter is optional. By leaving it blank, the cell size will be calculated automatically.

5. Click **OK**.

The gridded data created using the minimum curvature gridding method is saved in your working directory and displayed in the map display area.

Figure 1.19 Minimum Curvature grid



6. In the Table of Contents, drag the geochemistry point layer above the grid.

The holes in the grid correspond to areas of no sample data. Later in this lesson, you will re-grid the data using several advanced options. One of these options is the blanking distance, which is the distance within which missing grid cell values will be interpolated.

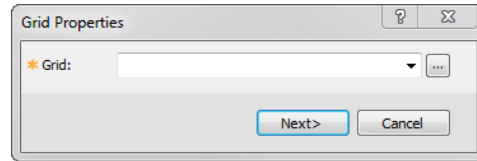
Now you will examine the properties of the grid that was just created.


To examine the grid properties:

1. On the Target Surface toolbar, click **Target Surface** then **Grid Properties**.

The Grid Properties dialog box opens.

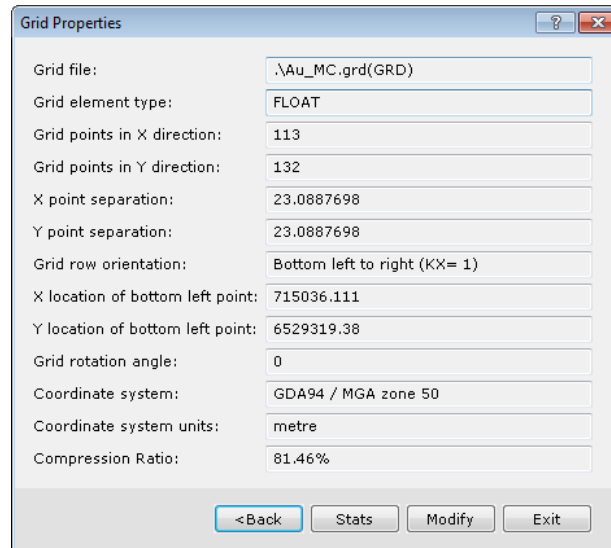
Figure 1.20 Grid Properties dialog box



2. Click the **Browse** button  and select **Au_MC.grd**.
3. Click **Next**.

The Grid Properties dialog box opens.

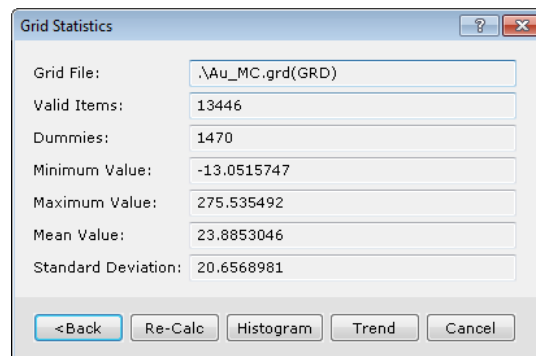
Figure 1.21 Grid Properties dialog box



4. Click **Stats**.

The Grid Statistics dialog box opens.

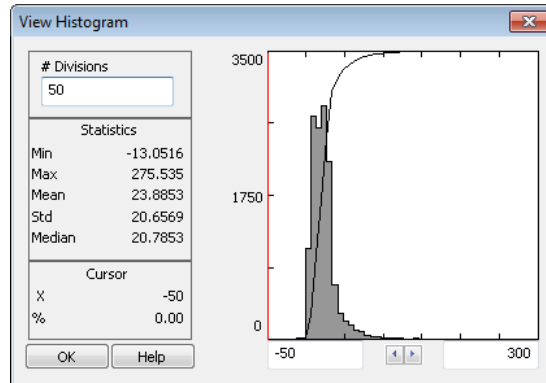
Figure 1.22 Grid Statistics dialog box



High and Low values for grid layers are also listed in the ArcMap Table Of Contents.


5. Click **Histogram**.

The View Histogram dialog box opens and displays the grid histogram.

Figure 1.23 View Histogram dialog box

Look at the statistics and the histogram. It appears that the data is logarithmically distributed. If you compare the minimum values of the Au grid to those of the Au point layer in the Table Of Contents, you can see a significant difference as well as the minimum of the grid being negative, which is unrealistic for geochemistry data. Because minimum curvature gridding attempts to interpolate as smoothly as possible between data points, it is not well suited to logarithmically distributed data. You will now run the minimum curvature gridding again, this time adjusting the grid cell size as well as advanced parameters such as the log option and blanking distance, in order to optimise the output grid.

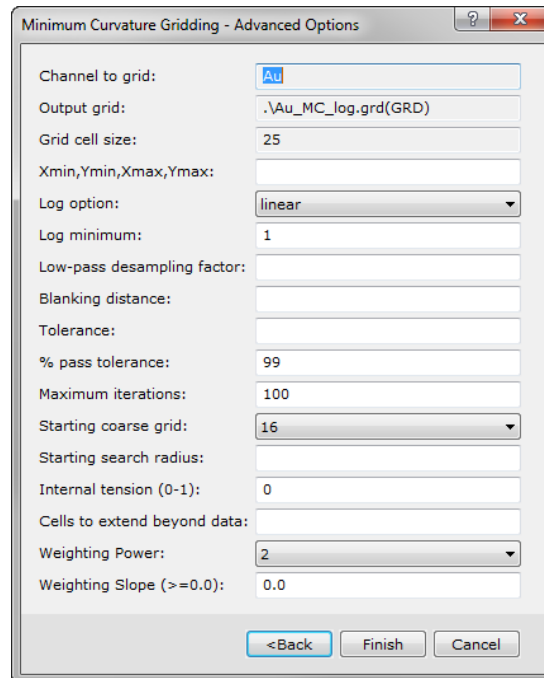
To grid data using the minimum curvature advanced options:

1. On the Target Surface toolbar, click the **Minimum curvature gridding from table in current document** button .
2. From the Select Table or Feature Class dialog box, select **geochemistry** and click **OK**.

The Minimum Curvature Gridding of attribute using feature positions in layer dialog box opens and remembers the parameters you used the first time.

3. For the **Name of new grid file**, enter **Au_MC_log.grd**.
4. For the **Grid cell size**, enter **25**.
5. Click **Advanced**.

The Minimum Curvature Gridding – Advanced Options dialog box opens.

Figure 1.24 Minimum Curvature Gridding – Advanced Options dialog box

The following are some of the key parameters that control the creation of the grid surface:

- **Log Option:** The logarithm (base 10) of the data can be gridded rather than the original data. Once gridded, the output grid can be stored as either the logarithmic data or it can be rescaled back to the original units. Gridding the log of the data can be a very effective way to reduce distortion due to highly skewed data such as geochemical data.
- **Blanking Distance:** All grid cells farther than the blanking distance from a valid point will be blanked out in the output grid. The default is the nominal sample interval, i.e. $2 * (\sqrt{\text{grid area} / \#\text{data points}})$. This parameter should be set to just greater than the maximum sampling interval through which interpolation is desired. If there are too many holes in the resulting grid, increase the blanking distance appropriately.
- **Tolerance:** The tolerance required for each grid cell (%). The default is 0.1 percent of the range of the data. Decrease the tolerance for a more accurate grid.
- **% Pass Tolerance:** The required percentage of points that must pass the tolerance. The default is 99.0 percent. Increase this percentage for a more accurate grid.
- **Maximum iterations:** Iterations at the finest coarseness level will stop when the maximum number of iterations is reached (default is 100). At each greater coarseness, the maximum iterations is reduced by 2. Increase this value if more iterations are required to produce a more accurate grid.
- **Internal tension:** The degree of internal tension (between 0 and 1). The default is no tension (0), which produces a true minimum curvature grid. Increasing

tension can be used to prevent overshooting of valid data in sparse areas, although curvature in the vicinity of real data will increase.

6. From the **Log option** list, select **log, save as linear**.

Because this is highly skewed geochemical data, the log option will be used. The log option transforms the data into log space, generates the grid based on the transformed data and outputs the resulting grid back in linear space.

7. For the **Blanking Distance**, enter **225**.

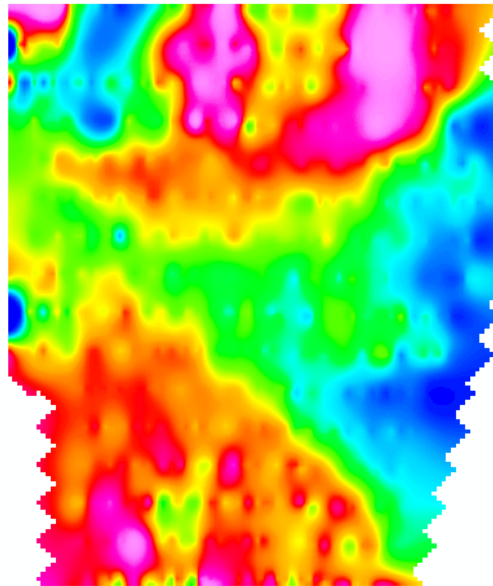
All grid cells within this distance from a valid point will be interpolated. Grid cells that are further than this distance, will be left blank in the output grid.

You will use the intelligent defaults for the remaining parameters.

8. Click **Finish**.

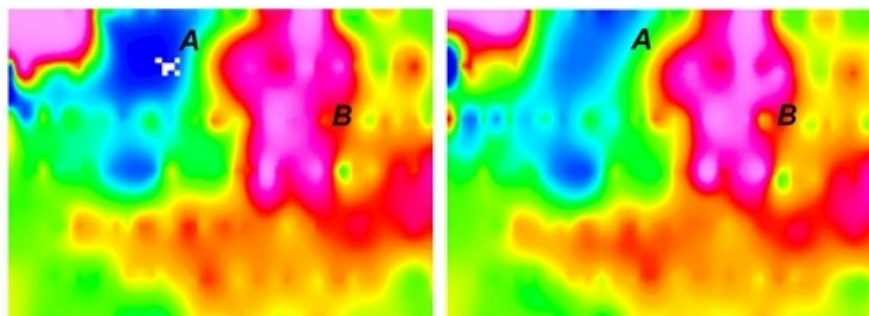
The gridded data created using the advanced options for the minimum curvature gridding method is saved in your working directory and displayed in the map display area.

Figure 1.25 Minimum Curvature grid with log option and blanking distance of 225 m



9. Zoom in to the upper-left corner of the grid.
10. Turn the display of the Au MC log grid on and off to compare the results.

Figure 1.26 Left, Au_MC.grd; right Au_MC_log.grd



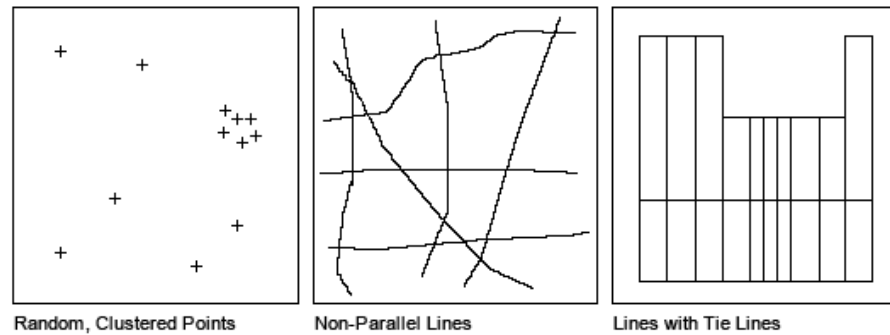


You can also use the tools on the Effects toolbar to compare the grids.

Kriging

Kriging is a geostatistical gridding technique for random data, non-parallel line data or orthogonal line data, as illustrated in the following figure:

Figure 1.27 Data distribution suitable for kriging



Use the kriging method if the data is variable between sample locations, known to be statistical in nature, poorly sampled or clustered. Kriging is ideally suited to geochemical or other geological sample-based data; it is rarely used with geophysical data, which tends to vary smoothly between data points..


It first calculates a variogram of the data showing the correlation of the data as a function of distance. The greater the distance between data points, the greater the variation between the points. Based on the variogram, you can select a model that best defines the variance of the data. Because kriging can be slow, the size of a dataset may be a limiting factor in choosing kriging.

Kriging has the following strengths:

- ideal for clustered data
- better control in anomalous or poorly sampled areas

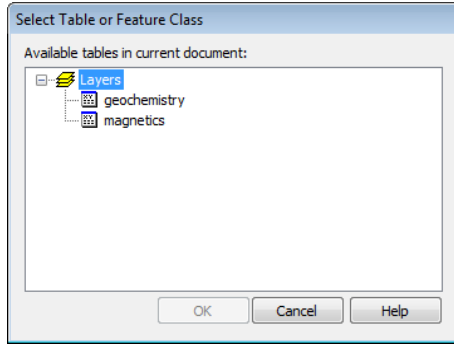
You will now use the kriging method. You will first compute the variogram, adjust the advanced options and then create the grid.

To compute the variogram:

1. On the Target Surface toolbar, click the **Kriging gridding from table in current document** button .

The Select Table or Feature Class dialog opens.

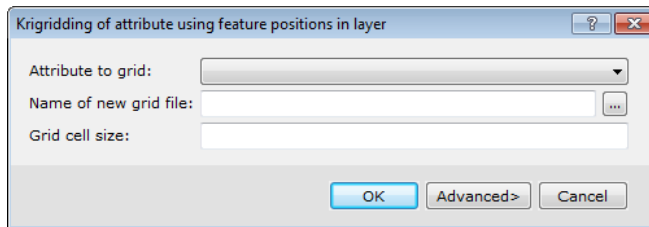
Figure 1.28 Select Table or Feature Class dialog



2. Select **geochemistry** and click **OK**.

The Krigging of attribute using feature positions in layer dialog opens.

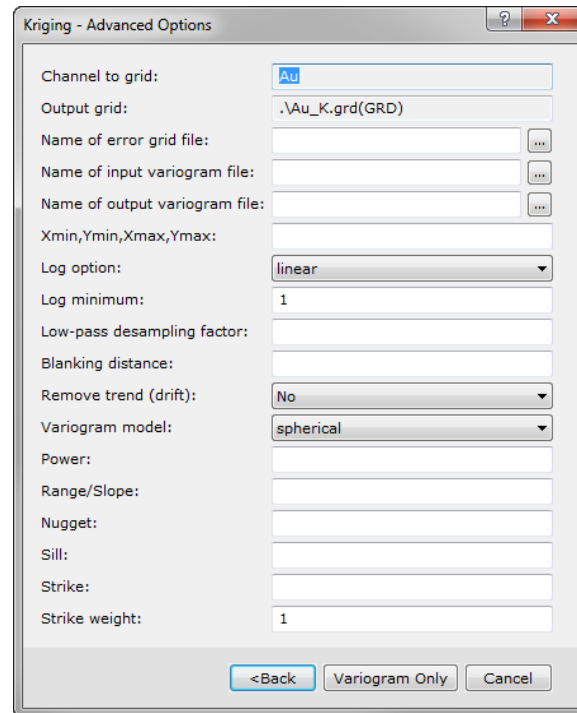
Figure 1.29 Krigging of attribute using feature position in layer dialog



3. For the **Attribute to grid**, select **Au**.
4. For the **Name of new grid file**, enter **Au_K**.
5. For the **Grid cell size**, enter **25**.
6. Click **Advanced**.

The Kriging - Advanced Options dialog opens.

Figure 1.30 Kriging - Advanced Options dialog



The following are some of the key parameters that control the creation of the grid surface:

- › **Range/Slope:** For spherical, Gaussian and exponential models, the range is the distance at which the variogram model reaches the sill value. Beyond the range, the data is uncorrelated. For the power model, this is the rate of climb, or slope for a linear model.
- › **Nugget:** The nugget is the average error in each data point and is indicated by the intersection of the variogram model with the $h=0$ axis. The default is 0.
- › **Sill:** This is the level at which the variogram becomes uncorrelated, or goes flat. The sill must be specified for the spherical, power and Gaussian models.
- › **Strike and Strike weight:** These parameters control anisotropic gridding.

7. For **Name of output variogram file**, enter **Au K**.

8. For **Log option**, select **log, save as linear**.

This option will grid in log space and output the results in linear space.

9. For the **Blanking Distance**, enter **225**.

10. For **Variogram model**, select **spherical**.


11. Click **Variogram Only**.

The variogram is plotted and displayed in the variogram window.

In general, the output variogram is good at matching the overall character of the data. Ideally, the red curve representing the variogram model should approximately match the black line representing the observed data.

Next, you will adjust the variogram parameters and then create the gridded data.

To adjust the variogram parameters:

1. On the Target Surface toolbar, click the **Kriging gridding from table in current document** button .
2. From the Select Table or Feature Class dialog, select **geochemistry** and click **OK**.

The Krigidding of attribute using feature positions in layer dialog opens and remembers the parameters you used the first time.

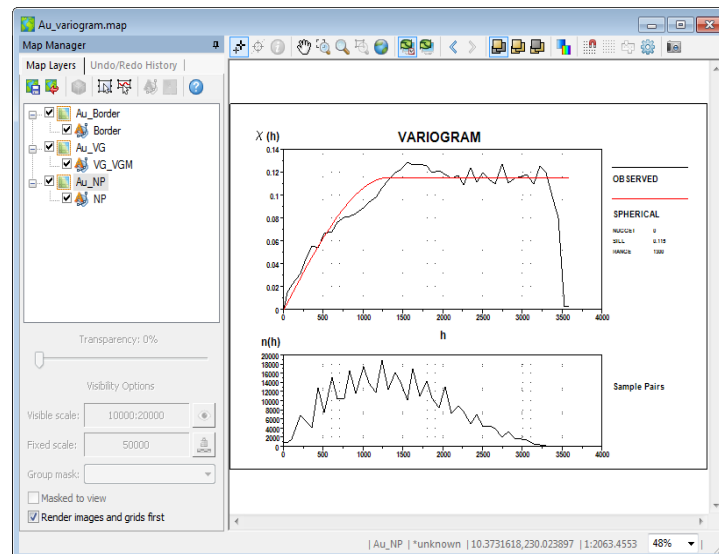
3. Click **Advanced**.
4. For **Range/Slope**, enter **1300**.
5. For **Sill**, enter **0.115**.
6. Click **Variogram Only**.

A File Validation window opens asking to overwrite the existing variogram.

7. Click **Yes**.


The variogram is updated based on the adjusted parameters.

Figure 1.31 Replotted variogram



The updated variogram has a better match to the observed data. Note that it is virtually impossible to exactly match the model to observed variance. It is acceptable for the model to approximate the variance, much like a trend line or "line of best fit". Now that the variance is accurately modelled, you will create the grid.

To create the grid based on the variogram:

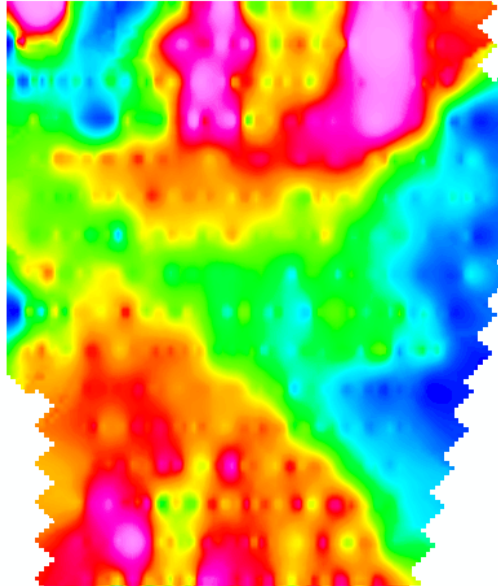
1. On the Target Surface toolbar, click the **Kriging gridding from table in current document** button .
2. From the Select Table or Feature Class dialog, select **geochemistry** and click **OK**.

The Krigidding of attribute using feature positions in layer dialog opens and remembers the parameters you used the first time.

3. Click **OK**.

The gridded data created using the kriging gridding method is saved in your working folder and displayed in the map display area.

Figure 1.32 Grid created using the kriging method



Compare the grids created using the two different gridding methods. Note that anomalies in the kriging grid are generally less continuous than in the minimum curvature grid, which is a product of honouring the variance in the data.



Now would be a good time to save your map document.

In this lesson you:

- > Created a grid using the minimum curvature method
- > Created a grid using the minimum curvature advanced options
- > Created a grid using the kriging method

Lesson 1.4 Creating Geosoft Contours

In this lesson you will:

- › Create multiple contour levels and output to a shapefile
- › Apply colours to the contours
- › Output contours to a Geosoft map layer
- › Create exact contours
- › Add a legend


Creating and Plotting Contours

Contours are lines drawn at specified intervals or multiples of intervals on your map based on a gridded dataset. After you create a grid, you may want to generate contours and display them on your map. The contouring tool in **Target for ArcGIS** is specially designed to handle the very large dynamic data ranges that characterise Earth Science datasets.

You can output the contours to a Geosoft map layer for display in the ArcMap window or make a shapefile (*.SHP) with Z attributes for use in ArcMap. You can also select different contour options and line styles.

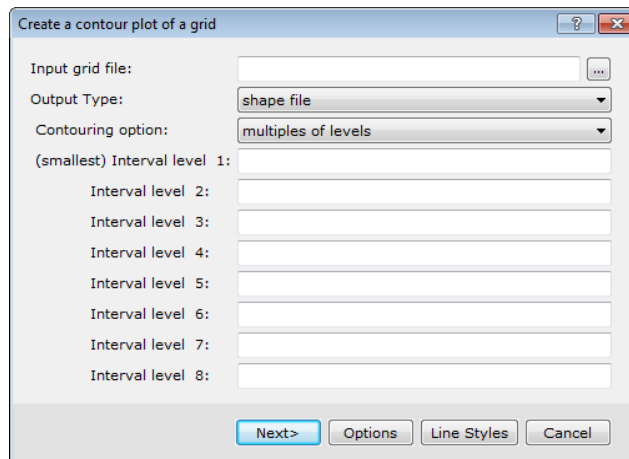
You will begin by creating multiple contour levels for magnetic data.


To create contours and output to a shapefile:

1. On the Target Surface toolbar, click the **Create Contours from grid on disk** button .

The Create a contour plot of a grid dialog opens.

Figure 1.33 Create a contour plot of a grid dialog



2. For the **Input grid file**, click the the **Browse** button .
3. From the **Geophysics** folder, select **mag_MC.grd** and click **Open**.
4. In the **Output Type** list, ensure **shape file** is selected.
5. In the **Contouring option** list, ensure **multiples of levels** is selected.

This option plots the levels and their multiples.

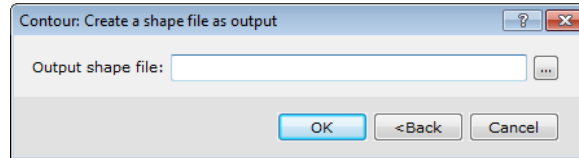
6. For **Interval level 1**, enter **100**.

You will create contours in 100 nT (nanotesla) intervals.

7. Click **Next**.

The Contour: Create a shape file as output dialog opens.

Figure 1.34 Contour: Create a shape file as output dialog



8. For **Output shape file**, enter **mag_100nT** and click **OK**.

The contour shapefile with Z attributes is saved in your working folder and displayed in the data frame.

Figure 1.35 Contours for mag_MC.grd




9. Using the **Zoom In** tool on the Tools toolbar, zoom in to the contour layer.

Figure 1.36 Contours for mag_MC.grd (zoomed)



In the Table Of Contents, there are two shapefiles; one representing the contour lines (mag_100nT_Inz) and a polygon layer (mag_100nT_pgz) representing lows in the data. Highs are marked with a star and lows are marked with a triangle. Notice with the contour line layer that the lines are not displayed in areas where the contours get really close together. These factors can be controlled using the contouring settings, which you will examine now while re-generating the contours.

To regenerate the contours:

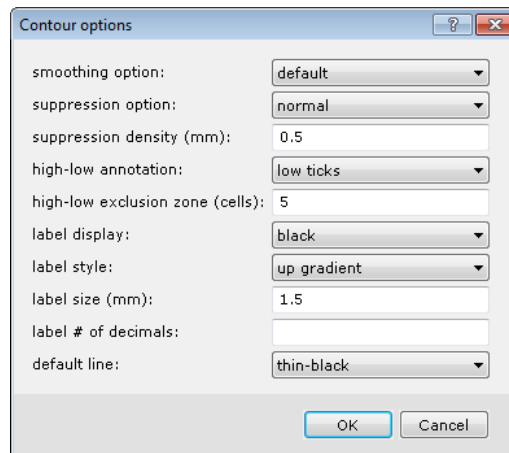
1. On the Target Surface toolbar, click the **Create Contours from grid on disk** button .

The Create a contour plot of a grid dialog opens and remembers the parameters you just used.

2. Click **Options**.

The Contour options dialog opens.

Figure 1.37 Contour options dialog



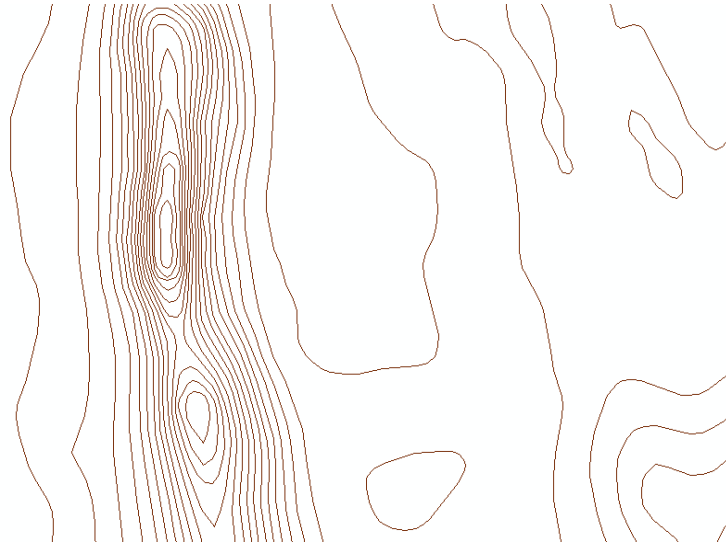
3. For **smoothing option**, select **smooth and refine**.
4. For **suppression density (mm)**, enter **0**.

A value of zero means that contours will be drawn regardless of how close together they are.

5. For **high-low annotation**, select **none**.
6. Click **OK**.
7. Click **Next**.
8. For the Output shape file, enter **mag_100nT_2** and click **OK**.

The contour shapefile is created, saved in your working folder and displayed in the data frame.

Figure 1.38 Contours without suppression (zoomed)



9. In the Table Of Contents, right-click and **remove** the original **mag_100nT_Inz** and **mag_100nT_pgz** layers.

Now you will adjust the layer properties to display the contours using graduated colours.

To apply colours to the contours:

1. In the Table Of Contents, right-click the **mag_100nT_2_Inz** layer and select **Properties**.

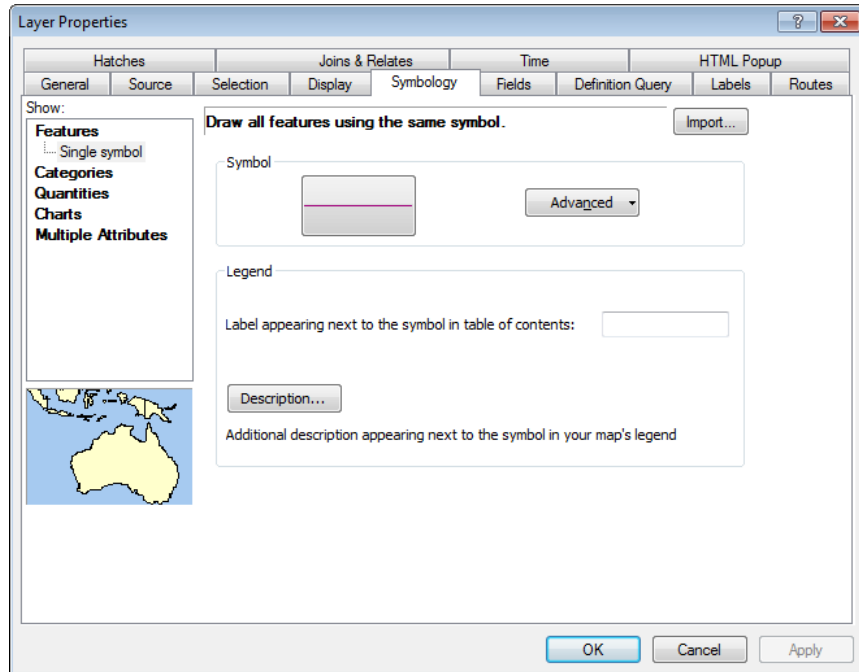
The Layer Properties dialog opens.



You can also open the Layer Properties dialog by double-clicking the layer in the Table Of Contents.

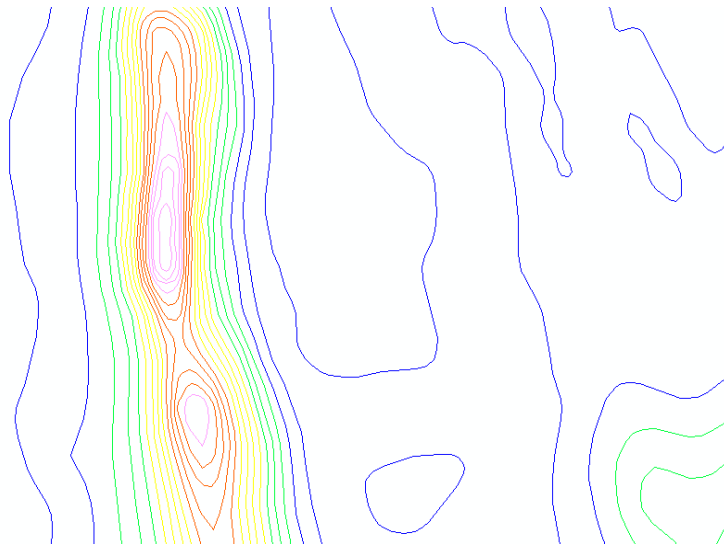
2. Click the **Symbology** tab.

Figure 1.39 Layer Properties Symboly tab



3. In the Show area on the left, click **Quantities**.
ArcMap automatically selects Graduated colors.
4. From the **Value** list, select **ZValue**.
The ZValue is the value from the grid that the contours were made from.
5. Click **OK**.
The contours are now coloured based on magnetic intensity.

Figure 1.40 Contours coloured by mag values (zoomed)




Shapefiles and Geosoft Map Layers

The **Target for ArcGIS** plotting tools provide two types of output; shapefiles (as you created in the previous exercise), and Geosoft map layers. The shapefile output contains the attributes of the data you are generating contours from. Geosoft map layers are static images ready for printing; they contain Geosoft's pre-defined annotations and legends and allow you to pre-define colours, patterns and line styles.

The Geosoft Map Layer is not a typical GIS layer. The ArcMap 'Identify' cursor, symbology and labelling tools are not available to use with this type of layer. You also cannot modify a Geosoft Map Layer; if changes are required, you must recreate it.

To output contours to a Geosoft map layer:

1. On the Target Surface toolbar, click the **Create Contours from grid on disk** button .

The Create a contour plot of a grid dialog opens and remembers the parameters you just used.

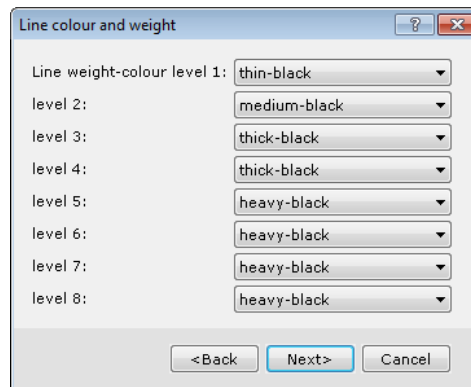
2. From the **Output Type** list, select **Geosoft map layer**.
3. For **Interval level 2**, enter **200**.

This will create a second interval level.

4. Click **Line Styles**.

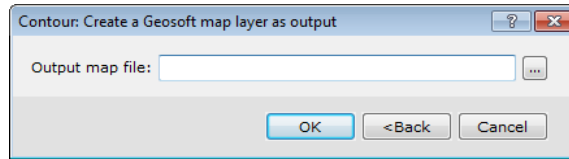
The Line colour and weight dialog opens.

Figure 1.41 Line colour and weight dialog



5. For **level 2**, select **medium blue**.
6. Click **Back**.
7. Click **Next**.

The Contour: Create a Geosoft map layer as output dialog opens.

Figure 1.42 Contour: Create a Geosoft map layer as output dialog


8. For **Output map file**, enter **mag_contours** and click **OK**.
The Geosoft contour layer is displayed.

Figure 1.43 Geosoft contour layer

The output map layer has contours every 100 nT. In addition, the level 2 contours specified at an interval of 200 nT have labels and are displayed with a thicker blue line style.

Note that you can create a contour at an exact level. This can be useful when you want to define anomalies or any other value in your data. For example, local knowledge tells us that the main geological units in our project area have a minimum magnetic response of 150nT. You will now create a 150nT contour from the magnetic grid to highlight these areas..

To create exact value contours:

1. On the Target Surface toolbar, click the **Create Contours from grid on disk** button .

The Create a contour plot of a grid dialog opens and remembers the parameters you just used.

2. From the **Output Type** list, select **shape file**.
3. From the **Contouring option** list, select **exact levels**.

This option plots only the exact stated contours.

4. For **Interval level 1**, enter **150** and **clear** the value of 200 from Interval level 2.
This will create a contour at exactly the 150 nT level.

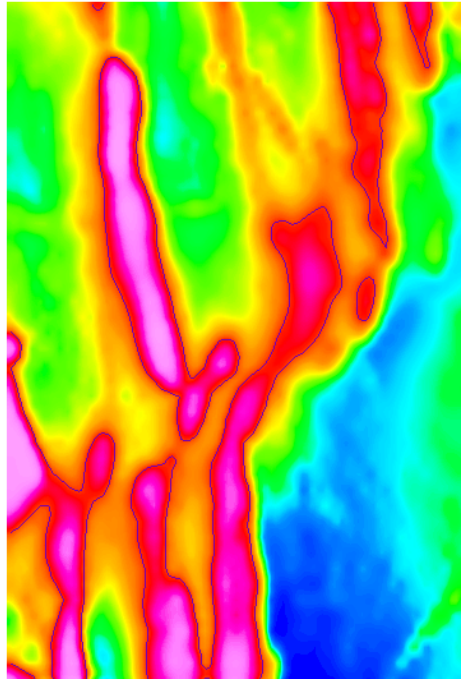
5. Click **Next**.

The "Contour: Create a shape file as output" dialog box opens.

6. For **Output shape file**, enter **mag_150nT** and click **OK**.

The shapefile **representing a value of exactly 150 nT** is saved in your working folder and displayed in the map display area. You may want to turn off the display of your other contour layers and turn on the display of the mag_MC grid layer.

Figure 1.44 Contours at 150 nT



Data View versus Layout View

ArcMap provides two ways to view your data:



- > **Data View:** Use this view for exploring, editing, displaying and querying your data.
- > **Layout View:** Use this view for designing your map and adding map elements such as a title, north arrow and scale bar.

To switch between Data View and Layout View:

- > From the **View** menu, select **Layout View**.

The entire map is displayed and the Layout toolbar is activated.



You can also switch between views by clicking the Data View button  and the Layout View button  on the lower left of the display window.

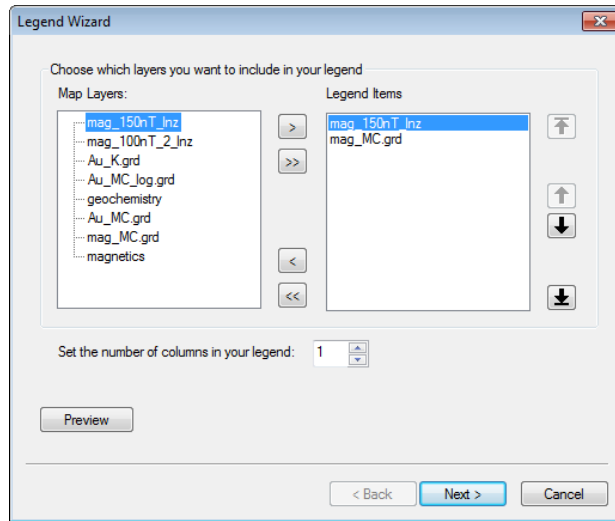
You will now add a legend and scale bar for your geochemical data.

To add a legend:

1. Ensure you are in **Layout View**.
2. From the **Insert** menu, select **Legend**.

The Legend Wizard opens. The Legend Items listed correspond to the layers in your map that are currently displayed.

Figure 1.45 Legend Wizard



3. Click **Preview** to see a preview of your legend.
4. Click **Finish**.

A Legend is added to your map and displayed using default properties. You may need to select it on the map and move it to a different location.


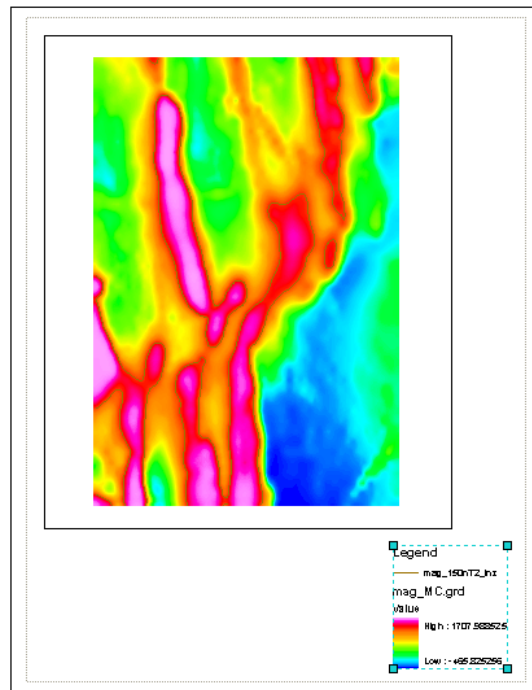
 *Double-click any map element in the Layout View to change its properties.*

Figure 1.46 Layout View of map with legend



Take some time now to explore the other map elements that you can add to your map, such as a title, north arrow and scale bar. Note that you can also change the map paper size, by using the "Change Layout" option on the ArcMap Map Layout toolbar.



Now would be a good time to save your map document.

In this lesson you:

- > Created multiple contour levels and output to a shapefile
- > Applied colours to the contours
- > Output contours to a Geosoft map layer
- > Created exact contours
- > Added a legend

Module 2: Managing Drillhole Data

Module 2 has Five Lessons:

Lesson 2.1	Creating a Drillhole Project and Importing Data	42
Lesson 2.2	Working with Geosoft Databases and Profiles	54
Lesson 2.3	Using QA/QC Tools	65
Lesson 2.4	Creating Composite Databases	69
Lesson 2.5	Using Drillhole Data Management Tools	76

Lesson 2.1 Creating a Drillhole Project and Importing Data

In this lesson you will:

- Create a drillhole project
- Import data
- Assign the coordinate system
- Save the databases

Creating an MXD

You will begin by making a new .MXD file for your drilling project.


To create an MXD:

1. From the **File** menu, select **New**.
2. Select **Blank Map** and click **OK**.
A new, untitled map is created.
3. From the **File** menu, click **Save**.
4. Locate the **Target for ArcGIS Training Data** folder.
This is the location where all your data created by Target for ArcGIS will be saved.
5. For the **File name**, enter **Drillhole Mapping.mxd** and click **Save**.

Geosoft Database Window

Target for ArcGIS stores your drillhole data within a series of Geosoft Databases. You can view this data in the Geosoft Database Window.

To open the Geosoft Database Window:

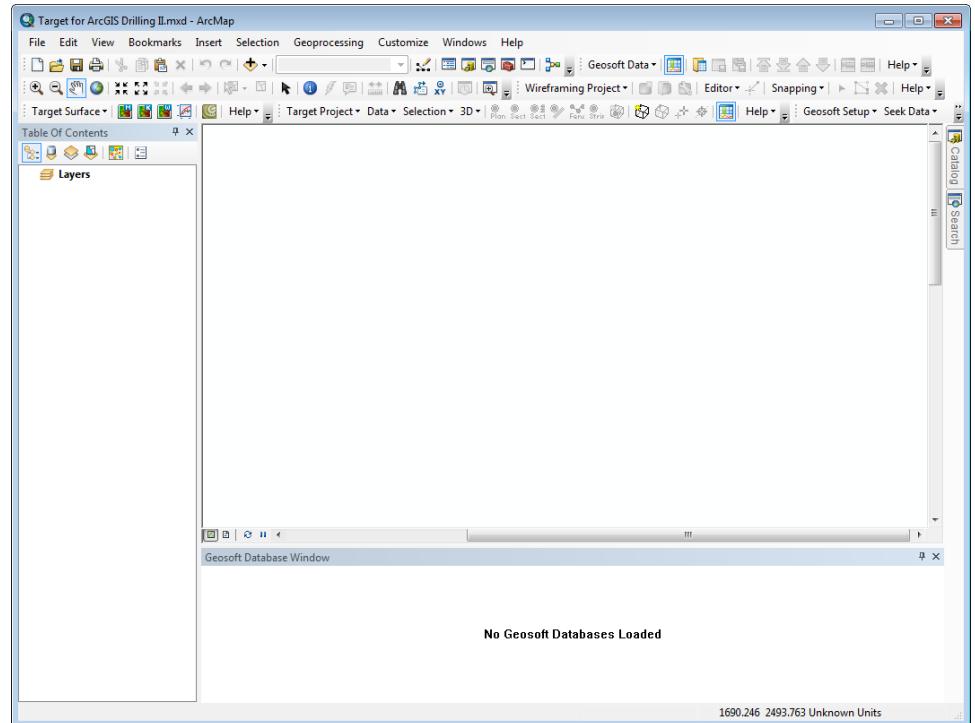
- On the **Target Drillhole** toolbar, click the **Show Geosoft Database Window** button .

The Geosoft Database Window opens. Because you have not yet imported drillhole data into the project, the window displays the “No Geosoft Databases Loaded” message.

You may wish to dock the Geosoft Database Window at the bottom of your ArcMap window.



To prevent a window from docking while moving it, hold down the CTRL key.

Figure 2.1 Geosoft Database Window docked at bottom of ArcGIS window

Creating a Drillhole Project

Target for ArcGIS uses Geosoft Databases to store and process drillhole data. Separate databases are created for each data type, which are linked by a common project name.

For example, in a project named "Prospect" the following databases could be created:

- Prospect_Collar.gdb (collar information)
- Prospect_Survey.gdb (hole dip-azimuth survey information)
- Prospect_Geology.gdb (geology information - the "Geology" label is user specified)
- Prospect_Assays.gdb (assay information - the "Assays" label is user specified)

When you first create a new drillhole project, an empty collar database is created and displayed in the Geosoft Database Window.

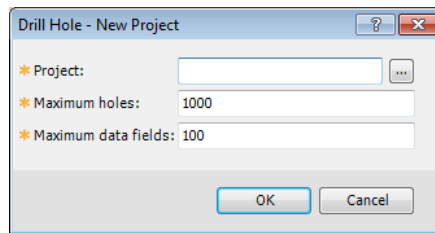
Target for ArcGIS scans the open project for all the databases currently loaded in the project to see which data is available. If you happen to close one of the project databases, it will not be visible to the system and will not be used when processing your data. While working with a **Target for ArcGIS** project, if you wish to hide a database, click the display of the Geosoft Database Window on and off instead of closing it so it can continue to be accessed by **Target for ArcGIS**.

To create a drillhole project:

1. From the **Target Project** menu on the Target Drillhole toolbar, select **New Project**.

The Drill Hole - New Project dialog opens.

Figure 2.2 Drill Hole - New Project dialog



2. For the **Project**, enter **Target**.

The Project folder displays your current working folder.

3. Use the default values for **Maximum holes** and **Maximum data fields per dataset**.

You should specify a number that is representative of the final estimated project size. This ensures you have enough space available in your project while not consuming excessive storage space. However, decreasing these values from the defaults will not enhance performance.



*If you have more holes and fields to import later on, you can grow the database to accommodate them by using the **Grow** option from the **Data then Maintenance** menu.*

4. Click **OK**.

The empty collar database with the project name Target_Collar.gdb is created and displayed.

Figure 2.3 Collar Database

Importing Data

The Drill Hole Import Wizard guides you through the import process and automatically displays your data in a spreadsheet window. You can import data in many different formats, including ASCII and XLS/database formats.

Each type of data (collar, survey, from-to and point) has certain required fields. You can include other fields as well, as long as the required fields are present. Each type of data and their required fields are outlined below.

Collar Data (Drillhole Locations)

Collar data must contain unique Hole ID, Easting (X), Northing (Y), Elevation (Relative Level), and Total Depth (EOH). The X, Y, and Depth data must all be in the same units. The following is an example of these fields in CSV or Excel format:

Figure 2.4 Example Collar Data

Hole_ID	East	North	RL	EOH	Dip	Azimuth
AXE001	10600	12400	367.7	38	-60	90
AXE003	10700	12400	367.6	60	-60	90
AXE005	10800	12400	367.5	60	-60	90

Hole_ID,East,North,RL,EOH,Dip,Azimuth,
AXE007,10900,12400,368.1,60,-60,90,
AXE009,11000,12400,368.3,52,-60,90,
AXE011,11100,12400,368.9,54,-60,90,
AXE013,11200,12400,369.7,37,-60,90,
AXE014,11200,12200,369.9,33,-60,90,

Survey Data

Survey data can be in one of two forms; a Dip-Azimuth survey or an Easting-Northing survey (less common). The required fields depend on the type of survey you have:

- › Dip-Azimuth survey: Hole ID, Depth, Azimuth, Dip
- › Easting-Northing survey: Hole ID, Depth, Easting, Northing, Elevation

Survey data includes depth from the collar, increasing from 0 depth. The X, Y and depth data must all be in the same units. A survey file is only required if a hole has actually been surveyed and the dip and azimuth change at depth. If a survey is not present, the collar dip and azimuth will be used. An example of a Dip-Azimuth survey file is shown in the following table:

Table 2.1 Example Survey Data

Hole_ID,Depth,Azimuth,Dip
AXE001,0,90,-60
AXE003,0,90,-60
AXE005,0,90,-60
AXE007,0,90,-60
AXE009,0,90,-60
AXE011,0,90,-60
AXE013,0,90,-60
AXE014,0,90,-60

From-To Data

Also known as Interval data, this data is acquired over specified depth intervals down the hole; for example, at a 1 m interval with a measurement starting at 0 m and extending to 139 m. From-To data must include the Hole ID, Depth From, Depth To and some data values. These data values can be assay results, lithology results from core logging, grain size, mineral content, etc. Note that for assay results you can include the units in the second row and these will be imported as well. Table 2.3 shows an example of From-To assay data and Table 2.4 shows an example of From-To lithology data.

Table 2.2 Example From-To Assay Data

Hole_ID,Sample,From,To,Au,As
AXE005,49040,50,54,-0.01,15
AXE005,49041,54,58,-0.01,25
AXE005,49042,58,60,0.04,20

Hole_ID,Sample,From,To,Au,As
AXE007,49043,0,2,0.01,15
AXE007,49044,2,6,-0.01,15
AXE007,49045,6,10,-0.01,15

Table 2.3 Example From-To Lithology Data

Hole_ID,From,To,Weathering,Rock,,,,,ppm,ppm
AXE011,19,21,Ls,S,,,*,*,*
AXE011,21,25,Ls,Sp,,,*,*,*
AXE011,25,31,Ls,Sp,,,*,*,*
AXE011,31,37,Ls,Sp,,,*,*,*
AXE011,37,45,Ls,Sp,,,*,*,*
AXE011,45,54,Ls,Mb,,,*

Point Data

Point data is data acquired in discrete measurements made at specific depths. It must include the Hole ID, Depth and some data values. The difference between Point data and From-To data is that Point data only has one depth per sample, while From-To has a depth interval. Below is an example of structural notes from the core logger, but this could be any point source measurements (e.g., Magnetic Susceptibility).

Table 2.4 Example Point Data

Hole ID,Depth,Type,Strike,Dip,
DD003,35.24,FOLIATION,356,20,
DD003,36.16,VEIN,19,25,
DD003,35.98,VEIN,15,65,
DD003,38.11,VEIN,59,40,
DD003,38.66,VEIN,343,45,
DD003,39.05,VEIN,264,60,

Now that you have created your drillhole project, you will import Collar, Survey, Structure, and two types of From-To data: geochemical assay results and logged geological rock codes.

Importing Drillhole Data

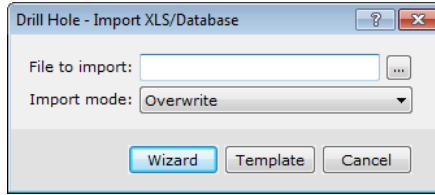
Target for ArcGIS has a multi-tab import tool that allows you to import all of your drillhole data at once. This tool also allows import of multiple groups of the same type of data, so you can import from-to data such as lithology, alteration and assays all at once. You will now import drillhole data from a Microsoft Access Database into Target for ArcGIS.

To import Collar data:

1. From the **Data** menu on the Target Drillhole toolbar, select **Import** then **XLS/Database**.

The Drill Hole - Import XLS/Database dialog opens.

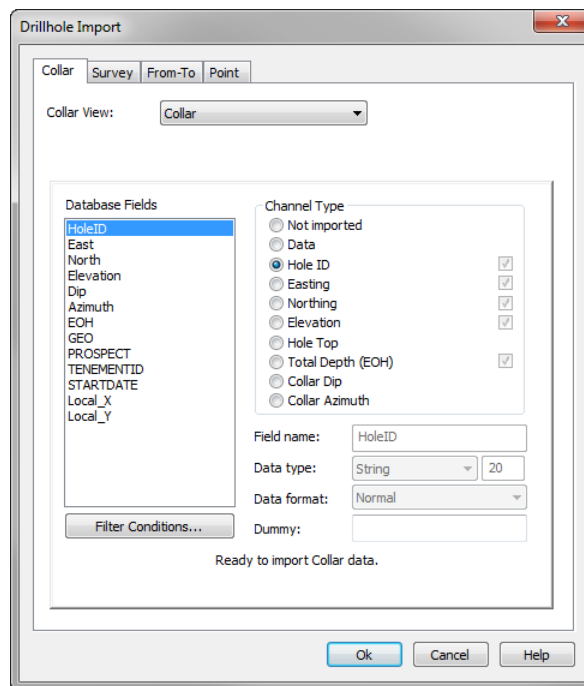
Figure 2.5 Drill Hole - Import XLS/Database dialog



2. For the **File to import**, click the **Browse** button (...).
3. Change the **File of type** from Files (*.xls) to **Files (*.mdb)**.
4. From the **Drilling Data** folder, select **Target Drilling.mdb** and click **Open**.
5. Click **Wizard**.

The Drillhole Import Wizard opens.

Figure 2.6 Drillhole Import Wizard



6. In the Collar tab, select **Collar** from the **Collar View** list.
7. Using the following table, select the appropriate **Channel Type** for each item in the **Database Fields** list.

Database Fields	Channel Type
HoleID	Hole ID
East	Easting
North	Northing
Elevation	Elevation
Dip	Collar Dip

Database Fields	Channel Type
Azimuth	Collar Azimuth
EOH	Total Depth (EOH)
GEO	Data
PROSPECT	
TENEMENTID	
STARTDATE	
Local_X	
Local_Y	

8. Click on the **OK** tab to continue importing data

Importing Survey Data

Various types of surveys (for example, acid, gyroscope and magnetic orientation) can be used to record hole depth (depth measured from the drillhole collar at the surface).

Now that you have imported the collar data, you will import the survey data.

To import Survey data:

After the previous exercise, the Drillhole Import Wizard should be open to the Survey tab

1. In the Survey tab, make sure to select **Survey** from the **Survey View** list and that **Dip-Azimuth** is the chosen **Survey Type**.

The Wizard updates the list of Database Fields associated with the Survey data. The **Survey type** is automatically detected as being Dip-Azimuth Survey data.

2. Select the appropriate **Channel Type** for each item in the **Database Fields** list.
3. Click on the **From-To** tab to continue importing.

Importing From-To Data

Next, you will import two types of From-To data: geochemical assay results and logged geological rock codes.

Importing Geochemical Assay Data


Geochemical assay data are typically acquired by obtaining core or rotary drill samples over specific depth ranges (From-To ranges) and sending samples to an assay laboratory. Numerical results are typically returned from the laboratory in electronic format and can be imported quickly. Before import, you must make sure that your data files contain the following information – HoleID, From, To and a series of Assay results.

To import geochemical assay data:

The Drillhole Import Wizard should be open to the **DH-Datab..**

1. In the From-To tab, select **Assay** from the **From-To View** list.

The Wizard updates the list of Database Fields associated with the Assay data.

2. Select the appropriate **Channel Type** for each item in the **Database Fields** list.
3. You can rename the **Data Name** by clicking the  icon next to the From-to View list. For this lesson, use the default **Assay**.

Importing Geological Data

Target for ArcGIS provides a versatile environment for importing a wide variety of geological data ranging from rock codes, extended text descriptions, alteration types, contact data such as fractures or faults, and other core log information that is important in your drilling project.

This type of information generally consists of text, but you can also add numerical values; for instance, if you want to retain numeric codes for statistical purposes or you want to run your own advanced classification strategies in **Target for ArcGIS**.

To import geological data:

The Drillhole Import Wizard should be open to the From-To tab..

1. Click the "Add" next to the From-To View dropdown. This adds a second From-To tab to the import wizard.
2. In the From-To tab, select **Geology** from the **From-To View** list.

The Wizard updates the list of Database Fields associated with the Geology data. The Types of Data to import is automatically detected as being From-To Data.

3. Using the following table, select the appropriate **Channel Type** for each item in the **Database Fields** list:

Database Fields	Channel Type
HoleID	Hole ID
From	Depth From
To	Depth To
Weathering	Data
Rock	
Foliation	

4. For the **Data Name**, use the default **Geology** and click on the **Point** tab to continue the import

Structure Data

Structural data is point data collected at specific depths down a drillhole. This data often represents planar structures (bedding, cleavage, schistosity, faults, joints and veins) or linear structures (planar structure intersections, mineral stretching, and other forms of lineation). In general, each structure type consists of an alpha and beta angle of measurement to determine the geological orientation of structures down the drillhole. In the structure data you will shortly import, the "alpha" represents the dip of the structure relative to the core reference line and the "beta" represents the dip direction of the structure relative to the core reference line.

To import structure data:

The Drillhole Import Wizard should be open to the **Point** tab .

1. In the Point tab, select **Structure** from the **Point View** list.

The Wizard updates the list of Database Fields associated with the structure data .

2. Select the appropriate **Channel Type** for each item in the **Database Fields** list.
3. For the **Data Name**, use the default **Structure** and click **OK**.

After importing all of your data, examine the databases and ensure all the data was imported correctly. Note how the data is stored differently in each database. In the collar database, data for each collar is stored on a separate row; other databases store each drillhole as an individual table or "line".

Collar data is now stored within the Target_Collar.gdb database. The coordinates are now stored in the DH_East, DH_North and DH_RL channels.

- Survey data is now stored within the Target_Survey.gdb database. DH_East, DH_North and DH_RL channels have also been added to this database; these channels are calculated from the collar and survey data.
- Assay data is now stored within the Target_Assay.gdb database. DH_East, DH_North and DH_RL channels have also been added to this database; these fields are calculated from the collar and survey data.
- Geology data is now stored within the Target_Geology.gdb database . DH_East, DH_North and DH_RL channels have also been added to this database; these fields are calculated from the collar and survey data.
- Structure data is now stored within the Target_Structure.gdb database . DH_East, DH_North and DH_RL channels have also been added to this database; these fields are calculated from the collar and survey data.



When the line header cell is selected, you can use the Page Up and Page Down keys to scroll through the lines.

Applying Filters When Importing Data

The Filter Conditions option in the Import wizard can be used to import data that meets a specific criteria.

Next, you will re-import the structure data, using the Fracture/Joint Set structure type as a filter condition, to create a new database with only Fracture/Joint Set data.

To import structure data with a filter condition:

1. From the **Data** menu on the Target Drillhole toolbar, select **Import** then **XLS/Database**.

The Drill Hole Import Wizard dialog box opens.

2. Click **Wizard**.

The Drill Hole Import Wizard opens.

3. From the **Database Table** list, select **Structure**.
4. For **Types of Data to import**, select **Point Data**.
5. Click **Next**.
6. Click **Filter Conditions**.

The Filter Conditions dialog box opens.

7. From the **Field** list, select **Structure_Type [WChar]**.
8. From the **Condition** list, select **equals**.
9. In the **Value** field, type **prt** and click **OK**.
10. Click the **Rename** icon.
11. For the Data Name, type **Fractures** and click **OK**.

A new database (Target_Fractures.gdb) is created that contains only intervals where the prt structure type (Fracture/Joint Set) is logged.

Assigning the Coordinate System

Now that you have imported your data and the coordinates from the collar table have been added to your databases, you will assign the coordinate system to all databases at once.



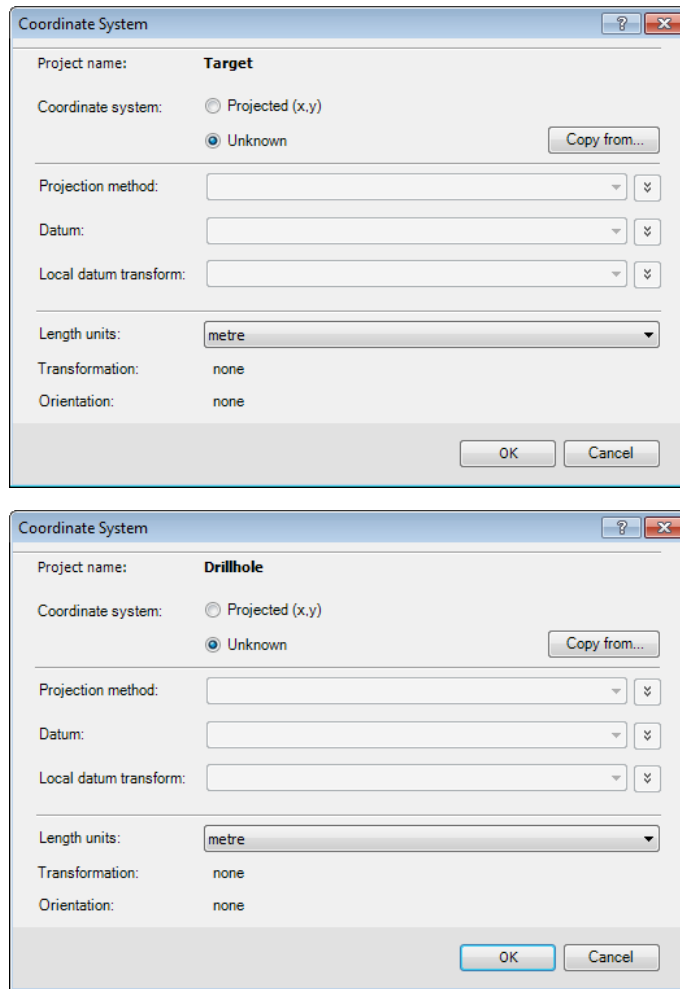
Drillhole coordinates and downhole depths must be in the same units. You can work with local coordinate systems and projected coordinate systems, but the DH_East and DH_North coordinates cannot have a geographic (long, lat) projection.

To set the coordinate system information:


1. From the **Data** menu on the Target Drillhole toolbar, select **Coordinate System**.

The Coordinate System dialog opens indicating the current coordinate system is unknown.



Figure 2.7 Coordinate System dialog



- For the Coordinate system, click **Projected (x,y)**.

 You can also copy coordinate system information from a Database, Grid, Voxel, Geosoft Projection, Esri Projection, Esri Coordinate System, Warp, Polygon, or GM-SYS Model.

- From the **Projection method** list, select **Map Grid of Australia zone 50**.

 For more information on the selected projection method or to create a custom projection, click the More button. 

- From the **Datum** list, select **GDA94**.

The Local datum transform defaults to [GDA94] (1m) Australia - onshore.

- Click **OK**.

The coordinate system is applied to the DH_East, DH_North and DH_RL channels in all four of the databases in your Drillhole project. These channels are the current X, Y and Z channels as indicated by the blue x, y and z markers in the channel header cells.



To ensure consistency between your databases and grids, you should define the coordinate system information for your database before creating maps from it.

Saving a Database

It is always a good habit to save changes you make to your databases, including new databases.

To save all databases:

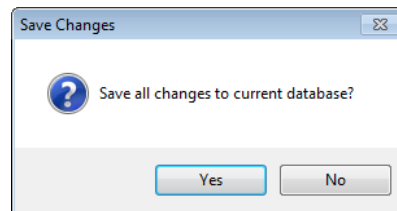
- From the **Target Project** menu on the Target Drillhole toolbar, select **Save Databases**.

To save changes to an individual database:

1. In the Geosoft Database Window, select the database you want to save.
2. From the **Geosoft Data** menu on the Geosoft Database toolbar, select **Commit Geosoft database changes**.

The Save Changes dialog opens.

Figure 2.8 Save Changes dialog



3. Click **Yes**.

The selected database is saved to your working directory.



*To undo changes, select **Discard Geosoft database changes** from the **Geosoft Data** menu. This will restore your database to its last saved state.*



If you close your project without saving it you will be prompted to save any modified documents.

In this lesson you:

- Created a drillhole project
- Imported data
- Assigned the coordinate system
- Saved the databases

Lesson 2.2 Working with Geosoft Databases and Profiles

In this lesson you will:

- > Examine the databases
- > Hide and view channels
- > Protect channels
- > View profiles
- > Use channel math tools
- > Display multiple profiles
- > Save changes to the database

Geosoft Databases

The Target drillhole project consists of a set of Geosoft Databases (*.GDBs). The database is organised in lines, channels and elements and stores all data “elements” of a particular type in individual “channels” (columns). Database “lines” are a collection of related “channels”.

In **Target for ArcGIS**, the collar database contains a list of holes and collar-related data stored in individual rows on one page or “line”. In the other databases, data is stored in lines representing different drillholes; one drillhole is visible at a time.

The figure below outlines the main characteristics of a Geosoft Database.

Figure 2.9 Geosoft Database characteristics

Line/Group header cell	Column (channel)	Current X, Y (and Z) column	Column header cell
✓ DH Collar	DH Hole	DH East	DH North
0.0	DD001	12102.0	9988.8
1.0	DD002	11697.3	9983.1
2.0	DD003	12100.0	9904.0
3.0	DD004	12000.0	10027.0
4.0	RC001	11401.5	10072.1
5.0	RC002	11498.4	10040.5
6.0	RC003	11597.6	10048.2
7.0	RC004	11696.5	10060.6
8.0	RC005	11697.1	9992.4
9.0	RC006	12200.9	9970.9

You can customise the database display to your specifications. You can also modify data within the database without actually committing to any changes until you decide to save the database. After you save the database, your changes are permanent.

Channel Header Cells are label cells used to identify the type of data contained in a spreadsheet column. The channel headers provide a visual indication of the current status of the displayed channel. A black triangle in the top left corner of the header cell indicates the channel is read-only and may not be modified.

Now you will examine the databases and learn how to manipulate and display data within them.

To select a line in a database:

1. In the Geosoft Database Window, select **Target_Assay.gdb**.

Data for drillhole AXE005 is currently displayed.

2. Click the **line/group header cell** in the top left corner of the spreadsheet.

3. **Right-click** and select **List**.

A list of line numbers corresponding to each drillhole in the current database is displayed.

4. Scroll down and select **SKA332**.

The spreadsheet window displays the data for drillhole SKA332.



You can also use the Go to First, Last, Previous and Next line/group in Geosoft database buttons on the Geosoft Database toolbar.



When the line header cell is selected, you can use the Page Up and Page Down keys to scroll through the drillholes.



A single asterisk () indicates a dummy or null value. A double asterisk (**) indicates the channel is not wide enough to display the values and should be resized.*

When you import data, all of your data is automatically displayed. You may want to hide certain channels from display in the Geosoft Database window.

To hide a channel from view:

1. In the **Target_Assay** database, click the **channel header cell** of the **Au** channel.

The Au channel is selected.

2. **Right-click** and select **Hide Column**.

The channel is hidden from view; however, the Target_Assay database still contains the data.



You can also hide the selected channel by pressing the Spacebar.

To view a channel in the database:

1. Click the **channel header cell** of the **empty** channel.

The empty channel is selected.

2. **Right-click** and select **List**.

A box appears beneath the empty channel header cell listing the available channels that currently are not being viewed in the spreadsheet window.

3. Select **Au** and click **OK**.

The channel is now displayed in the spreadsheet.



If you know the name of the data channel, you can position the cursor in an empty channel header cell, type the name of the channel and press Enter.

After importing your data into a database, there may be a channel you want to protect. When channels are protected, they cannot be edited.

To protect a channel:

- > **Right-click** the **Au** channel header cell and select **Protected**.

The black triangle appears in the channel header cell and the channel is protected. You can also select Protect All and all channels will be protected from editing.

To remove channel protection:

- > **Right-click** the **Au** channel header cell and **clear the check mark** beside **Protected**.

The black triangle disappears and the channel is no longer protected. You can also select Protect None and all channel protections will be removed.

Channel Statistics

Statistical information about your data can be very useful for general quality control and to ensure the values are in range with what you would expect.

You will now calculate statistics for the Au channel.

Table 2.5 Methods for calculating statistics

To:	Do this:
Calculate statistics for selected cells	Click and drag to select the desired cells, right-click and select Statistics .
Calculate statistics for a selected line of the database	Click the channel header cell two times , right-click and select Statistics .
Calculate channel statistics for all of the selected lines in the database	Click the channel header cell three times , right-click and select Statistics .

The Stat Report dialog opens.

Figure 2.10 Stat Report dialog

Property	Value
Channel:	Au
Line(s):	SKA332
Fid range:	8328 to 8348
Number of items:	21
Number of dummies:	0
Minimum:	0.01
Maximum:	0.05
Mean:	0.01
Length-weighted mean:	0.01
Length:	61.00
Standard deviation:	0.01
Arithmetic sum:	0.23

Buttons: OK, Save Stats

You will now edit the properties of the Au channel to display more decimal places.

To edit the channel properties:

1. **Right-click** the **Au** channel header cell and select **Edit**.

The Edit Channel dialog opens.

Figure 2.11 Edit Channel dialog

Name	Au
Label	Au
Data type	Double
Array Size	1
Class	Assay
Units	
<input type="checkbox"/> Protected	
Display	
Format	Normal
Field width	10
Decimals	2

Buttons: OK, Cancel, Help

2. For **Decimals**, enter **3** and click **OK**.

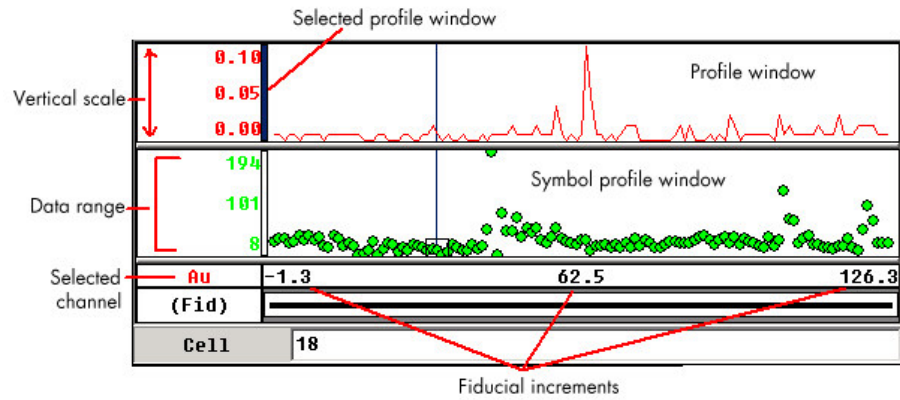
The Au values are now displayed with three decimal places.

Profile Window

The Profile window shows a graphical representation of the data in your Geosoft Database. The profile appears directly below its corresponding database in a profile window. You can display up to five profile windows and up to a total of 128 profiles.

The figure below outlines the main characteristics of the Profile window.

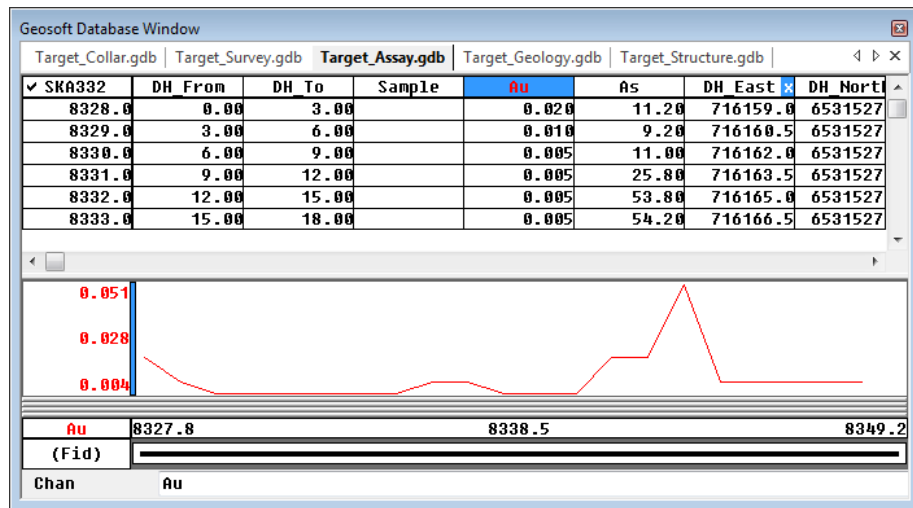
Figure 2.12 Profile window



To display a profile:

1. Undock the Geosoft Database Window by double-clicking on the title bar. The window undocks and can be resized as necessary.
2. Right-click in the **Au** channel and select **Show Profile**. The Profile Window opens below the spreadsheet.

Figure 2.13 Profile of Au channel



Profile windows are dynamically linked to their corresponding database. When you select a value or range of values in either the database or profile window, they are also highlighted in the other window.

To remove a profile:

- > Right-click in the "**Mag**" channel header cell and select **Remove Profile**. The profile is no longer displayed.

Creating a New Channel

You can create new channels in your database to store the results of mathematical expressions or other processing operations. You should create a new channel for storing any changes you make to the original imported data.

In this lesson, you will create a new channel to store the results of a mathematical expression.

To create a new channel:

1. In the Geosoft Database Window, ensure the **Target_Assay.gdb** database is selected.
2. Right-click in the DH_East channel header cell and select **Insert Column**.
A new empty channel appears to the left of the DH_East channel.
3. In the empty channel header cell, type **Au_As** and press **ENTER**.

The Create Channel dialog opens.

Figure 2.14 Create Channel dialog

4. For **Label**, enter **Au/As**.
When plotting channel information, this optional Channel Label will be used instead of the Channel Name.
5. For **Decimals**, enter **4**.
6. Click **OK**.

The new channel is added and is filled with dummy, or null, values indicated with a single asterisk.

Figure 2.15 Assay database with new empty channel

SKA332	DH From	DH To	Sample	Au	As	Au As	DH East
8328.0	0.00	3.00		0.020	11.20	*	716159
8329.0	3.00	6.00		0.010	9.20	*	716160
8330.0	6.00	9.00		0.005	11.00	*	716162
8331.0	9.00	12.00		0.005	25.80	*	716163
8332.0	12.00	15.00		0.005	53.80	*	716165
8333.0	15.00	18.00		0.005	54.20	*	716166
8334.0	18.00	21.00		0.005	74.60	*	716168
8335.0	21.00	24.00		0.005	58.40	*	716169
8336.0	24.00	27.00		0.010	66.20	*	716171
8337.0	27.00	30.00		0.010	41.00	*	716172
8338.0	30.00	33.00		0.005	147.00	*	716174
8339.0	33.00	36.00		0.005	124.00	*	716175
8340.0	36.00	39.00		0.005	47.20	*	716177
8341.0	39.00	42.00		0.020	57.40	*	716178
8342.0	42.00	45.00		0.020	90.20	*	716180

Chan Au_As



A single asterisk (*) indicates a dummy or null value. A double asterisk (**) indicates the channel is not wide enough to display the values and should be resized.

Computing a Mathematical Expression

Now that you have added a new empty channel, you will use the advanced status bar feature in the spreadsheet window to apply a mathematical expression to the data selected in your spreadsheet window. You can select a portion of a channel, an entire channel on a single line, or the same channel on all lines in your database. In this example, the contents of the new channel will be the results of dividing the Au channel by the As channel.

To compute a new channel of data:

1. Click the **Au_As** channel header cell **three times**.
This entire channel is selected in the database.
2. Press the **equal sign (=)** on your keyboard.
This changes the status bar at the bottom of the spreadsheet window to "Formula=".
3. In the **Formula=** box, type **Au/As** and press **ENTER**.
The values are calculated and output to the Au_As channel.

Figure 2.16 Assay database with new calculated channel of data

The screenshot shows a window titled 'Geosoft Database Window' with several tabs: 'Target_Collar.gdb', 'Target_Survey.gdb', 'Target_Assay.gdb', 'Target_Geology.gdb', and 'Target_Structure.gdb'. The 'Target_Assay.gdb' tab is active, displaying a table with the following columns: 'SKA332', 'DH From', 'DH To', 'Sample', 'Au', 'As', 'Au_As', and 'DH East'. The 'Au_As' column is highlighted in blue. Below the table, a 'Chan' field is visible with 'Au_As' selected.

SKA332	DH From	DH To	Sample	Au	As	Au_As	DH East
8328.0	0.00	3.00		0.020	11.20	0.0018	716159
8329.0	3.00	6.00		0.010	9.20	0.0011	716160
8330.0	6.00	9.00		0.005	11.00	0.0005	716162
8331.0	9.00	12.00		0.005	25.80	0.0002	716163
8332.0	12.00	15.00		0.005	53.80	0.0001	716165
8333.0	15.00	18.00		0.005	54.20	0.0001	716166
8334.0	18.00	21.00		0.005	74.60	0.0001	716168
8335.0	21.00	24.00		0.005	58.40	0.0001	716169
8336.0	24.00	27.00		0.010	66.20	0.0002	716171
8337.0	27.00	30.00		0.010	41.00	0.0002	716172
8338.0	30.00	33.00		0.005	147.00	0.0000	716174
8339.0	33.00	36.00		0.005	124.00	0.0000	716175
8340.0	36.00	39.00		0.005	47.20	0.0001	716177
8341.0	39.00	42.00		0.020	57.40	0.0003	716178
8342.0	42.00	45.00		0.020	90.20	0.0002	716180

Channel Math Expression Builder

Now you will use the Channel Math Expression Builder to compute new data. With the Channel Math Expression Builder you can create, save, load and execute math expressions on data that is stored in database channels.

In this example, you will use a true/false expression to create a new channel named High_Au representing Au values higher than 10 ppm; if the value is less than 10 ppm, it will receive a dummy value.

To use the Channel Math Expression Builder:

1. From the **Data** menu on the Target Drillhole toolbar, select **Utilities** then **Channel Math**.

The Channel Math Expression Builder dialog opens.

Figure 2.17 Channel Math Expression Builder dialog

The screenshot shows the 'Channel Math Expression Builder' dialog box. It has two steps:

Step 1: Type in a mathematical expression. Use your own variable names or use the button below to insert default variable names.

Expression: C0 =

Example: C0=C1+C2

Buttons: Operators, Insert Channel Variable, Clear

Step 2: Assign actual channels to the variable names used above.

Assign channels: * C0 =

Common tasks:

Expression file: Load Save

Buttons: OK Cancel

2. From the list of **Common tasks**, select **True/False statement**.

The Expression box at the top updates with an example mathematical expression.

- In the **Expression** box, change the value of 5000 to **10**.

In this example, all Au values greater than 10 will be output to the new channel.

- In the **Assign channels** box, click in the list beside the **C0** channel and enter **High_Au**.

A new channel called High_Au will be created when this expression is computed.

- From the list beside the **C1** channel, select **Au**.

The expression that will be computed $[C0 = (C1 > 10) ? (C1) : (DUMMY);]$ can be read as follows: the new High_Au channel (C0) will contain values of Au (C1) that are greater than 10 ppm; otherwise, a dummy value will be output.

- Click **OK**.

A new channel named High_Au containing the results of this expression is added to your Target_Assay database.

Figure 2.18 Assay database with new High_Au channel

SKA332	As	Au	As	DH East	DH North	DH RL	Mask	High Au
8328.0	11.20	0.0018		716159.0	6531527.3	399.2	1	*
8329.0	9.20	0.0011		716160.5	6531527.3	396.6	1	*
8330.0	11.00	0.0005		716162.0	6531527.3	394.0	1	*
8331.0	25.00	0.0002		716163.5	6531527.3	391.4	1	*
8332.0	53.00	0.0001		716165.0	6531527.3	388.8	1	*
8333.0	54.20	0.0001		716166.5	6531527.3	386.2	1	*
8334.0	74.60	0.0001		716168.0	6531527.3	383.6	1	*
8335.0	58.40	0.0001		716169.5	6531527.3	381.0	1	*
8336.0	66.20	0.0002		716171.0	6531527.3	378.4	1	*
8337.0	41.00	0.0002		716172.5	6531527.3	375.8	1	*
8338.0	147.00	0.0000		716174.0	6531527.3	373.2	1	*
8339.0	124.00	0.0000		716175.5	6531527.3	370.6	1	*
8340.0	47.20	0.0001		716177.0	6531527.3	368.0	1	*
8341.0	57.40	0.0003		716178.5	6531527.3	365.4	1	*
8342.0	90.20	0.0002		716180.0	6531527.3	362.8	1	*

Chan High_Au

Next, you will use tools in the profile window to display and compare the values in the High_Au and Au channels to visually identify lines that have high Au values.

To compare the Au values and the High_Au values:

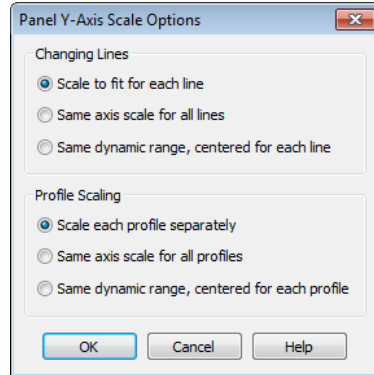
- Go to line **SKC242**.
- Right-click in the **High_Au** channel and select **Show Symbol Profile**.

This displays the High_Au values with a default symbol style.

- Right-click in the **Au** channel and select **Show Profile**.
- Right-click in the profile window and select **Y Axis Options**.

The Panel Y-Axis Scale Options dialog opens.

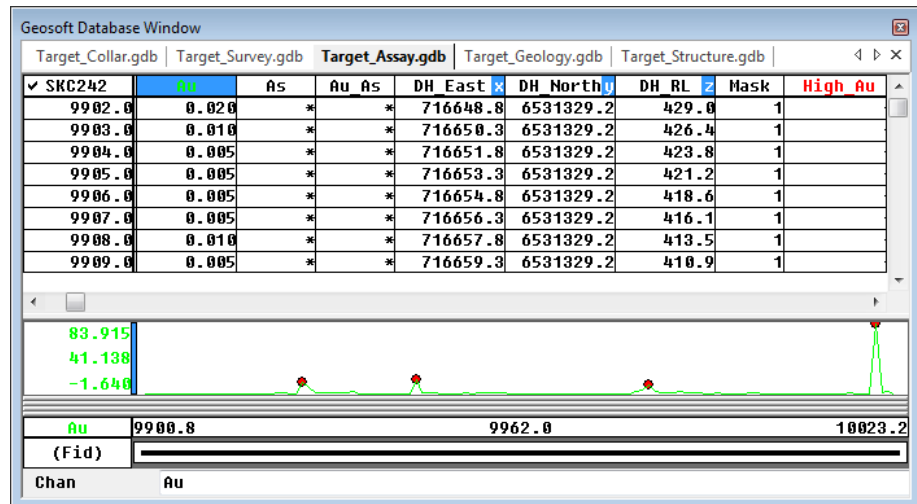
Figure 2.19 Panel Y-Axis Scale Options dialog



5. In the Profile Scaling section, select **Same axis scale for all profiles** and click **OK**.

Although the High_Au and Au channels have different dynamic ranges, they are displayed using the same axis scale.

Figure 2.20 Comparing High_Au and Au profiles



6. Click in the line header cell and use the **Page Up** and **Page Down** keys to move to different lines in the Geosoft Database.

Lines that have High_Au values are more easily identified by examining the profiles.

Saving a Database

Changing how data appears in the spreadsheet (i.e., showing or hiding channels) does not alter the data in your database; however, editing (i.e., adding or deleting channels or altering data values) does change your underlying database.

You should save changes to your database whenever you perform a major editing or processing step. If you process your data and the results are not as you expected, you can restore your database to its previously saved state.

To save changes to your database:

1. From the **Geosoft Data** menu on the Geosoft Database toolbar, select **Commit Geosoft database changes**.

The Save Changes message window opens asking if you want to save all changes to the current database.

2. Click **Yes**.

Your database changes are now saved.



*To undo changes, select **Discard Database Changes** from the **Database** menu. This will restore your database to its last saved state.*

In this lesson you:

- › Examined the databases
- › Hid and viewed channels
- › Protected channels
- › Viewed profiles
- › Used channel math tools
- › Displayed multiple profiles
- › Saved changes to the database

Lesson 2.3 Using QA/QC Tools

In this lesson you will:

- > Run QAQC tools on all data
- > Run QAQC tools on the From-To data

Quality Control and Quality Assurance Tests

Target for ArcGIS provides tools to perform Quality Control and Quality Assurance (QAQC) tests on each of the five different database types (Collar Data, Dip/Azimuth Survey, Easting-Northing Survey, From-To Data, and Point Data). For each database type you can select different QAQC options.



The QA/QC tool also reports unregistered holes in each database. These are drillholes in the databases that do not correspond to any entry in the collar table.

An important point to note about the QAQC tools is that they are designed to look for various errors in your datasets and report them to you; the tools themselves do not fix these errors for you but indicate to you where you may need to investigate your data.

The following table lists the QA/QC tool options for each data type:

Table 2.6 QA/QC Tools

Data Type	QA/QC Tools
Collar data	Dummy East, North or RL Duplicated Locations Reversed Easting/Northing Dip out of range -90 to 90 Azimuth out of range -360 to 360 Dip sign discrepancy Top of hole depth less than 0 Hole depth less than 0 Hole depth less than top of hole
Dip-azimuth survey	Dummy Depth Depth less than hole top Depth greater than hole bottom Duplicated Depths Out-of-sequence Depths Dummy Dip or Azimuth Dip out of range -90 to 90 Azimuth out of range -360 to 360 Dip sign discrepancy Max curvature - degrees per metre (or feet)
East-North survey	Dummy East, North or RL Duplicated Locations Reversed Easting/Northing

Data Type	QA/QC Tools
	Broken trend in RL Max curvature - degrees per metre (or feet)
From-To data	Dummy From or To From greater than To From equals To From or To less than hole top From or To greater than hole bottom Duplicated From-To intervals Out-of-sequence Depths Overlapping intervals Missing intervals Duplicated sample numbers
Point data	Dummy Depth Depth less than hole top Depth greater than hole bottom Duplicated Depths Out-of-sequence Depths Duplicated sample numbers

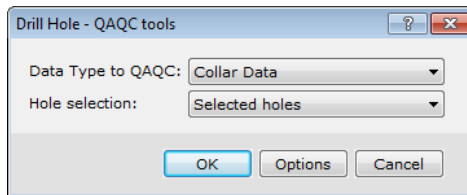
You will run the QAQC tools first on all data and then you will change some of the From-To options and run the tools again.

To run the QAQC tools on all data:

1. From the **Data** menu on the Target Drillhole toolbar, select **QA/QC**.

The Drill Hole - QAQC tools dialog opens.

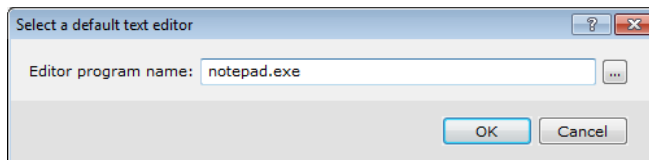
Figure 2.21 Drill Hole - QAQC tools dialog



2. From the **Data Type to QAQC** list, select **All Data**.
3. From the **Hole selection** list, select **All holes**.
4. Click **OK**.

If this is your first time editing a file in **Target for ArcGIS**, the Select a default text editor dialog opens.

Figure 2.22 Select a default text editor dialog



5. Click the **Browse** button to select a default text editor or click **OK** to select the default.

The QAQC.log file opens in a text editor.

Figure 2.23 QAQC.log file

```

QAQC.log - Notepad
File Edit Format View Help
Collar Data-----

Dip-Azimuth Survey Data-----

East-North Survey Data-----
    There is no East-North survey database.

From-To Data-----
Database: c:\geosoft training data\Target_Assay.gdb
AXE005 [line 0] : Missing From/To interval at hole top [0.00 : 50.00].
AXRC013 [line 1928] : Missing From/To interval at hole top [0.00 : 34.00].
AXRC016 [line 2233] : Missing From/To interval at hole top [0.00 : 34.00].
AXRC027 [line 3711] : Missing From/To interval [36.00 : 40.00].
AXRC030 [line 4319] : Missing From/To interval at hole bottom [189.00 : 190.00].
AXRC036 [line 5213] : Missing From/To interval [153.00 : 154.00].
AXRC046 [line 5924] : Missing From/To interval at hole top [0.00 : 34.00].
SKA041 [line 7098] : Missing From/To interval at hole bottom [81.00 : 83.00].
SKA052 [line 7146] : Missing From/To interval at hole bottom [87.50 : 88.00].
SKA053 [line 7172] : Missing From/To interval at hole bottom [75.50 : 76.00].
SKA065 [line 7467] : Missing From/To interval at hole bottom [72.00 : 73.00].
SKA070 [line 7580] : Missing From/To interval at hole bottom [75.00 : 78.00].
SKC255 [line 11057] : Missing From/To interval [39.00 : 42.00].
SKC380 [line 14601] : Missing From/To interval [87.00 : 90.00].
SKC386 [line 14878] : Missing From/To interval at hole top [0.00 : 36.00].
SKC389 [line 14942] : Missing From/To interval at hole top [0.00 : 36.00].
SKC390 [line 15016] : Missing From/To interval at hole top [0.00 : 36.00].
SKC391 [line 15100] : Missing From/To interval at hole top [0.00 : 42.00].
SKC392 [line 15138] : Missing From/To interval at hole top [0.00 : 42.00].
SKC392 [line 15150] : Missing From/To interval [54.00 : 55.00].
SKC393 [line 15245] : Missing From/To interval at hole top [0.00 : 36.00].
  
```

Examine the contents of this report. It shows that there are no errors in the Collar Data, the Dip-Azimuth Survey Data, nor the Structure data, and that an East-North Survey database was not imported.

However, the From-To data returned several “errors”. Errors occur in the Geology database; most of these are caused by missing and overlapping intervals. There are numerous drillholes with overlapping intervals and it may be necessary for the data logger to return to the source and make the necessary corrections.

The Assay database also has numerous missing intervals and a few errors where the final “To” of the “From-To” data is greater than the total hole bottom.

Errors in the databases should be corrected in the original source data rather than in the Geosoft Databases. That way, any colleagues using the same data will have access to the latest, updated data. As long as the channel names do not change and additional channels are not created, drillhole data can be re-imported into **Target for ArcGIS** by selecting the Refresh Project tool from the Target Project menu. Any new drillholes that are added to the source data will also be imported into the drillhole project.

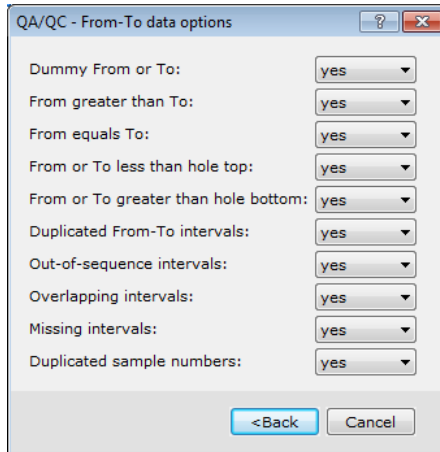
Next, you will run the QAQC tools on the From-To data and select some specific options.

To run the QA/QC tools on the From-To data:

1. From the **Data** menu on the Target Drillhole toolbar, select **QA/QC**.
The Drill Hole - QA/QC tools dialog opens.
2. From the **Data Type to QA/QC** list, select **From-To Data**.
3. Click **Options**.

The QA/QC - From-To data options dialog opens.

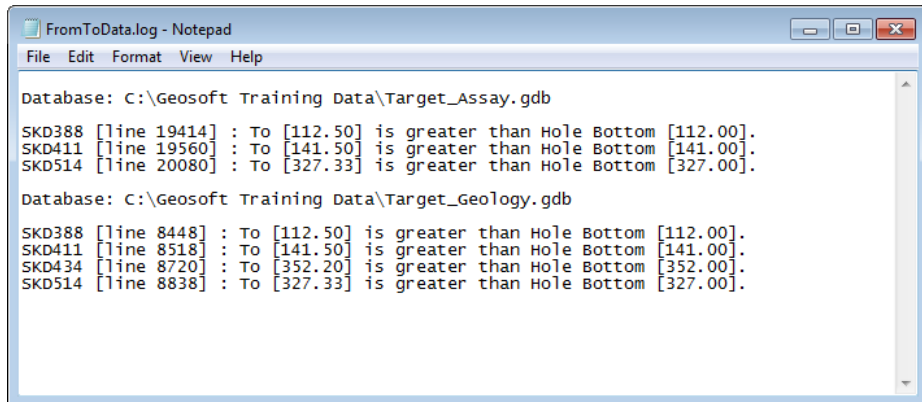
Figure 2.24 QA/QC - From-To data options dialog



4. For **Overlapping intervals** and **Missing intervals**, select **no**.
5. Click **Back**.
6. Click **OK**.

The FromToData.log file opens in a text editor. The QA/QC tools are run only on the From-To data.

Figure 2.25 FromToData.log file



Now would be a good time to save your map document.

In this lesson you:

- > Ran QAQC tools on all data
- > Ran QAQC tools on the From-To data

Lesson 2.4 Creating Composite Databases

In this lesson you will:

- › Create a composite database using the fixed interval method
- › Create a composite database using the lithological interval method
- › Create a composite database using the best-fit lithological interval method
- › Create a composite database using the significant intersections method

Composite Databases

A composite database is a database where the From–To data intervals have been re-calculated to an alternative interval. This practice is common when dealing with geochemical data; for example a geologist might want to re-calculate assay values over larger intervals than those originally sampled, perhaps to suit a certain deposit style or geotechnical scenario. The composite database will include all the selected lines from the input database and the From, To, Mask and all the assay channels from the input database.

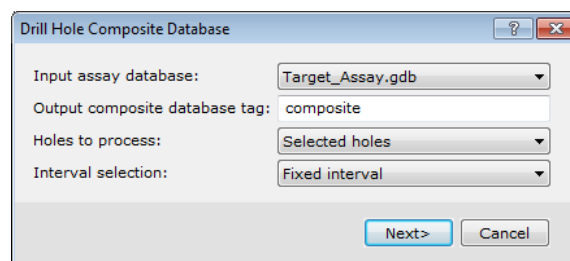
In this lesson, you will create a composite database from the Assay database. The original Assay database has inconsistent 'From-To' intervals; you will make the intervals an even 10 m with the average assay value calculated over this new interval. In this lesson, you will create composite databases based on lithology, best-fit lithological interval and significant intersections.

To create a composite database using fixed intervals:

1. From the **Data** menu on the Target Drillhole toolbar, select **Composite database**.

The Drill Hole Composite Database dialog box opens.

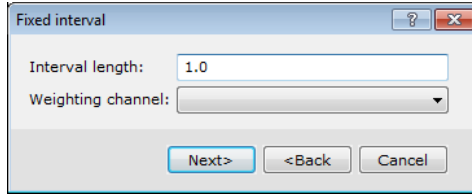
Figure 2.26 Drill Hole Composite Database dialog box



2. Ensure the **Input assay database** is **Target_Assay.gdb**.
For the remaining parameters, use the defaults.
3. Click **Next**.

The Fixed interval dialog box opens.

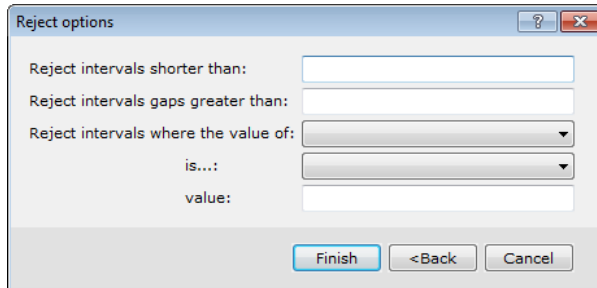
Figure 2.27 Fixed interval dialog box



4. For the **Interval length**, enter **10** and click **Next**.

The Reject options dialog box opens. For this lesson there is no need to specify any rejection options.

Figure 2.28 Reject options dialog box



5. Click **Finish**.

The composite From-To assay database is created.

Figure 2.29 Target_composite.gdb

	DH From	DH To	Au As	Au	As	High Au	Mask	DH East
AXE005	0.0	10.0	*	*	*	*	1	716677.
	10.0	20.0	*	*	*	*	1	716682.
	20.0	30.0	*	*	*	*	1	716687.
	30.0	40.0	*	*	*	*	1	716692.
	40.0	50.0	*	*	*	*	1	716697.
	50.0	60.0	0.0006	0.012	20.00	*	1	716702.

The assay values are now averaged over intervals of 0-10 m, 10-20 m, 20-30 m, etc. This can be useful for many reasons, but in particular it is useful for labelling. In Module 3, you will plot assay values down drillholes on a section map; if the values are pulled from the original assay database (which has samples at every 1-2 m), the section map can become very cluttered.

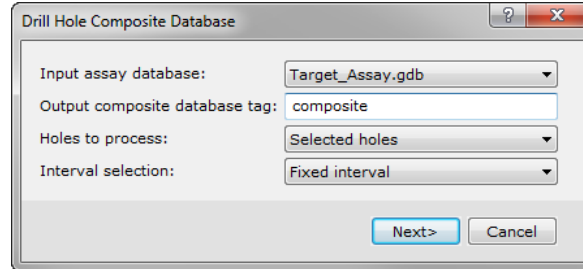
Databases can also be composited based on rock codes. For example, a geologist might want to determine assay values within a specified geological unit. The tool can also be used to merge adjacent intervals that have the same rock code. You will now do the latter using the lithological interval method.

To create a composite database using lithological intervals:

1. From the **Data** menu on the Target Drillhole toolbar, select **Composite database**.

The Drill Hole Composite Database dialog opens.

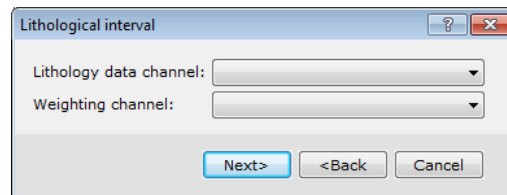
Figure 2.30 Drill Hole Composite Database dialog



2. From the **Input assay database** list, select **Target_Assay.gdbDrillhole_Geology.gdb**.
3. For the **Output composite database tag**, enter **litho_composite**.
4. From the **Interval selection** list, select **Lithological interval** and click **Next**.

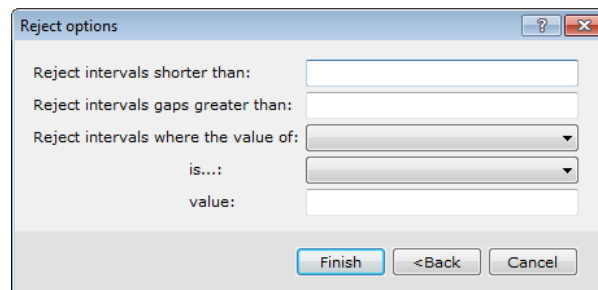
The Lithological interval dialog opens.

Figure 2.31 Lithological interval dialog



5. From the **Lithology data channel** list, select **Rock [Geology]** and click **Next**. The Reject options dialog opens. For this lesson there is no need to specify any rejection options.

Figure 2.32 Reject options dialog

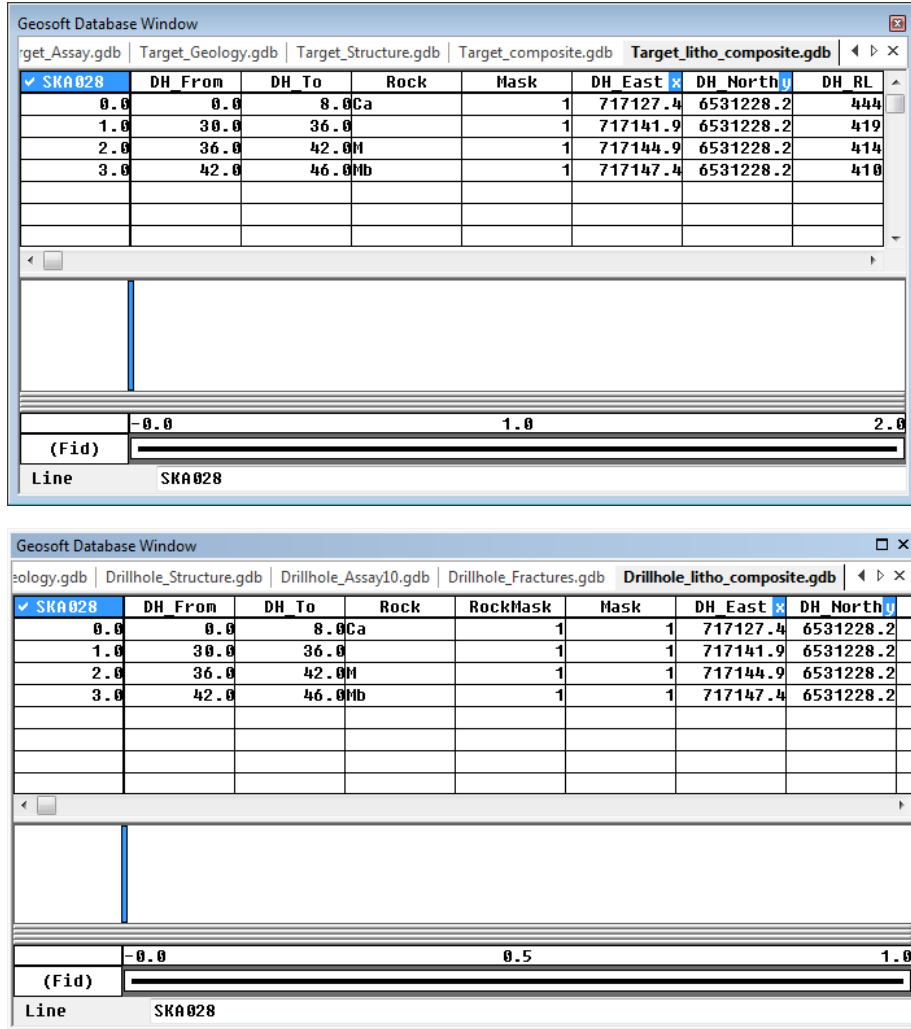


6. Click **Finish**.

The Drillhole_litho_composite.gdb database is created.

The Target_litho_composite from-to assay database is created.

Figure 2.33 Lithological interval composite database



The new database contains composited lithology and assay values that have been averaged over the composited interval. This average is weighted according to the length of the constituent intervals in the input database. The output database is also useful for creating less cluttered labelling on section maps.

 *Now would be a good time to save your map document.*

Best-Fit Lithological Interval Composite Databases

The best-fit lithological interval combines aspects of the fixed interval and the lithology interval composite databases. It requires an assay database and a lithology database.

The composite database is created by breaking the assay data into user-defined fixed intervals. Lithology is then matched to those intervals as best as possible. If a lithological interval is shorter than the user-defined interval value, then it is output "as-is". A lithological interval is subdivided if it is larger than the user-defined interval.

To create a best-fit composite database:

1. From the **Data** menu in the Target Drillhole toolbar, select **Composite Database**.

The Drill Hole Composite Database dialog opens.

2. For the **Input assay database** select **Target_Assay.gdb**.
3. For the **Output composite database tag**, enter **best_fit_composite**.
4. For the **Interval selection**, select **Best-fit lithological interval**.
5. Click **Next**.

The Best-fit Lithological interval dialog opens. The Rock [Geology] lithology data channel is already selected as this channel was just used to create the lithological interval composite database.

Figure 2.34 Best-fit Lithological interval dialog

6. For the **Interval length**, enter **10** and click **Next**.

The Reject options dialog opens. For this lesson there is no need to specify any rejection options.

7. Click **Finish**.

The Target_best_fit_composite from-to assay database is created.

Figure 2.35 Best-fit composite database

AXRC051	DH From	DH To	Rock	Au	As	RockMask	Mask
0.0	0.0	2.0	X	*	*	1	1
1.0	2.0	11.0		*	*	1	1
2.0	11.0	13.0	Ca	*	*	1	1
3.0	13.0	18.0		*	*	1	1
4.0	18.0	25.3	Ca	*	*	1	1
5.0	25.3	32.7	Ca	*	*	1	1
6.0	32.7	40.0	Ca	0.00	*	1	1
7.0	40.0	47.0		0.01	*	1	1
8.0	47.0	54.0		0.02	*	1	1
9.0	54.0	61.0		0.84	*	1	1
10.0	61.0	62.0	Uac	2.94	*	1	1
11.0	62.0	67.5	Hb	0.83	*	1	1
12.0	67.5	73.0	Hb	0.31	*	1	1
13.0	73.0	80.0	Uac	0.04	*	1	1
14.0	80.0	88.0	Hb	0.29	*	1	1
15.0	88.0	96.0	Uac	0.03	*	1	1
16.0	96.0	100.0	Hb	0.03	*	1	1



When the line header cell is selected, you can use the Page Up and Page Down keys to scroll through the lines.

Significant Intersections Composite Databases

You can use this tool to identify significant grade intersections in your assay database. You are able to define significant intersection criteria (i.e. minimum composite length and minimum composite grade). The tool uses the specified cutoff grade to divide assay intervals into high- and low-grade intervals. Adjacent high-grade intervals (those greater than or equal to the cutoff grade) will be composited. If the resulting composited interval satisfies the defined significant intersection criteria, it is added to the newly created database. You may also specify dilution zone criteria to include short, low grade intervals in the compositing. The resulting database will include the new from-to intervals, the length of the interval, and the composited data for all your assay channels. The database will also contain a channel with labels you can use when plotting maps.

To create a composite database using significant intersections:

1. From the **Data** menu in the Target Drillhole toolbar, select **Composite Database**.

The Drill Hole Composite Database dialog opens.

2. From the **Input assay database** list, select **Target_Assay.gdb**.
3. For the **Output composite database tag**, enter **significant_int_composite**.
4. From the **Interval selection** list, select **Significant intersections**.
5. Click **Next**.

The Significant intersection options dialog opens.

Figure 2.36 Significant intersection options dialog

6. For the **Primary assay data channel**, select **Au [Assay]**.
7. For **Minimum composite grade**, enter **0.3**.

Output only composite intervals whose composite (weighted mean value) grade is equal to or greater than this value.

8. For **Cutoff grade**, enter **0.1**.

The Cutoff grade must be less than the Minimum composite grade, which in this case is 0.3.

Only individual from-to intervals where the primary assay data channel's grade is greater than or equal to this value will be included in the composite interval.

9. For **Clip values higher than**, enter **100**.

If the clip limit is defined, any values above the clip limit are set to the clip limit value.



Any missing intervals, or any intervals whose grade is recorded as a dummy "", are assigned the "Grade for missing assays".*

10. Click **Finish**.

The Target_significant_int_composite.gdb database is created.

Figure 2.37 Significant intersections composite database

Line	Hole ID	DH From	DH To	Length	Au	Au Labels
0.0	AXRC035	67.00	68.00	1.0	0.84	1.0m @ 0.84 Au
1.0	AXRC035	71.00	73.00	2.0	1.22	2.0m @ 1.22 Au
2.0	AXRC035	80.00	82.00	2.0	1.23	2.0m @ 1.23 Au
3.0	AXRC035	88.00	89.00	1.0	1.38	1.0m @ 1.38 Au
4.0	AXRC035	95.00	96.00	1.0	0.90	1.0m @ 0.90 Au
5.0	AXRC035	99.00	100.00	1.0	2.55	1.0m @ 2.55 Au
6.0	AXRC035	102.00	103.00	1.0	2.15	1.0m @ 2.15 Au
7.0	AXRC035	136.00	138.00	2.0	2.34	2.0m @ 2.34 Au
8.0	AXRC035	140.00	141.00	1.0	5.95	1.0m @ 5.95 Au
9.0	AXRC035	160.00	161.00	1.0	1.27	1.0m @ 1.27 Au
10.0	AXRC035	163.00	164.00	1.0	1.50	1.0m @ 1.50 Au
11.0	AXRC035	166.00	167.00	1.0	2.65	1.0m @ 2.65 Au



Now would be a good time to save your project.

In this lesson you:

- > Created a composite database using the fixed interval method
- > Created a composite database using the lithological interval method
- > Created a composite database using the best-fit lithological interval method
- > Created a composite database using the significant intersections method

Lesson 2.5 Using Drillhole Data Management Tools

In this lesson you will:

- > Create, edit and delete a drillhole
- > Create and edit a rock code file
- > Set global drillhole preferences

Target for ArcGIS provides several drillhole management tools, including tools to create, edit and delete drillholes, edit rock and structure codes and set project preferences.

Adding a Proposed Hole to a Drillhole Project

With the New Drill Hole tool, you can manually add new drillholes to your project.

There are a number of different ways of manually defining a new drill hole:

- > based on the collar and end-of-hole locations
- > based on the collar location, dip and azimuth
- > based on setting a target location to be intersected from a given dip and azimuth

In this lesson, you will define the location for a new proposed hole.

To create a new drillhole:

1. From the **Data** menu on the Target Drillhole toolbar, select **New drill hole**.

The New Drill Hole dialog box opens.

Figure 2.38 New Drill Hole dialog

2. For **Drill Hole name**, enter **SKC350**.
3. For **Location**, enter the following:
 - > X: **716756.5**
 - > Y: **6531283.4**
 - > Z: **441.2**

4. For **Dip**, enter **-60.0**.
5. For **Azimuth**, enter **90.0**.
6. For **Planned depth**, enter **170**.
7. Click **OK**.

Note how the fields that are checked to "calculate" update (in this case the End of Hole coordinates) as you define your drillhole attributes. The new proposed drillhole is added to your collar and survey databases.

Edit Drill Hole

With **Target for ArcGIS** you can edit drillhole collar information and save the changes to your project database. By using this editing tool, any edits you make to a drillhole will be propagated to all relevant databases in the drillhole project. Here, you will edit the proposed drillhole and move it 5 m to the east.

To edit a drillhole:

1. From the **Data** menu on the Target Drillhole toolbar, select **Edit drill hole**.
The Edit Drill Hole dialog opens.
2. From the **Drill Hole name** list, select **SKC350**.

The Edit Drill Hole dialog displays the drillhole collar information for your proposed hole SKC350.

Figure 2.39 Drill Hole dialog

3. In the Collar Coordinates section, change the **Location X** value to **716761.5**.
4. Click **OK**.

The edited location for the proposed drillhole is now updated in both the Target_Collar and Target_Survey databases.



If you are sharing data with colleagues, you should edit the original source data rather than making edits in the Geosoft Databases. Once changes are made to the source data tables, they can be re-imported into your project using the Refresh Project tool from the Target Project menu.

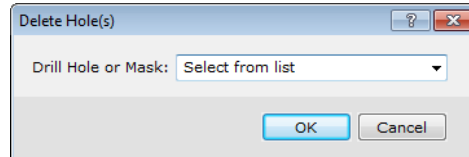
Delete Drill Hole

Target for ArcGIS enables you to delete a drillhole from your project collar database. The tool will remove all data associated with the selected hole(s) from each database in the project.

To delete a drillhole:

1. From the **Data** menu on the Target Drillhole toolbar, select **Delete drill hole**.
The Delete Hole(s) dialog opens.

Figure 2.40 Delete Hole(s) dialog



2. From the **Drill Hole or Mask** list, select **SKC350** and click **OK**.
A message window opens indicating the drillhole has been removed from the project.

Next, you will examine the tool for editing rock codes.

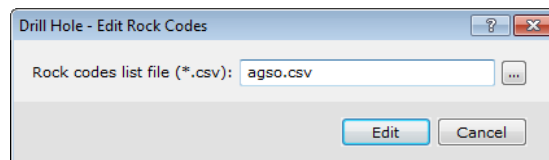
Editing Rock Codes

Rock codes and their associated patterns are usually used with From-To data to represent lithology and other logged rock properties on maps. You can use the editing tool to edit an existing rock code file, or to create a new one.

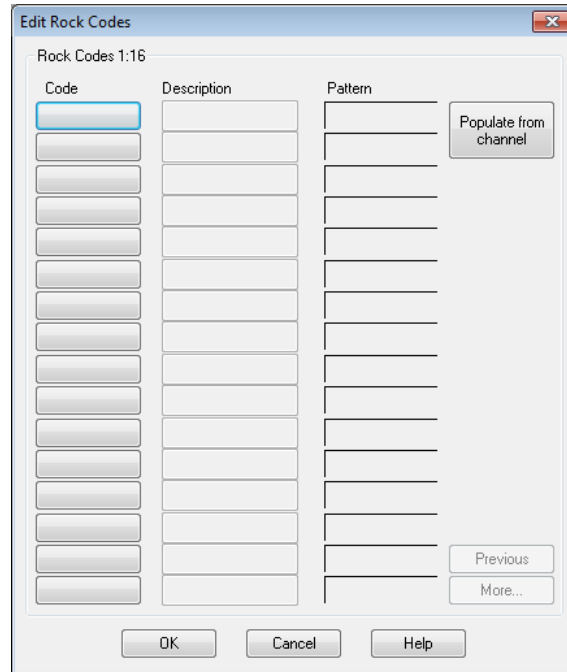
To create a new rock code file:

1. From the **Data** menu on the Target Drillhole toolbar, select **Edit rock codes**.
The Drill Hole - Edit Rock Codes dialog opens listing the default agso.csv rock code file or which ever rock code file that was used in your last drillhole project.

Figure 2.41 Drill Hole - Edit Rock Codes dialog

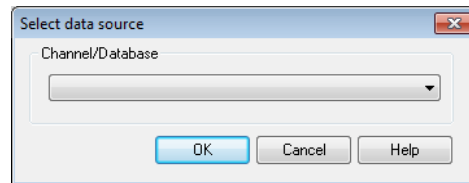


2. For the **Rock codes list file (*.csv)**, click on the browse button.
3. Navigate to your "Logos and Legends" enter **New rock codes.csv** as the filename.
This will create a new rock code .csv file.
4. Click **Edit**.
The Edit Rock Codes dialog opens.

Figure 2.42 Edit Rock Codes dialog

5. Click **Populate from channel**.

The Select data source dialog opens. This lets you select a channel from which the rock code file will be automatically populated.

Figure 2.43 Select data source dialog

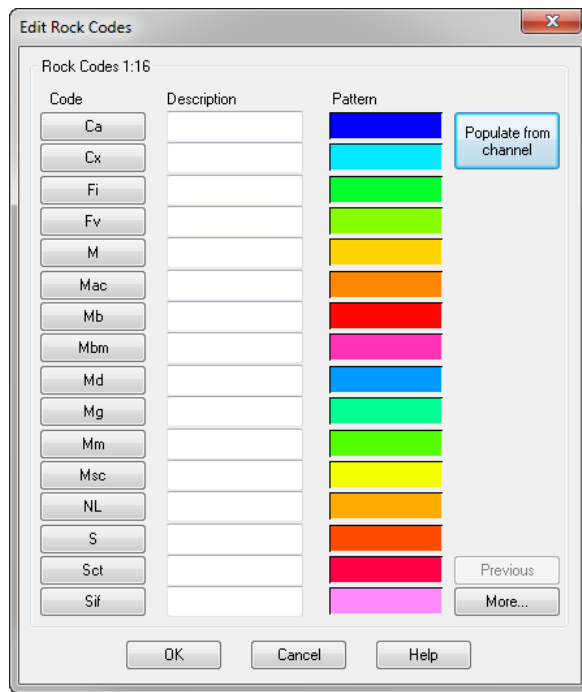
6. From the **Channel/Database** list, select **Rock [Geology]** and click **OK**.

This is the Rock Channel from the Geology database.

A message window opens indicating there are new items to import into the rock code file.

7. Click **Yes**.

The Edit Rock Codes dialog is populated with the rock codes from the Rock channel in the Target_Geology.gdb database.



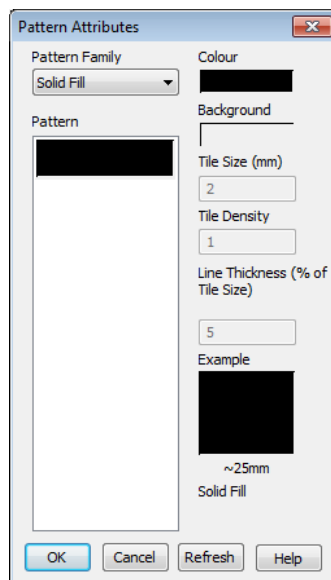
Now you will assign descriptions and patterns for the rock codes.

To edit rock codes:

1. Click in the **Description** box beside the Ca rock code and type **Alluvium - depositional**.
2. Click in the **Pattern** box beside the Ca rock code.

The Pattern Attributes dialog opens. You can use this to select the Pattern Family (Solid Fill, Objects, Hatches and Geology), Colour, Background Colour, Pattern, Size, Density and Thickness.

Figure 2.44 Pattern Attributes dialog



3. From the **Pattern Family** menu, select a category of patterns.

4. Select a **Pattern** and click **OK**.
5. Once you have edited the Description and Pattern for a few rock codes, click **OK**.

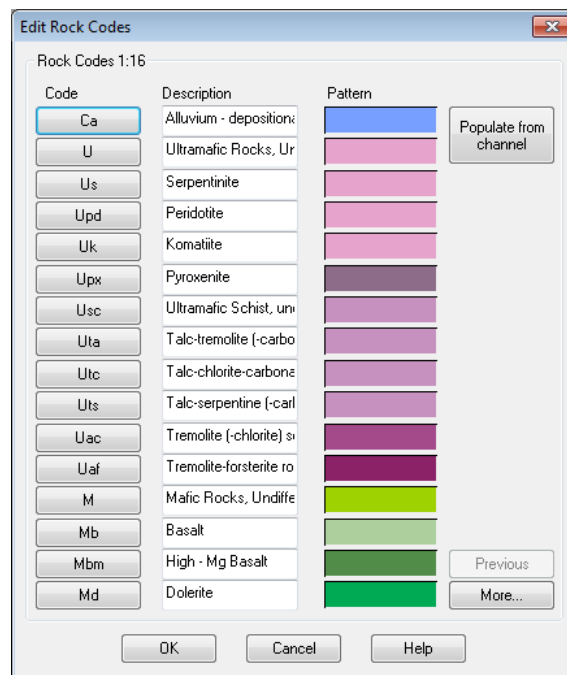
Because there are a number of rock codes in this file, you will now view an existing rock code file. You will then specify this as your default rock code file.

To view an existing rock code file:

1. From the **Data** menu on the Target Drillhole toolbar, select **Edit rock codes**.
The Drill Hole - Edit Rock Codes dialog opens.
2. For the **Rock codes list file (*.csv)**, click **Browse**.
3. From the **Logos and Legends** folder in the Geosoft Training Data folder, select **Lithology Rock Code.csv** and click **Open**.
4. Click **Edit**.

The Lithology rock codes are displayed in the Edit Rock Codes dialog. Edits can now be made to this file, although it is not necessary for this lesson.

Figure 2.45 Lithology Rock Code.csv displayed in the Edit Rock Codes dialog



Structure Code File

The Structure Code file is similar to a Rock Codes file, except that only solid colours can be specified, not patterns. Structure data can be represented on maps as one of several types of symbol; the structure code file dictates the colour of those symbols.

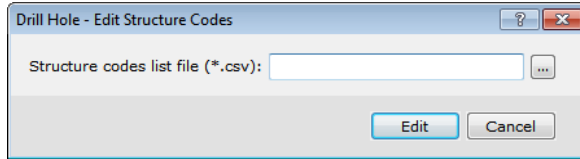
Examine the Structure_Type channel in the Drillhole_Structure database. You will now view the associated structure code file which provides a description and colour for each of these structure types.

To view the structure code file:

1. From the **Data** menu on the Target Drillhole toolbar, select **Edit structure codes**.

The Drill Hole - Edit Structure Codes dialog opens.

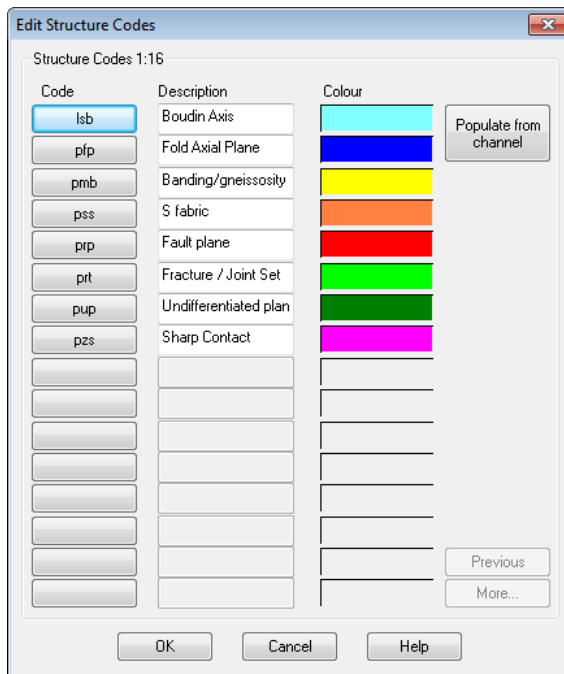
Figure 2.46 Drill Hole - Edit Structure Codes dialog



2. For the **Structure codes list file (*.csv)**, click the **Browse** button.
3. From the **Logos and Legends** folder, select **Structure Codes.csv** and click **Open**.
4. Click **Edit**.

The Edit Structure Codes dialog opens. It lists the code, description and colour for each structure code.

Figure 2.47 Edit Structure Codes dialog



Note the structure type described as a Fracture/Joint Set and its code, prt.

5. Click **OK**.

Drillhole Preferences

Drillhole preferences are global settings that allow you to customise various aspects how data is processed and visualised within Target for ArcGIS. Once set, they apply to all **Target for ArcGIS** projects that you create.

To set drillhole preferences:

1. From the **Target Project** menu on the Target Drillhole toolbar, select **Preferences**.

The Drill Hole Preferences dialog opens.

Figure 2.48 Drill Hole Preferences dialog

Drill Hole Preferences dialog box showing the following settings:

- Rock codes list file (*.csv): agso.csv
- Structure codes list file (*.csv):
- Resurveying method: Radius of curvature
- Order for polynomial fit: 5
- Hole discretization interval: 0.1
- Hole survey dip angle sign (downward): negative
- Right/left for posted data: hole reference
- N-S sections face...: west
- Auto-convert negatives on import?: yes
- Use mask channel for plotting/export?: no
- Default mask channel: Mask
- Azimuth magnetic declination correction: 0
- Angled section axis origin: Center

Drill Hole Preferences dialog box showing the following settings:

- Rock codes list file (*.csv): agso.csv
- Structure codes list file (*.csv): structcodes.csv
- Resurveying method: Radius of curvature
- Order for polynomial fit: 5
- Hole discretization interval: 0.1
- Hole survey dip angle sign (downward): negative
- Right/left for posted data: hole reference
- N-S sections face...: west
- Auto-convert negatives on import?: yes
- Use mask channel for plotting/export?: no
- Default mask channel: Mask
- Azimuth magnetic declination correction: 0
- Angled section axis origin: Center

2. For **Rock codes list file (*.csv)**, click **Browse**.
3. From the **Logos and Legends** folder, select **Lithology Rock Code.csv** and click **Open**.

By selecting your default rock code file here, you will not need to browse for it while creating section maps, strip logs or 3D views.

4. For **Structure codes list file (*.csv)**, click **Browse**.
5. From the **Logos and Legends** folder, select **Structure Codes.csv** and click **Open**.

By selecting your default rock and structure code files here, you will not need to browse for them while creating section maps, strip logs or 3D views.

6. Click **OK**.



Now would be a good time to save your map document.

In this lesson you:

- > Created, edited and deleted a drillhole
- > Created and edited a rock code file
- > Set global drillhole preferences

Module 3: Mapping Drillhole Data

Module 3 has Seven Lessons:

Lesson 3.1	Selecting Drillholes	86
Lesson 3.2	Creating a Plan Map	89
Lesson 3.3	Creating a Section Map	106
Lesson 3.4	Creating a Stacked Section Map	123
Lesson 3.5	Creating a Fence Diagram	129
Lesson 3.6	Creating a Strip Log	134
Lesson 3.7	Creating a 3D Map	147

Lesson 3.1 Selecting Drillholes

In this lesson you will:

- › Select drillholes
- › Change the page size and orientation

Selecting Drillholes

A key requirement for geologists and geochemists evaluating and presenting drillhole data is the ability to select or deselect various combinations of drillholes. The following selection methods are available:

- › **Selecting (or Deselecting) All Holes:** Allows you to quickly select or deselect all holes and enables quick development of plan maps or resetting of drillholes prior to making new selections.
- › **Selecting Holes by Name or Name Mask:** Enables batch selection of multiple holes with systematic drillhole identifiers. This requires a standard drillhole naming scheme for identifying related holes.
- › **Selecting Holes from List:** Enables single or multiple selection from a list.
- › **Selecting Holes via a Selection Tool:** Enables custom selection of single or multiple holes from a list of all the holes in the database. You can select interactively by drawing a polygon around or clicking holes individually.

You can also select or mask data by code or range:

- › **Masking Data by Code:** Enables the creation or update of a mask channel based on values matched from any other channel.
- › **Masking Data by Range:** Enables the setting of a mask channel dependent on whether values in a specified channel are within a specified range.

In addition, you can also save and load your selections:

- › **Save and/or Load Selections:** Enables you to save the current hole selections, or load previous hole selections from a selection file (*.sel).

Selecting Holes using the Hole Selection Tool

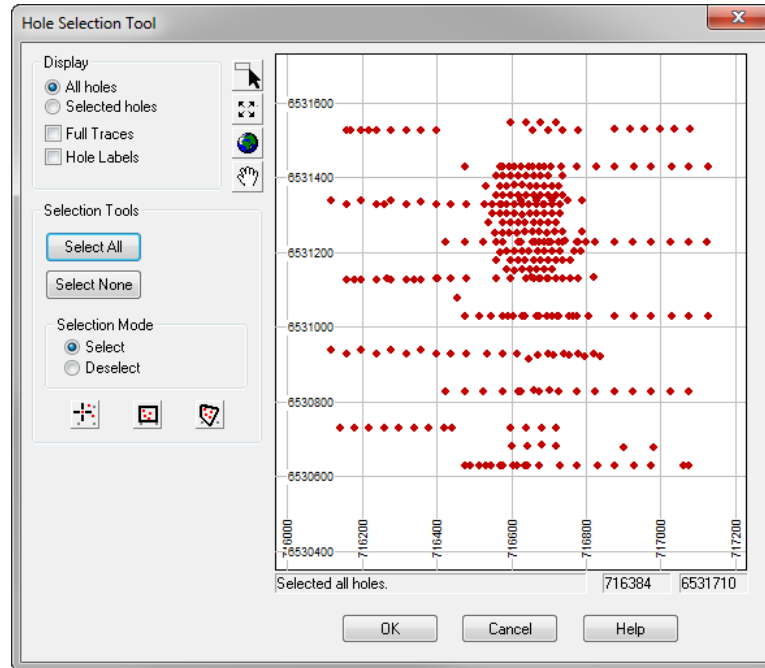
The Hole Selection Tool enables you to interactively select single or multiple holes from a plan view of the drillhole project. You can view all holes, selected holes and holes including full traces or labels.


You will now use the Hole Selection Tool to select the holes to be plotted on your plan map.

To select holes using the Hole Selection Tool:

1. From the **Selection** menu on the Target Drillhole toolbar, select **Selection tool**.
The Hole Selection Tool dialog opens. By default, all the holes are selected.

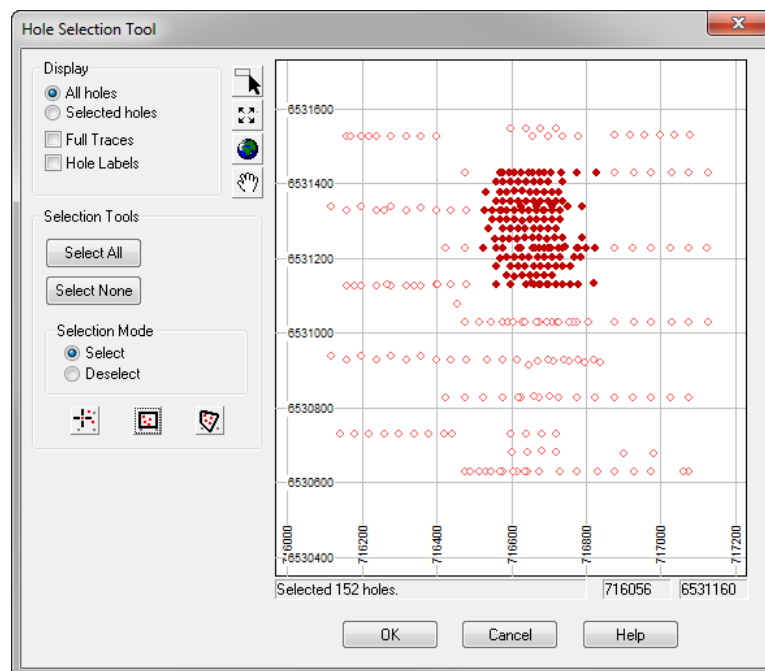
Figure 3.1 Hole Selection Tool dialog



2. To clear the selected holes, click **Select None**.
3. Ensure the **Selection Mode** is set to **Select**.
4. Click the **Select/Deselect using rectangle tool** button .
5. **Click** and **move** your mouse to define the size and location of the rectangle selection tool, as shown in the figure below.

The drillholes within the rectangle you defined are now selected.

Figure 3.2 Selected drillholes



6. Click **OK**.

To change the page size and orientation:

1. From the ArcMap **File** menu, select **Page and Print Setup**.
The Page and Print Setup dialog opens.
2. From the **Paper Size** list, select either **11x17** or **A3**.
3. For **Orientation**, select **Landscape**.
4. For **Map Page Size**, select the **Use Printer Paper Settings** option.
5. Select the **Scale Map Elements proportionally to changes in Page Size** option and click **OK**.
6. **Save** the map document (.mxd) file.

In this lesson you:

- › Selected drillholes
- › Changed the page size and orientation

Lesson 3.2 Creating a Plan Map

In this lesson you will:

- › Create a Geosoft Map Layer plan map
- › Change the collar symbol and hole trace parameters
- › Plot a grid
- › Add a legend and titles
- › Create a plan map in shapefile format
- › Symbolise the shapefiles and add map elements

Plan Maps

Plan maps provide a surface view of drillhole collars, drillhole traces, graticules, and surface grids. You can also include a title block, list of the holes, map information, north arrow and scale bar. Plan maps allow you to quickly display drillhole collars and traces, to help establish subsurface drilling coverage. Plan maps can also be used to define the location and orientation of section maps.

The Target for ArcGIS plotting tools provide two types of output; shapefiles and Geosoft Map Layers. The Shapefile output creates points, lines, and polygons to represent data in your maps which you can then modify using the tools available within ArcMap and create ESRI-style map layouts.

Geosoft Map Layers are static images that are ready for printing; they contain Geosoft's pre-defined annotations and legends, and allow you to pre-define colours, patterns and line styles. The Geosoft Map Layers is not a typical GIS layer. The ArcMap 'Identify' cursor, symbology and labelling tools are not available for Geosoft Map Layers. You cannot modify a Geosoft Map Layer; if changes are required, you must create a new one.

Since ArcMap is designed to function as a single-map environment, you will save the Target for ArcGIS Drilling.mxd file as a new .mxd file in which you will create the plan map.

To create a new .MXD:

1. From the **File** menu, select **New**.
2. Select **Blank Map** and click **OK**.
A new, untitled map is created.
3. From the **File** menu, click **Save**.
4. For the **File name**, type **Plan Map.mxd** and click **Save**.

The new map document opens with the existing drillhole project open. The drillhole selection for your project is also maintained, as well as the page setup parameters for the MXD itself.




You can also click the New Map File button on the Standard toolbar.

Geosoft Map Layer Output

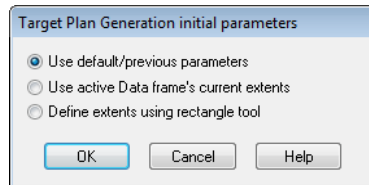
You will now create a plan map in Geosoft Map Layer format. You will begin by creating a plan map of the selected drillholes using the default parameters. Then, you will adjust various parameters to change the display of the map.

To create a plan map:

1. On the Target Drillhole toolbar, click the **Generate Target Plan Plot** button .

The Target Plan Generation initial parameters dialog opens.

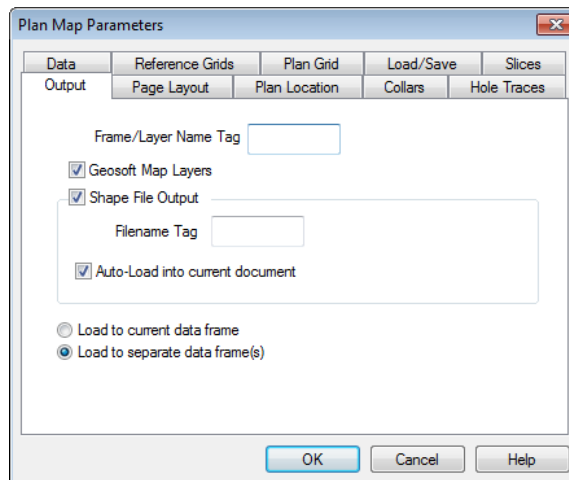
Figure 3.3 Target Plan Generation initial parameters dialog



2. Select **Use default/previous parameters** and click **OK**.

The Plan Map Parameters dialog opens. Both Geosoft Map Layer and Shape File outputs are selected by default. In this lesson you will start by only creating a Geosoft Map Layer.

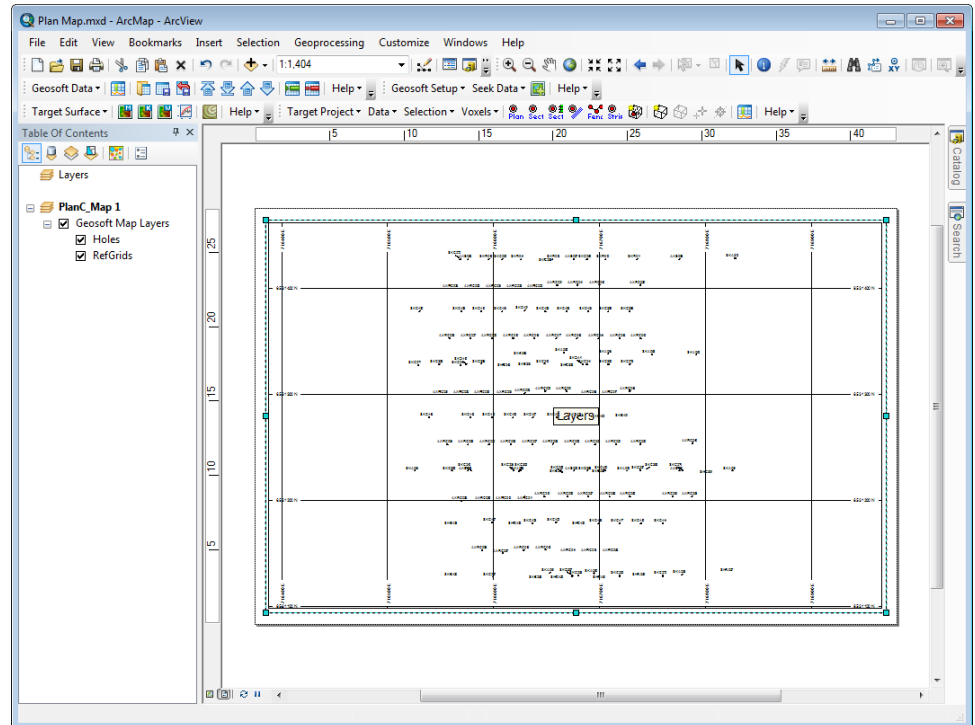
Figure 3.4 Plan Map Parameters dialog



3. For **Frame/Layer Name Tag**, enter **Map 1**.
4. Clear the **Shape File Output** option.
5. Click **OK**.

The PlanC_Map 1 data frame is added to the Table Of Contents; it contains the Holes and RefGrids layers which are parts of the Geosoft Map Layer output.

Figure 3.5 Default Plan Map



The Layers data frame is not needed and can be removed.

To remove the default data frame:

- In the ArcMap Table Of Contents, right-click the **Layers** data frame and select **Remove**.

There are several parameters available to help you customise the display of your plan map. The following table describes each tab in the Plan Map Parameters dialog:

Table 3.1 Plan Map Parameters dialog tabs

Tab	Description
Output	Use this tab to specify the output of the plan map as either Geosoft Map Layer or shapefile format.
Page Layout	Use this tab to specify the page layout of the plan plot(s), including the map components, legend, and titles.
Plan Location	Use this tab to specify the location of the plan map. By default, this is defined by the extent of the selected holes.
Collars	Use this tab to select specific symbols for plotting hole collars. These options only affect Geosoft Map Layer output.
Hole Traces	Use this tab to define the hole traces on the plan map. The hole traces can be modified in the shapefile output;

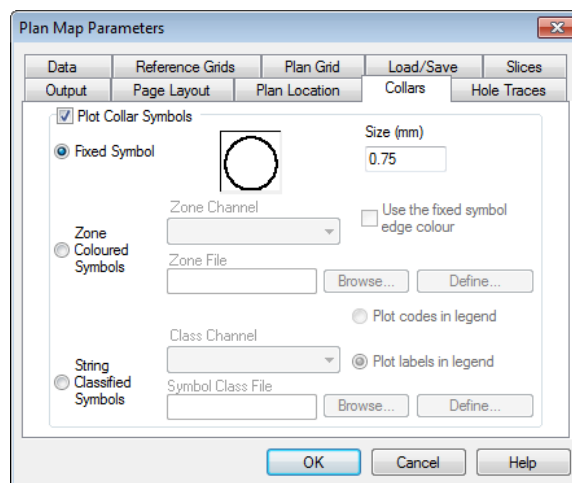
Tab	Description
	line width and colour options only apply to Geosoft Map Layer output.
Data	Use this tab to specify the data and plot type for your map.
Reference Grids	Use this tab to specify the parameters for displaying reference grids of local eastings and northings on the plan map. These options only affect Geosoft Map Layer output.
Plan Grid	Use this tab to plot a plan grid on the map.
Load/Save	Use this tab to load and/or save map and data plotting parameters to a file for sharing or for reuse.
Slices	Use this tab to plot a slice from a voxel, 3D DXF or 3D Shapefile on the plan map.

The collar locations are plotted using a circle symbol by default. You will now reopen the Plan Map Parameters dialog and change the way the collars are displayed on the Geosoft Map Layer.

To change the collar symbols:

1. On the Target Drillhole toolbar, click **Generate Target Plan Plot**.
The Target Plan Generation initial parameters dialog opens.
2. Select **Use default/previous parameters** and click **OK**.
The Plan Map Parameters dialog opens.
3. Click the **Output** tab.
4. For **Frame/Layer Name Tag**, enter **Map 2**.
5. Click the **Collars** tab.

Figure 3.6 Plan Map Parameters Collars tab



6. For **Size (mm)**, enter **2**.

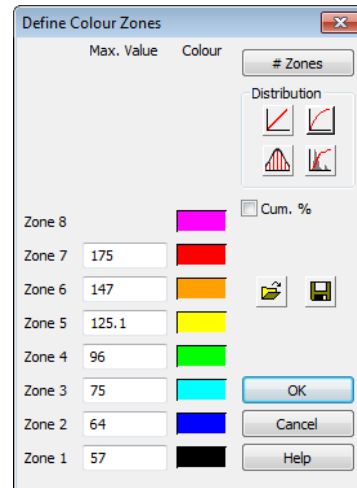
7. Select the **Zone Coloured Symbols** option.
8. From the **Zone Channel** list, select **DH_Bottom**.

DH_Bottom is a channel created in your Collar Database that is based on the total depth values there were imported into the drillhole project. Each collar will be coloured based on these values.

9. Click **Define**.

The Define Colour Zones dialog opens. Here you can specify the number of zones, the maximum value of each zone, the colour to represent each zone and the distribution of the data.

Figure 3.7 Define Colour Zones dialog



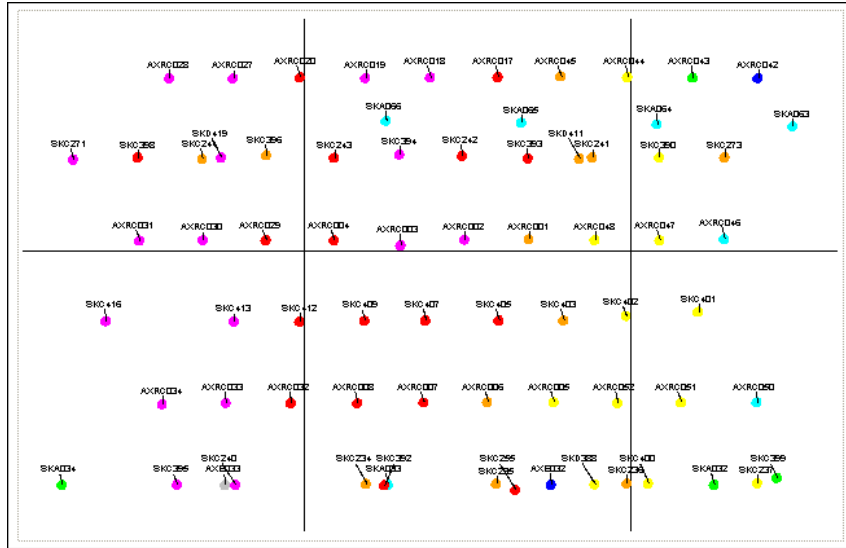
10. For **Zone 1**, click the black colour box and change the colour to **grey**.
11. Click **OK**.

A Zone File called DH_DH_Bottom.itr will be created and saved in your project folder. This file saves the zones and colours you define. You can use this file again later on if you want to use the same parameters in another map.

12. Click **OK**.

A new data frame, PlanC_Map 2, is added to the Table Of Contents. The collars are now plotted using coloured symbols based on the drillhole depth.

Figure 3.8 Plan Map with Zone Coloured Collars (zoomed)



The PlanC_Map 1 data frame is no longer needed and can be removed.

To remove the data frame:

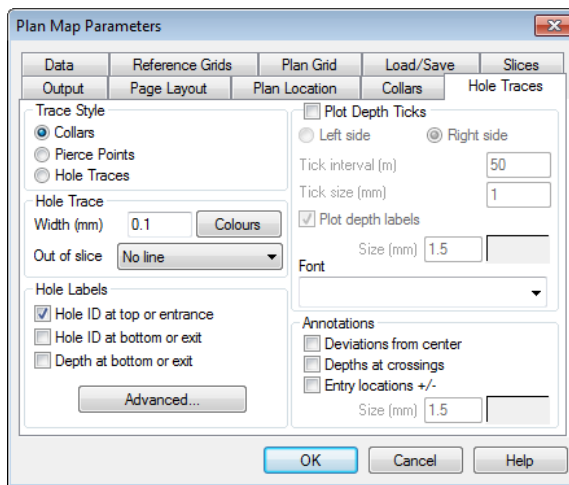
- In the ArcMap Table Of Contents, right-click the **PlanC_Map 1** data frame and select **Remove**.

Now you will adjust the parameters for the Hole Traces.

To change the hole trace parameters:

1. Open the **Plan Map Parameters** dialog
2. Click the **Output** tab.
3. For **Frame/Layer Name Tag**, enter **Map 3**.
4. Click the **Hole Traces** tab.

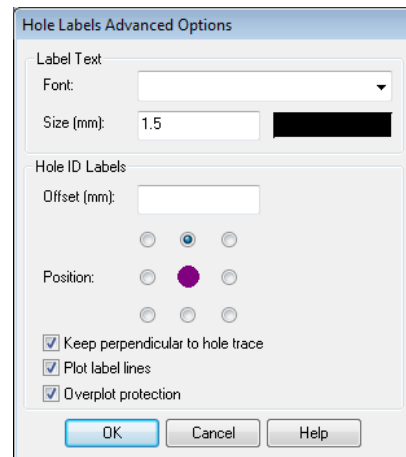
Figure 3.9 Plan Map Parameters Hole Traces tab



5. For the **Trace Style**, select **Hole Traces**.
6. In the Hole Labels section, click **Advanced**.

The Hole Labels Advanced Options dialog opens.

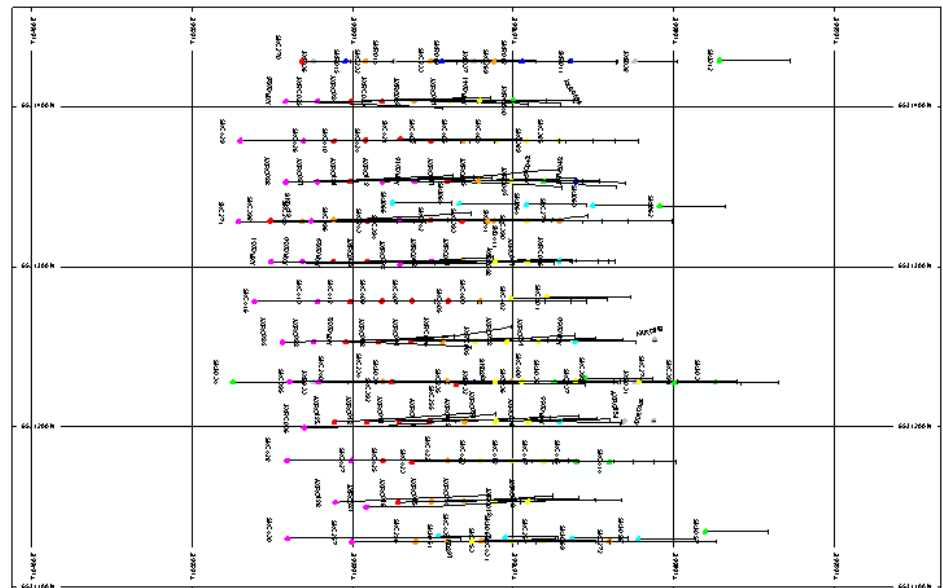
Figure 3.10 Hole Labels Advanced Options dialog



7. For Label Text **Size (mm)**, enter **2.0**.
8. Clear the **Plot label lines** option.
9. Click **OK**.
10. Clear the **Plot Depth Ticks** option.
11. Click **OK**.

A new data frame, PlanT_Map 3, is added to the Table Of Contents and displays the hole traces for each drillhole.

Figure 3.11 Plan Map with Hole Traces



The PlanC_Map 2 data frame is no longer needed and can be removed.

To remove the data frame:

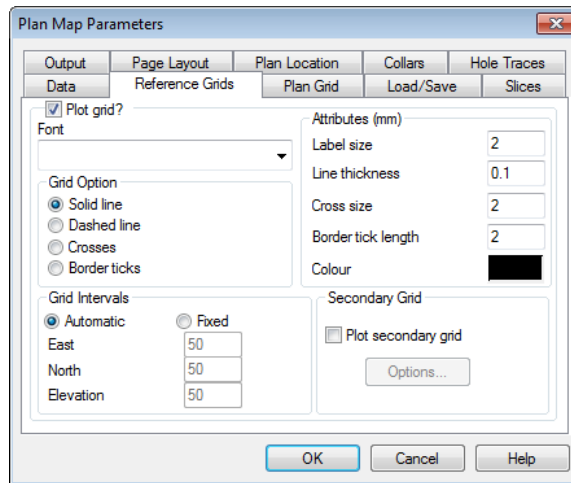
- In the ArcMap Table Of Contents, right-click the **PlanC_Map 2** data frame and select **Remove**.

Next you will change the display of the Reference Grid and display a topography grid on the plan map.

To change the display of the Reference Grid and display topography:

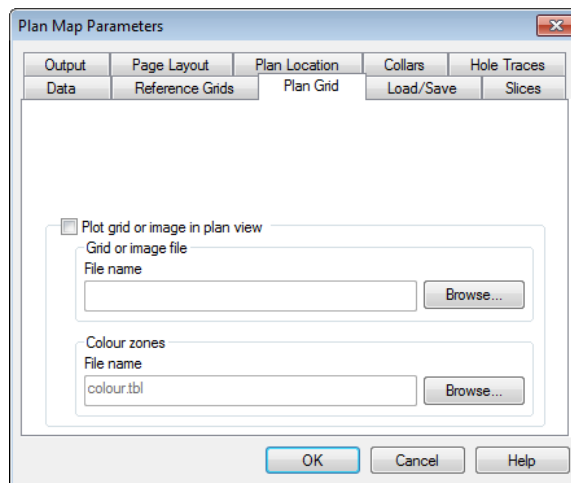
1. Open the **Plan Map Parameters** dialog .
2. Click the **Output** tab.
3. For **Frame/Layer Name Tag**, enter **Map 4**.
4. Click the **Reference Grids** tab.

Figure 3.12 Plan Map Parameters Reference Grids tab



5. For the Grid Option, select **Border ticks**.
6. Click the **Plan Grid** tab.

Figure 3.13 Plan Map Parameters Plan Grid tab

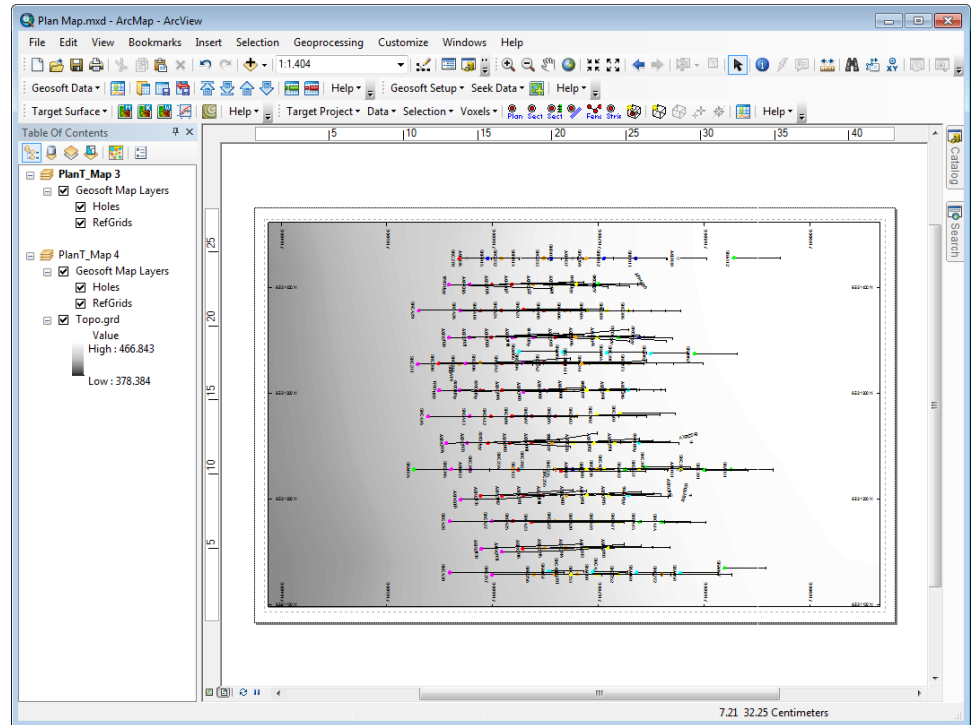


7. Select the **Plot grid or image in plan view** option.
8. For the **Grid or image file name**, click **Browse**.
9. From the **Topography** folder, select **Topo.grd** and click **Open**.
10. For the **Colour zones file name**, click **Browse**.

11. Within the Geosoft Colour Table Directory, navigate to the monochromatic folder and select **grey.tbl** and click **Open**.
12. Click **OK**.

A new data frame, PlanT_Map 4, is added to the Table Of Contents; the reference grid is now displayed with border ticks instead of solid lines and the topography grid is displayed.

Figure 3.14 Plan map with topography grid



The PlanT_Map 3 data frame is no longer needed and can be removed.

To remove the data frame:

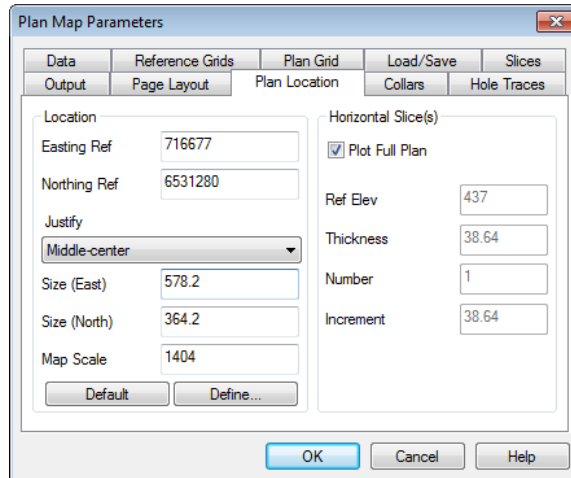
- › In the ArcMap Table Of Contents, right-click the **PlanT_Map 3** data frame and select **Remove**.

Finally, you will specify the map scale and add a legend and titles to your map.

To specify the scale and add a legend and titles:

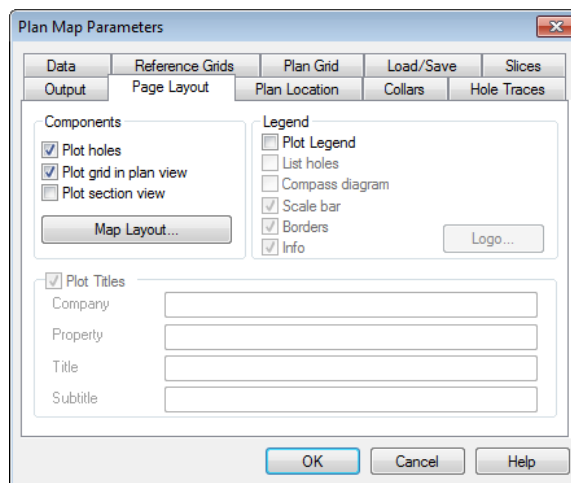
1. Open the **Plan Map Parameters** dialog.
2. Click the **Output** tab.
3. For **Frame/Layer Name Tag**, enter **Map 5**.
4. Click the **Plan Location** tab.

Figure 3.15 Plan Map Parameters Plan Location tab



5. For **Map Scale**, enter **1500**.
6. Click the **Page Layout** tab.

Figure 3.16 Plan Map Parameters Page Layout tab



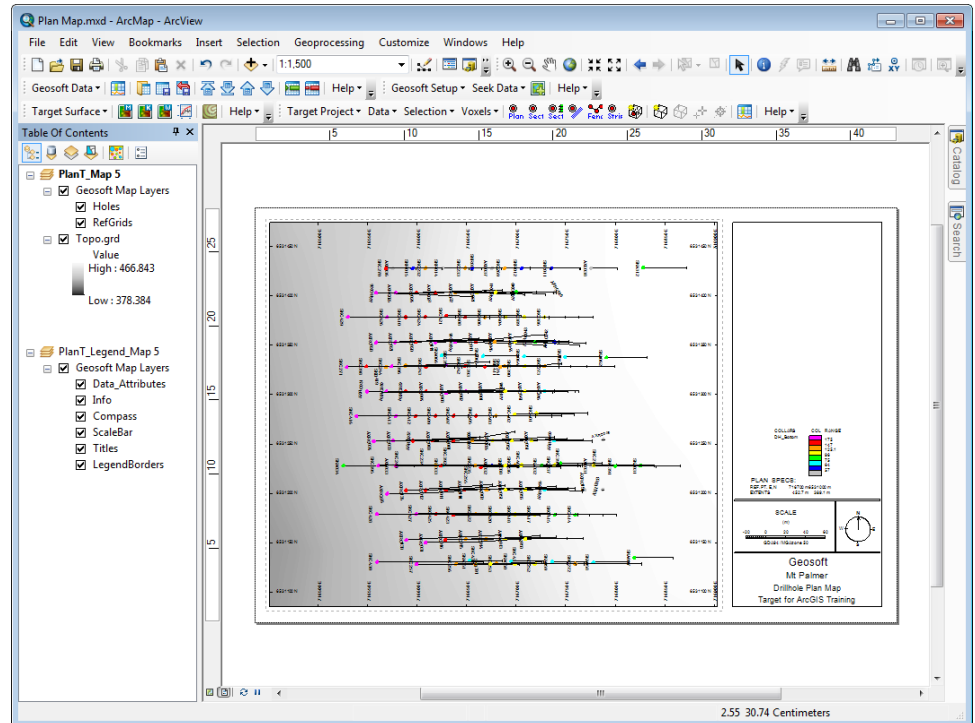
7. Select the **Plot Legend** and **Compass diagram** options.
8. Select **Plot Titles** and enter the following information:
 - > Company: **Geosoft**
 - > Property: **Mt Palmer**
 - > Title: **Drillhole Plan Map**
 - > Subtitle: **Target for ArcGIS Training**
9. Click **OK**.

Two new data frames are added to the Table Of Contents; PlanT_Map 5 and PlanT_Legend_Map 5. The legend data frame contains several Geosoft Map Layers for which you can turn the display off and on.

The PlanT_Map 4 data frame is no longer needed and can be removed.

10. In the ArcMap Table Of Contents, right-click the **PlanT_Map 4** data frame and select **Remove**.

Figure 3.17 Plan Map with legend and titles



Now would be a good time to save your map document.

Shapefile Output

You will now create a plan map in shapefile format.

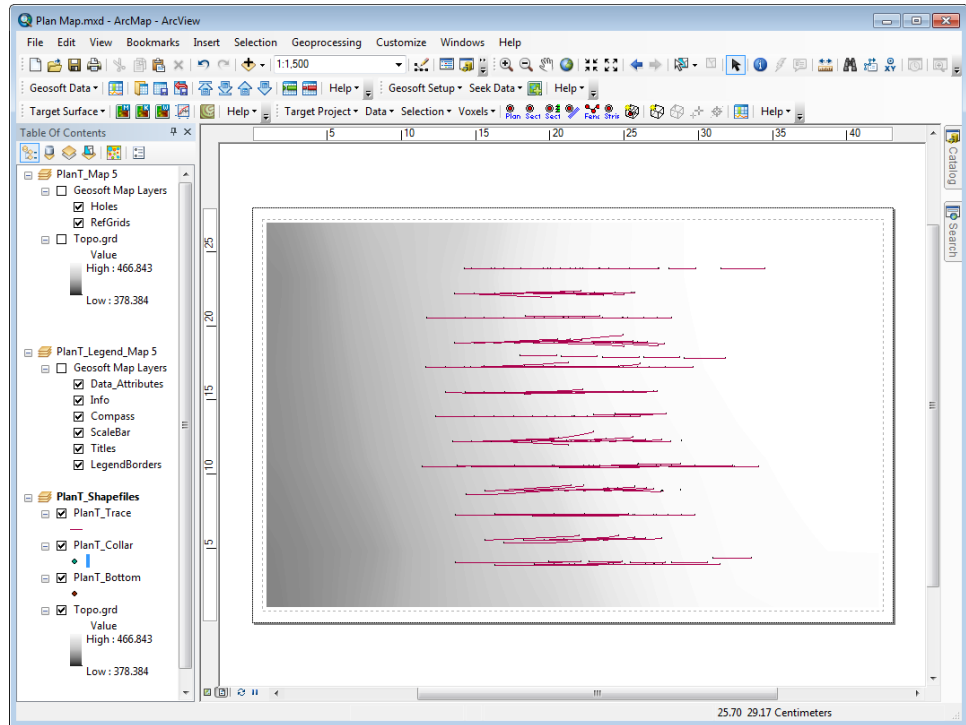
To create a plan map in shapefile format:

1. Open the **Plan Map Parameters** dialog.
All the parameters you used to create the Geosoft Map Layer are remembered.
2. Click the **Output** tab.
3. For **Frame/Layer Name Tag**, enter **Shapefiles**.
4. Clear the **Geosoft Map Layer** option and select the **Shape File Output** option.
5. Click the **Page Layout** tab and clear the **Plot Legend** option.
6. Click the **Reference Grids** tab and clear the **Plot grid** option.
7. Click **OK**.

Three shapefiles are created in the new PlanT_Shapefiles data frame: PlanT_Collar and PlanT_Bottom are point layers representing the locations of the drillhole collars and the bottom of the traces, respectively; the PlanT_Trace is a line layer representing the hole traces.

8. Turn off the display of the PlanT_Map 5 and PlanT_Legend_Map 5 data frames.

Figure 3.18 Plan Map shapefile output



Currently, the PlanT_Collar layer is displayed with the default symbology. You will now edit the symbology to display this layer similarly to the Geosoft Map Layer.

To change the symbology of the collar shapefile:

1. Double-click the **PlanT_Collar** layer in the Table Of Contents.
The Layer Properties dialog opens.
2. Click the **Symbology** tab.
3. In the Show area on the left, click **Quantities**.
The Graduated colors option is automatically selected.
4. From the **Value** list, select **Depth**.
The Symbol, Range and Label columns update to show the data classified based on the natural statistical groupings of data values.
5. From the **Classes** list, select **8**.
6. Click the **Symbol** column header and select **Properties for All Symbols**.
The Symbol Selector dialog opens.
7. Select **Circle 1** and for **Size** enter **12**.
8. Click **OK**.
9. Referring to the table below, change the Symbol colours and enter the following Range values.

Symbol Colour	Range Maximum
grey	57

Symbol Colour	Range Maximum
dark blue	64
light blue	75
green	97
yellow	125.8
orange	147
red	175
magenta	230

10. Click **OK**.

These symbol colours and range values now correspond to those in the Geosoft Map Layer.

Next you will change the symbology of the hole traces and trace bottoms.

To change the symbology of the hole traces and trace bottoms:

1. Double-click the **PlanT_Trace** layer.

The Layer Properties dialog opens.

2. On the **Symbology** tab, click the **Symbol** button showing you the current default symbol.

The Symbol Selector dialog opens.

3. Change the line colour to **Black** and click **OK**.

4. Click **OK**.

The hole traces are now displayed as black lines.

5. Change the symbology for the **PlanT_Bottom** layer to a **black cross**.



It is a good idea to change the symbology of the shapefiles once you are happy with the output from Target for ArcGIS; otherwise, if you recreate them your shapefiles may be overwritten and will be displayed with the default symbology.



Before inserting map elements on your map, ensure you are in Layout View mode and that PlanT_Shapefiles is the active data frame.

To switch between Data View and Layout View:

> From the **View** menu, select **Layout View**.

The entire map is displayed and the Layout toolbar is activated.



You can also switch between views by clicking the Data View button  and the Layout View button  on the lower left of the display window.

You will now add a legend and scale bar for your geochemical data.

To activate a data frame:

- > In the ArcMap Table Of Contents, right-click the **Plan Map Shapefiles** data frame and select **Activate**.

 You can also activate a data frame by pressing the **ALT** key and clicking the name of the data frame.

 To refresh the active view, click the **Refresh View** button  in the lower left of the display window or press **F5** on your keyboard.

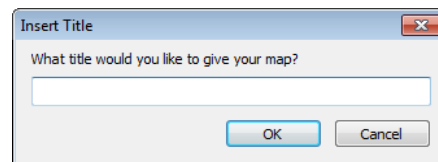
You will now insert a title, legend, north arrow and scale bar to your map using the ArcMap tools.

To insert a title:

1. From the **Insert** menu, select **Title**.


The Insert Title dialog opens.

Figure 3.19 Insert Title dialog



2. Type **Mt Palmer Plan Map** and click **OK**.

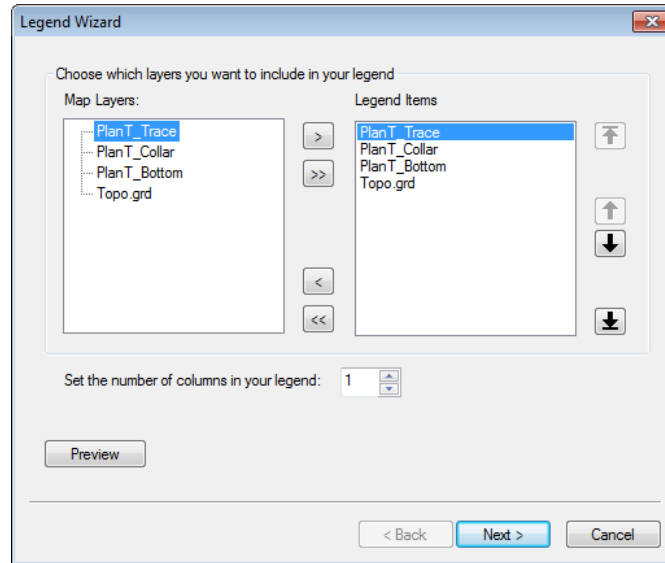
You can resize and reposition the title as necessary. You can also use the Symbol Selector to change the text properties.


 For any of the map elements you insert, you can double-click them on the map and a **Properties** dialog opens where you can make changes to their display.

To add a legend:

1. From the **Insert** menu, select **Legend**.

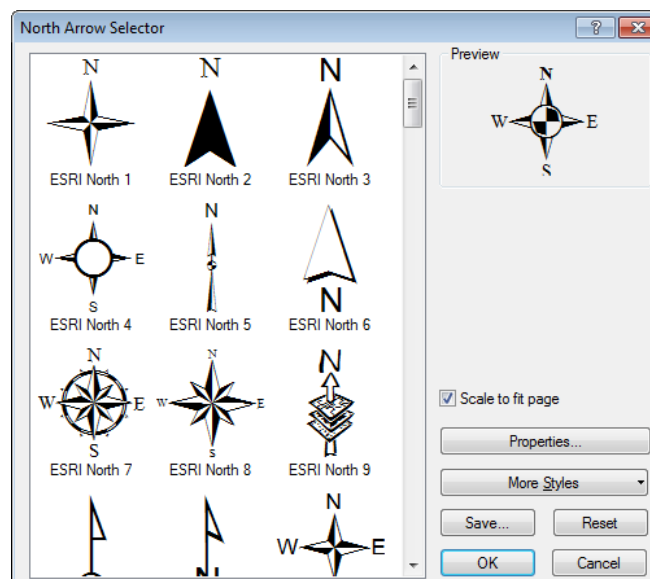
The Legend Wizard opens. You can use this wizard to control which layers are included in your legend as well as the size, spacing and text properties.

Figure 3.20 Legend Wizard

2. Click **Next** in each window to use the defaults.
You can also use the Preview button to help adjust the display properties.
 3. Click **Finish**.
The legend is added to your map.
 4. Using the Select Elements tool , you can select the legend on your map and resize and reposition as necessary.
- You will now insert a north arrow.

To insert a north arrow:

1. From the **Insert** menu, select **North Arrow**.
The North Arrow Selector dialog opens.

Figure 3.21 North Arrow Selector dialog

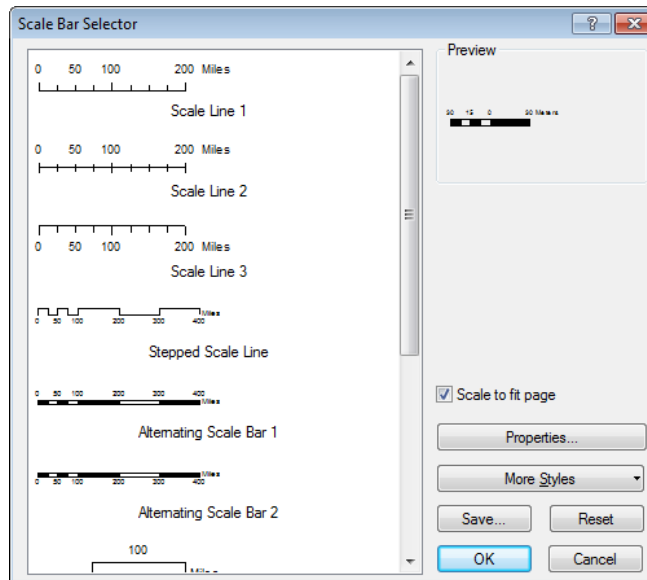
2. Select a north arrow from the list and click **OK**.
The selected north arrow is added to your map. You can resize and reposition it as necessary.

Next, you will insert a scale bar.

To insert a scale bar:

1. From the **Insert** menu, select **Scale Bar**.
The Scale Bar Selector dialog opens.

Figure 3.22 Scale Bar Selector dialog



2. Select a scale bar style from the list and click **OK**.
The selected scale bar is added to your map and shows the scale based on the extent of your data frame. You can reposition the size and location of the scale bar as necessary; the scale bar intervals will update automatically when you resize it.

As you insert map elements on your map, they may overlap the data displayed in your PlanT_Shapefiles data frame. You will now resize the data frame to improve the map layout.

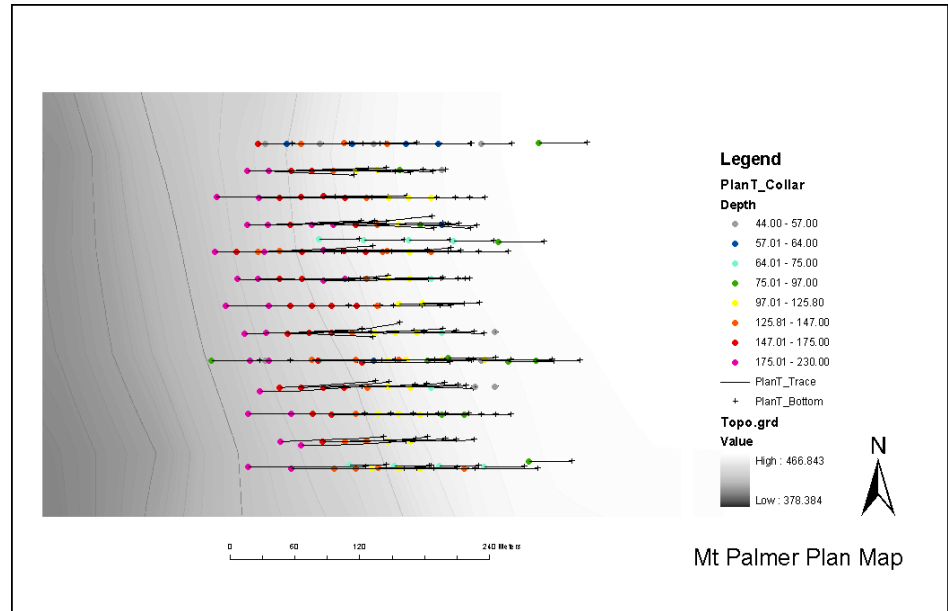
To resize the Plan Map Shapefile data frame:

1. Click on the map to select the PlanT_Shapefiles data frame.
2. Click and drag the right selection handle towards the left side of your map.

As you resize your data frame, your scale bar is automatically updated.

Your map should look similar to the one below.

Figure 3.23 Plan Map created from shapefile output



Now would be a good time to save your map document.

In this lesson you:

- > Created a Geosoft Map Layer plan map
- > Changed the collar symbol and hole trace parameters
- > Plotted a grid
- > Added a legend and titles
- > Created a plan map in shapefile format
- > Symbolised the shapefiles and added map elements

Lesson 3.3 Creating a Section Map

In this lesson you will:


- › Select drillholes for the section map
- › Create a Geosoft Map Layer section map
- › Change the display of the collar symbols and hole traces
- › Plot topography
- › Plot rock patterns, a bar plot, a grid and post values
- › Add a legend and plan view
- › Create a section map in shapefile format

Section Maps

Section maps allow you to visualise any of your drillhole data on a cross-section. Section maps can be generated in any orientation, in terms of strike and dip.

Since ArcMap is designed to function as a single-map environment, you will begin by creating a new map file in which you will create the section map.


To create a new .MXD:

1. On the Standard toolbar, click the **New Map File** button . The New Document dialog opens.
2. Click **OK**. A new, untitled map with the Target Project loaded, the drillholes selected and the correct page setup is created.
3. Save the map as **Section Map.mxd**.

Geosoft Map Layer Output

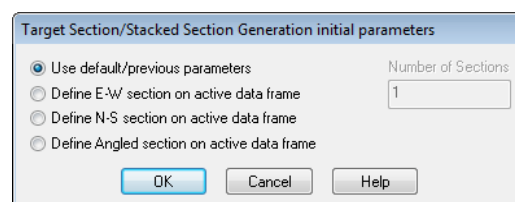
You will now create a section map in Geosoft Map Layer format.

To create a section map:

1. On the Target Drillhole toolbar, click the **Generate Target Section Plot** button .

The Target Section/Stacked Section Generation initial parameters dialog opens.

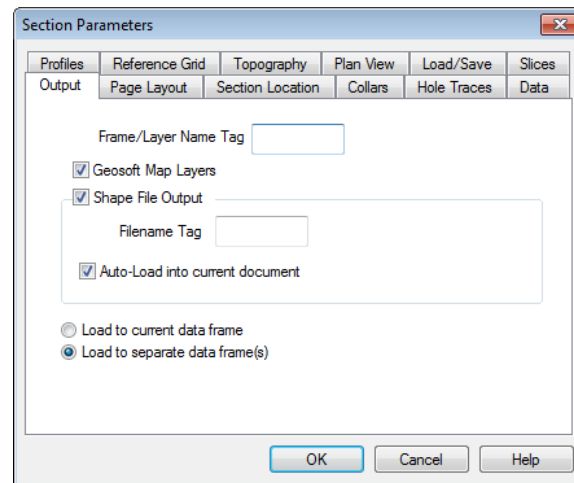
Figure 3.24 Target Section/Stacked Section Generation initial parameters dialog



2. Select **Use default/previous parameters** and click **OK**.

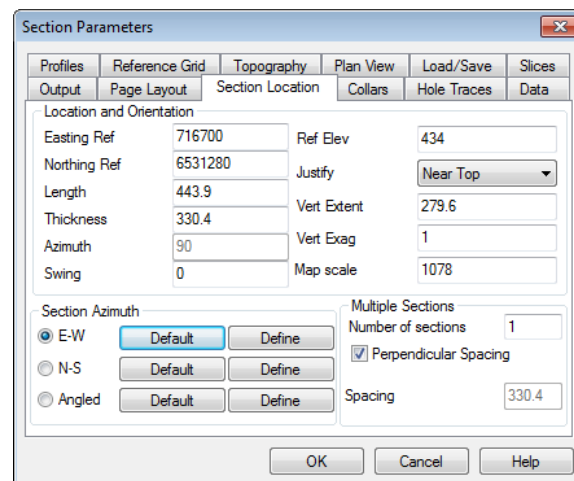
The Section Parameters dialog opens. Both Geosoft Map Layer and Shape File outputs are selected by default. In this lesson you will start by only creating a Geosoft Map Layer.

Figure 3.25 Section Parameters dialog



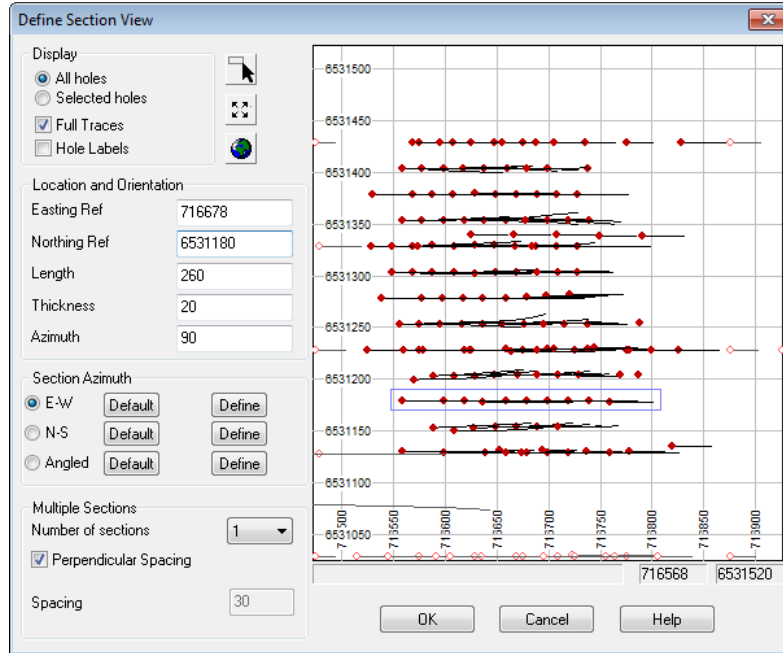
3. For **Frame/Layer Name Tag**, enter **Map 1**.
4. Clear the **Shape File Output** option.
5. Click the **Section Location** tab.

Figure 3.26 Section Parameter Section Location tab



6. Beside E-W in the Section Azimuth section, click **Define**.
The Define Section View dialog opens.
7. Click the **Zoom into selected area** button and zoom to the currently selected holes.
8. Beside E-W in the Section Azimuth section, click **Define**.
9. **Click** and **move** your mouse to define the size and location of the section.
Define your section as narrow as possible making sure to include the hole traces, as shown in the figure below.

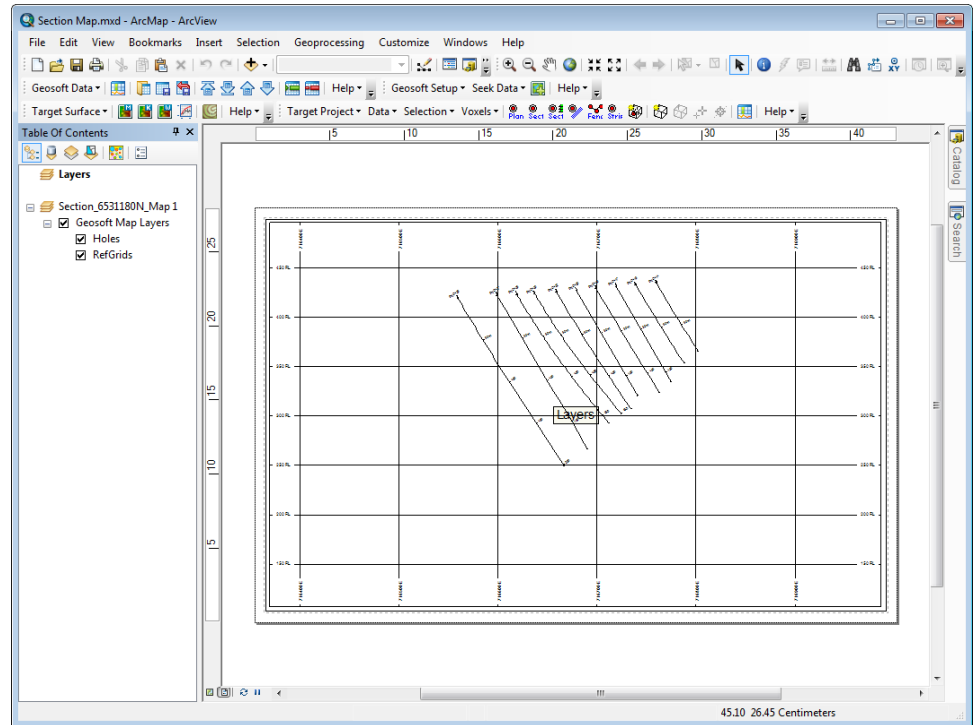
Figure 3.27 Defining the section location



10. Click **OK**.
11. For the Northing Ref, enter **6531180**.
12. For Map Scale, enter **1500**.
13. Click **OK**.

The Section_6531180N_Map 1 data frame is added to the Table Of Contents; it contains the Holes and RefGrids layers which are parts of the Geosoft Map Layer output.

Figure 3.28 Section Map



The Layers data frame is not needed and can be removed.

To remove the default data frame:

- › In the ArcMap Table Of Contents, right-click the **Layers** data frame and select **Remove**.

There are several parameters available to help you customise the display of your section map. The following table describes each tab in the Section Parameters dialog.

Table 3.2 Section Parameters dialog tabs

Tab	Description
Output	Use this tab to specify the output of the plan map as either Geosoft Map Layers or shapefile format.
Page Layout	Use this tab to specify the page layout of the section plot (s), including the map components, legend, and titles.
Section Location	Use this tab to specify the location of the section plot(s). A section plot is a slice of the ground, which is normally vertical but may also swing away from the vertical.
Collars	Use this tab to select specific symbols for plotting hole collars.
Topography	Use this tab to plot topography on the section plot(s).
Plan View	Use this tab to plot a plan view on the section plot(s).

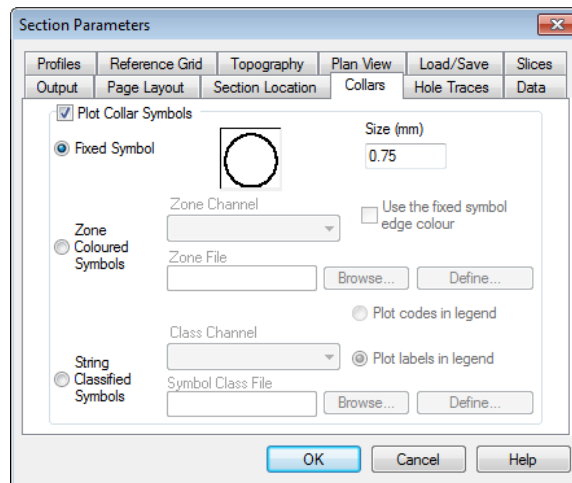
Tab	Description
Load/Save	Use this tab to load and/or save map and data plotting parameters to a file.
Slices	Use this tab to plot a slice from a voxel, 3D DXF or 3D Shapefile on the section map.
Hole Traces	Use this tab to define the hole traces of the section plot (s).
Data	Use this tab to specify the data and plot type for your map.
Profiles	Use this tab to plot profiles of gridded data on the section plot(s).
Reference Grid	Use this tab to specify the parameters for a reference grid displayed on the section plot(s).

By default, the collar locations and depth ticks are plotted. Next, you will reopen the Section Parameters dialog and plot the collar symbols with different colours based on who logged the data.

To change the display of the collar symbols:

1. Open the **Section Parameters** dialog and click the **Output** tab.
2. For **Frame/Layer Name Tag**, enter **Map 2**.
3. Click the **Collars** tab.

Figure 3.29 Section Parameters Collars tab

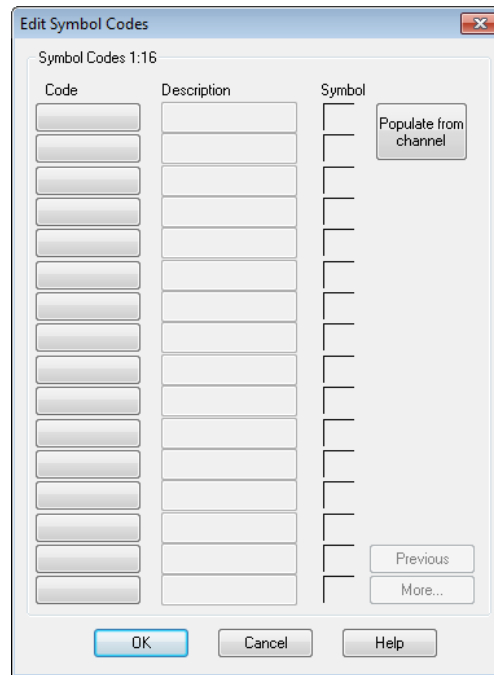


4. For **Size (mm)**, enter **2.0**.
5. Select the **String Classified Symbols** option.
6. From the **Class Channel** list, select **GEO** and click **Define**.
The Select collar symbol classification file name dialog opens.
7. For the **File name**, enter **SymbolCodes**.
This will automatically create a new .csv file with this name.

8. Click **Save**.

The Edit Symbol Codes dialog opens.

Figure 3.30 Edit Symbol Codes dialog



9. Click **Populate from channel**.

The Select data source dialog opens.

10. From the **Channel/Database** list, select **GEO** and click **OK**.

A message window opens indicating there are new items to import into the symbol code file.

11. Click **Yes**.

12. For each Code, click in the **Description** box to enter the following text and click in the **Symbol** box to select the following fill colours:

Code	Description	Symbol
NG	Natalie Green	green
SD	Sara Deschamps	orange
TM	Tara Macey	blue

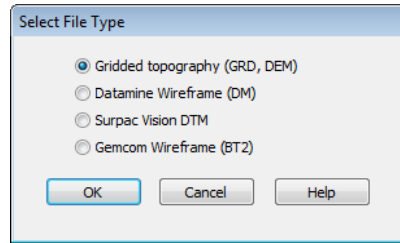
13. Click **OK**.

14. Click **OK**.

A new data frame, Section_6531180N_Map 2, is added to the Table Of Contents. The collars are now plotted using coloured symbols based on who logged the data.

The Select File Type dialog opens.

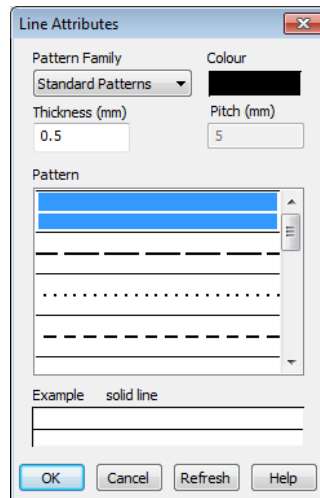
Figure 3.33 Select File Type dialog



5. Select **Gridded topography (GRD, DEM)** and click **OK**.
6. From the **Topography** folder, select **Topo.grd** and click **Open**.
7. Click in the **Line Style** box.

The Line Attributes dialog opens.

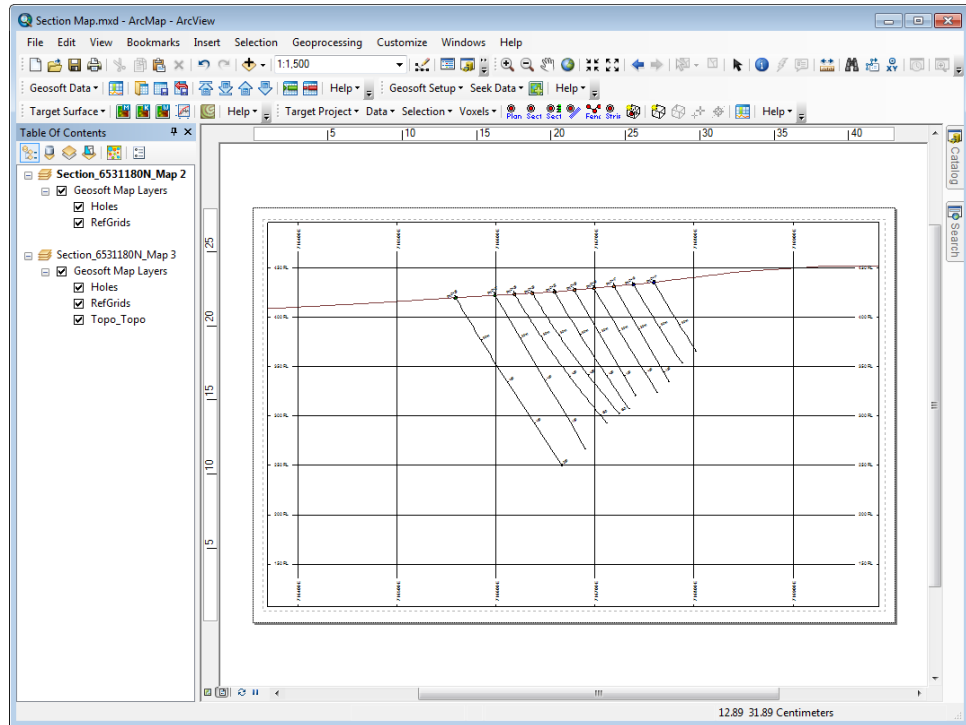
Figure 3.34 Line Attributes dialog



8. Click the **Colour** box, select **brown** and click **OK**.
9. Click **OK**.

A new data frame, Section_6531180N_Map 3, is added to the Table Of Contents; the topography profile is displayed and listed as a Geosoft Map Layer.

Figure 3.35 Section Map with Topography



The Section_6531180N_Map 2 data frame is no longer needed and can be removed.

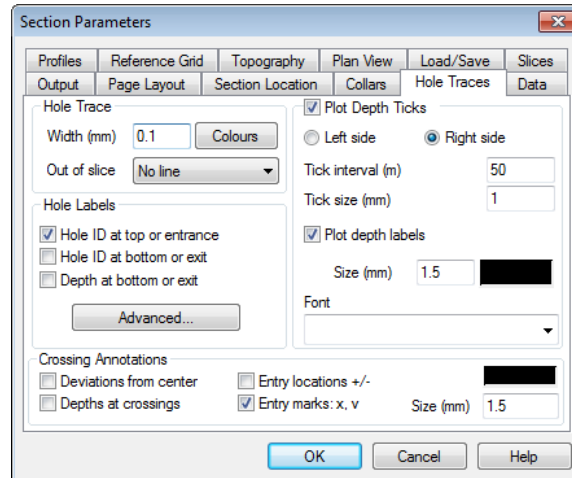
To remove the data frame:

- In the ArcMap Table Of Contents, right-click the **Section_6531180N_Map 2** data frame and select **Remove**.

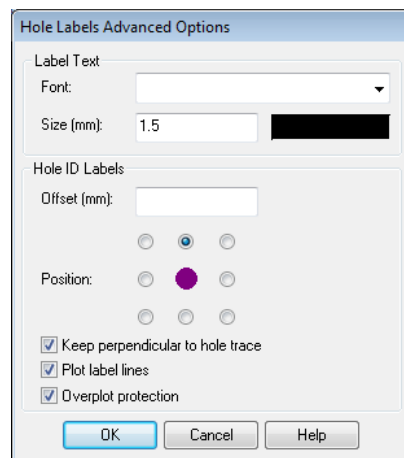
Now you will change the hole trace parameters to remove the label lines and depth ticks.

To change the hole traces parameters:

1. Open the **Section Parameters** dialog and click the **Output** tab.
2. For **Frame/Layer Name Tag**, enter **Map 4**.
3. Click the **Hole Traces** tab.

Figure 3.36 Section Parameters Hole Traces tab

- In the Hole Labels section, click **Advanced**.
The Hole Labels Advanced Options dialog opens.

Figure 3.37 Hole Labels Advanced Options dialog

- For Label Text **Size (mm)**, enter **2.0**.
- Clear the **Plot label lines** option.
- Click **OK**.
- Clear the **Plot Depth Ticks** option.
- Click **OK**.

A new data frame, Section_6531180N_Map 4, is added to the Table Of Contents; the depth ticks and label lines are no longer plotted.

- Remove** the **Section_6531180N_Map 3** data frame.

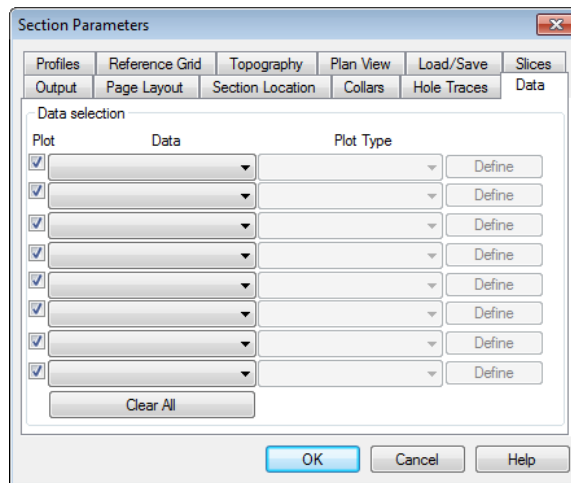
Next, you will plot four different types of data on your section map. First you will plot rock patterns, then a bar plot, a grid and finally post values.

To plot rock patterns on the section map:

- Open the **Section Parameters** dialog and click the **Output** tab.
- For **Frame/Layer Name Tag**, enter **Map 5**.

- Click the **Data** tab.

Figure 3.38 Section Parameters Data tab



- From the first **Data** list, select **Rock (Geology)**.
- From the **Plot Type** list, select **Rock patterns**.

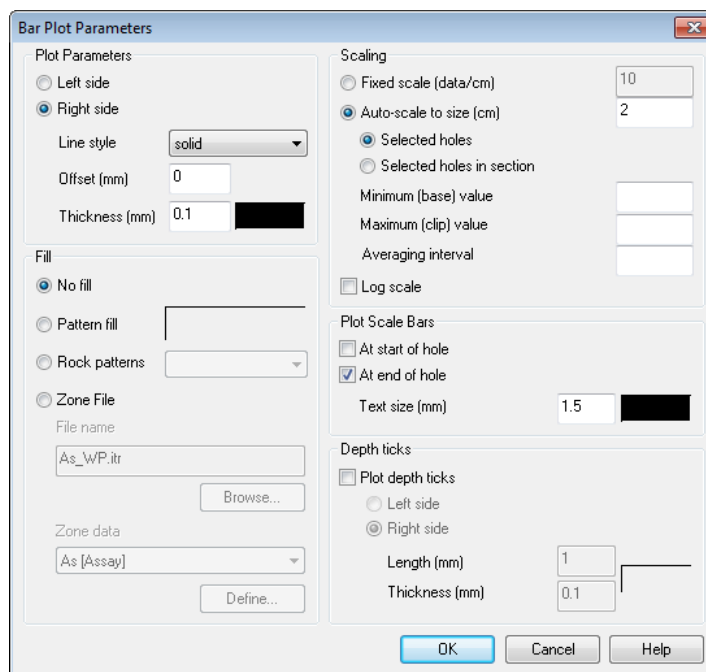
Recall that you earlier used the Project Preferences to set the default rock code file to be "Lithology Rock Code.csv". If you click on the Define button next to the Data field, you will see that this file has been loaded automatically.

To plot a bar plot of Au assay values:

- From the second **Data** list, select **Au (Assay)**.
- From the **Plot Type** list, select **Bar plot** and click **Define**.

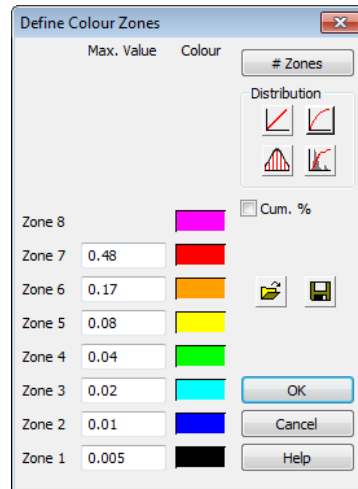
The Bar Plot Parameters dialog opens.

Figure 3.39 Bar Plot Parameters dialog



3. In the Plot Parameters section, select **Left side** and for **Thickness (mm)**, enter **0.05**.
4. In the Fill section, select **Zone File**.
The Au_WP.itr zone fill will be created automatically.
5. Click **Define**.
The Define Colour Zone dialog opens.

Figure 3.40 Define Colour Zone dialog



6. Select the **Cum. %** option.
This displays the data as cumulative percentages.
7. Enter the following values for each zone:

Zone	Value
7	99
6	97
5	95
4	90
3	83
2	75
1	50

8. For Zone 1, click the black colour box and change the colour to **grey**.
9. Click **OK**.
10. In the Plot Scale Bars section of the Bar Plot Parameters dialog, clear the **At the end of hole** option.
11. Click **OK**.

To plot gridded data:

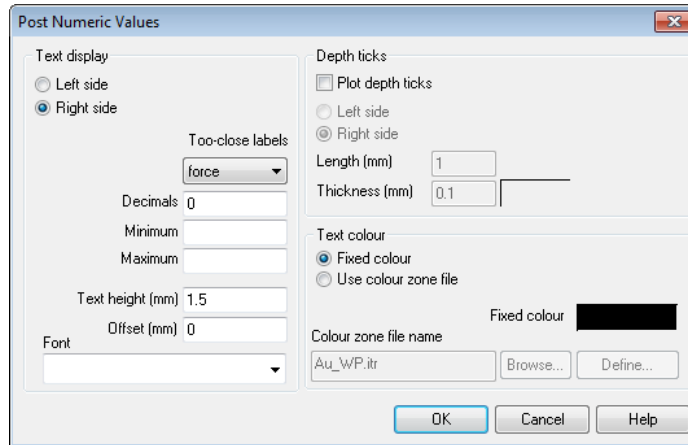
1. From the third **Data** list, select **Au (Assay)**.

- From the **Plot Type** list, select **Grid: Min. curve**.

To plot post values:

- From the fourth **Data** list, select **Au (composite)**.
These are the Au values from the composite database you created earlier.
- From the **Plot Type** list, select **Post values** and click **Define**.
The Post Numeric Values dialog opens.

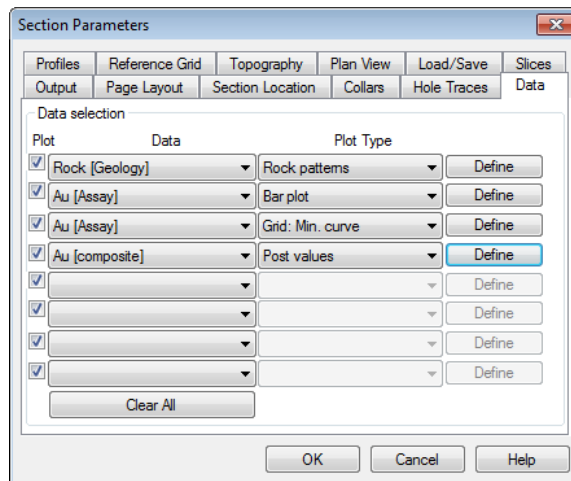
Figure 3.41 Post Numeric Values dialog



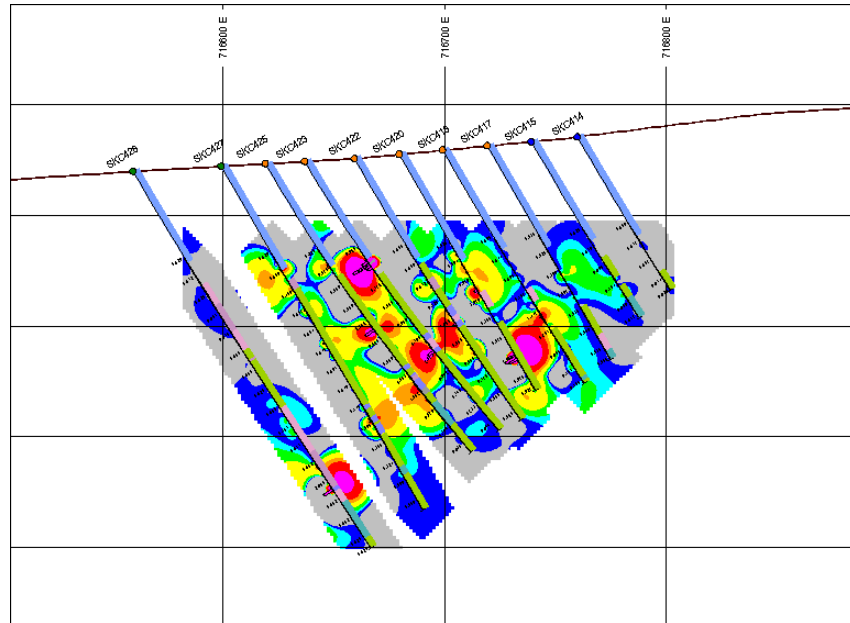
- In the Text display section, select **Left side**.
- For **Decimals**, enter **3**.
- For **Text height**, enter **1**.
- Click **OK**.

All four data plots are now setup in the Section Parameters dialog.

Figure 3.42 Section Parameters dialog with four data types to be plotted



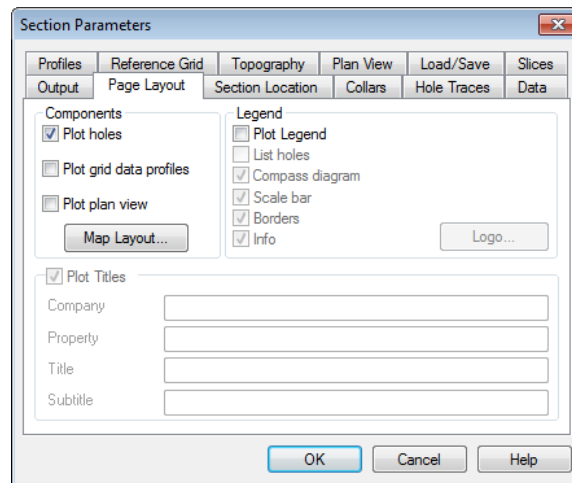
- To view the four data plots on the section map, click **OK**.
A new data frame, Section_6531180N_Map 5, is added to the Table Of Contents; the four data plots are added as separate layers that can be turned off and on.
- Remove** the **Section_6531180N_Map 4** data frame.

Figure 3.43 Section Map with data plotted (zoomed)

Next, you will add a legend and a plan view to your map.

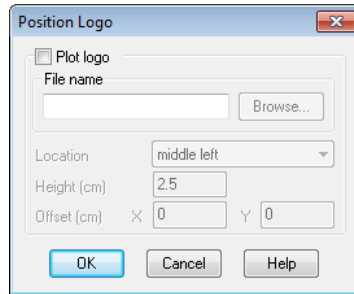
To add a legend:

1. Open the **Section Parameters** dialog and click the **Output** tab.
2. For **Frame/Layer Name Tag**, enter **Map 6**.
3. Click the **Page Layout** tab.

Figure 3.44 Section Parameters Page Layout tab

4. In the Legend section, select **Plot legend** and click the **Logo** button.
The Position Logo dialog opens.

Figure 3.45 Position Logo dialog



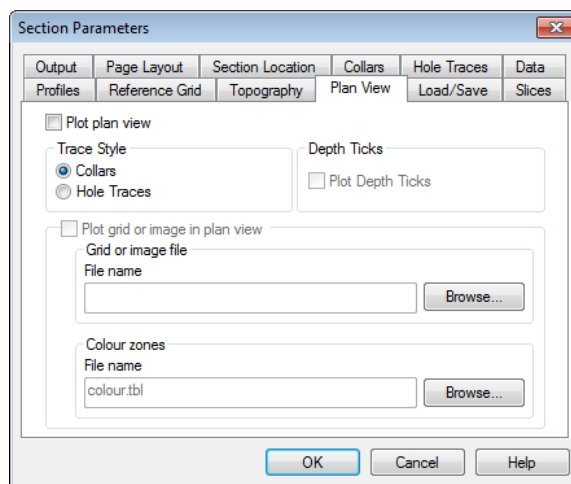
5. Select **Plot logo** and click **Browse**.
6. Change the Files of type to **JPEG Image (*.jpg)**.
7. From the Logos and Legends folder, select **geosoft-logo.jpg** and click **Open**.
8. For the logo Location, select **top centre** and click **OK**.

Next, you will set up to display the plan view on your section map to plot the hole traces and display an image.

To change the plan view parameters:

1. Click the **Plan View** tab.

Figure 3.46 Section Parameters Plan View tab

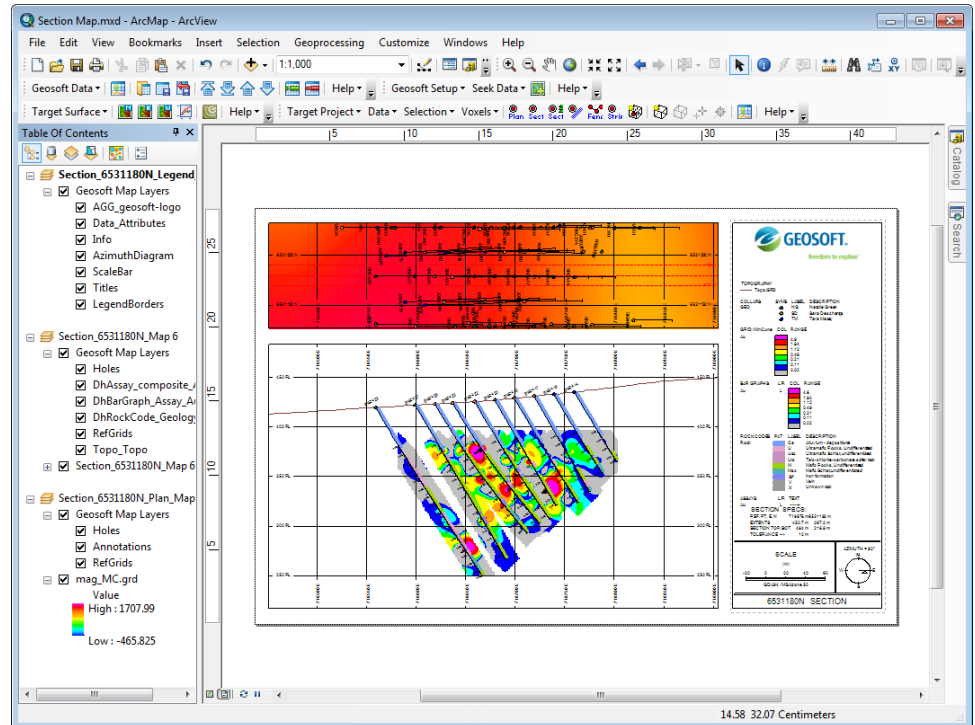


2. Select **Plot plan view**.
3. For **Trace Style**, select **Hole Traces**.
4. Select the **Plot grid or image in plan view** option.
5. For the **Grid or image file name**, click **Browse**.
6. From the Geophysics folder, select **mag_MC.grd** and click **Open**.
7. Click **OK**.

Three new data frames are added to the Table Of Contents; one for the section plot data, one for the legend and one for the plan map.

8. **Remove** the **Section_6531180N_Map 5** data frame.

Figure 3.47 Section Map



Shapefile Output

You will now create a section map in shapefile format.

To create a section map in shapefile format:

1. Turn off the display of the three existing Map 6 data frames.
2. Open the **Section Parameters** dialog.
 - All the parameters you used to create the Geosoft Map Layer are remembered.
3. Click the **Output** tab.
4. For **Frame/Layer Name Tag**, enter **Shapefiles**.
5. Clear the **Geosoft Map Layer** option and select the **Shape File Output** option.
6. For the Shape File Output **Filename Tag**, enter **Shapefile**.
7. Click the **Page Layout** tab and clear the **Plot Legend** option.
8. Click the **Reference Grid** tab and clear the **Plot grid** option.
9. Click the **Plan View** tab and clear the **Plot plan view** option.
10. Click **OK**.

Eight shapefiles are created and displayed on your map; two polygon layers represent rock codes and the Au bar plot, two line layers represent the hole traces and the topography, three point layers represent the collar locations, drillhole bottoms and the Au post values, and finally, a raster layer representing the gridded Au data.

Take some time now to use the ArcMap tools to symbolise these shapefiles and insert map elements. Resize the data frame as necessary.



Now would be a good time to save your map document.

In this lesson you:

- > Selected drillholes for the section map
- > Created a Geosoft Map Layer section map
- > Changed the display of the collar symbols and hole traces
- > Plotted topography
- > Plotted rock patterns, a bar plot, a grid and post values
- > Added a legend and plan view
- > Created a section map in shapefile format

Lesson 3.4 Creating a Stacked Section Map

In this lesson you will:

- Create a stacked section map
- Plot structural ticks
- Trace the sections on a plan map

Creating a Stacked Section Map

Using the drillhole stacked section option, you can plot multiple sections on a single map.

In this lesson, you will generate multiple sections on a single map using the parameters defined for the previous single section you made earlier.

Since ArcMap is designed to function as a single-map environment, you will create and save a new .mxd file with portrait orientation in which you will create the stacked section map. A portrait page layout is preferred for stacked section maps.

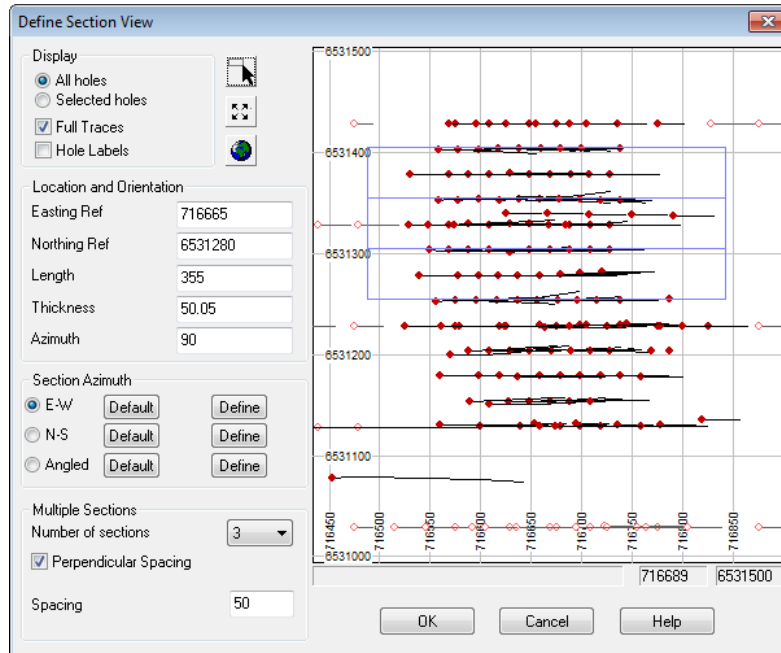
To change the page orientation:

1. From the ArcMap **File** menu, select **Page and Print Setup**.
The Page and Print Setup dialog opens.
2. For **Orientation**, select **Portrait**.
3. Click **OK**.

To create a stacked section map:

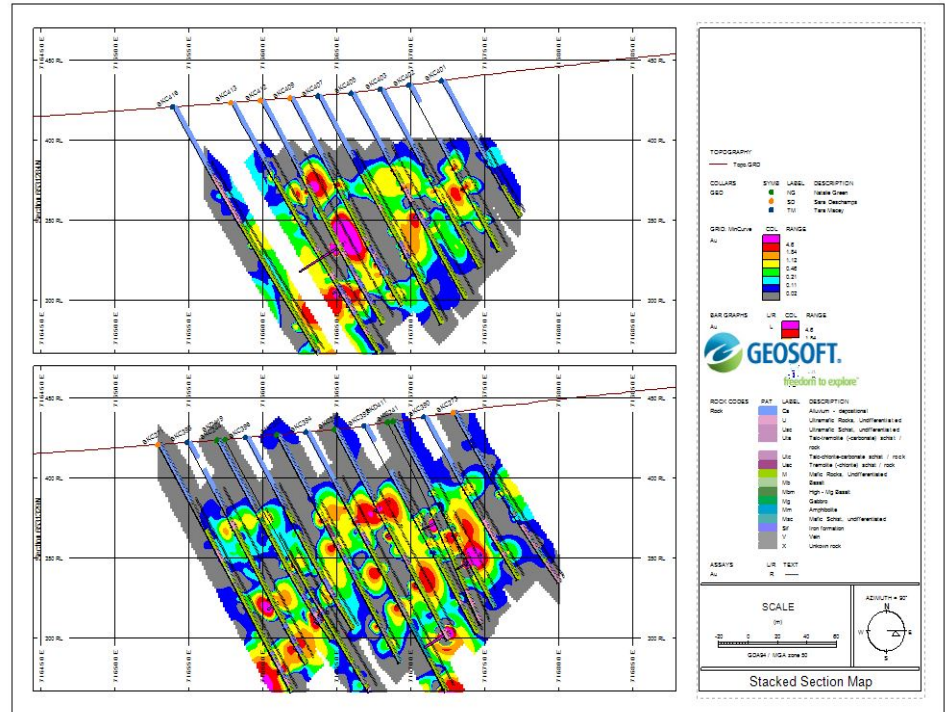
1. On the Target Drillhole toolbar, click the **Generate Target Stacked Section Plot** button.
2. Select **default/previous parameters** and click **OK**.
The Section Parameters dialog opens.
3. Click the **Load/Save** tab.
4. In the Load/Save Map Parameters section, click the **Load map parameters from a file** button.
5. From the Geosoft Training Data folder, select **TfA_Section.ins** and click **Open**.
6. Click the **Section Location** tab.
7. For **Number of sections**, enter **2**.
8. In the Section Azimuth section, click **Define** beside E-W.
9. Define the section location based on the figure below.

Figure 3.48 Selecting holes for stacked section map



10. Click **OK**.
11. For **Easting Ref**, enter **716662**.
12. For **Northing Ref**, enter **6531284**.
13. For **Thickness**, enter **15**.
14. For **Spacing**, enter **45**.
15. For **Map scale**, enter **1500**.
16. Click the **Page Layout** tab.
17. In the Plot Legend section, click the **Logo** button and deselect the **Plot logo** checkbox in the Position Logo dialog.
18. Click **OK**.
19. In the Plot Titles section, change the **Title** to **Stacked Section Map**.
20. Click **OK**.

Figure 3.49 Stacked section map



Structure Data

Structural data is point data collected at specific depths down a drillhole. This data often represents planar structures (bedding, cleavage, schistosity, faults, joints and veins) or linear structures (planar structure intersections, mineral stretching, and other forms of lineation). In general, each structure type consists of an alpha and beta angle of measurement to determine the geological orientation of structures down the drillhole. In the structure data you will shortly import, the "alpha" represents the dip of the structure relative to the core reference line and the "beta" represents the dip direction of the structure relative to the core reference line.

Structural Ticks

At each location in the hole, the structural tick is plotted to indicate the orientation of a structure that crosses the axis of the hole. For section plots, the intersection of the structure with the section plane is determined and a line is drawn, centred on the hole, to show the intersection.

Using the dip and dip direction channels of the structure table, you will add structural ticks to the section which will be coloured based on the structure type as defined in the structure codes file you examined earlier.

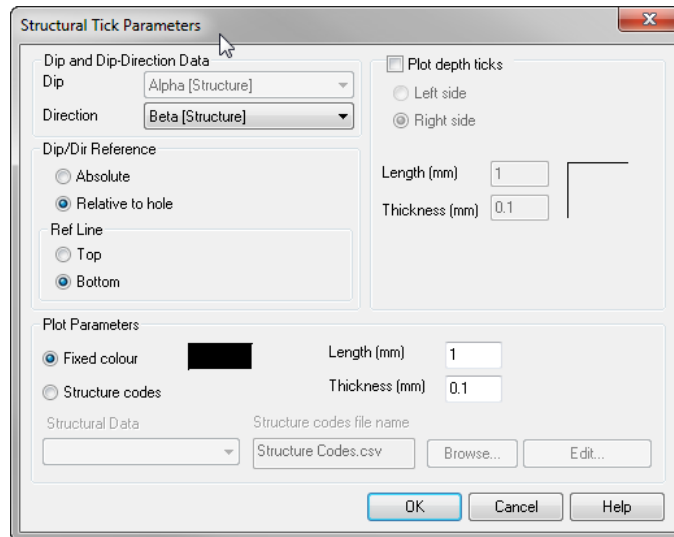
To plot structural ticks:

1. Open the Section Parameters dialog and click the **Data** tab.
2. Deselect the first four **Data** rows that are plotted.
3. From the fifth **Data** row, select **Alpha [Structure]**.

4. From the **Plot Type** list, select **Structural ticks**.
5. Click **Define**.

The Structural Tick Parameters dialog opens.

Figure 3.50 Structural Tick Parameters dialog



6. From the **Direction** list, select **Beta [Structure]**.



For "Absolute" Dip/Dir Reference, the dip is measured downward from the horizontal (e.g. 90 degrees is vertical) and the direction is the azimuth angle, measured clockwise from North.



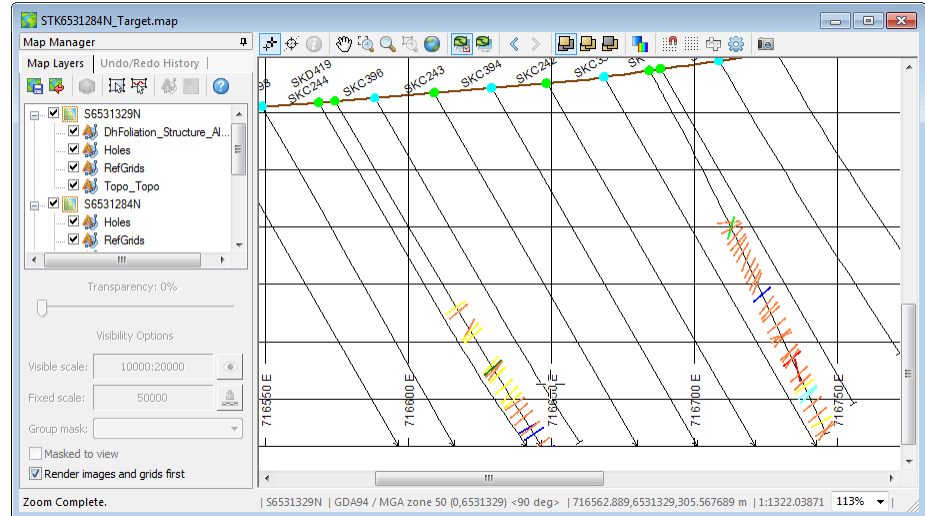
For "Relative to hole", dip is measured with respect to the hole axis, with zero degrees parallel to the hole axis and 90 degrees being perpendicular to the hole axis. Direction is measured clockwise looking down the hole axis, from either the top or bottom reference point.



In cases of vertical holes, Dip and Dip Direction is always interpreted using the "absolute" reference.

7. In the **Plot Parameters** section, select **Structure codes**.
8. For **Length (mm)**, enter **5**.
9. For **Thickness (mm)**, enter **0.4**.
10. From the **Structural Data** list, select **Structure_Type [Structure]**.
11. Ensure the Structure code name file is **Structure Codes.csv**.
12. Click **OK** and **OK** again.

The result should look similar to the figure below.

Figure 3.51 Stacked section map with structure ticks plotted (zoomed)

You may want to change the transparency of the other layers.

To trace the sections on a plan map:

1. Open the .mxd file containing your mineralised plan map.
2. Hide all data frames except for those pertaining to the plan map.
3. On the **Target Drillhole** toolbar, click the **Trace Target section into current data frame** button.

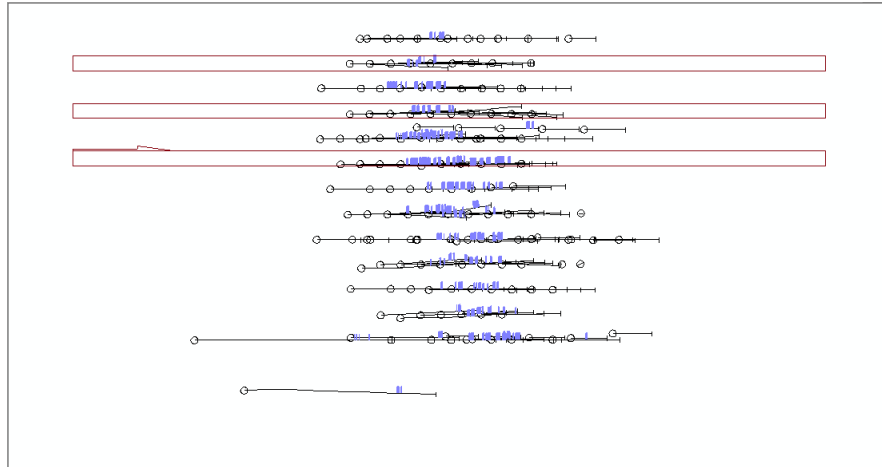
The Draw a section dialog opens.

Figure 3.52 Draw a section dialog

4. For **Section job name**, click the **Browse** button and select **TfA_stacked.ins**.
5. For **Output shape file name**, enter **Stacked_Sections**.
6. For **Plot section**, select **perimeter(s)**.

The outside boundary of the stacked sections are plotted on the plan map.

Figure 3.53 Plan Map with stacked section perimeter lines



Now would be a good time to save your map document.

In this lesson you:

- > Created a stacked section map
- > Plotted structural ticks
- > Traced the sections on a plan map

Lesson 3.5 Creating a Fence Diagram

In this lesson you will:


- > Select holes
- > Create a fence diagram
- > Plot geology polygons
- > Add a legend and plan view

Fence Diagrams

A limiting factor in using section maps is the fact they can only be oriented in a straight line. . Fence diagrams provide you with the means to plot a cross-section from any combination of drillholes, regardless of how they are located relative to one another. Fence diagrams are particularly useful in regional exploration, where you might want to interpret lithology between widely-separated and offset drillholes in your project area.

Since ArcMap is designed to function as a single-map environment, you will begin by creating a new map file in which you will create the fence diagram.

To create a new .MXD:

1. On the Standard toolbar, click the **New Map File** button .
The New Document dialog opens.
2. Click **OK**.
A new, untitled map with the Target Project loaded, the drillholes selected and the correct page setup is created.
3. Save the map as **Fence Diagram.mxd**.

To select holes and create a fence diagram:


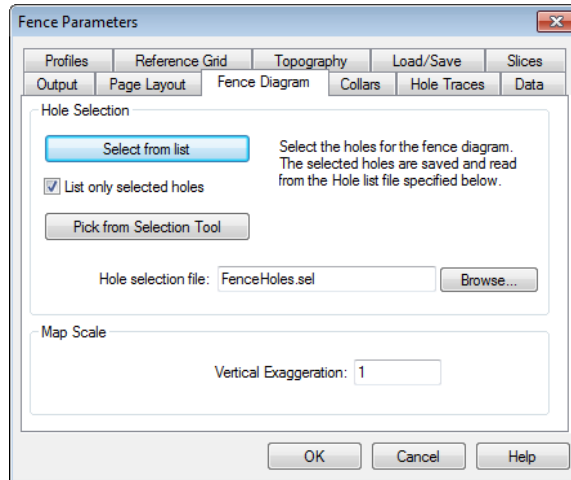
1. On the Target Drillhole toolbar, click the **Generate Target Fence Diagram Plot** button .
The Fence Parameters dialog opens.

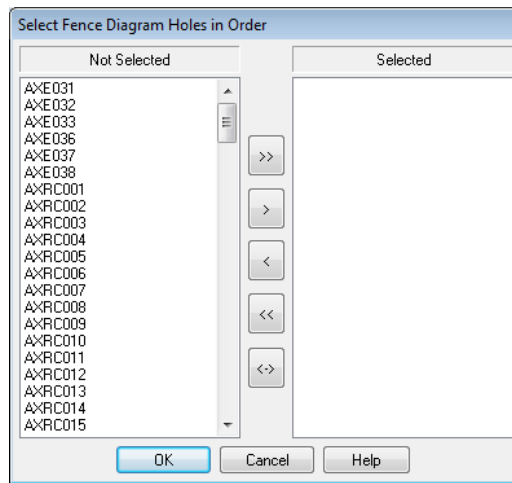
Figure 3.54 Fence Parameters dialog




2. Click **Select from list**.

The Select Fence Diagram Holes in Order dialog opens.

Figure 3.55 Select Fence Diagram Holes in Order dialog

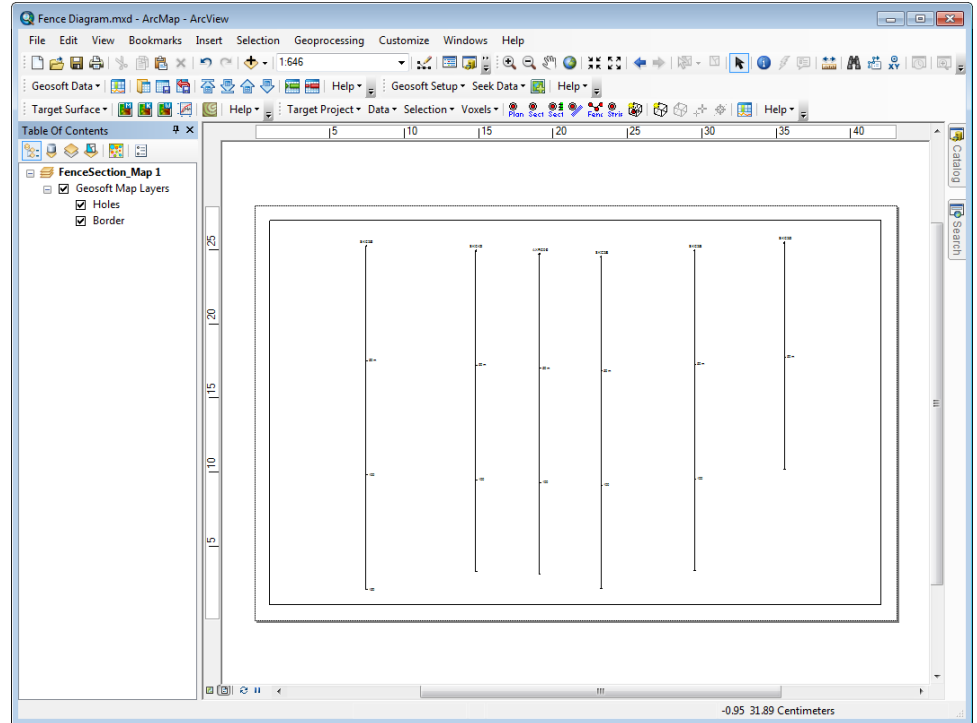


3. From the **Not Selected** list, select **SKC255** and click the  button.
This hole is now listed in the Selected list.
4. Repeat step 3 to select the following holes: **SKC422**, **AXRC015**, **SKC256**, **SKC253** and **SKC252**.
5. Click **OK**.
6. Click the **Output** tab.
7. For **Frame/Layer Name Tag**, enter **Map 1** and click **OK**.

The FenceSection data frame is added to the Table Of Contents; it contains the Holes and Borders layers which are parts of the Geosoft Map Layer output.

8. **Remove** the **Layers** data frame.

Figure 3.56 Default Fence Diagram

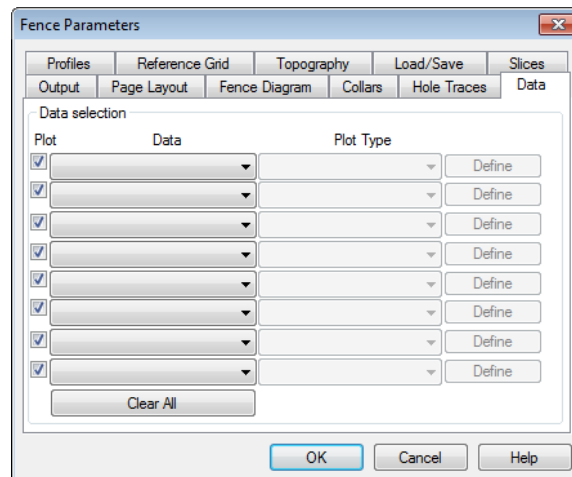


Next, you will reopen the Fence Parameters dialog box and interpolate the geology between the drillholes.

To plot geology polygons:

1. Open the **Fence Parameters** dialog and click the **Output** tab.
2. For **Frame/Layer Name Tag**, enter **Map 2**.
3. Click the **Data** tab.

Figure 3.57 Fence Parameters Data tab

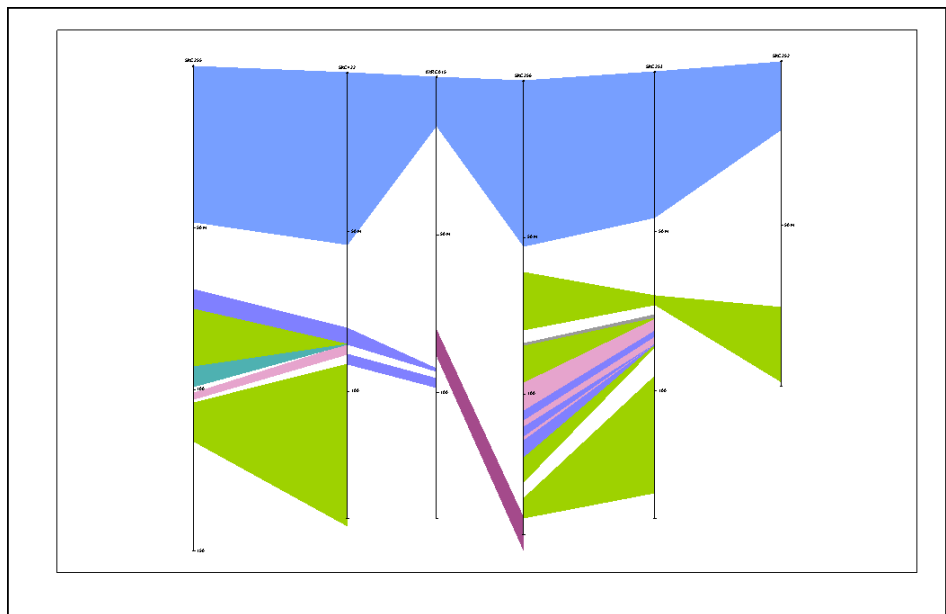


4. From the first **Data** list, select **Rock (Geology)**.
5. From the **Plot Type** list, select **Geology polygons** and click **OK**.

A new data frame, FenceSection_Map 2, is added to the Table Of Contents; the geology polygons are now plotted on the fence diagram.

6. Remove the FenceSection_Map 1 data frame.

Figure 3.58 Fence Diagram with geology polygons

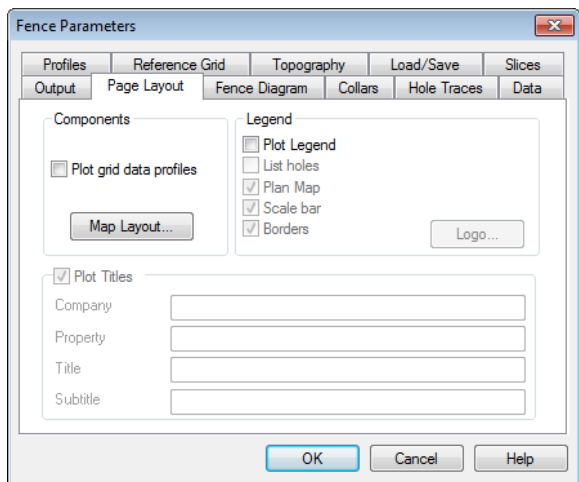


Next, you will add a legend and plan map to the fence map.

To add a legend and plan map:

1. Open the Fence Parameters dialog and click the Output tab.
2. For Frame/Layer Name Tag, enter Map 3.
3. Click the Page Layout tab.

Figure 3.59 Fence Parameters Page Layout tab

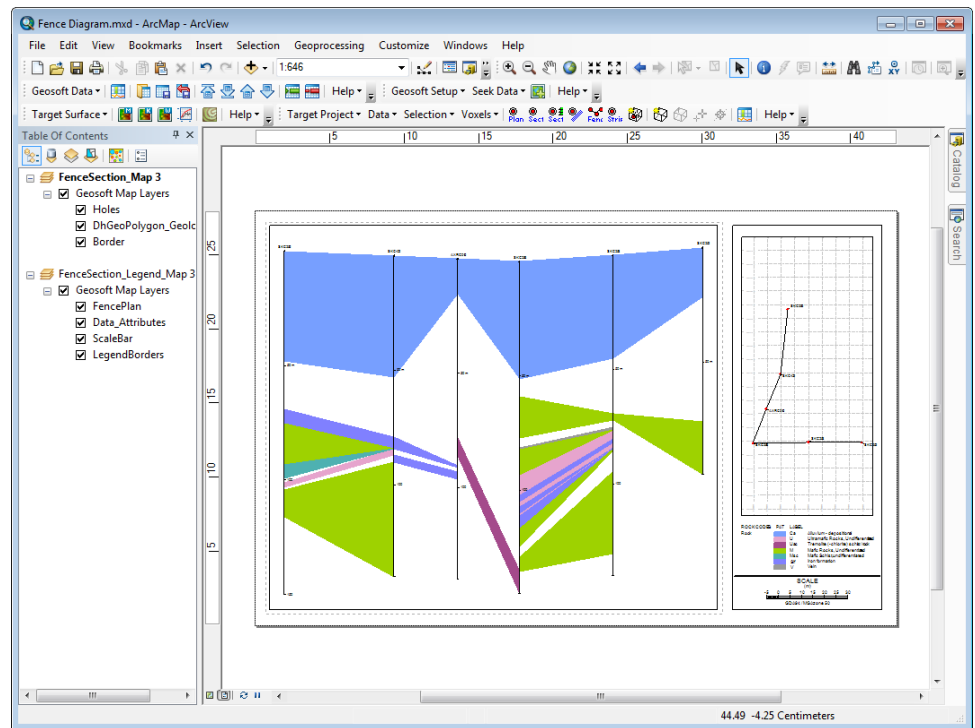


4. Select Plot legend and click OK.

Two new data frames are added to the Table Of Contents; one for the fence diagram and one for the legend.

5. Remove the FenceSection_Map 2 data frame.

Figure 3.60 Fence Diagram with legend



Now would be a good time to save your map document.

In this lesson you:

- Selected holes
- Created a fence diagram
- Plotted geology polygons
- Added a legend and plan view

Lesson 3.6 Creating a Strip Log

In this lesson you will:


- > Create a strip log
- > Plot data on the strip log
- > Add a legend
- > Create a fracture density plot
- > Create a stereonet
- > Create a tadpole plot
- > Create strip logs for a range of holes

Strip Logs

Strip logs are useful for visualising data from one drillhole, side by side.. The drillhole trace is plotted vertically on the strip log and the associated data are plotted at regular or customised intervals. You can plot up to 32 different drillhole datasets on a strip log.

Since ArcMap is designed to function as a single-map environment, you will begin by creating a new map file in which you will create the strip log.

To create a new .MXD:

1. On the Standard toolbar, click the **New Map File** button .
The New Document dialog opens.
2. Click **OK**.
A new, untitled map with the Target Project loaded, the drillholes selected and the correct page setup is created.
3. Save the map as **Strip Log.mxd**.

Geosoft Map Layer Output

You will now create a strip log for a single drillhole, in Geosoft Map Layer format output. Following that, you will adjust various parameters to change the display of the strip log.

To create a strip log:


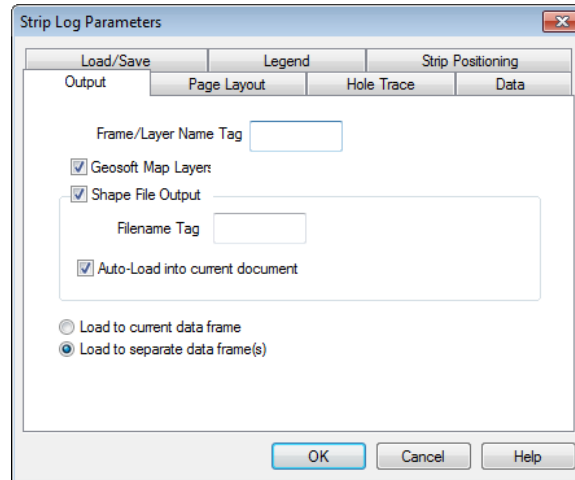
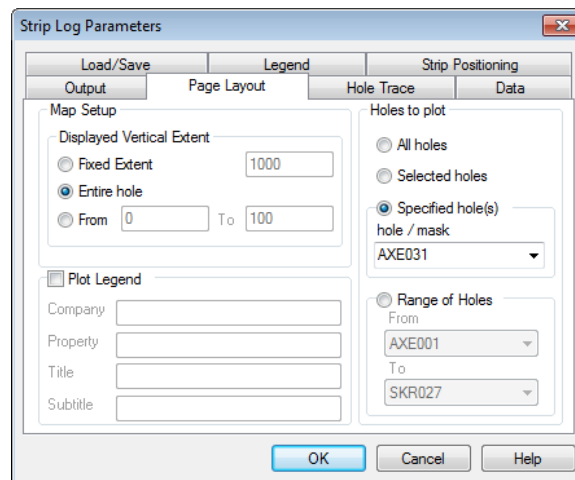
1. On the Target Drillhole toolbar, click the **Generate Target Striplog Plot** button .
The Strip Log Parameters dialog opens.

Figure 3.61 Strip Log Parameters dialog

2. For **Frame/Layer Name Tag**, enter **Map 1**.
3. Clear the **Shape File Output** option.
4. Click the **Page Layout** tab.

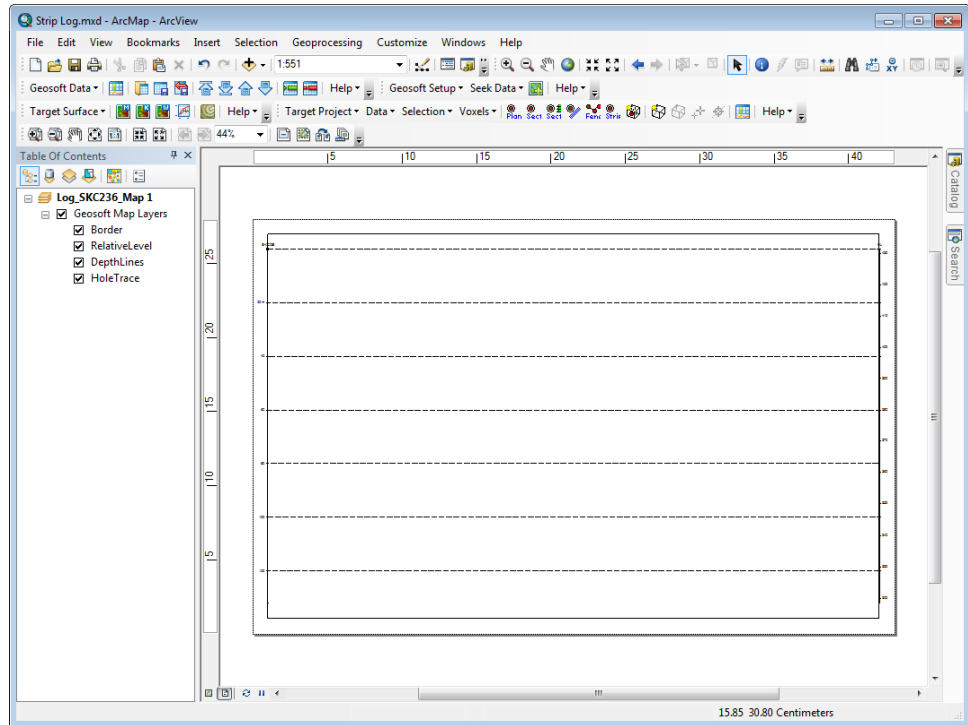
Figure 3.62 Strip Log Parameters Page Layout tab

5. In the Holes to plot section, select **Specified hole(s)**.
6. From the **hole/mask** list, select **SKC236**.
7. Click **OK**.

The Log_SKC236_Map 1 data frame is added to the Table Of Contents; it contains the Border, RelativeLevel, DepthLines and HoleTrace Geosoft Map Layers.

8. **Remove** the **Layers** data frame.

Figure 3.63 Strip Log map

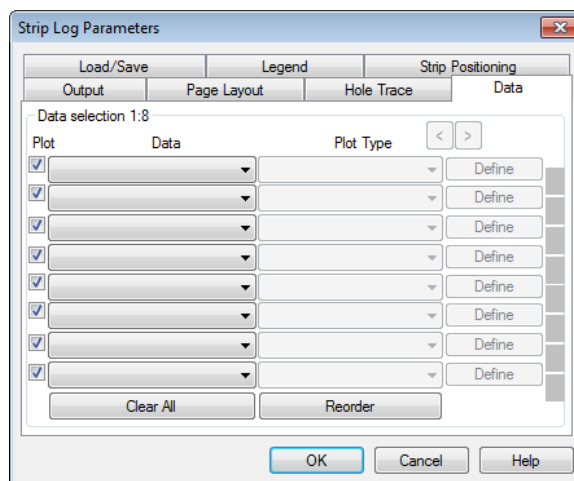


Next, you will plot four different types of data on your strip log. First you will plot downhole lithology as rock patterns, then Au assay data as a bar plot, post values and a profile.

To plot rock patterns on the strip log:

1. Open the **Strip Log Parameters** dialog and click the **Output** tab.
2. For **Frame/Layer Name Tag**, enter **Map 2**.
3. Click the **Data** tab.

Figure 3.64 Strip Log Parameters Data tab



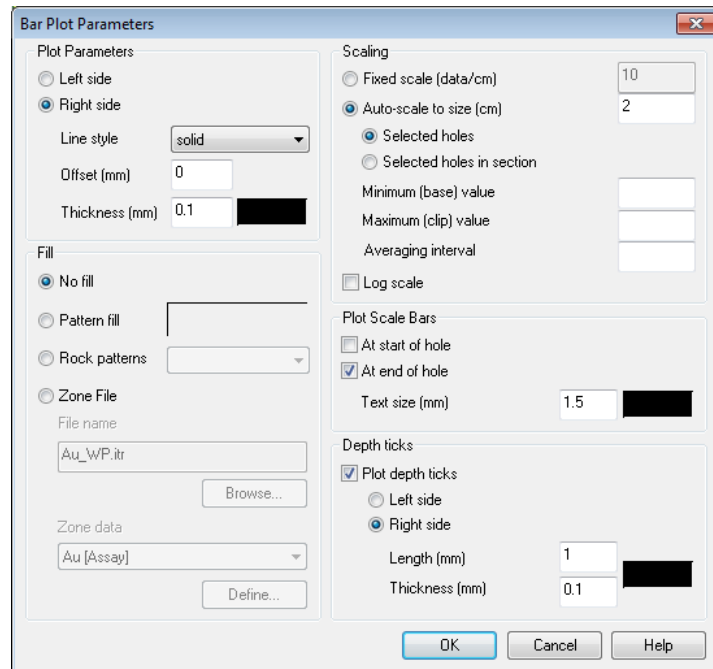
4. From the first **Data** list, select **Rock (Geology)**.
5. From the **Plot Type** list, select **Rock patterns**.

To plot a bar plot of Au assay values:

1. From the second **Data** list, select **Au (Assay)**.
2. From the **Plot Type** list, select **Bar plot** and click **Define**.

The Bar Plot Parameters dialog opens.

Figure 3.65 Bar Plot Parameters dialog



3. In the Plot Parameters section, for **Thickness (mm)**, enter **0.05**.
4. In the Scaling section, enter an **Auto-scale to size (cm)** value of **10**.
5. In the Fill section, select **Zone File**.

The Au_WP.itr zone fill will be used automatically.

6. In the Depth ticks section, clear the **Plot depth ticks** option.
7. Click **OK**.

To plot post values:

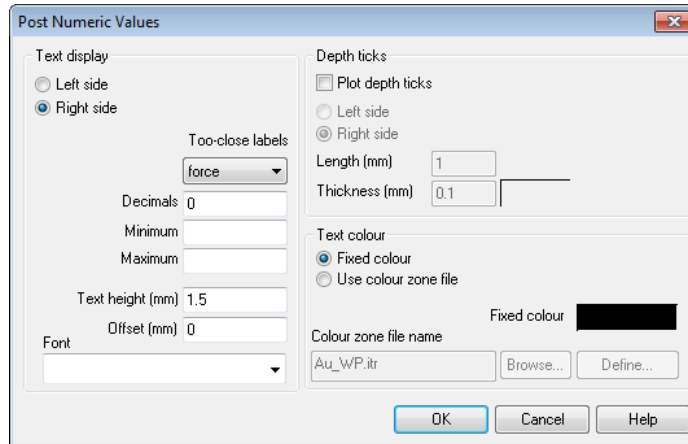
1. From the third **Data** list, select **Au (composite)**.

These are the Au values from the composite database you created earlier.

2. From the **Plot Type** list, select **Post values** and click **Define**.

The Post Numeric Values dialog opens.

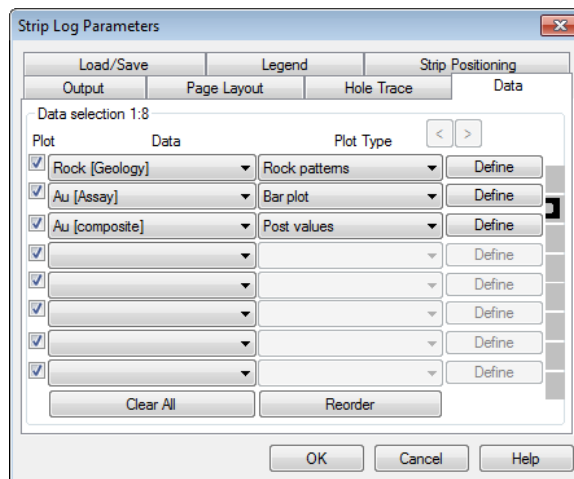
Figure 3.66 Post Numeric Values dialog



3. In the Text display section, select **Left side**.
4. For **Decimals**, enter **3**.
5. For **Text height**, enter **2**.
6. For **Text colour**, select **Use colour zone file**.
7. Click **OK**.
8. In the Strip Log Parameters dialog, click the **Link** button straddling the bar plot and post values data selections.

This will link the hole traces so that these two data plots will be plotted down the same vertical hole. The dialog should appear as follows:

Figure 3.67 Strip Log Parameters Data tab with link

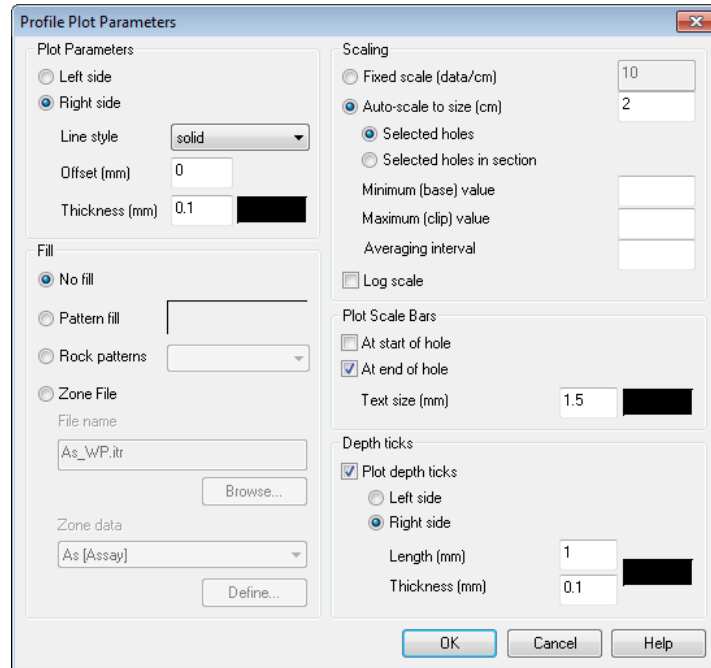


To plot a profile:

1. From the fourth **Data** list, select **As (Assay)**.
2. From the **Plot Type** list, select **Profile** and click **Define**.

The Profile Plot Parameters dialog opens.

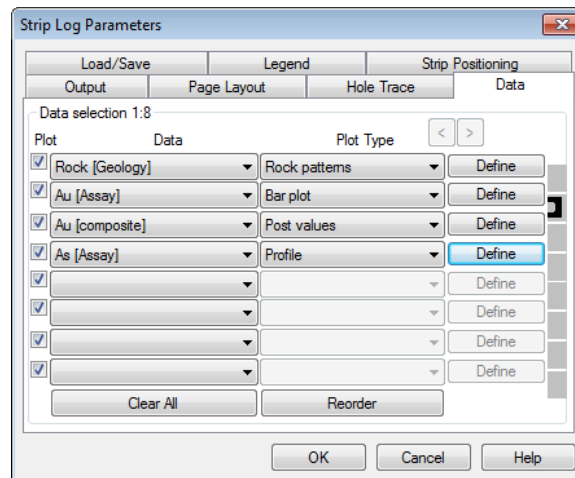
Figure 3.68 Profile Plot Parameters dialog



3. In the Plot Parameters section, for **Thickness (mm)**, enter **0.05**.
4. Click the black colour box and change the profile line colour to **green**.
5. In the Scaling section, enter an **Auto-scale to size (cm)** value of **20**.
6. In the Depth ticks section, clear the **Plot depth ticks** option.
7. Click **OK**.

All four data plots are now setup in the Section Parameters dialog.

Figure 3.69 Strip Log Parameters dialog with four data types to be plotted

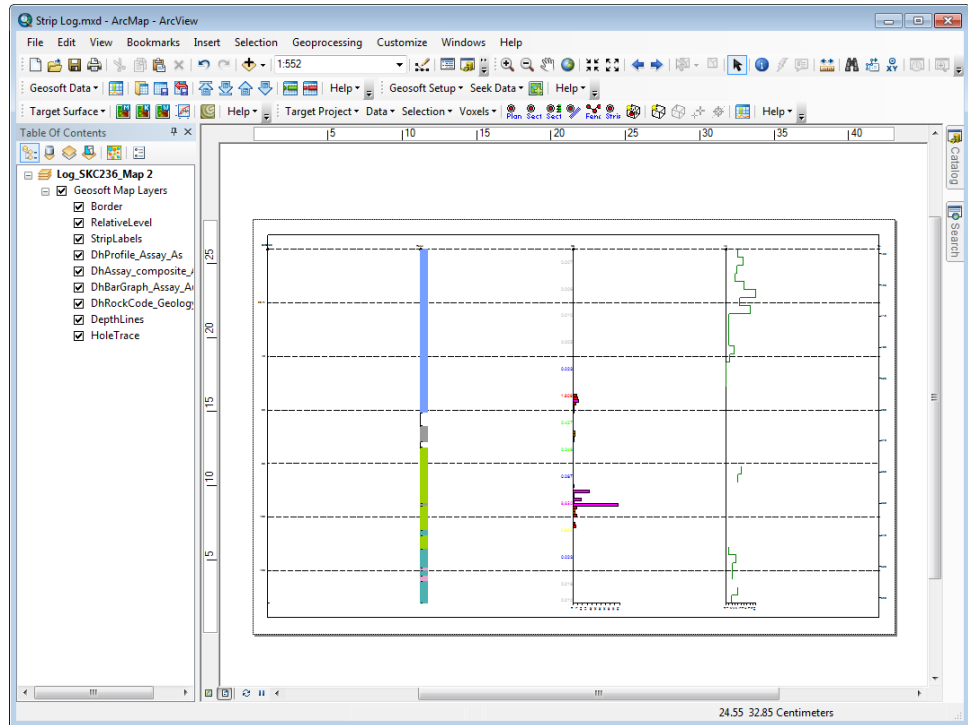


8. To view the four data plots on the strip log, click **OK**.

A new data frame, Log_SKC236_Map 2 is added to the Table Of Contents; the data plots are added to your strip log as Geosoft Map Layers.

9. **Remove the Log_SKC236_Map 1 data frame.**

Figure 3.70 Strip Log map with data plotted

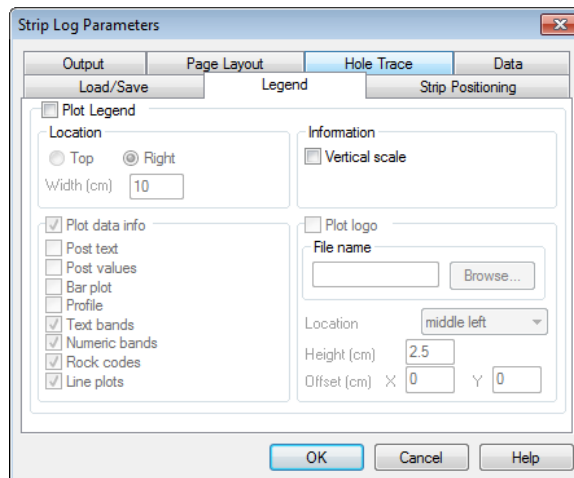


Next, you will add a legend to the strip log.

To add a legend:

1. Open the **Strip Log Parameters** dialog and click the **Output** tab.
2. For **Frame/Layer Name Tag**, enter **Map 3**.
3. Click the **Legend** tab.

Figure 3.71 Strip Log Parameters Legend tab

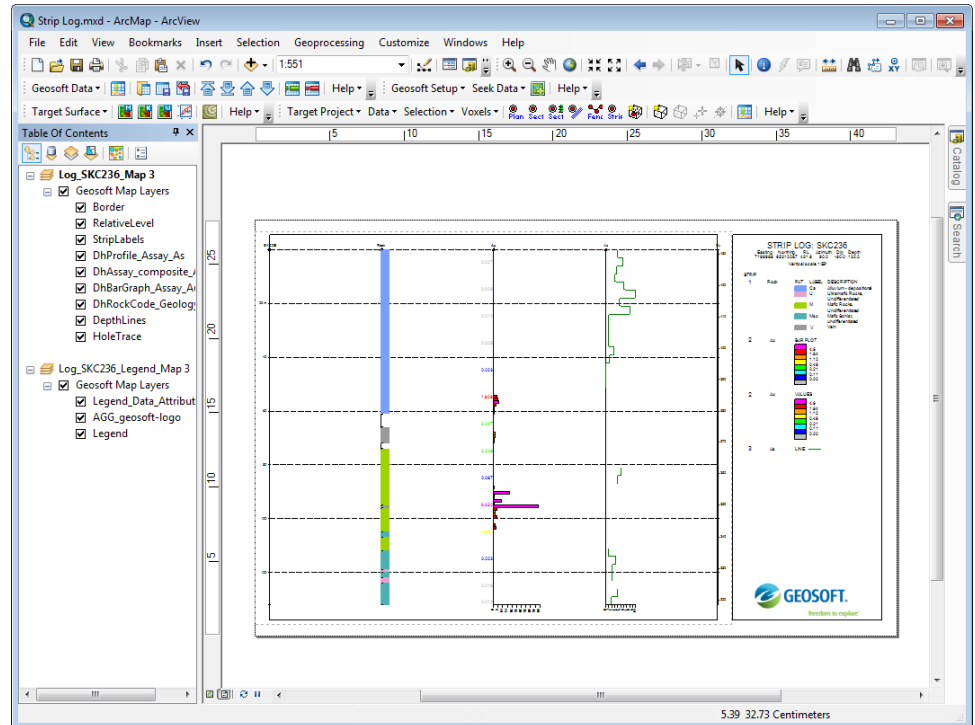


4. Select **Plot Legend**.
5. In the Information section, select **Vertical Scale**.
6. In the Plot data info section, select **Post text**, **Post values**, **Bar plot** and **Profile**.

Leave the remaining options selected as well.

7. Select **Plot logo** and click **Browse**.
8. Change the Files of type to **JPEG Image (*.jpg)**, select **geosoft-logo.jpg** from the Logos and Legends folder and click **Open**.
9. For the logo Location, select **bottom centre** and click **OK**.
Two new data frames are added to the Table Of Contents; one for the strip log and one for the legend.
10. **Remove the Log_SKC236_Map 2** data frame.

Figure 3.72 Strip Log map with legend information



As with plan maps and section maps, strip logs can also be created in shapefile format.




Now would be a good time to save your map document.

Fracture Density Plots

The fracture density plot is a form of "rose" plot, essentially a radial histogram with the height (radius) of the bins showing the relative frequency of the points as a function of dip-direction. The number of points in each interval is summed, and the radius of the bin with the highest number of items is scaled to the entered radius. Note that it is the radius of the petal which is proportional to the number of items, not the area.

You will now create a Fracture Density plot from a single drillhole within the Target_Structure.gdb database.

To create a new .MXD:

1. On the Standard toolbar, click the **New Map File** button .
The New Document dialog box opens.
2. Click **OK**.
A new, untitled map with the Target Project loaded, the drillholes selected and the correct page setup is created.
3. Save the map as **Rose Plot.mxd**.

To create a fracture density plot:


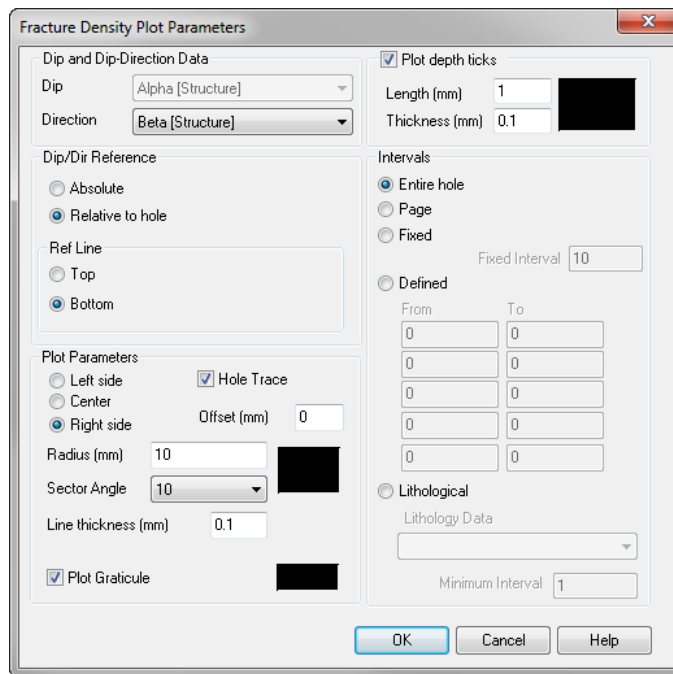
1. On the Target Drillhole toolbar, click the **Generate Target Striplog Plot** button .
The Strip Log Parameters dialog box opens.
2. Click the **Data** tab.
3. Deselect the second, third, and fourth data rows.
4. From the fifth **Data** list, select **Alpha [Structure]**.
5. For **Plot Type**, select **Fracture density** and click **Define**.
The Fracture Density Plot Parameters dialog box opens.

Figure 3.73 Fracture Density Plot Parameters dialog box



6. In the Plot Parameters section, enter a **Radius (mm)** of 15.
7. Change the black box next to Radius and Sector Angle to **red**.
8. Clear the **Plot depth ticks** option and click **OK**.
9. **Remove** the **Layers** data frame.


Stereonet

Stereonets are used in structural geology to depict the angular relationships between geologic structures; they are a way of representing 3-dimensional directions on a 2-dimensional surface. The colour of the dots shows the structure type being measured; the dip direction is shown in its orientation from North or the top of the stereonet. Steeply dipping planes have poles that are closer to the edge of the stereonet; shallow dipping planes have poles that are closer to the centre of the stereonet.

Two stereographic projections are commonly used; equal-area projection (Schmidt net) and equal-angle projection (Wulff net). The net is a projection from the centre point of a sphere onto the surface of the sphere thus allowing you to visualise the dip and dip direction of measured features.

You will now create a Stereonet for one of the drillholes on your project.


To create a new .MXD:

1. On the Standard toolbar, click the **New Map File** button .

The New Document dialog box opens.
2. Click **OK**.

A new, untitled map with the Target Project loaded, the drillholes selected and the correct page setup is created.
3. Save the map as **Stereonet.mxd**.

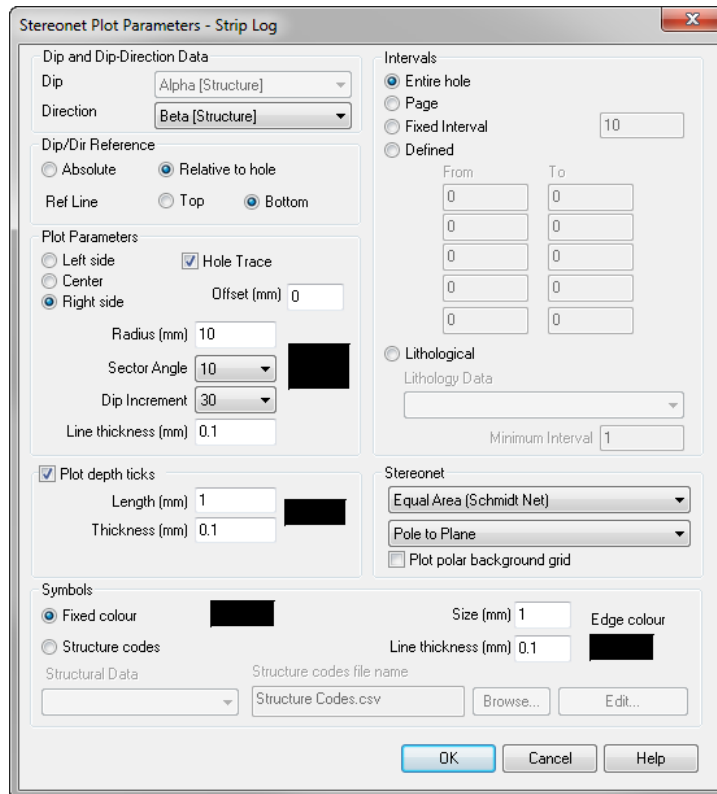
To create a stereonet:

1. On the Target Drillhole toolbar, click the **Generate Target Striplog Plot** button .

The Strip Log Parameters dialog box opens.
2. From the sixth **Data** list, select **Alpha [Structure]**.
3. From the **Plot Type** list, select **Stereonet** and click **Define**.

The Stereonet Plot Parameters - Strip Log dialog box opens.

Figure 3.74 Stereonet Plot Parameters - Strip Log dialog box



4. In the Plot Parameters section, enter a **Radius (mm)** of 15.
5. Change the colour of the black box to **grey**.
6. Clear the **Plot depth ticks** option.
7. In the Symbols section, select **Structure codes**.
8. From the **Structural Data** list, select **Structure_Type [Structure]**.
9. In the Intervals section, select **Defined** and enter the following values:

From	To
0	60
60	90
90	110

10. **Remove** the **Layers** data frame.

Tadpole Plot

A tadpole plot is a graph of dipmeter results. The dip of the measured structure is plotted as a dot on a depth/dip graph and the dot tail indicates the dip direction (e.g., the tail points vertically upwards from the dot for a northerly dip direction).

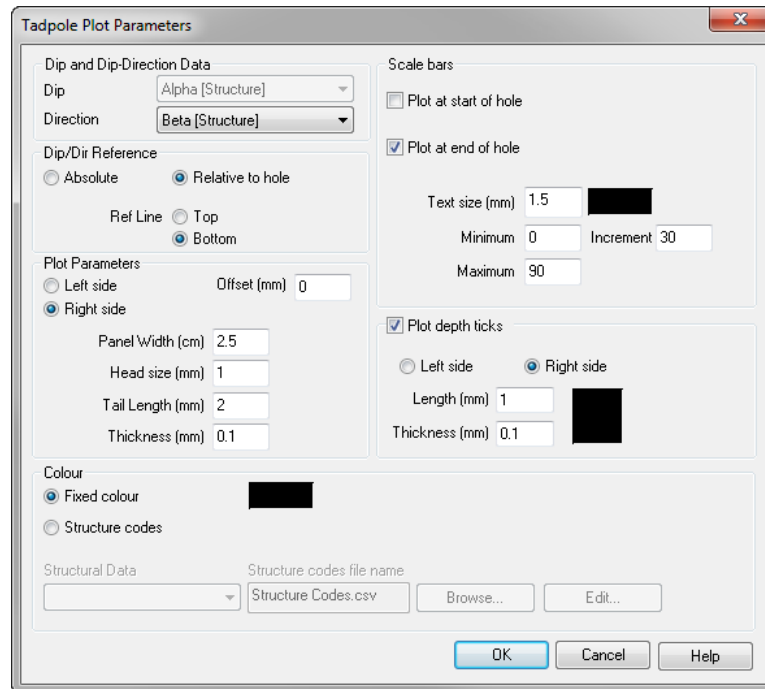
You will now create a Tadpole plot from a drillhole in your project.

To create a tadpole plot:

1. From the seventh **Data** list, select **Alpha [Structure]**.

- For the **Plot Type**, select **Tadpole plot** and click **Define**.
The Tadpole Plot Parameters dialog opens.

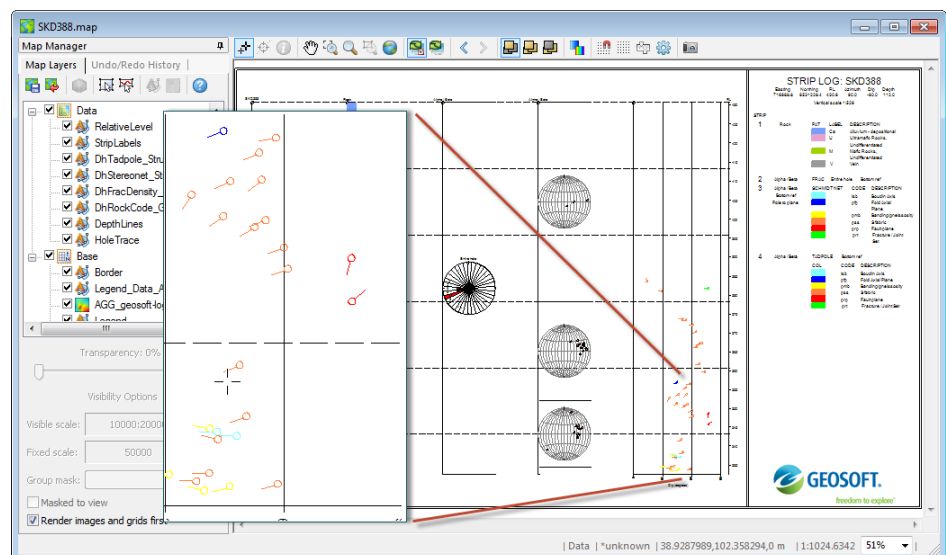
Figure 3.75 Tadpole Plot Parameters dialog



- In the Plot Parameters section, enter a **Panel Width (cm)** of 5.
- In the Colour section, select **Structure codes**.
- From the **Structural Data** list, select **Structure_Type [Structure]**.
- Clear the **Plot depth ticks** option and click **OK**.
- In the Strip Log Parameters dialog, click **OK**.

The tadpole plot is displayed on the strip log map.

Figure 3.76 Tadpole Plot



As with the stereonet, the colour of each tadpole symbol represents a structure type. Its location on the graph along the X axis describes the dip of the structural feature that each tadpole symbol represents. The tails show the dip direction. If a tail is pointing straight up, the dip direction would be due North.



*You can also create strip logs for a range of holes, where you select **Range of Holes** and specify a range in the **Holes to plot** section.*

In this lesson you:

- > Created a strip log
- > Plotted data on the strip log
- > Added a legend
- > Created a fracture density plot
- > Created a stereonet
- > Created a tadpole plot
- > Created strip logs for a range of holes

Lesson 3.7 Creating a 3D Map

In this lesson you will:

- Create a 3D map
- Use the Geosoft 3D Viewer
- Plot rock patterns and a bar plot
- Plot gridded data
- Export to a 3D PDF


3D Drillhole Maps

Geosoft's 3D Viewer enables you to display drillhole, surface and other data types in an interactive, three-dimensional environment. Drillholes are displayed in their "true" three-dimensional location and can have up to two different data types plotted along their trace.

In this lesson, you will create a simple 3D View and explore the navigation tools within.

Since ArcMap is designed to function as a single-map environment, you will begin by creating a new map file in which you will create the 3D map.

To create a new .MXD:

1. On the Standard toolbar, click the **New Map File** button .
The New Document dialog opens.
2. Click **OK**.
A new, untitled map with the Target Project loaded, the drillholes selected and the correct page setup is created.
3. Save the map as **3D Map.mxd**.

To create a 3D map:


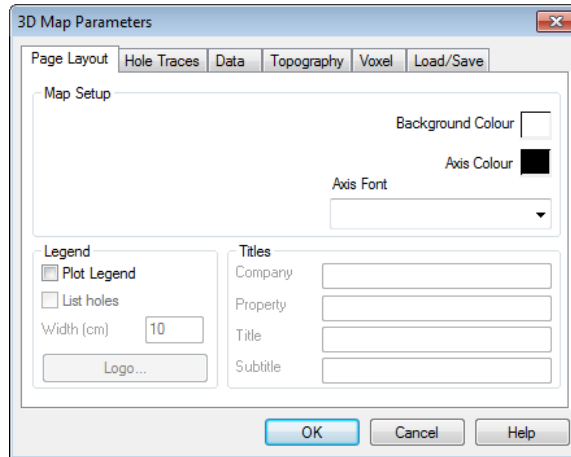
1. On the Target Drillhole toolbar, click the **Generate 3D Drillhole Plot** button .
The 3D Map Parameters dialog opens.

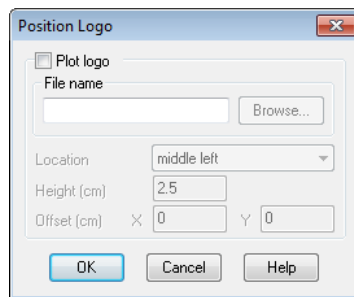
Figure 3.77 3D Map Parameters dialog



2. Open the Page Layout tab, select the **Plot Legend** option and click the **Logo** button.

The Position Logo dialog opens.

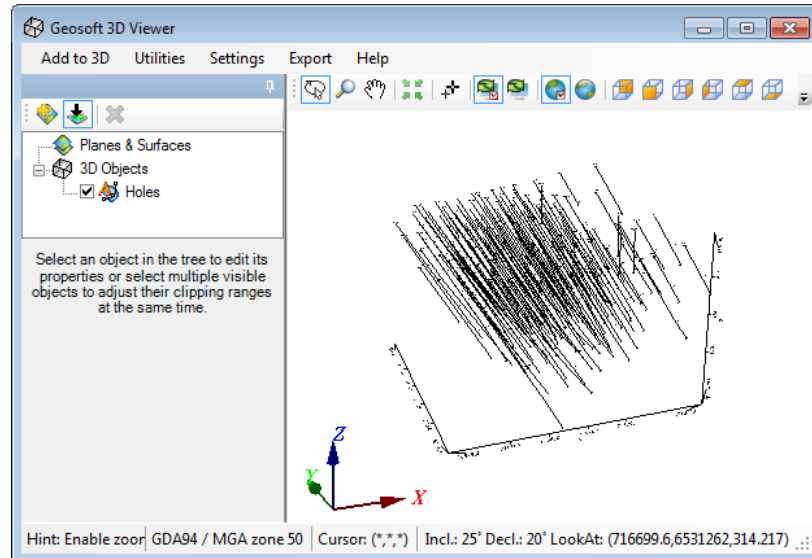
Figure 3.78 Position Logo dialog



3. Select **Plot logo** and click **Browse**.
4. Change the Files of type to **JPEG Image (*.jpg)**, select **geosoft-logo.jpg** from the Logos and Legends folder and click **Open**.
5. For the logo Location, select **top centre** and click **OK**.
6. Enter the following titles:
 - > Company: **Geosoft**
 - > Property: **Mt Palmer**
 - > Title: **3D Map**
 - > Subtitle: **Target for ArcGIS Training**
7. Click **OK**.

The Geosoft 3D Viewer opens.

Figure 3.79 Geosoft 3DViewer



The following interactive tools are provided to navigate the display of your data in 3D:

Table 3.3 3D Manager Tools






















Icon	Tool	Description
	Add Plane	Click this button to add any number of planes to the 3D view.
	Section View	Click this button in order to create a section view, where you must first position the Reference Plane where you want to create the new drillhole, then draw the proposed drillhole on the plane.

Table 3.4 3D Navigation Tools

Icon	Tool	Description
	Shadow Cursor On	To toggle the visibility of the shadow cursor On/Off
	Pan	To move the entire displayed view
	Zoom In/Out	To zoom in and out of the displayed view
	Rotate	To rotate the displayed view 360 degrees in all directions
	Centre to Window	To centre the current 3D view in the 3D Viewer display window
	Automatic Zoom On/Off	To automatically adjust the 3D view as objects are selected/deselected in the tree control

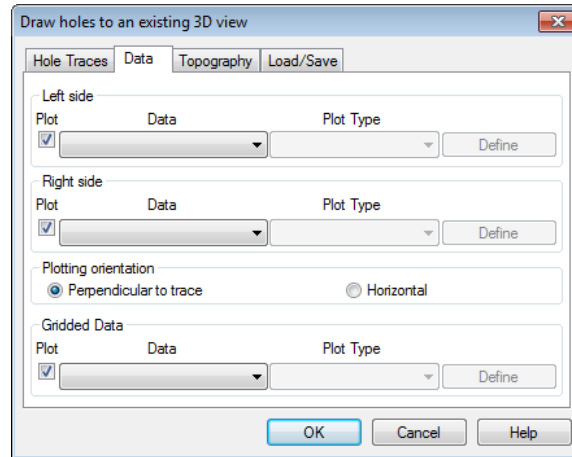
Icon	Tool	Description
	Zoom to Full Extents	To centre the 3D view based on the currently selected objects in the tree control
	Automatic Redraw On	To refresh/redraw the 3D view automatically after changes are made
	Redraw	To refresh/redraw the 3D view in the 3D Viewer window
	North View	To set the observation point for the current view to directly north (looking south)
	South View	To set the observation point for the current view to directly south (looking north)
	East View	To set the observation point of the current view to directly east (looking west)
	West View	To set the observation point of the current view to directly west (looking east)
	Top View	To set the observation point of the current view to directly above (looking down)
	Bottom View	To set the observation point of the current view to directly below (looking up)
	User Defined View	To specify the inclination and azimuth of the view
	Perspective View/ Orthographic View	To select to display as either a perspective view or an orthographic view.
	Create Snapshot	To create a snapshot of your current view.

 *Right-click in the 3D Viewer for quick access to some of these navigation tools. You can also zoom in and out using the track wheel on your mouse.*

Take some time now to experiment with the 3D navigation tools.

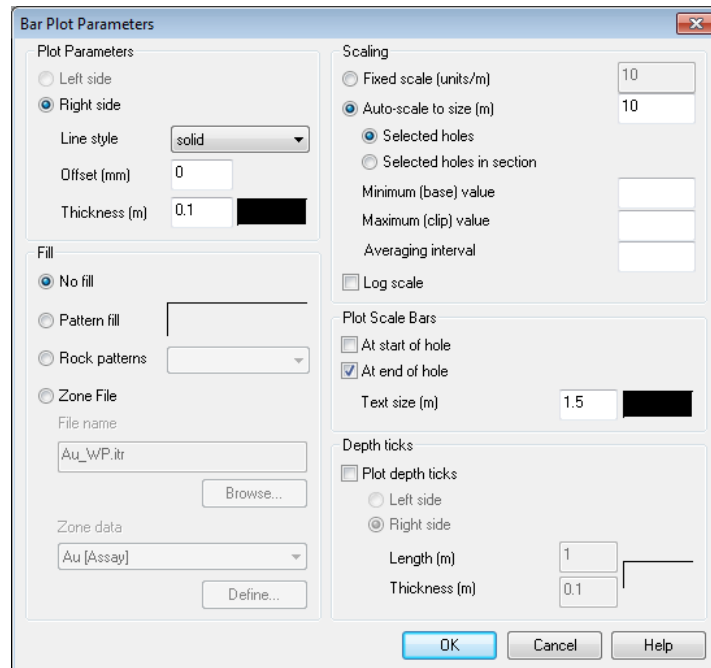
To plot data in the Geosoft 3D Viewer:

1. From the **Add to 3D** menu in the Geosoft 3D Viewer, select **Drillholes**.
The Draw holes to an existing 3D view dialog opens.
2. Click the **Data** tab.

Figure 3.80 Draw holes to an existing 3D view Data tab

3. From the **Data** list in the Left side section, select **Rock [Geology]**.
4. From the **Plot Type** list, select **Rock patterns**.
5. From the **Data** list in the Right side section, select **Au [Assay]**.
6. From the **Plot Type** list, select **Bar plot** and click **Define**.

The Bar Plot Parameters dialog opens.

Figure 3.81 Bar Plot Parameters

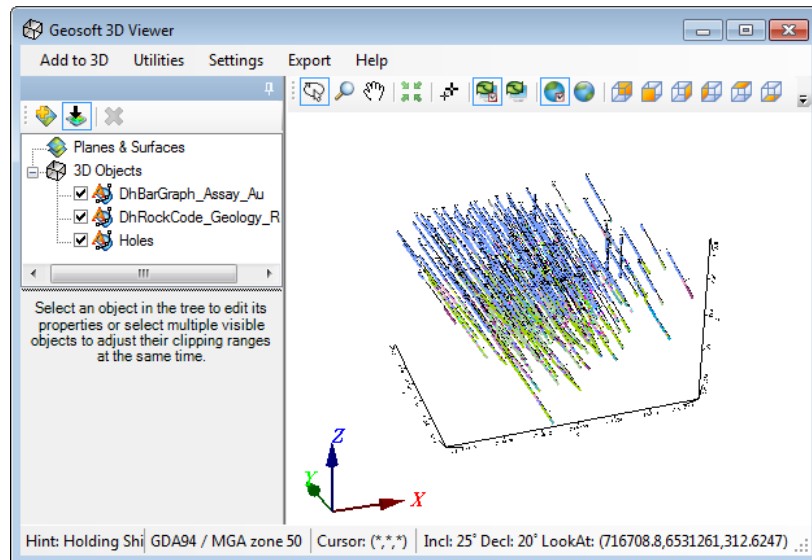
7. For **Thickness**, enter **0.05**.
8. In the Fill section, select **Zone File**.
9. For **Auto-scale to size (m)**, enter **10**.
10. In the Plot Scale Bars section, clear the **At the end of hole** option and click **OK**.

The Bar Plot Parameter dialog closes.

11. Click **OK**.

The rock patterns and bar plots are displayed in the Geosoft 3D Viewer.

Figure 3.82 Geosoft 3D Viewer with data plotted

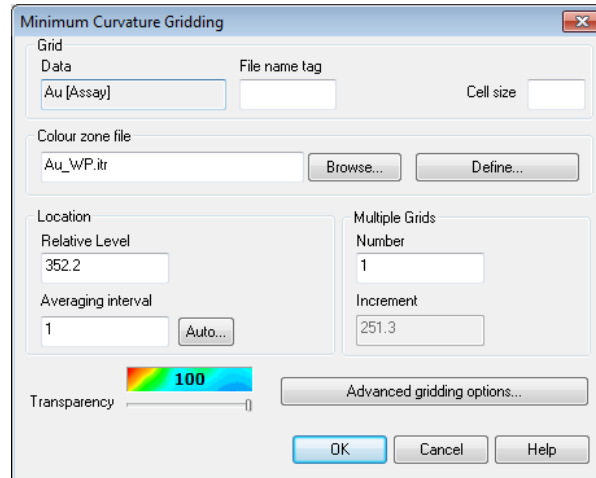


In 3D, only two sets of data can be plotted down hole along the trace (Left side and Right side). When the data appears in the Geosoft 3D Viewer, the point-of-view will always remain the same, no matter how the image is rotated, panned or zoomed.

Next you will plot gridded data by interpolating the drillhole values at a specified relative level or elevation. You will create three grids at 50 metre intervals.

To plot gridded data:

1. From the **Add to 3D** menu in the Geosoft 3D Viewer, select **Drillholes**.
The Draw holes to an existing 3D view dialog opens.
2. Click the **Data** tab.
3. From the **Data** list in the Gridded Data section, select **Au [Assay]**.
4. From the **Plot Type** list, select **Grid: Min. curve** and click **Define**.
The Minimum Curvature Gridding dialog opens.

Figure 3.83 Minimum Curvature Gridding dialog

5. For **Averaging interval**, enter **10**.

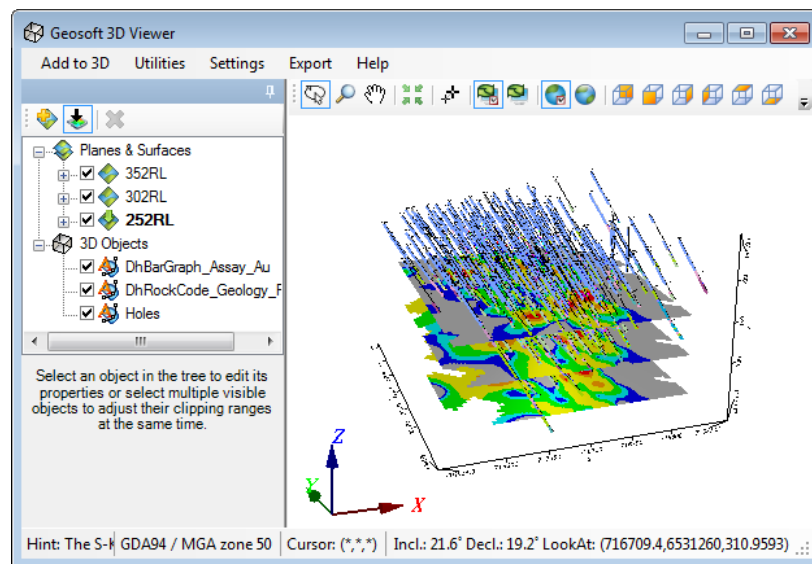
The grid's values are derived from the values in the holes where the holes intersect the grid plane. Because these values may vary quite rapidly with location down the hole, an averaging interval is used to sample the hole in the area of the intersection, and smooth out this variation.

6. In the Multiple Grids section, for the **Number** of grids enter **3**.
7. For **Increment**, enter **-50**.
8. Click **OK**.

The Minimum Curvature Gridding dialog closes.

9. Click **OK**.

The three grids are created and displayed in your Geosoft 3D Viewer.

Figure 3.84 Geosoft 3D Viewer with three grids

Exporting a 3D View to a 3D PDF

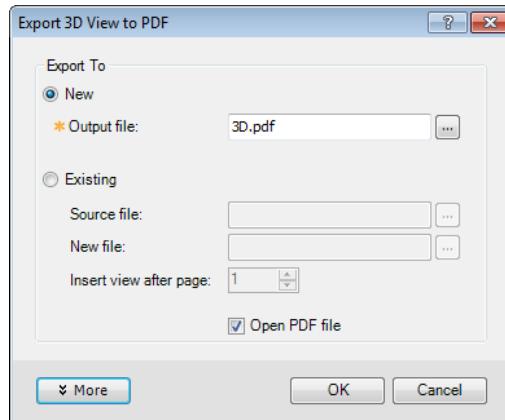
By exporting your current 3D View to a standard Adobe PDF file, you can share your 3D subsurface models with colleagues who do not have Geosoft software installed. The 3D interactive tools, such as zoom, rotation, layer controls, and transparency are included in the exported PDF file.

To export to a 3D PDF:

1. From the **Export** menu in the Geosoft 3D Viewer, select **3D PDF**.

The Export 3D View to PDF dialog opens.

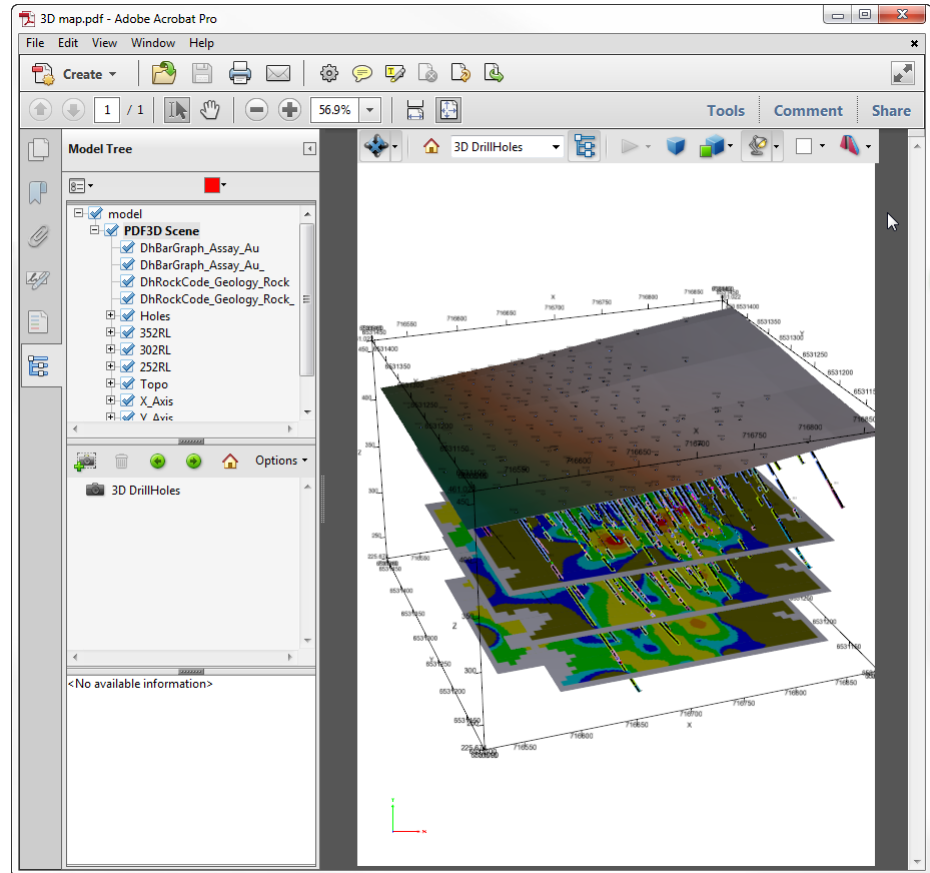
Figure 3.85 Export 3D View to PDF dialog



2. Click **OK**.

The 3D PDF opens in your default PDF viewer.

Figure 3.86 3D PDF



You can use these interactive tools on the 3D Toolbar to change the zoom, rotation, layer controls, transparency, lighting and many other options for your PDF 3D View:

Figure 3.87 3D Toolbar



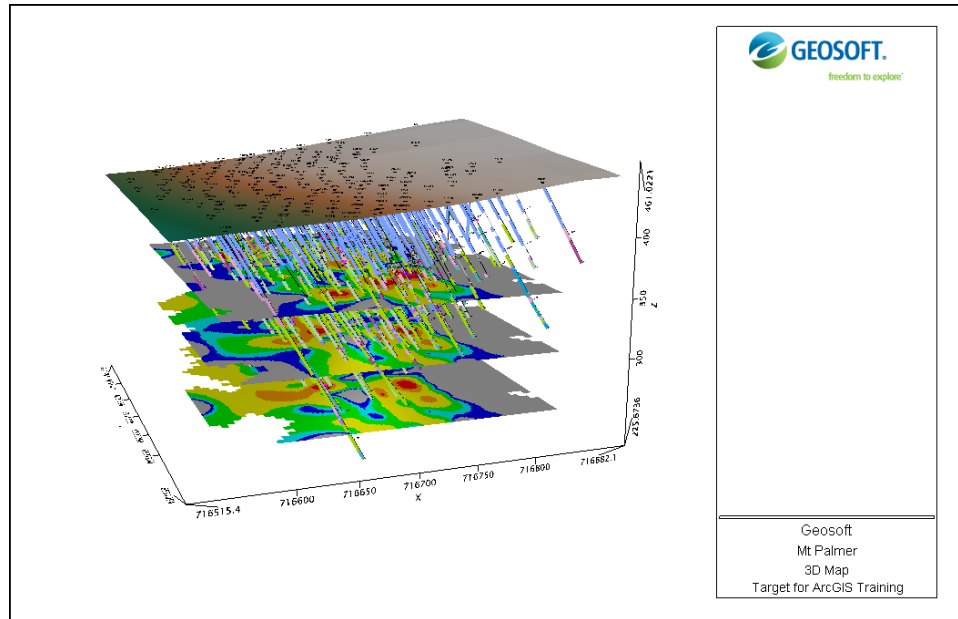
Now would be a good time to save your project.

To view the static 3D map:

- > Close the **Geosoft 3D Viewer**.

The Geosoft 3D Viewer closes and a static view of the data manipulated in the Geosoft 3D Viewer is displayed on the map.

Figure 3.88 3D Map



Now would be a good time to save your map document.

In this lesson you:

- > Created a 3D map
- > Used the Geosoft 3D Viewer
- > Plotted rock patterns and a bar plot
- > Plotted gridded data
- > Exported to a 3D PDF

Module 4: Implicit Modelling

Module 4 has Five Lessons:

Lesson 4.1	Creating Voxels	158
Lesson 4.2	Creating Isosurfaces	164
Lesson 4.3	Creating a Lithology Voxel and Contact Surface from Geology	170
Lesson 4.4	Extracting Slices from a Voxel	175

Lesson 4.1 Creating Voxels

In this lesson you will:

- › Create a voxel
- › View voxel statistics
- › Clip a voxel to a topography layer
- › Window a voxel
- › Apply a colour zone file to a voxel

3D Gridding

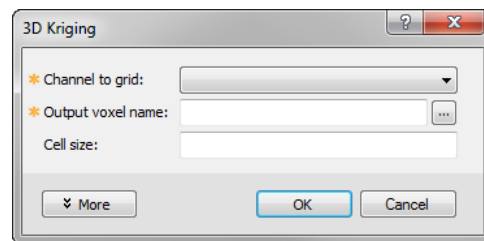
The 3D Gridding option uses either basic statistical kriging or inverse distance weighted interpolation algorithms to create a 3D voxel grid from 3D data contained in a Geosoft Database (GDB file). To create a voxel grid you will need to have a data set that contains X, Y, and Z fields that locate the data within a coordinate system in which Z coordinates are interpreted as positive up.

To create a 3D voxel grid model using Kriging:

1. From the **Voxel** menu in the 3D Viewer, select **3D Gridding** then **Kriging**.

The 3D Kriging dialog opens.

Figure 4.1 3D Kriging dialog



2. Click **More**.

The 3D Kriging dialog expands to show additional options.

Figure 4.2 Expanded 3D Kriging dialog

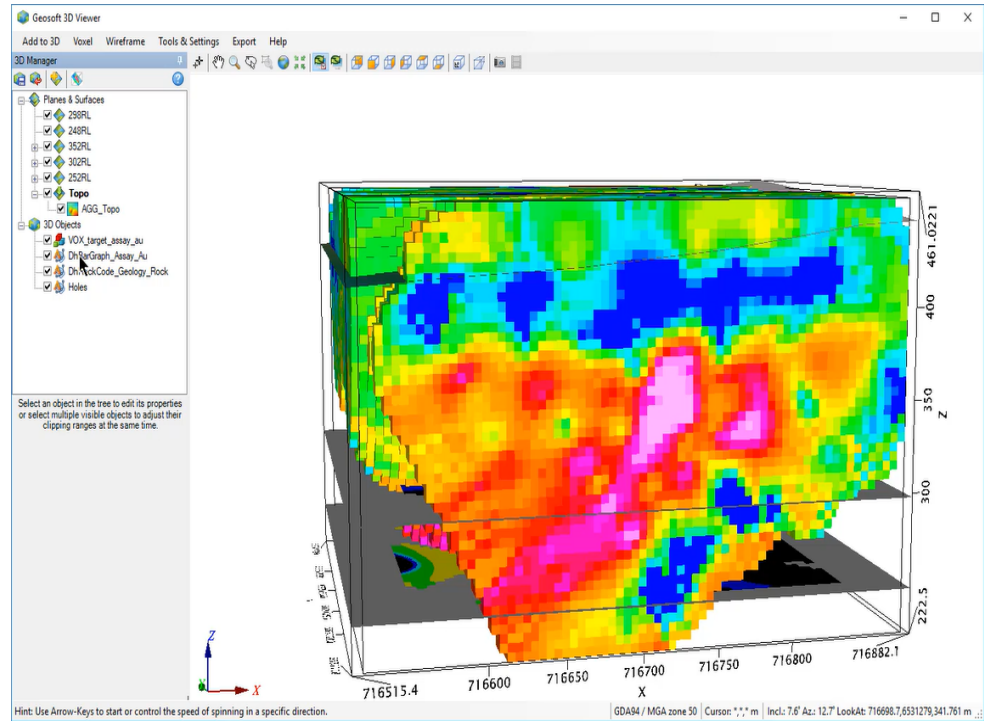
3. From the **Channel to grid** list, select **Au [Assay]**.
The Output voxel file will be named Target_Assay_Au.geosoft_voxel by default.
4. For **Cell size**, enter **5**.
5. For **Blank distance**, enter **10**.
6. For **Log option**, select **log, save as linear**.
7. For **Log minimum**, enter **0.01**
8. Click the **Variogram** tab.
9. For **Range**, enter **55** and for **Sill**, enter **0.44**.



You can optionally view the variogram to visually inspect the result of the advanced parameters on the model.

10. Click **OK**.
The voxel is displayed in the Geosoft 3D Viewer and listed in the 3D Manager tree list.

Figure 4.3 Voxel in 3D Viewer

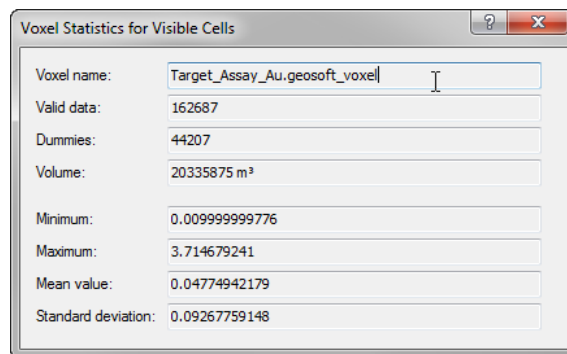


To view voxel statistics:

1. In the 3D Manager tree list, select the **VOX_target_assay_au** 3D Object.
2. On the **Attributes** tab displayed below, click **Visible Cell Statistics**.

The Voxel Statistics for Visible Cells dialog opens and displays the Maximum, Minimum, Mean, Standard Deviation, and Volume.

Figure 4.4 Voxel Statistics for Visible Cells dialog

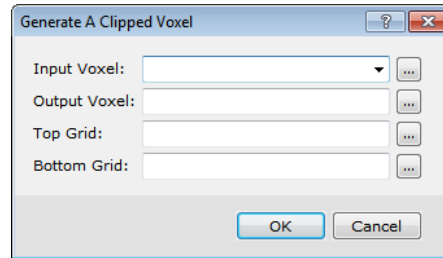


The statistics depend on the gridding parameters you entered (i.e. blanking distance, cell size, log option/minimum, advanced Kriging parameters).

To clip a voxel to the topography:

1. From the **Voxel** menu in the 3D Viewer, select **Voxel Topography Clip**.

The Generate A Clipped Voxel dialog opens.

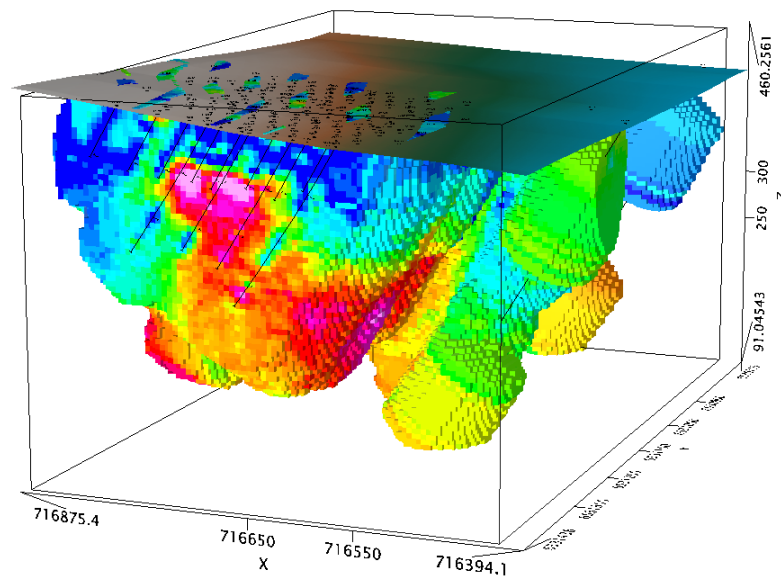
Figure 4.5 Generate A Clipped Voxel dialog

2. For the **Input Voxel**, select **Target_Assay_Au.geosoft_voxel**.
3. For the **Output Voxel**, enter **Au_Topo_Clip**.
4. For the **Top Grid**, click the **Browse** button.
5. From the Topography folder, select **Topo.grd** and click **Save**.
6. Click **OK**.

A new 3D Object called VOX_au_topo_clip is created.

7. To view the new clipped voxel, turn off the display of the unclipped voxel **VOX_target_assay_au**.

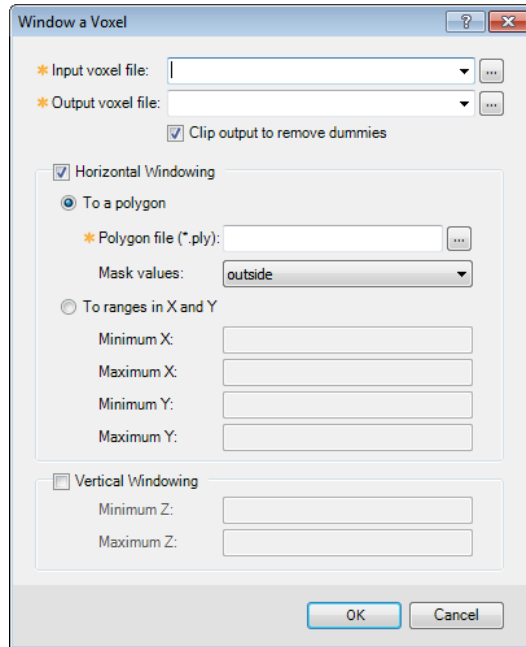
Your clipped voxel is now visible. Voxel cells that intersect with the topography surface are not clipped and are visible; voxel cells above the surface have been removed.

Figure 4.6 3D Viewer with voxel clipped to topography

To window a voxel to a boundary:

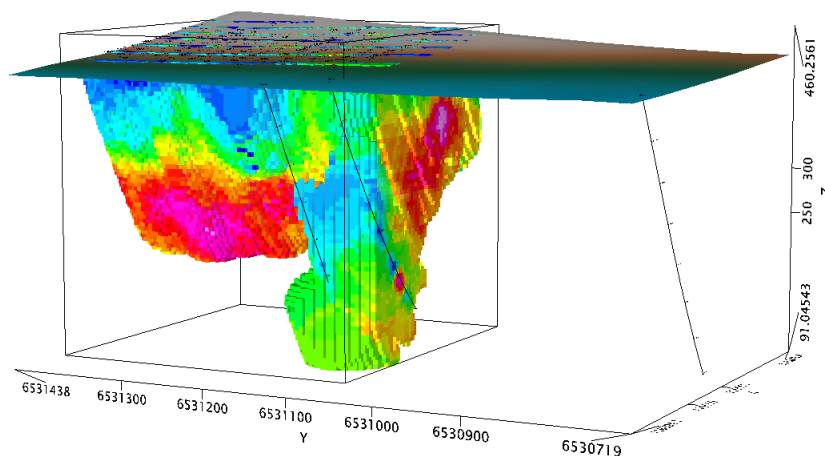
1. From the **Voxel** menu, select **Window a Voxel**.
The Window a Voxel dialog opens.

Figure 4.7 Window a Voxel dialog



2. For the **Input voxel file**, select **Au_Topo_Clip.geosoft_voxel**.
The **Output voxel file** field will be automatically populated with **Au_Topo_Clip_win.geosoft_voxel**.
 3. Ensure the **Horizontal Windowing** and **To a shapefile** options are selected.
 4. For **Shape file (*.shp)**, click the **Browse** button.
 5. From the **Drilling Data** folder, select **DH_Selection.shp** and click **Open**.
 6. Click **OK**.
- The output voxel VOX_au_topo_clip_win is clipped to the extents of the .shp file and is added to the 3D Viewer. Turn off the display of the Topo layer and the VOX_au_topo_clip 3D Object to see the clipped object.

Figure 4.8 3D Viewer with windowed voxel



In this lesson you:

- › Created and displayed a voxel
- › Viewed voxel statistics
- › Clipped a voxel to a topography layer
- › Windowed a voxel
- › Applied a colour zone file to a voxel

Lesson 4.2 Creating Isosurfaces

In this lesson you will:

- › Determine an isosurface value
- › Create and display an isosurface
- › Append an isosurface to the geosurface file
- › Create multiple isosurfaces from a voxel
- › Export isosurfaces to a DXF file

Isosurfaces

An isosurface is a 3-dimensional surface that passes through points of equal value. Isosurfaces can be thought of as 3D contours. Isosurfaces are written to a geosurface file that is able to save many isosurfaces extracted from the same voxel. This allows you to copy a single file with all its isosurfaces to another location, or to easily share your results with colleagues to collaborate on projects.

Using Geosoft tools, there are three different ways to create isosurfaces from a voxel:

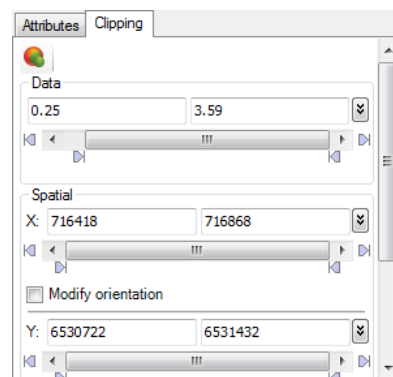
- › Visually clip the minimum data values from a voxel and automatically extract an isosurface at the specified clip value.
- › Create a single isosurface at a specified value.
- › Automatically create multiple isosurfaces using a linear, log or user-specified distribution.

To determine the isosurface value:

1. In the 3D Objects tree in the 3D Manager, select **VOX_au_topo_clip_win**.
2. Click the **Clipping** tab.
3. In the Data section, use your cursor to move the bottom slider to the right to increase the value to **0.25**.


This interactively modifies the low value of the Data clip region. Only the voxel cells valued at 0.25 ppm or greater will be visible.

Figure 4.9 Voxel Data Clipping



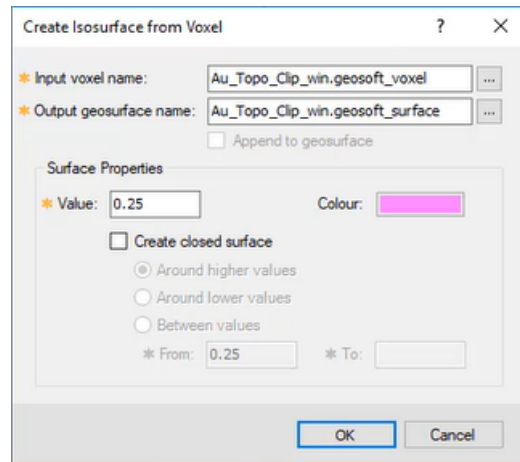
You can also type in specific values.

To create and display the isosurface:

1. Click on the **Define isosurface using data clipping values** icon within the clipping pane ().

The Create Isosurface from Voxel dialog opens and is automatically populated with the Input voxel name, the Output geosurface name, the Surface Properties value (this is the minimum data clip value you specified) and the Colour.

Figure 4.10 Create Isosurface from Voxel dialog

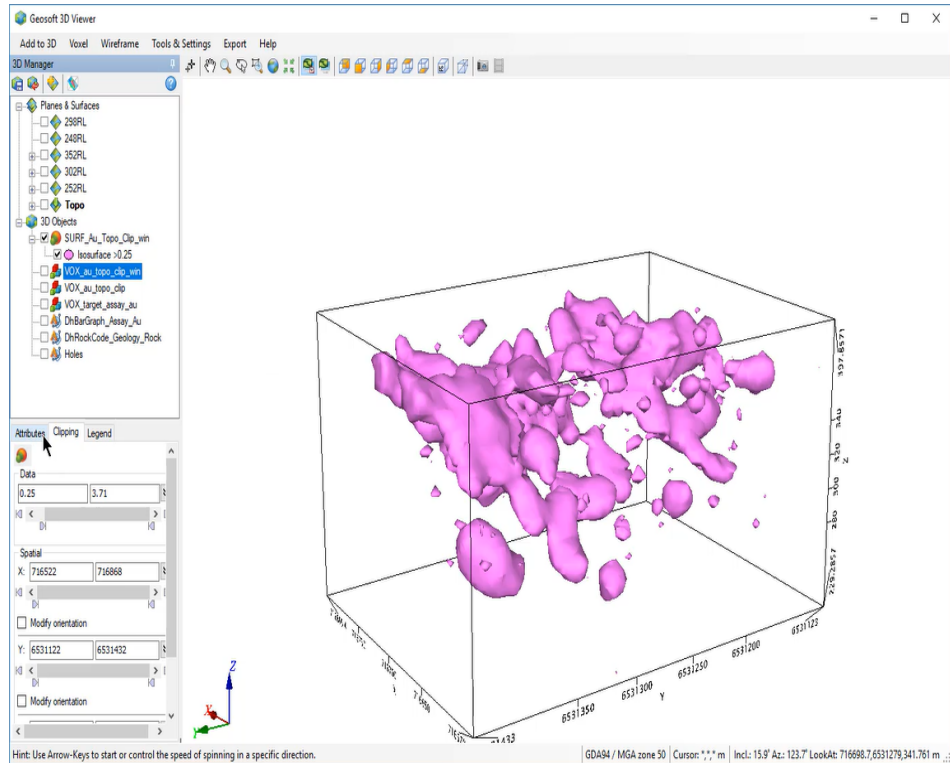


2. Change the **Colour** to **pink**.
3. Specify **0.25** for **Value**.
4. Check the **Create closed surface** check box.
5. Click **OK**.

The geosurface (SURF_Au_Topo_Clip_win) and the isosurface (Isosurface >0.25) are added to the 3D Viewer.

6. Turn off the display of the **VOX_au_topo_clip_win** layer to see the isosurface more clearly.

Figure 4.11 Isosurface at a value of 0.015



To change the attributes of the isosurface:

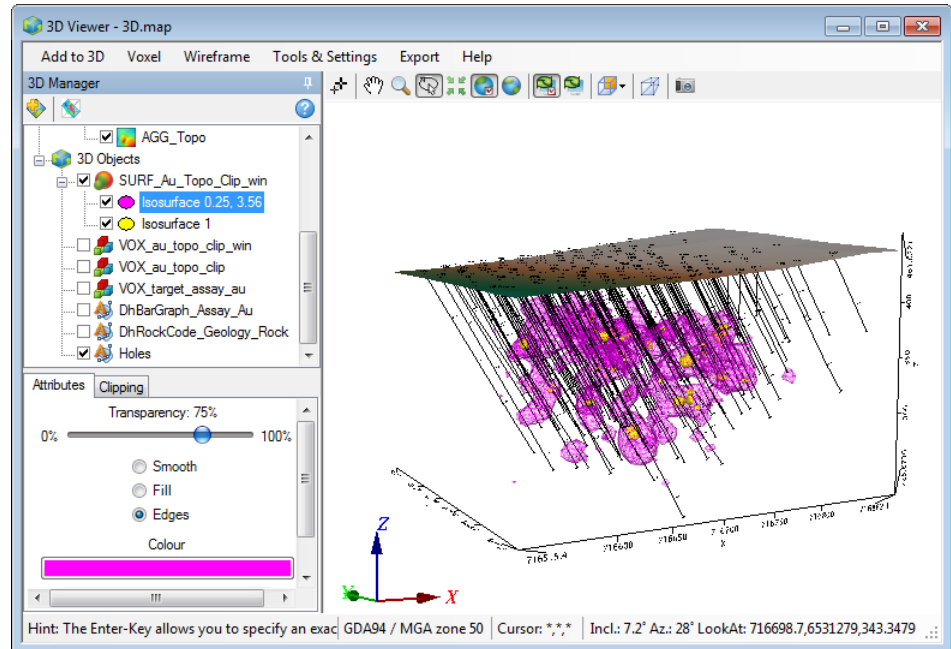
1. Select the **Isosurface_0.25** 3D Object and click the **Attributes** tab.
2. Using the slide bar, change the **Transparency** to **75%**.
3. Select the **Edges** option.

To append another isosurface to the geosurface file:

1. From the **Add to 3D** menu, select **Isosurface** then **Create Isosurface**.
The Create Isosurface from Voxel dialog reopens.
2. Click the **Create closed surface** checkbox so it is no longer checked.
3. Change the **Surface value** to **1**, and change the **Colour** to **yellow**.
4. Ensure the **Append to geosurface** option is selected.

Isosurface_1 is appended to SURF_au_topo_clip_win and you now see the two isosurfaces, one inside the other, in the 3D Viewer.

Figure 4.12 Isosurfaces in 3D



1. Click the **Create closed surface** checkbox so it is no longer checked.

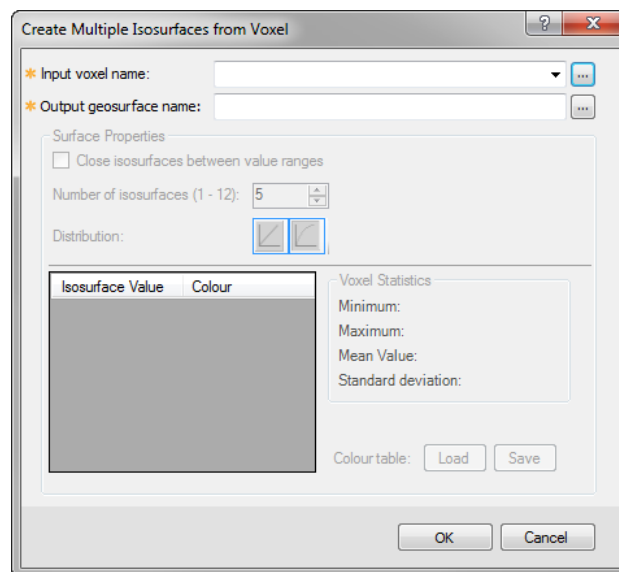
You will now automatically extract 5 isosurfaces using a log distribution from the `au_topo_clip_win.geosoft_voxel`.

To create multiple isosurfaces:

1. From the **Add to 3D** menu, select **Isosurface** and click **Create Multiple Isosurface**.

The Create Multiple Isosurfaces from Voxel dialog opens.

Figure 4.13 Create Multiple Isosurfaces from Voxel dialog

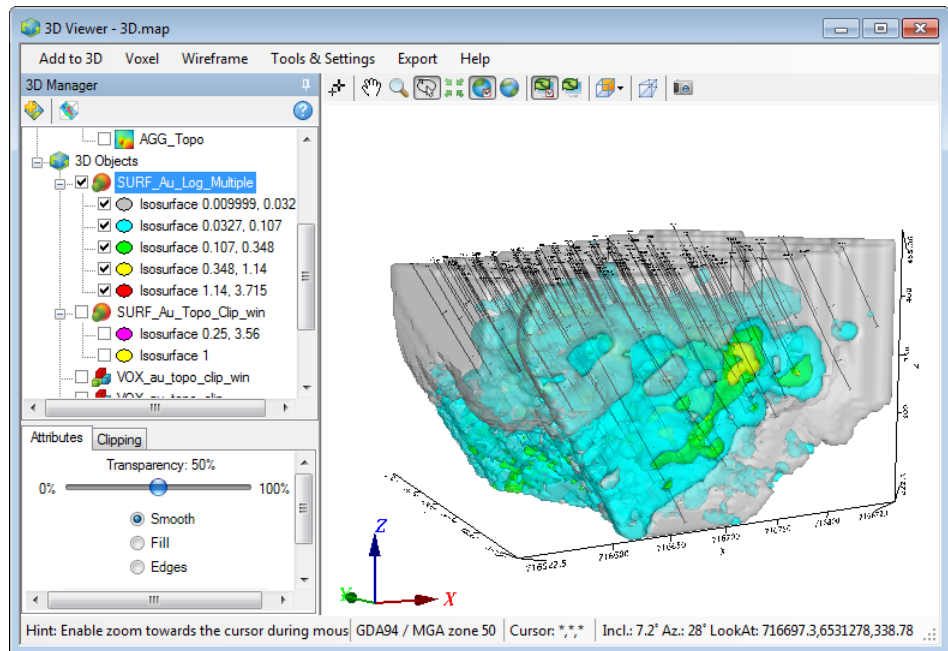


2. For **Input voxel**, select `Au_Topo_Clip_win.geosoft_voxel`.
3. Change the **Output geosurface name** to `Au_Log_Multiple.geosoft_surface`.

4. Select the **Close isosurfaces between value ranges** option.
5. For **Distribution**, click **Use a log distribution**.
The Log Transform dialog opens.
6. Use the default values and click **OK**.
7. In the Create Multiple Isosurfaces from Voxel dialog, click **OK**.

SURF_au_log_multiple and its five multiple isosurfaces are added to the 3D Viewer. All of the isosurfaces are automatically displayed with 50% transparency except for the highest value which is opaque.

Figure 4.14 Multiple Isosurfaces in 3D



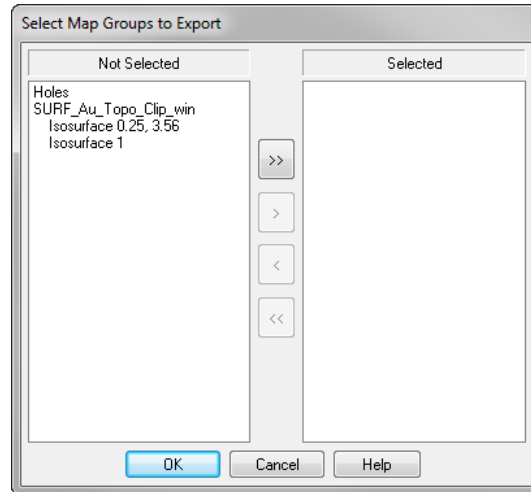
8. Turn off the display of the SURF_au_topo_clip_win layer.
 1. Select the **Close isosurfaces between value ranges** option.


You will now export the first two isosurfaces you created to an AutoCAD DXF file.

To export isosurfaces to a AutoCAD 3D DXF file:

1. Turn off the display of the SURF_au_log_multiple layer and turn on the display of the SURF_au_topo_clip_win layer.
2. From the **Export** menu in the 3D Viewer, select **AutoCAD DXF File**.

The Select Map Groups to Export dialog opens. Only 3D vector layers that are visible in the 3D Viewer are listed.

Figure 4.15 Select Map Groups to Export dialog

3. From the **Not Selected** list on the left, select **SURF_au_topo_clip_win**.
4. Click the right arrow button  .
The geosurface and its isosurfaces are added to the Selected list.
5. Click **OK**.
The Export to 3D DXF File dialog opens.
6. For the **DXF file**, enter **Au_Orebodies.dxf** and click **OK**.
These map groups are saved to a single DXF file and added to your project folder.

In this lesson you:

- > Determined an isosurface value
- > Created and displayed an isosurface
- > Appended an isosurface to the geosurface file
- > Created multiple isosurfaces from a voxel
- > Exported isosurfaces to a DXF file

Lesson 4.3 Creating a Lithology Voxel and Contact Surface from Geology

In this lesson you will:

- Create a lithology voxel
- View lithology voxel statistics
- Create a contact surface from geology

Creating a Lithology Voxel

A lithology voxel is a voxel based on codes. First, contact surfaces between each lithology unit are created; then the spaces between the surfaces are filled in with the voxel blocks.

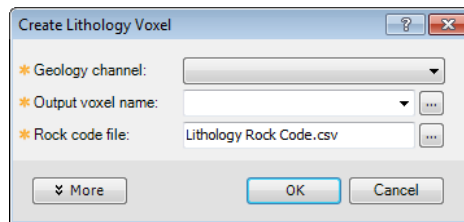
In this lesson, you will create a lithology voxel based on the codes that are in the Rock channel of the Geology database and reference the lithology rock codes.csv file to colour the voxel cells.

To create a lithology voxel:

1. From the **Voxel** menu in the 3D Viewer, select **Create Lithology Voxel**.

The Create Lithology Voxel dialog opens.

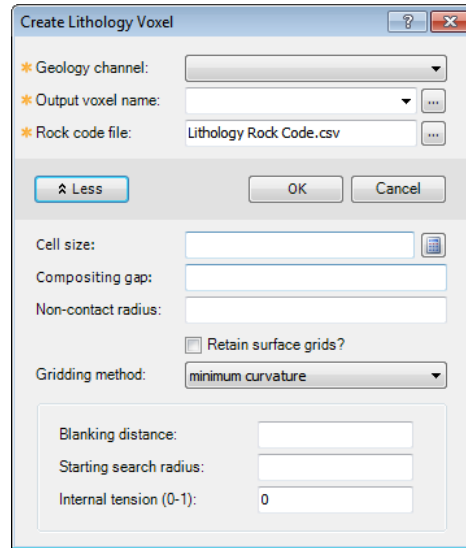
Figure 4.16 Create Lithology Voxel dialog



2. Click **More**.

The Create Lithology Voxel dialog expands to show additional options.

Figure 4.17 Expanded Create Lithology Voxel dialog

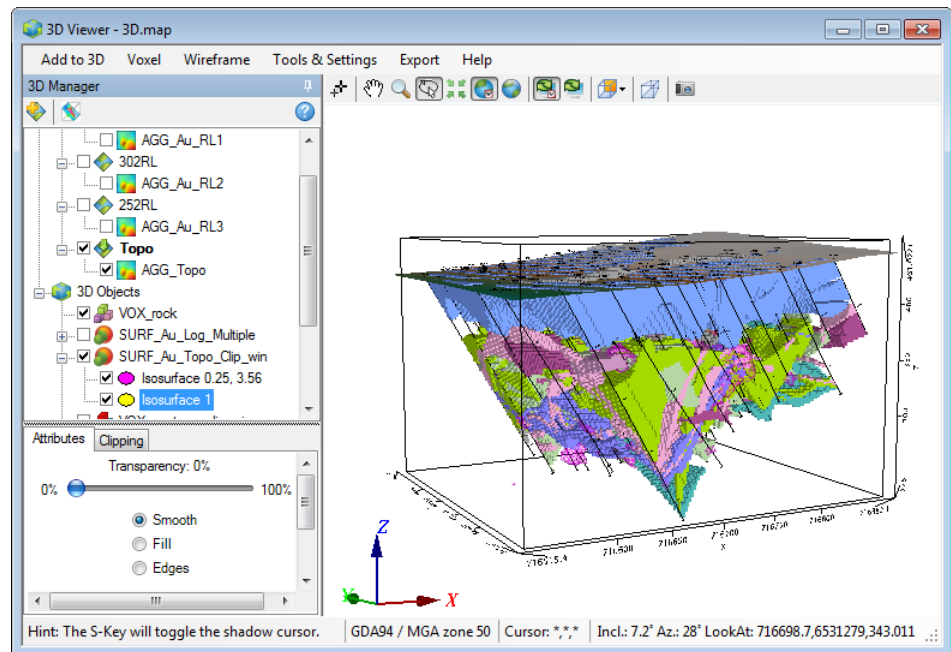


3. For the **Geology channel**, select **Rock [Geology]**.
The Output voxel name will be Rock.geosoft_voxel by default.
4. Check the **Retain surface grids** box.
5. For **Cell size**, enter **2** and click **OK**.
The lithology voxel is displayed in the 3D Viewer.



Note: It may take a while before the results are displayed in the 3D Viewer.

Figure 4.18 Lithology Voxel



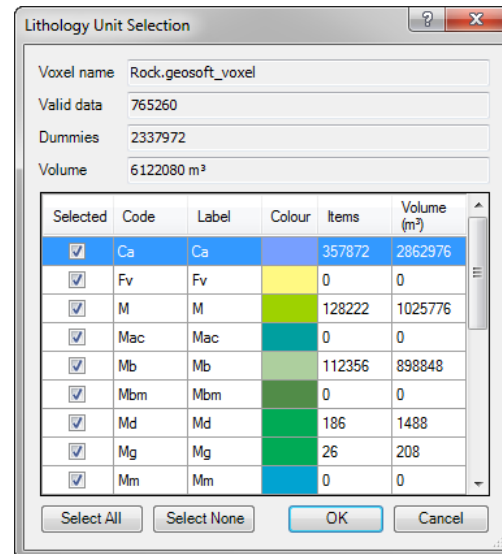
Next, you will open the Lithology Unit Selection tool. You can use this tool to view basic statistical information about the data in the lithology voxel and to turn the display of lithology units on and off.

To view lithology voxel statistics:

1. In the 3D Objects tree list, select the **VOX_rock** lithology voxel.
2. On the **Clipping** tab below, click **Lithology Unit Selection**.

The Lithology Unit Selection dialog opens. The Volume (m³) column displays the total volume of all items (cubes) with the given lithology within the current XYZ clipping region of the voxel.

Figure 4.19 Lithology Unit Selection dialog



The Volume calculation is based on your chosen cell size and interpretation distance. It is not detailed enough to be used for resource calculations or modelling.

3. Click in the **Selected** column to turn the display of lithology units on and off in the 3D Viewer.
4. Click **OK**.

Creating a Contact Surface

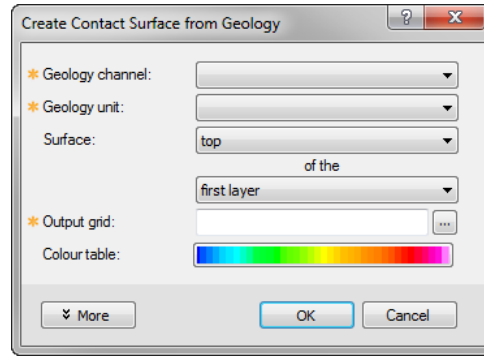
A contact surface is a sub-surface topography of the top or bottom of any lithology unit. A contact surface is useful for identifying folding features and can be added to a section map as sub-surface topography. You do not need to make a lithology voxel before creating a contact surface as it looks solely at the downhole lithology data.

In this lesson, you will create a contact surface to represent the top of the mineralised zone identified earlier as the Sif lithology unit.

To create a contact surface from geology:

1. From the **Voxel** menu in the 3D Viewer, select **Create Contact Surface from Geology**.

The Create Contact Surface from Geology dialog opens.

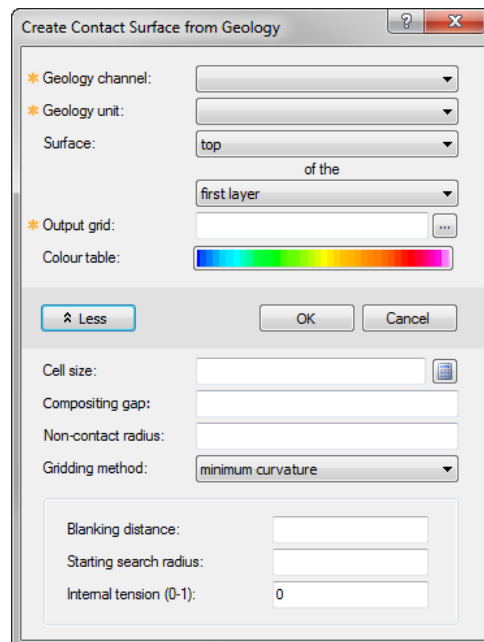
Figure 4.20 Create Contact Surface from Geology dialog

2. For the **Geology channel**, select **Rock [Geology]**.
3. For the **Geology unit**, select **Sif**.

The grid file Rock_Sif_top_first.grd (grd) will be generated based on using the default Surface settings of top of the first layer.

4. For the **Colour table**, click the **Colour ramp**.
5. For **Category**, select **monochromatic**.
6. Select the **blue.tbl** colour table and click **OK**.
7. Click **More**.

The Create Contact Surface from Geology dialog expands to show additional options.

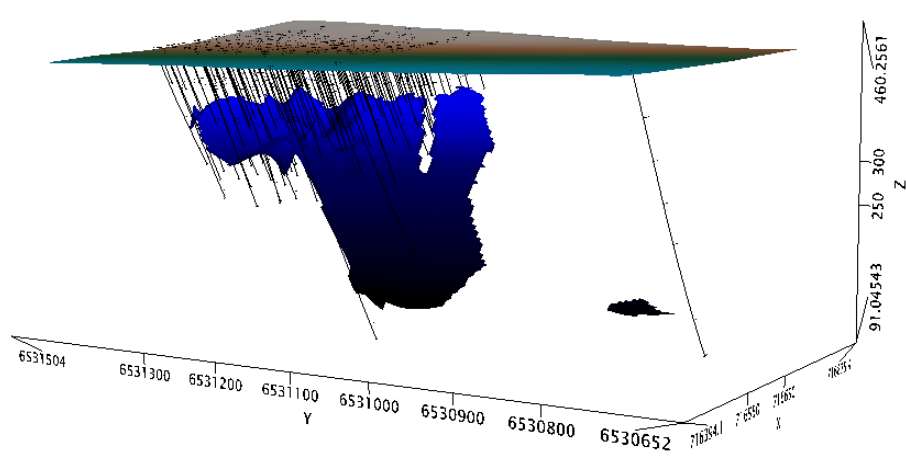
Figure 4.21 Create Contact Surface from Geology More dialog

8. For **Cell size**, enter **2** and click **OK**.
9. For **Compositing gap**, enter **2**.
10. For **Non-contact radius**, enter **0**.

11. For **Blanking distance**, enter **500**.
12. Click **OK**.

The contact surface is added to the 3D Viewer. You may want to turn off the display of other layers to see the contact surface more clearly.

Figure 4.22 Contact surface from geology



To create a contact surface from geology:

1. Select the appropriate colour table and click **OK**.

In this lesson you:

- > Created a lithology voxel
- > Viewed lithology voxel statistics
- > Created a contact surface from geology

Lesson 4.4 Extracting Slices from a Voxel

In this lesson you will:

- › Interactively extract a slice from a voxel

Extracting Slices from a Voxel

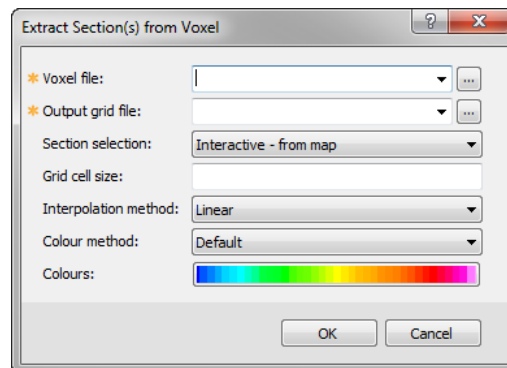
In addition to slicing voxels from within the Section tool, you can interactively extract voxel slices in any orientation from your plan map. A new Geosoft Grid (.grd) file representing the voxel slice will be generated and automatically displayed in the 3D viewer.

To interactively extract a slice from a voxel:

1. From the **Add to 3D** menu in the 3D Viewer, select **Grids**, then **Extract Sections(s) from Voxel**.

The Extract Section(s) from Voxel dialog opens.

Figure 4.23 Extract Section(s) from Voxel dialog



2. For the **Voxel file**, select **Au_Topo_Clip.geosoft_voxel**.
3. For the **Output grid file**, enter **Voxel_Slice**.
4. For the **Section selection**, select **Interactive - from drill project**.
5. For the **Grid cell size**, enter **5** and click **OK**.

The Define Section View window opens.

6. Click one of the **Define** buttons and draw a line to define the slice.

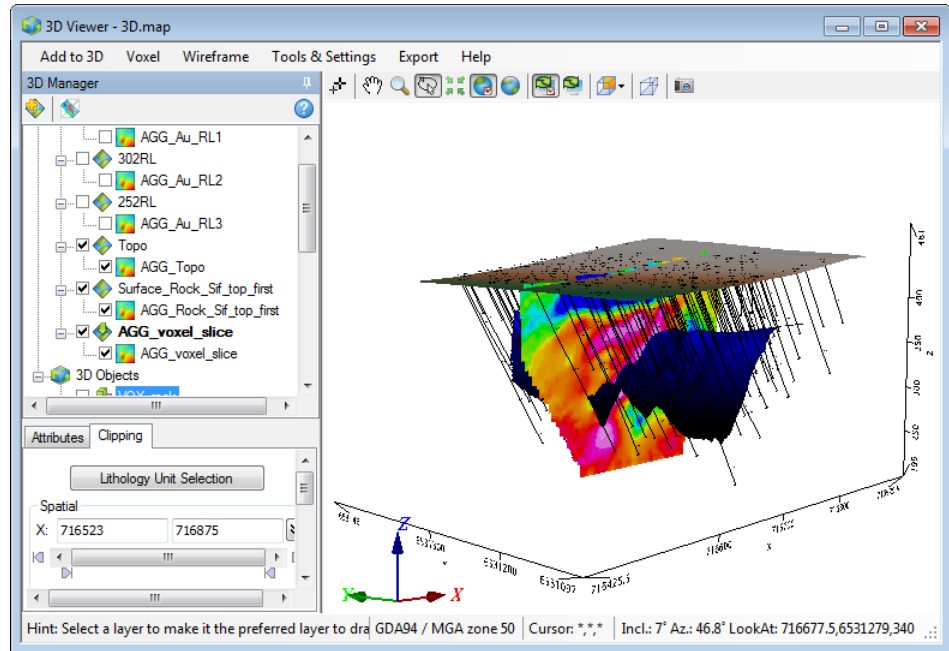


You can also enter specific Location and Orientation information.

7. Click **OK**.

Your voxel slice is added to the 3D Viewer.

Figure 4.24 3D Viewer with voxel slice



In this lesson you:

- › Interactively extracted a slice from a voxel

Module 5: Explicit Modelling

Module 5 has Four Lessons:

Lesson 5.1	Creating an fGDB and Adding Feature Classes	178
Lesson 5.2	Digitizing Interpretations on Section Maps	182
Lesson 5.3	Editing Feature Classes	186
Lesson 5.4	Wireframing Interpretations	190

Lesson 5.1 Creating an fGDB and Adding Feature Classes

In this lesson you will:

- › Create an interpretations geodatabase
- › Add a polygon feature to the geodatabase
- › Add a polyline feature to the geodatabase

Explicit Modelling

Geological Interpretation and Wireframing

Explicit modelling in enables you to build 3D models from 2D interpretations drawn on section or plan maps. There are four main parts to the modelling workflow:

- › Create a series of horizontal or vertical section maps containing the data to support your interpretation.
- › Digitize two-dimensional (2D) interpretations on section or plan maps.
- › Open the 2D interpretations in the 3D Viewer and wireframe (connect) them together to form 3D volumes and/or surfaces.
- › Use the analysis tools to extract information from the volumes and surfaces.

Interpretations Geodatabases

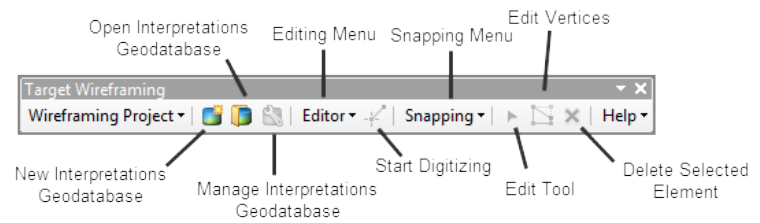
Target Interpretations Geodatabases are a special version of an Esri File Geodatabase that store digitized interpretations drawn on section maps. Both polygon and polyline features can be stored in the same file geodatabase. Interpretations geodatabases store the following information:

- › XYZ locations
- › Coordinate system
- › Attributes including feature names and feature templates



The coordinate system for the Interpretations Geodatabase will automatically be set based on the current Target drillhole database.

You can use the options available on the Target Wireframing toolbar to create an interpretations geodatabase, digitize interpretations, and manage the features and interpretations.

Figure 5.1 Target Wireframing toolbar options

The Target Wireframing toolbar can be docked or it can remain in a floating state.

To open the Wireframing map document:

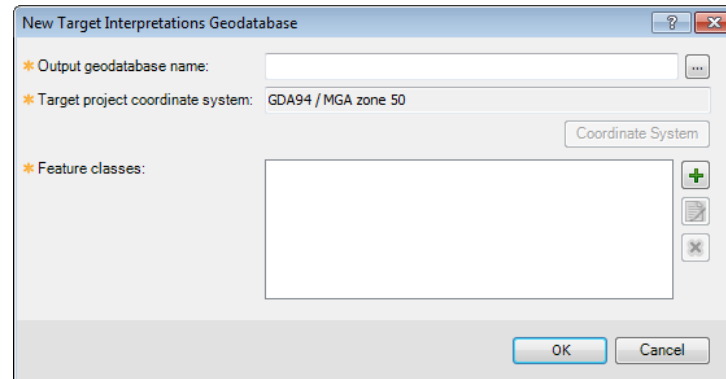
1. From the ArcMap **File** menu, select **Open**.
2. From the Geosoft Training Data folder, select **Wireframing.mxd** and click **Open**.

The Wireframing map document opens in ArcMap.

To create an interpretations geodatabase:

1. From the **Wireframing Project** menu on the Target Wireframing toolbar, select **New Geodatabase** or click the **New Interpretations Geodatabase** button.

The New Target Interpretations Geodatabase dialog opens.

Figure 5.2 New Target Interpretations Geodatabase dialog

2. For **Output geodatabase name**, enter **Palmer_interp**.

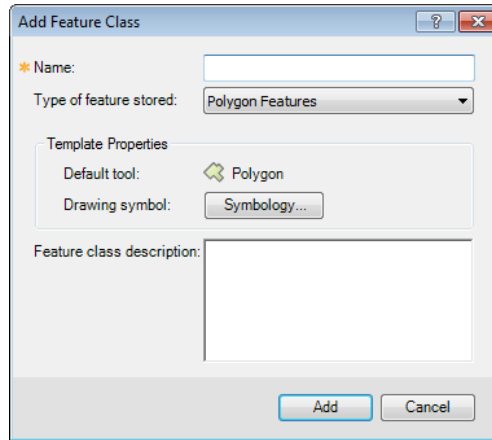
The Coordinate system is automatically set to GDA94 / MGA zone 50 based on the projection of the drillhole project.

To add a Line feature class to a geodatabase:

1. To the right of the Feature classes box, click the **Add new feature class** button.

The Add Feature Class dialog opens.

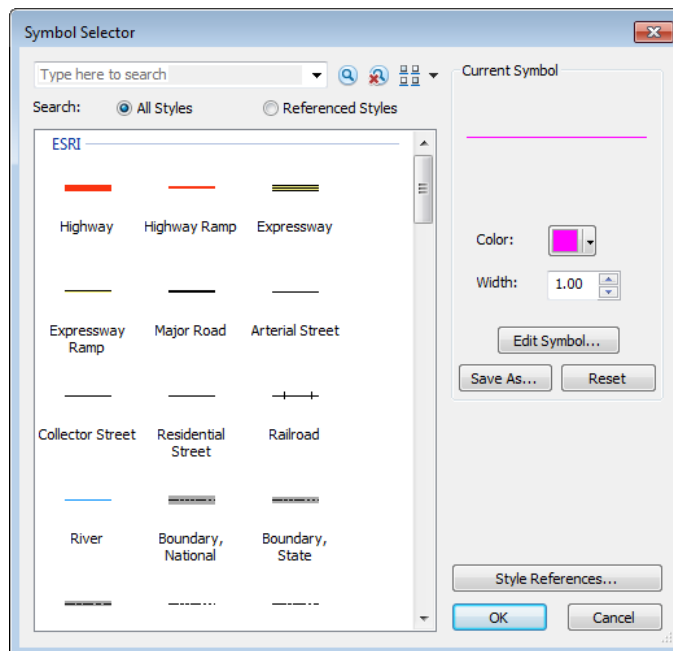
Figure 5.3 Add Feature Class dialog



2. For **Name**, enter **Top_Iron**.
3. From the **Type of feature stored** list, select **Line Features**.
4. In the Template Properties area, click **Symbology**.

The Symbol Selector opens.

Figure 5.4 Symbol Selector



5. For **Colour**, select **orange** and click **OK**.
6. For **Feature class description**, enter **Top of iron formation** and click **Add**.
Top_Iron is added to the list of Feature classes stored in the Interpretations Geodatabase.

To add a polygon feature class to a geodatabase:

1. To the right of the Feature classes box, click the **Add new feature class** button.

The Add Feature Class dialog opens.

2. For **Name**, enter **Alluvium**.
The Type of feature stored is Polygon Features by default.
3. Click the **Symbology** button, change the Fill Colour and Outline Colour to **blue** and click **OK**.
4. For **Feature class description**, enter **Depositional layer** and click **Add**.
Alluvium is added to the list of Feature classes stored in the Interpretations Geodatabase.
5. Click **OK**.



You can manage your feature classes by clicking the Manage Interpretations Geodatabase button on the Target Wireframing Toolbar.



To open an existing geodatabase to continue digitizing, click the Open Interpretations Geodatabase button on the Target Wireframing toolbar. The interpretations in the geodatabase will be automatically displayed on the dataframes on which they were drawn.

In this lesson you:

- Created an interpretations geodatabase
- Added a polygon feature to the geodatabase
- Added a polyline feature to the geodatabase

Lesson 5.2 Digitizing Interpretations on Section Maps

In this lesson you will:

- › Digitize polyline interpretations on section maps
- › Digitize polygon interpretations on section maps

Snapping

Snapping can make digitizing more efficient and accurate. If snapping is on when digitizing an interpretation on a section map, a red circle will appear around the tip of the cursor when it comes within 10x10 pixels of the specified Snap To item. The following table describes the three snapping options and their snapping circle colours:

Table 5.1 Snapping Options

Snap To Option	Description	Snapping Circle Colour
True Drillhole Locations	The cursor will snap to the true X, Y, Z location along the drillhole trace from the interval or point in the specified drillhole database.	Pink
Projected Map Groups	The cursor will snap to the X, Y, Z location of the specified map group projected onto the centre plane of the section.	Red
Geostrung Features	The cursor will snap to the X, Y, Z location of the specified geostring feature. Polygon and polyline features can snap to other polygon and polyline features including to polygons and polylines of the same feature.	Orange when snapping to a vertex; Red when snapping to an edge

To digitize geological polyline features on a section map:

1. Make the Section_6531260N dataframe Active and switch to the Data View.
2. From the **Editor** menu on the Target Wireframing toolbar, select **Start Editing**.
3. From the **Snapping** menu, select Snap to DH Database and from the Select DH Database menu choose Litho_composite.
4. Click **Start Digitizing** and select the Top_Iron feature class in the Create Features sidebar.
5. Hover the digitization cursor over different rock codes along the section trace.

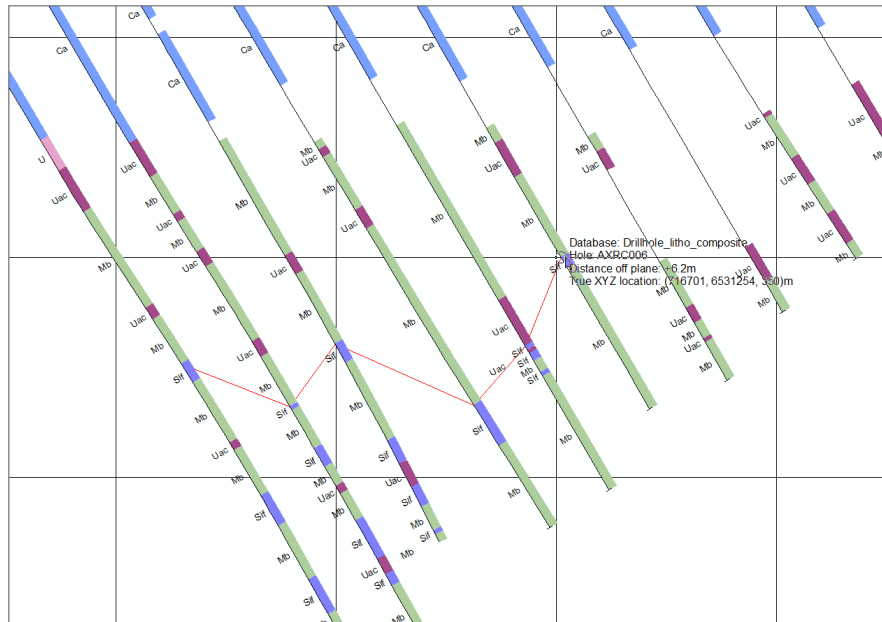
If you allow the snap cursor to rest for a moment, you will see the following information reported in a tooltip:

- The Hole ID and specified Snap To database.
 - The distance away from the centre plane of the section.
 - The true X, Y, Z location.
6. Digitize a polyline along the top of the iron formation.

This is the purple rock pattern along the drillhole trace and the Rock code is Sif.
 7. To end the polyline, right-click and select Finish Sketch.
 8. Click the Editor menu in the Target Wireframing toolbar and click Save Edits.

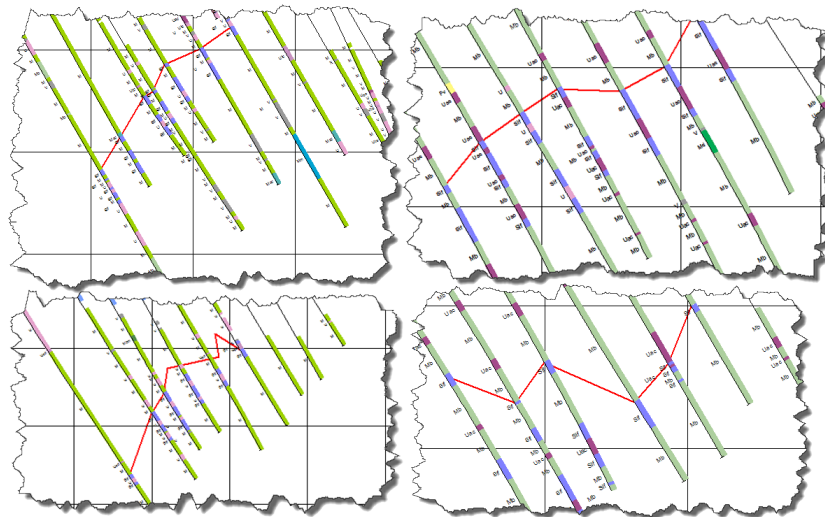
If you switch dataframes without saving your edits you will be prompted to do so.

Figure 5.5 Litho_composite database and S6531260N section



9. Repeat this process digitizing a polyline for the top of the iron formation on the following section map dataframes:
 - Section_6531285N
 - Section_6531310N
 - Section_6531331N
 - Section_6531380N

Figure 5.6 Top_iron interpretations on section maps



You can double-click to end a polyline or polygon.

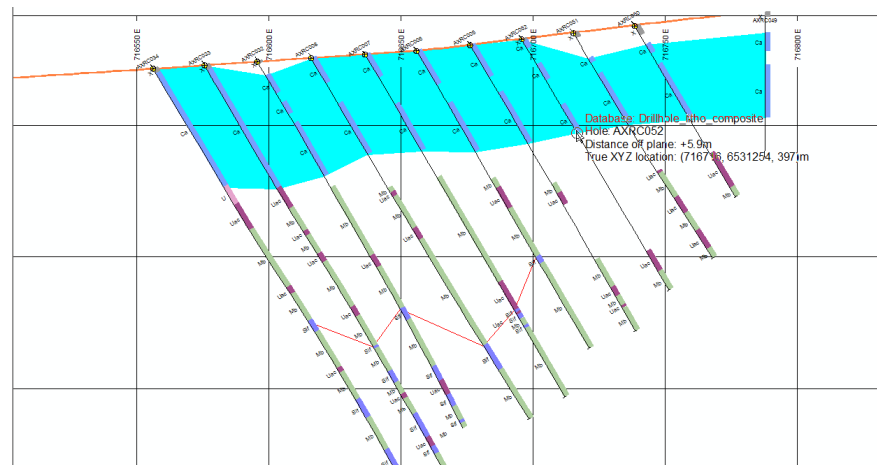


Digitizing more points will create smoother interpretations which will also create smoother wireframe surfaces.

To digitize geological polygon features on a section map:

1. Activate the Section_6531260N dataframe to open it in the Data View.
2. From the **Create Features** list, select **Alluvium**.
3. Digitize a polygon for the alluvium layer (the blue rock pattern marked with rock code Ca just below the collars), snapping to the top and the bottom of this layer.
4. To close the polygon, right-click and select **Finish Sketch**.

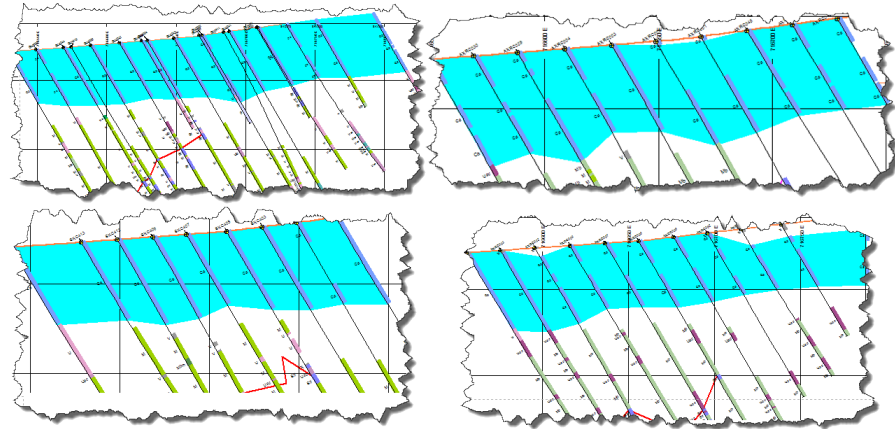
Figure 5.7 Alluvium and Top_Iron polygon interpretation on S6531260N.map



5. Repeat this process digitizing an alluvium polygon on the following section maps:

- > Section_6531285N.map
- > Section_6531310N.map
- > Section_6531331N.map
- > Section_6531380N.map

Figure 5.8 Alluvium polygon and Top_Iron polyline interpretations on section maps



In this lesson you:

- > Digitized polyline interpretations on section maps
- > Digitized polygon interpretations on section maps

Lesson 5.3 Editing Feature Classes

In this lesson you will:

- › Change the colour of a feature class
- › Add a feature class into the geodatabase
- › Digitize a polyline interpretation
- › Delete an interpretation from a section map
- › Delete a feature class from an interpretations geodatabase
- › View a summary of the interpretations on section maps
- › View the interpretations in the 3D Viewer

Editing Feature Classes

The Manage Interpretations Geodatabase tools in the Wireframing toolbar allow you to edit your geodatabase and all its associated feature classes and interpretations. Using these tools, you can add and delete features, or change the properties of an interpretation after it has been digitized.

To change the colour of a feature class:

1. On the Wireframing toolbar, click **Manage Interpretations Geodatabase**.
The Manage Feature Classes dialog opens.
2. Select Top_iron and click the **Edit selected feature class** button.
3. Click the **Symbology** button.
The Symbol Selector dialog opens.
4. From the Color dropdown menu, select Red from the Color palette.
5. Click **OK** on the Symbol Selector dialog.
6. In the Edit Feature Class dialog, click **Save**.
7. In the Manage Geostring dialog, click **OK**.
The colour of the Top_iron feature automatically updates on the section maps.

Next, you will add a new feature to the geostring.

To add another feature class to the geodatabase:

1. On the Wireframing toolbar, click **Manage Interpretations Geodatabase**.
2. Click the **Add new feature class** button.
The Add Feature Class dialog opens.
3. For **Name**, type **temp**.
4. Click the **Symbology** button and change the **Colour** to **pink**.
5. For **Type of feature stored**, select **Line Features**.
6. Click **Add**.
7. Click **OK**.


Next, you will digitize a polyline interpretation on S6531285N.map.

To digitize a polyline on a section map:

1. In the Project Explorer, double-click **S6531285N.map** to make it the current section map or right-click and select **Activate**.
2. On the Target Wireframing toolbar, click on the **Editor** dropdown menu and select **Start Editing**.
3. Click on the Snapping menu and ensure **Snap to DH Database** is selected and under the **Select DH Database** submenu, ensure **Geology** is selected.
4. On the **Create Features** tools tab on the right, ensure **temp** is selected.
5. Digitize a polyline line along the top of the first bright green rock pattern (this is the rock code "M").
6. To end the polyline, **right-click** and select **Finish Sketch**.
7. On the Target Wireframing toolbar, click on the **Editor** dropdown menu and select **Save Edits** and then **Stop Editing**.

Next, you will delete the polyline that you digitized on the section map.

To delete an interpretation:

1. On the Target Wireframing toolbar, click on the **Editor** dropdown menu and select **Start Editing**.
2. On the Geostings toolbar, click the **Edit Tool**.
3. Click to select the blue polyline interpretation.
Once selected it will be bounded by a cyan border.
4. On the Geostings toolbar, click the **Delete Selected Interpretation(s)** button ().
5. On the Target Wireframing toolbar, click on the **Editor** dropdown menu and select **Stop Editing**.



You can select multiple interpretations by holding the Control key on your keyboard.

Next, you will delete the temp feature from the geodatabase.

To delete a feature class from an Interpretations Geodatabase:

1. On the Target Wireframing toolbar, click on the **Wireframing Project** dropdown menu and select **Manage Geodatabase**.
2. From the **Feature Classes** list, select **temp**.
3. Click the **Delete selected feature class** button, then **Yes**.
4. Click **OK**.



It is not necessary to delete interpretations from a feature before deleting the feature. This was only done so you would learn how to perform both tasks.

To edit the vertices of an interpretation:

1. In the Editor menu select **Start Editing**.

2. Click the **Edit** Tool in the Wireframing toolbar and select any of your interpretations.
3. In the Wireframing toolbar, select the **Edit Vertices** tool.
4. Place your mouse cursor over a vertex on your selected interpretation and click and drag to move it. You can also remove or add a vertex by right-clicking and choosing the appropriate tool.
5. Right-click and select **Edit Vertices** to turn off the editor.
6. In the Editor menu, select **Save Edits** and then **Stop Editing**.

To view the geostring in the 3D Viewer:

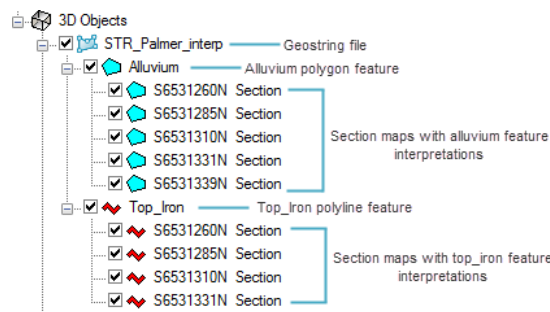
1. Open the Layout View for the Wireframing MXD.
2. From the Target Drillhole toolbar, click **Activate 3D Viewer**.


The Geosoft 3D Viewer opens.


3. From the **Add to 3D** menu, select **Interpretations Geodatabase**.
4. For **Geodatabase**, browse and select **Palmer_interp.gdb** and click **OK**.

Your section interpretations are displayed in the 3D Viewer. In the tree list, the geodatabase is listed as the top node under 3D Objects. Within the item, the sections that have interpretations for the feature class are listed.

Figure 5.9 Geostring in 3D Viewer tree list



 You can turn the visualization off for some of the other map groups to help you see the geostring file more clearly.

 Viewing the geostring alongside other data may help indicate which interpretations require editing. Changes to interpretations are done on the section map and the geostring will automatically update in the 3D Viewer.

In this lesson you:

- > Changed the colour of a feature class
- > Added a feature class into the geodatabase
- > Digitized a polyline interpretation
- > Deleted an interpretation from a section map

- › Deleted a feature class from an interpretations geodatabase
- › Viewed a summary of the interpretations on section maps
- › Viewed the geostring in the 3D Viewer

Lesson 5.4 Wireframing Interpretations

In this lesson you will:

- › Start a wireframing session
- › Join polygons to form a closed surface wireframe
- › Add ends to a closed surface
- › Save a surface to a geosurface file
- › View the total surface area and total volume for a surface
- › End a wireframing session

Wireframing

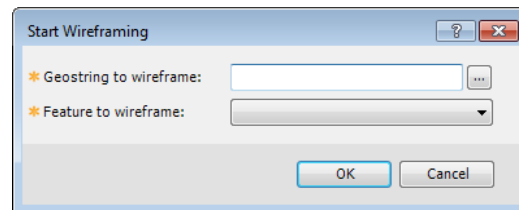
Wireframing is the process of joining 2D interpretations to form 3D surfaces. In **for ArcGIS**, this process is streamlined, efficient, and intuitive. After starting a wireframing session, you click interpretations in the 3D Viewer to form the wireframe. All the required tools are found in the Wireframing Tools panel that displays when a wireframing session is in progress.

To start a wireframing session:

1. From the **Wireframe** menu, select **Start Wireframing**.

The Start Wireframing dialog opens.

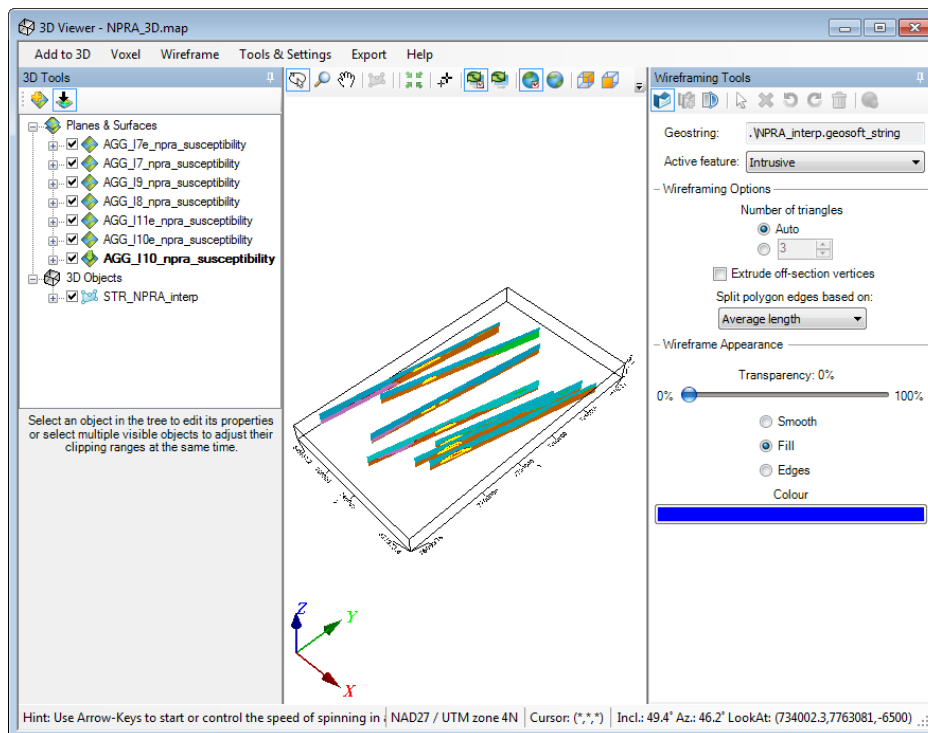
Figure 5.10 Start Wireframing dialog



2. For **Geodatabase to wireframe**, select **Palmer_interp.gdb**.
3. For **Feature to wireframe**, select **Top_iron**.
4. Click **OK**.

The Wireframing Tools are displayed on the right side of the 3D Viewer; note that you are in Wireframing Mode. The top_iron interpretation polygons are bounded in yellow.

Figure 5.11 3D Viewer with Wireframing Tools displayed



To create an open surface by wireframing polylines:

1. Click a polyline in the 3D model.
The selected polyline will be bounded by a cyan border.
2. Click an adjacent polyline to wireframe to the first polyline you selected.
You now have the beginning of a wireframe surface.
3. Continue selecting the remaining polylines to add them to the surface.



Wireframe bodies and surfaces are automatically assigned the same colour of the feature class. You can optionally change the colour of the wireframe created from an active feature.

To create a closed surface by wireframing polygons:

1. On the Wireframing Tools panel, change the **Active feature** to **Alluvium**.
2. In the 3D Viewer, **click** to select the closest polygon to you in the 3D view.
The selected polygon is bounded by a cyan border.
3. **Click** the adjacent polygon.

You now have the beginning of a wireframe body. Both ends are bounded in cyan meaning you could continue adding to the wireframe from either side.



If you have interpretations that belong to the same active feature that need to be treated as separate wireframes, click the Add New Wireframe Body button. For instance, the same rock type may occur in two separate areas (separated by a fault) or may bifurcate.



While wireframing, you can use the Undo/Redo buttons on the Wireframing Tools panel or use the Ctrl+Z and Ctrl+Y shortcut keys, respectively.

To create ends on the wireframe body:

1. On the Wireframing Tools panel in the 3D Viewer, click **Create Wireframe Ends**.

The options on the Wireframing Tools panel update and the two open ends in the wireframing body are bounded in yellow.

2. For **Ends Options**, select **Conical**.
3. Click an open end to close it.
4. Click the remaining open end to close the body.



Ends must be added before saving the wireframes to a geosurface file. If you attempt to save before adding ends, you will be asked if you would like to auto-close the wireframe with flat ends.

Geosurfaces


Geosurfaces are 3D vector files that store the results of wireframing or isosurface creation. The files can be saved and shared with others, and also used in voxel math expressions and VOXI constraint building.

Geosurfaces store the following information:

- > XYZ locations
- > Coordinate system
- > Attributes including feature names and colours

The coordinate system for the geosurface is automatically set based on the geostring used to create the wireframe. Isosurfaces, created from a voxel, are also stored in geosurface files.

To save the wireframes to a geosurface file:

1. On the Wireframing Tools panel, click the **Save Wireframes as Geosurface File** button ()
2. For **File name**, enter **palmer** and click **Save**.

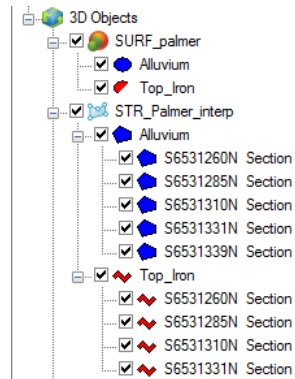
The surface Alluvium is added to the 3D Objects tree.



You also have the option of saving the Wireframe as a MultiPatch Feature class in the Interpretations Geodatabase.

3. Under Wireframing Tools, change the **Active Feature** to **Top_iron**.
4. Click **Save Wireframe as a Geosurface File**.
5. Select the **palmer.geosoft_surface** file and click **Save**.

The surface Top_Iron is added to the same geosurface and is added to the 3D Objects tree.

Figure 5.12 palmer and Top_iron geosurface in 3D Viewer tree list

Next, you will view the Total Surface Area and the Total Volume for the .

To view the Total Surface Area for the Top_iron surface:

1. In the **3D Objects tree**, select the **Top_iron** surface.
2. Click **Properties** in the Attributes tab.

The Geosurface Properties dialog opens.

Information about this surface, including Total Surface Area is given. The transparency, rendering, and colour can be changed to alter how the surface displays.

To view the Total Surface Area and Total Volume for the surface:

1. In the **3D Objects tree**, select the .
2. Click **Properties** in the Attributes tab.

The Geosurface Properties dialog opens.

Information about this surface, including Total Surface Area ("Area") and Total Volume ("Volume") are displayed.

3. Click **Close**.

To stop wireframing:

- From the **Wireframe** menu in the 3D Viewer, click **Stop Wireframing**.
The Wireframing Tools are removed from the 3D Viewer.

In this lesson you:

- Started a wireframing session
- Joined polylines to form an open surface wireframe
- Joined polygons to form a closed surface wireframe
- Added ends to a closed surface
- Saved a surface to a geosurface file
- Viewed the total surface area and total volume for a surface
- Ended a wireframing session

