Tensor Research

ModelVision™ v17.0 Tutorials

Contents

Tutorial 1 – Introducing ModelVision

Introduction

The purpose of this tutorial is to provide an overview of ModelVision. Subsequent tutorials illustrate basic and advanced features available in the software.

This tutorial concentrates on the import and display of geophysical data in ModelVision. It is recommended that this tutorial be reviewed briefly, and then progress onto other tutorials that address more advanced topics.

Within the tutorials are individual sections on:

- How to use the tutorials
- Import of data and supported formats
- Displaying data in a variety of forms and how to control the displays
- Development of basic models
- Advanced modelling
- Regional and residual design and manipulation
- Inversion and its uses
- Use of AutoMag

Aim

This tutorial is aimed at providing an overview of ModelVision and presenting the concepts associated with importing and displaying geophysical data. A number of basic import data formats are supported for single and multiple line types and these are presented with examples. Additional import descriptions are given for grid, point and borehole datasets.

Intended User Level

This tutorial is intended for viewing by beginners and inexperienced users to provide an overview for interested parties and a good refresher for experienced users who have not used ModelVision for some time.

ModelVision Overview

ModelVision is a geophysical modelling package for the display, analysis and modelling of magnetics and gravity data. ModelVision provides a powerful suite of data display tools for maps and cross-sections. These views enable you to visualize your models in the context of field data. You can import other data sets such as drillhole logs, radiometrics, basement surfaces, terrain and vector graphics for visualization in the modelling context.

Models include conventional body types such as dykes, tabular, polygons, spheres, ellipsoids and cylinders. Complex 3D faceted shapes can be built from cross-section and map views. This combination of shapes allows discrete anomalies to be analyzed as simple mineral exploration targets or modelling of basins for petroleum exploration. Bodies can be combined to build complex models such as intrusions within a layered environment.

With the synthetic survey option you can create an atlas of magnetic profiles and maps to simulate expected responses. These atlases can be used for pre-survey planning or assistance in the interpretation process.

Generalized geophysical inversion allows you to optimize forward model solutions without spending large amounts of time on what is normally regarded as minor geological refinement. If you have physical property constraints or drillhole control, you can use inversion to better define the shape of a particular anomalous source.

The concept of In-line filters allows you to look at your data and model results through the same filter. A first vertical derivative filter is used to better separate overlapping anomalies and assist with the removal of the regional magnetic field. Significant improvements in depth accuracy can be expected with this technique. In noisy areas of laterite, an in-line upward continuation filter can be used to help reduce the noise.

When examining your models in a map view, you can enable a range of display types such as images, contours, stacked profiles, grid profiles, sample points, drillholes and vector graphic files. Cross-section views are used to produce detailed depth sections where you can simultaneously model gravity and magnetic data, display drillholes, plot individual and composite body responses, turn on in-line filters and display topography. Map and section views can be plotted to scale or at A4 presentation scale. A simple perspective presentation is included for the display of models and drillhole data.

Regional magnetic and gravity fields can be implemented on individual profiles or as two dimensional polynomial surfaces. These surfaces are supported directly in the modelling and can be used as a tool for simple regional-residual separation.

Line oriented convolution and FFT filters are implemented for standard geophysical tasks. These help to analyze the information content of your data prior to modelling. A powerful multi-channel display option is included to help with the visualization of the filter output. You can use this tool to zoom and pan long lines of data. A set of 2D convolution filters compatible with the ER Mapper image processing system are also included.

Useful utilities such as gridding, synthetic surveys, traverse extraction, channel calculator, interpolation, data maintenance, statistics and reporting provide a well rounded environment for the geophysical interpreter. Grids of virtually any size can be computed from either point or line data. The calculator interface provides a simple visualization for manipulation of channel data in line, point, grid and drillhole data sets. You can also use it to sample gravity survey grid data onto aeromagnetic flight lines.

A CAD tool (Layout Window) is included that allows you to layout your interpretation in a composite view. You can add interpretive information as overlays to any of the standard views. A comprehensive suite of drawing and annotation tools allow you to produce professional interpretation displays for inclusion in your reports. Use any size page that is supported by your printer or plotter.

You can import data from a variety of data standards such as ER Mapper, Geosoft, ASEG-GXF or use the external data base link option to design direct links to your in-house database system. Similarly you can export your profile and grid data in a variety of industry standard formats.

Project Definitions

ModelVision uses projects to manage various analysis exercises and a file called MVPROJ.INI records your entries for later use. Initially, a project is required to define the various properties of the data. Binary session files (.SES) can also be used to store data, models and displays if required.

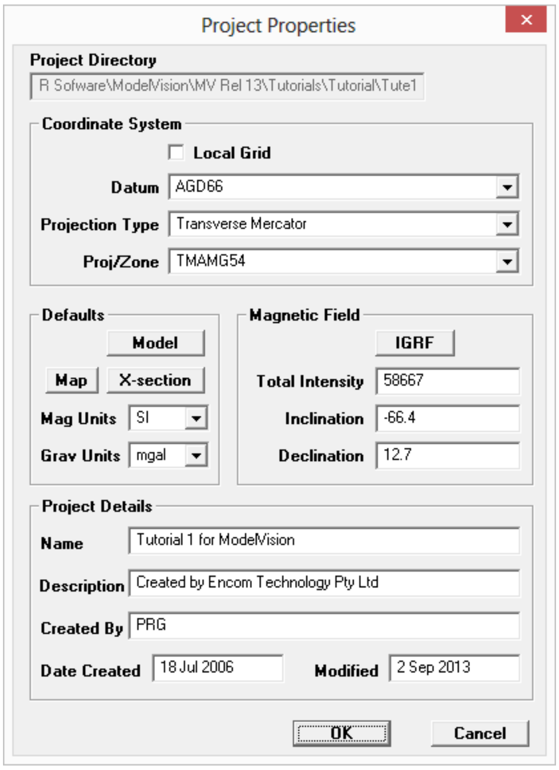
Project files contain information relating to:

- Data projection, datum and spheroid
- Location of the project (a project folder)
- Magnetic field information for modelling
- Units to be used

- Default information regarding models and displays
- Project description information

Select the **File>New>Project** option and enter the information as required. As an example, assume the required parameters are as below:

Datum:	Australian Geodetic Datum 1966 (AGD66)
Projection:	Transverse Mercator
Zone:	AMG Zone 54 (TMAMG54)



Note that the entries for the Earth's magnetic field are set to be:

Field Intensity :	58667
Inclination:	-66.4 degrees
Declination:	12.7 degrees East

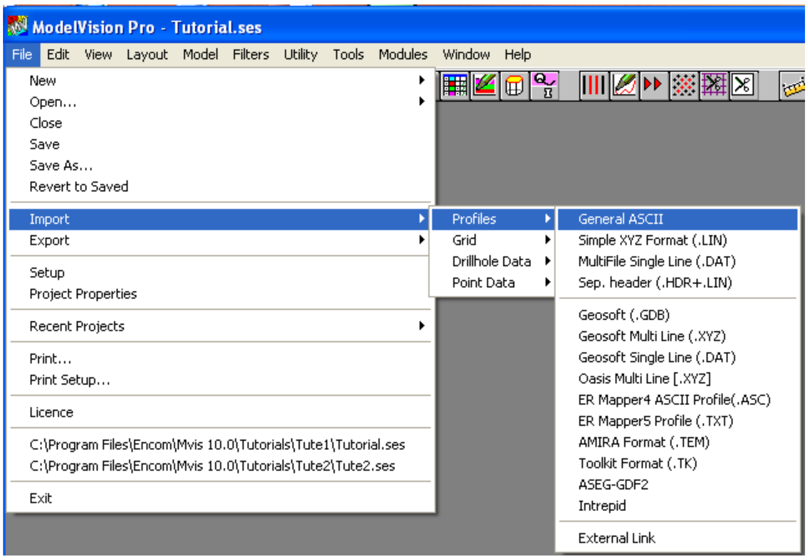
Once a project is defined in ModelVision, it is then possible to import line, grid, point or drillhole data.

Importing Data into ModelVision

ModelVision can import the following data types:

- Binary session files of saved data (program snapshots)
- ASCII profile data
- ASCII point data
- ASCII drillhole data
- Grids of various formats
- External data from databases (e.g. Geosoft, Intrepid etc)

ASCII data to be manipulated, displayed and modelled can be loaded by the **File>Import** option. Selection of this option displays a second menu list containing ASCII formats and other possible sources of data. This list for survey lines is shown in the following screen:



ModelVision can import ASCII line oriented and point based data. Grid files, drillhole data and information held in databases or external software programs can also be imported.

Profile Data

Eight ASCII line oriented data file types are available for import. These fall into the following two categories:

Single line – Files that contain multi-column data but with columns relating to only one profile or traverse.

Multi line – Combined traverses of multi-column data can be read using this style of input. A number of standard file formats are available and suitable for importing into ModelVision.

Single-line formats

File types that contain data from single data traverses and are supported by ModelVision include:

- TOOLKIT format files
- Geosoft single format files
- ER Mapper ASCII traverse files

Data contained in these formats have one or more header lines followed by data records. Usually the header record(s) contains information about the line name, data columns or title text. The data records may be formed by a number of columns that contain the location and variable channels of input data. The various data values may lie one under the other for each record or, depending on the format, the values may be placed in any column of the record provided their order is maintained.

Multi-line formats

Combined traverses of multi-column data can be read using this style of input. A number of standard file formats are available and suitable for importing into ModelVision. Supported Multi-Line data file formats include the following:

- Simple XYZ Encom line format (LIN)

- XYZ line format but with a separate header (LIN and HDR)
- Multi line Geosoft XYZ file (XYZ)

These data formats, similarly to the single line formats, may have descriptive header records followed by data arranged in column or open format. Alternatively, in the case of the multi-column dataset with no header, an external header can be used.

Drillhole Data

Downhole data can be imported into ModelVision by reading a file that contains X, Y and Z data location coordinates plus sensor channels. The X and Y coordinates refer to horizontal positioning as for a conventional line oriented dataset and the Z data contains depth information beneath a nominated datum. The default sense of direction is positive down for increasing depth.

Point Data

Certain data collection procedures acquire data at random locations. Examples of this are regional gravity surveys or geochemical assaying projects. With these data types there are no line-related methods of collating the data and so each record of the input data file is dealt with individually. There is a requirement that point data sets have an associated number similar to the line specification. This number is referred to as a ‘point set identifier’ and is used to associate records with similar properties (for example, gravity datasets, gold assay datasets, magnetic solution points etc). The input data type is of a definable XYZ format where the X and Y coordinates specify the easting and northing location of the sample point. A number of Z channels corresponding to measurements at each sample location can be loaded.

Grid Import

ModelVision is capable of importing various grid formats, some of that are written in ASCII although most are in binary. In most grid formats, the first few lines of the file are assigned to a header. The header contains information relating to origin, extent, numbers of rows and columns, grid mesh size, rotation etc.

ModelVision supports the following grid formats:

- Encom
- ER Mapper (ERS and image BIL file - Versions 3.x, 4.x and 5.x)
- INTREPID (similar to ER Mapper grids)
- Geosoft binary grid format
- ASEG GXF format
- Geopak binary grid
- USGS binary grid format.

Displaying Data in ModelVision

ModelVision displays data in Map, cross-section, 3D and graph formats selected from the View menu. When you first load line, point or grid data into ModelVision it will open a view of the data for you.

Multi-track is the only view that does not display the model, as it is used for general multi-channel data presentation. Each view is opened in a separate window and you can have multiple views of the same type. A change that is made to a body in one window will be reflected in other views. For example, if a tabular body is shifted in a profile cross-section it will also shift along the line of section in any map view.

The stacked profile, contour and grid profile display formats are map presentations. These formats allow both data and models to be shown together for interactive modelling. The X-section display format allows data channels plus a cross-section of the modelled bodies to be displayed.

In each of the model display windows, bodies created and edited are shown in their correct geographic location and orientation. As editing functions are performed, other model display windows are updated to reflect the changes.

The individual presentation formats available are described below. With each presentation format is an interactive tutorial associated with a dataset. To run the tutorial, execute ModelVision and then import and present the data using this display format.

New Layout

A new Print Layout can be displayed using this menu option which allows you to organize your active map, x-section or perspective display windows into a report style with titles and captions.

Multi-Track

Multi-channel display windows are available to display data channels imported into or created in ModelVision. Many options are available to control display attributes.

X-Section

Model cross-section and data display of magnetic or gravity channels. Additional channels such as residual or elevation tracks can be included in the displayed tracks.

Perspective

A perspective view of models created in model windows.

Map>Stacked Profile

A map view where the selected data channels are displayed for all loaded lines.

Map>Points

A map view of points representing measurement location not contrained to lines.

Map>Contour

A contour map view of a ModelVision grid or an imported grid.

Map>Grid Profiles

A stacked profile map view of a grid. The data is derived from the grid as a series of column or row stacked profiles. This presentation is useful for modelling purposes and is very fast to display.

Map>Grid Images

A basic image display of a selected grid of data. Sun angle enhancement can be applied to highlight anomalies. Image presentations are always presented first in map displays with vector style displays drawn onto the image.

Multi-Track Display

Multi-channel display windows are available to display data channels imported into or created by ModelVision. Many options are available to control display attributes.

Note Throughout the tutorials you will be required to load certain datasets. If you are running an unlicensed or evaluation version of ModelVision, this will not be possible as this function is disabled for demonstration purposes. You will, however, be able to use the **File>Open** option and load an equivalent dataset from a .SES (binary) project file. If for example `TUTE.TK` is required, there will be a `TUTE1.SES` file that can be loaded with the **File>Open** option to import the data.

Tutorial Procedure

For this part of the tutorial, you will investigate both importing and display of loaded data into a Multi-track display. For this exercise the data file `TUTE1.TK` will be used. This data file is similar to that shown below:

Line 45670N Bodalla West (1987)

EAST	NRTH	FID	MAG	K	U	TH
442420.0	7829846.0	7864.0	2008.29	86.80	23.90	41.80
442419.9	7829855.0	7863.0	2008.42	86.80	23.90	41.80
442419.8	7829864.0	7862.0	2008.54	86.80	23.90	41.80
442419.6	7829872.0	7861.0	2008.67	86.80	23.90	41.80
442419.5	7829881.0	7860.0	2008.81	86.80	23.90	41.80
442419.4	7829890.0	7859.0	2008.95	86.80	23.90	41.80
442419.3	7829899.0	7858.0	2009.09	86.80	23.90	41.80
442419.1	7829907.0	7857.0	2009.23	86.80	23.90	41.80
442419.0	7829916.0	7856.0	2009.37	64.90	12.00	47.70
442419.0	7829926.0	7855.0	2009.52	64.90	12.00	47.70
442419.0	7829936.0	7854.0	2009.66	64.90	12.00	47.70
442419.0	7829946.0	7853.0	2009.82	64.90	12.00	47.70
442419.0	7829956.0	7852.0	2009.96	64.90	12.00	47.70
442419.0	7829966.0	7851.0	2010.11	64.90	12.00	47.70
442419.0	7829976.0	7850.0	2010.23	64.90	12.00	47.70
442419.0	7829986.0	7849.0	2010.34	64.90	12.00	47.70
442419.0	7829996.0	7848.0	2010.42	52.50	24.90	20.40
442419.0	7830004.0	7847.0	2010.48	52.50	24.90	20.40
442419.0	7830011.0	7846.0	2010.54	52.50	24.90	20.40
442419.0	7830019.0	7845.0	2010.59	52.50	24.90	20.40

Step 1

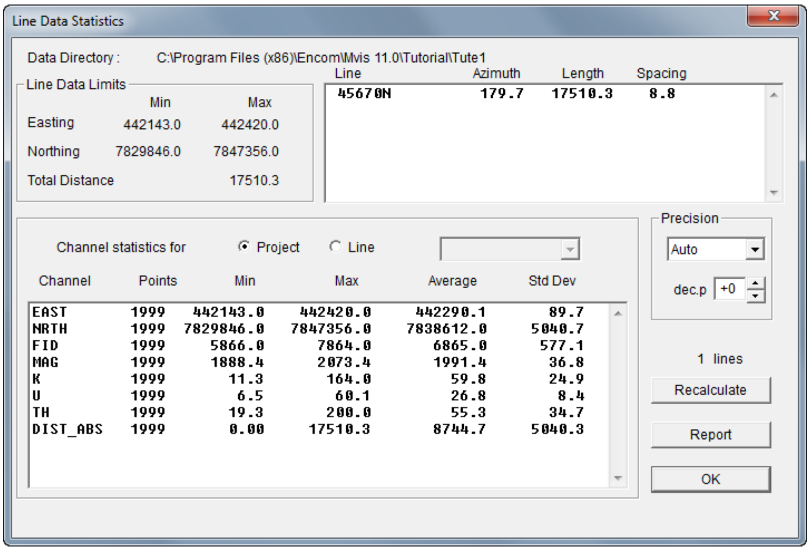
Use the **File>Open>Project** menu option to open the project file for this tutorial located in: `..\TUTORIAL\TUTE1`

Using the **File>Import>Profile>Toolkit Format** menu option import the file `TUTE1.TK` (located in the same folder as the project file). This single line of data has been extracted from a typical airborne survey and is a sequential line file containing 1999 data records. Contained in the file are AMG coordinates, Fiducials, Magnetics and the spectrometer channels of Potassium (K), Uranium (U) and Thorium (Th).

The aim of this exercise is to present the data in various forms and to illustrate the ease with that various relationships between multi-channel information can be derived. The following tasks should be undertaken to make you familiar with analysis of this type of data.

Step 2

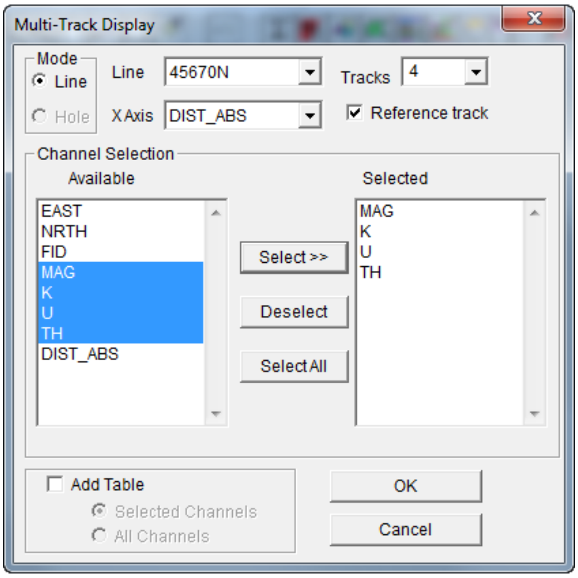
After loading, the line data statistics dialog will appear which can be manually accessed via the **Utility>Statistics>Line Data** menu option. This dialog shows you line extents, channels ranges etc. Note that a distance (DIST_ABS) column has been computed automatically from the northing and easting data. This channel is used as the horizontal scale for plotting and display purposes.



When you close the statistics dialog, you will be given the option to display selected channels in a stacked profile map. This is useful when loading a full survey, but not pertinent for a single line of data. Cancel this option and go to the next step.

Step 3

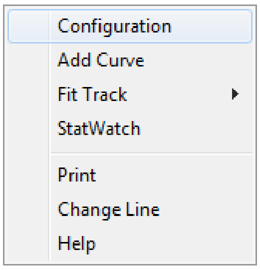
Display the magnetics and radiometrics using the **View>Multi-Track** menu option to check the integrity of the data. Activate the Reference track check box to allow the data in the reference track to be scrolled, zoomed and panned.

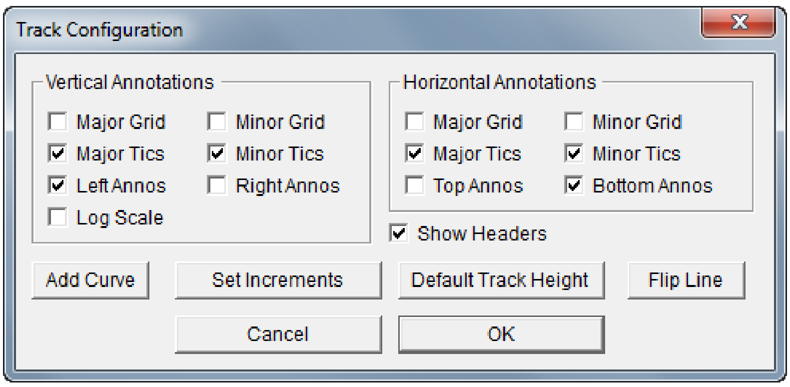


When the channels are selected the profile will be displayed. Experiment with the Zoom In/Zoom Out and Pan toolbar options on one of the channel tracks before returning the track to its full size by positioning the cursor over one of the tracks and clicking the right mouse button to display the pop-up menu options. Select the Fit **Track>Horizontal** option to adjust the appearance of the track.

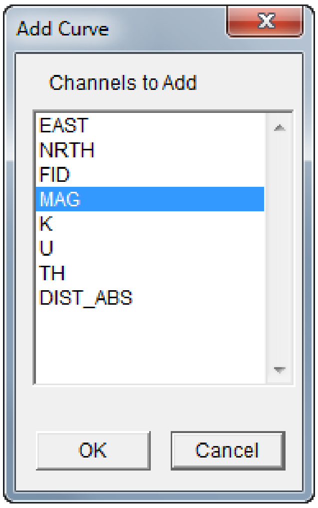
Step 4

Position the cursor in the Th (Thorium) track and click the right button. A pop-up menu will appear with various options available including the Configuration utility. Select this option and a new dialog called the Track Configuration will appear which controls the display attributes of tracks. This dialog can be used to add a new curve to a track. The Add Curve option is also available in the track pop-up menu.



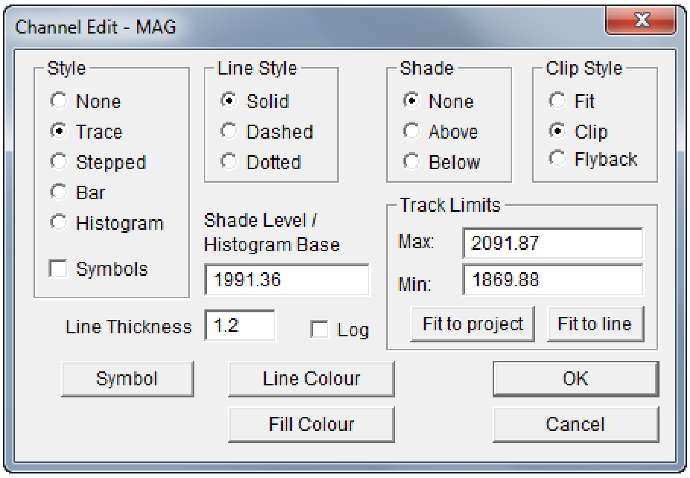


Use the **Add Curve** button to add a second channel of Magnetics.



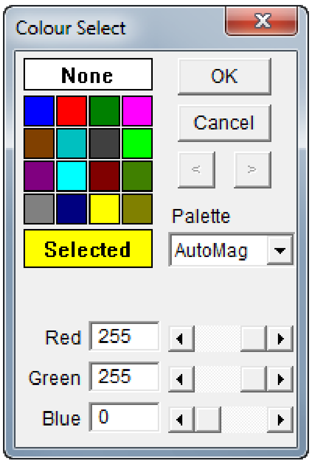
Step 5

To modify the appearance of the traces in each track, place the cursor in the left hand ‘curve-name’ box of the original MAG track and double click the left mouse button. A configuration dialog will appear as below.



Modify the various options as required. Remember that the modifications being made apply only to the selected curve. The fill or trace colour can be altered using the dialog.

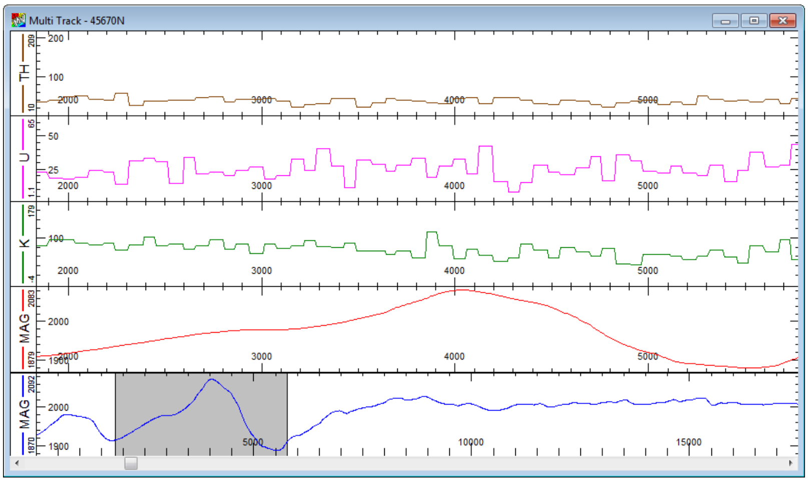
To display the station positions along each track activate the Symbols check box on this Channel Edit dialog and select the Symbol button to specify the appearance of the symbols.



Also within this dialog the above and below colours for the profiles can be altered as can scaling of the various channels.

You may wish to experiment with some of the other settings controlling the display of the curves. Remember also that you can 'drag-and-drop' curves from one track to another and resize the tracks to enlarge one track over another.

Zooming, scrolling and panning are controlled by the **Reference track** (the greyed track at the base of the display), the scroll bar across the bottom and the **Fit** button on the Display toolbar. Zooming in to an area on the reference track will resize the other tracks in the display to this zoomed area but the reference track will remain at the original scale with the zoomed area highlighted.



Place the cursor in the Reference Track and click the right mouse button to display the **Fit Track** pop-up menu item. This option allows the track to be fitted in both vertical and horizontal directions.

Clearing Memory and Data

When you are finished processing or interpreting a dataset in ModelVision, you can save the data, display formats etc in a binary session file (use the **File>Save** or **Save As...** options). This operation saves a project but it does not clear memory, data and models etc. To clear the memory and reset defaults to prepare for a new dataset, use the **File>Close** or **File>New** option.

Display a Cross-Section

This part of the tutorial involves loading a multi-line dataset and displaying a selected line as a cross-section. Cross-sections (referred to in this reference manual and the software as "x-sections") are used primarily for modelling (either gravity or magnetics).

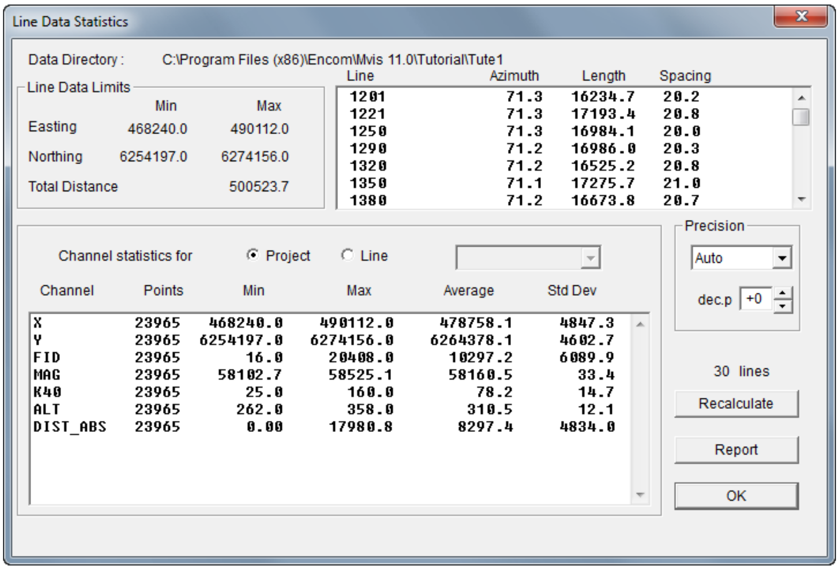
Step 6

Import the file `TUTE.HDR` (located in the `..\TUTORIAL\TUTE1` folder) and use the **File>Import>Profiles>Sep. Header (HDR + LIN)** option. This import style is used for large data files where it is not desirable to edit the data file, but simply to create a Header file that describes the contents of the columns of the data. As an example of this format, examine the `TUTE.HDR` and `TUTE.LIN` files in an editor.

Note If ModelVision is running in Demonstration Mode, import the file `TUTE1.SES` using the **File>Open>Session** option.

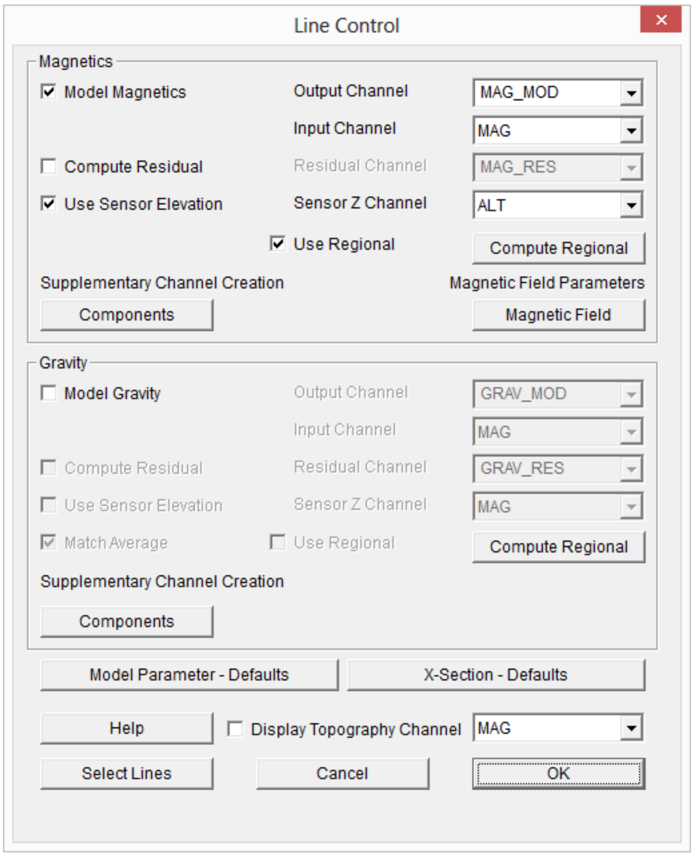
Step 7

Once the data has been imported, examine the statistics in the Line Data Statistics dialog which automatically appears. To open this dialog at any other time in ModelVision use the **Utility>Statistics>Line Data** menu option.



Step 8

Prior to starting modelling, you need to provide ModelVision with information about the channels that will be used to define the sensor type(s) and sensor location. Use the **Model>Line Control** menu option to open the modelling control dialog. Turn on the *Model Magnetics* and *Use Regional* check boxes magnetic data input channel is set to *MAG* and the Sensor Z Channel is *ALT*.

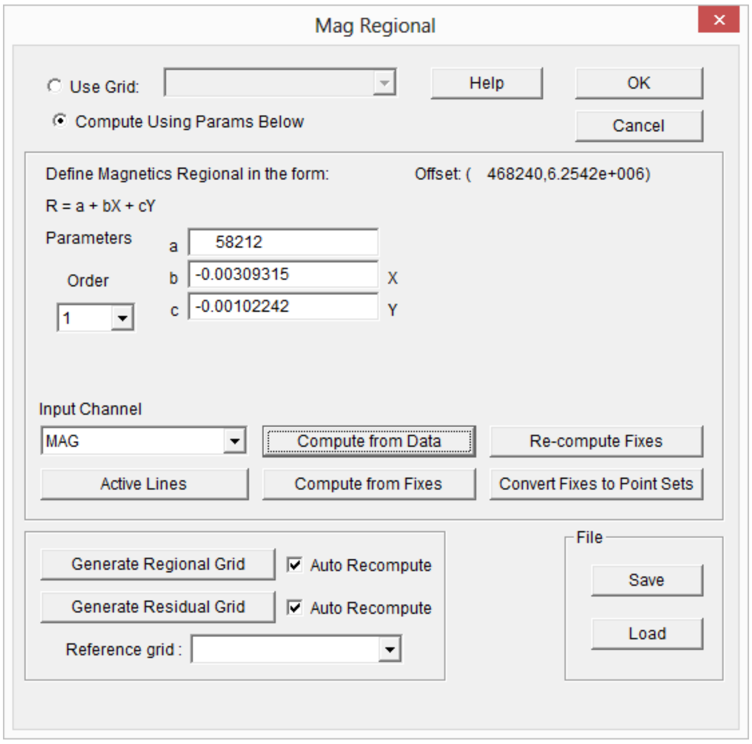


Click on the **Select Lines** button in the **Line Control** dialog and activate Line 1500 for modelling.

Now activate the **Use Regional** tick box and select the Compute Regional button in the Line Control dialog to access the regional setup (see below). Select the *Active Lines* button and select all lines available in the list for the regional. Make sure that the regional *Order* is set to 1, then select the *Compute from Data* option. This fits a first order polynomial (slant line) to the data. This line will be too high for normal modelling and you will need to apply a manual adjustment to the level using the fix points that will be visible in the cross-section view.

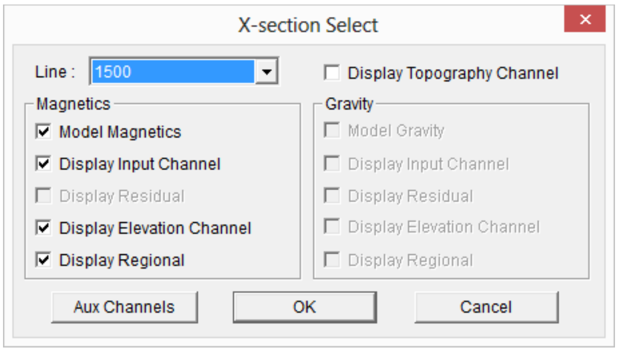
You can experiment with the polynomial order and you will see the formula and coefficient fields change each time. You need to select *Compute from Data* each time you change the polynomial *Order*.

Note that you must activate the regional for each line you wish to model with a cross-section, but this can be done each time you wish to activate a new line. You can also compute regionals for lines that are not modelled. In this exercise we will only model the one line.

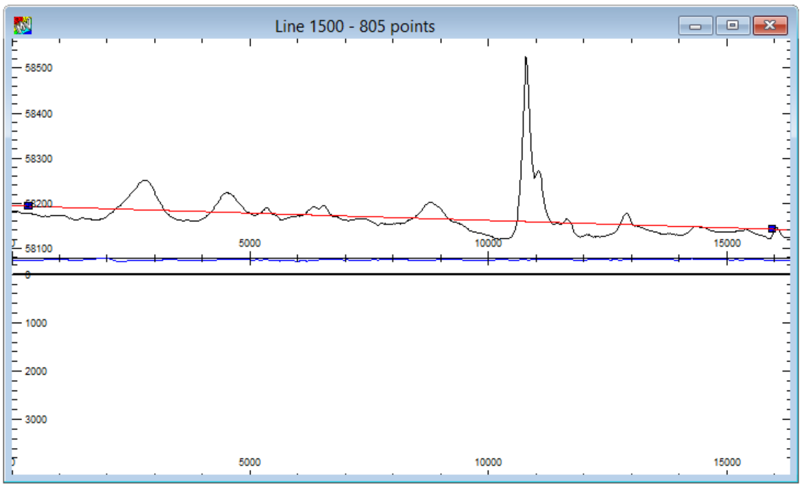


Step 9

Use the **View>X-Section** menu option to select Line **1500** and ensure that the *Model Magnetics*, *Display Input Channel, Display Elevation Channel and Display Regional* check boxes are selected.

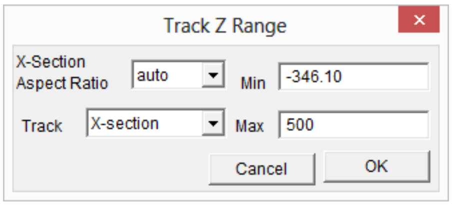


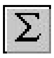
Once the cross-section is displayed, a profile will be shown with observed data in the top track and a 'slice' vertically through the earth beneath. This lower track represents a cross-section of the subsurface where bodies can be created and edited for modelling. In this example we are modelling depth below the ground surface (black line) and the altimeter channel provides us with a vertical position for the sensor (blue line). In most airborne surveys you will have access to a sensor elevation channel and a ground surface elevation channel and this allows your models to have absolute elevation values.



Note that the first order regional magnetic field is shown as a red sloping line with two square fix points positioned at either end of the profile. The fix points can be selected and by holding the left mouse down, dragged to a new position. When model computation is in immediate mode, the regional will be recalculated with each new position. For more information on managing 2D regionals refer to Chapter 6 in the **Geophysical Interpreters Guide**.

Change the depth of your cross-section to 500 metres as this will provide more sensitivity for visual editing of the models. To access this control, position the cursor inside the section below the thick black horizontal line representing the elevation channel and use a right-mouse click to access the configuration pop-up menu and select the *Track 0>Vertical Range* sub-menu.





The **Compute** button on the Model toolbar is used to compute the forward model response curve after a series of edits.

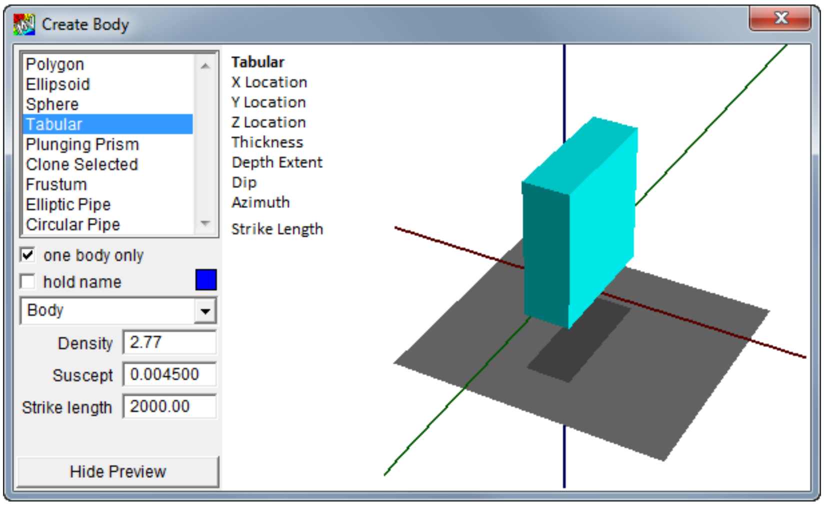
The Manual/Immediate Mode button allows the user to toggle between **Manual** and **Immediate** compute modes.

When highlighted green the button is in **Immediate** compute mode.

Set the computation mode to **Immediate** for this tutorial.

Adding Models to a Cross-Section

Once a cross-section is created it is possible to then introduce individual bodies to form a model. The creation of bodies is accomplished from the **Create Body** button on the Model toolbar. This dialog can also be opened from the **Model>Body Operations>Create Body** menu option.



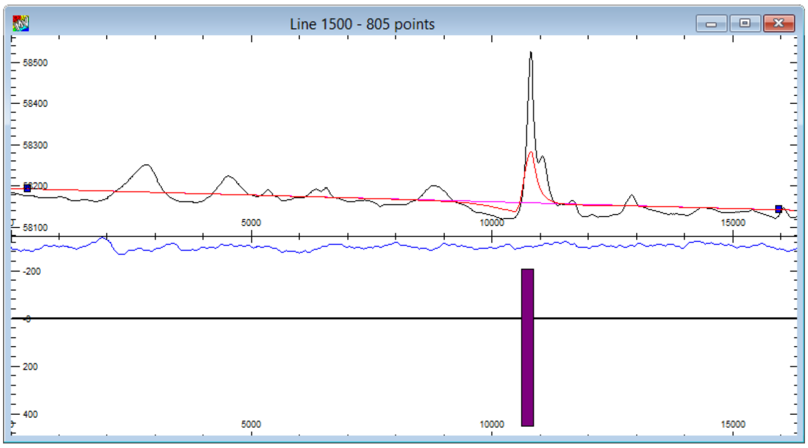
Any of the listed body types can be created in a cross-section, but the *Tabular* body is used in this example.

Step 10

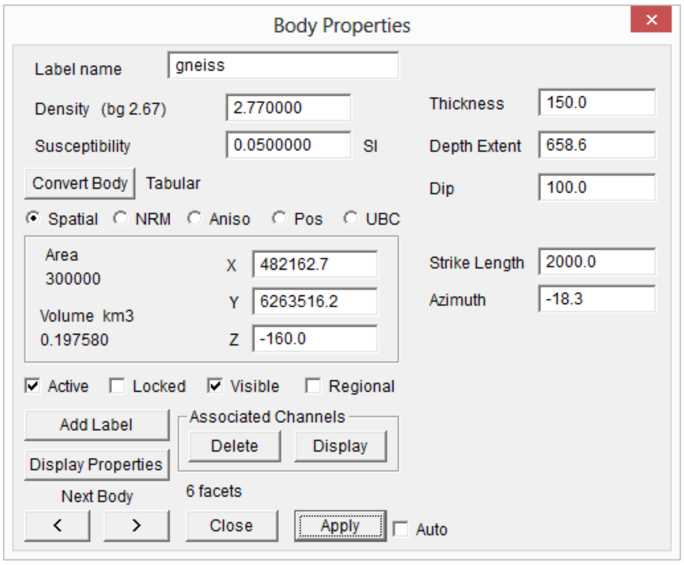
Highlight the *Tabular* body shape from the **Create Body** dialog and click on the colour square to change the colour of the body. Also enter a susceptibility value of *0.01* and a strike length of *2000m*.

Step 11

Move the cursor into the cross-section window and directly beneath the large anomaly drag out a narrow but deep rectangle and release the mouse button to reveal a body similar to that illustrated below.



Double mouse click on the tabular body in the cross-section window to display the **Body Properties** dialog. Edit the *Susceptibility, Thickness*, *Depth Extent, Dip* and *Z* value of the body to match that illustrated in the dialog below and click **Apply** and **Close** to return to the cross-section window.

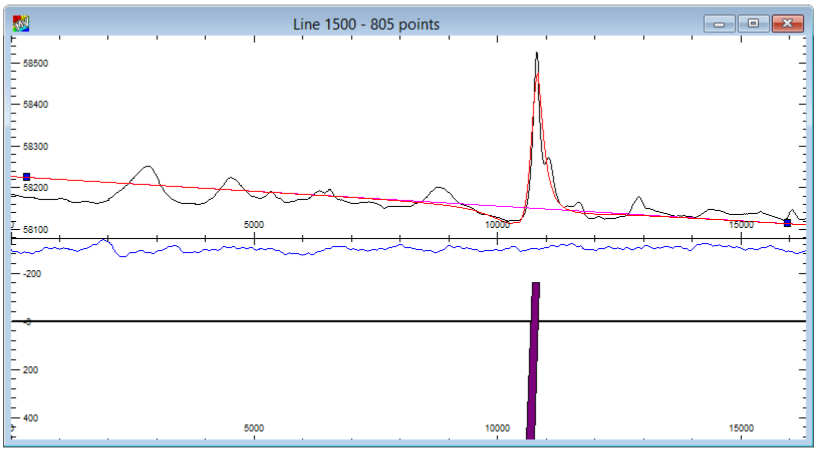


Step 12

Adjust the regional fix points to set the model regional level. After computing the resulting body should like similar to that illustrated below. To move the body along the x-axis, left mouse click on the body to activate the edit mode (identified by the black nodes in the corner of the body) and then left click (hold) and drag the body to the new position.

Step 13

Save this session as a new file called `TUTE_NEW.SES` by selecting **the File>Save As…** menu option. Save this file to the same folder as the Project file i.e. `..\TUTORIAL\TUTE1`.



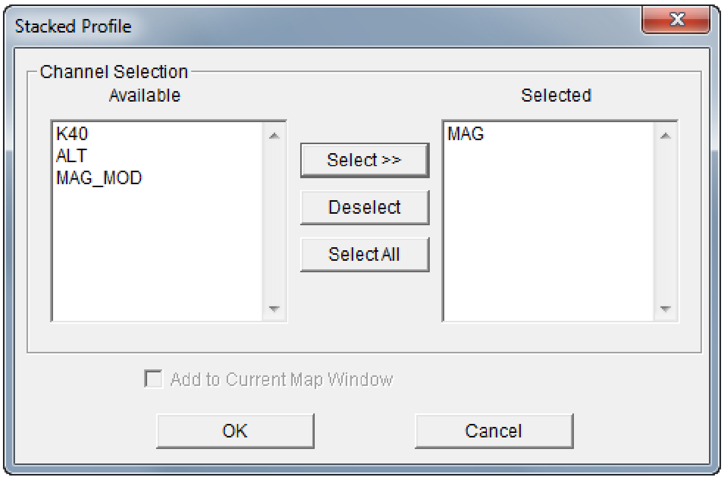
Stacked Profile Displays

A stacked profile of data lines can be produced in ModelVision using the **View>Map>Stacked Profile** option.

For this tutorial exercise, you can continue with the session already saved or clear all windows by selecting the Close button in the top right hand corner of each open window.

Step 14

After importing the dataset (see above) selecting the **View>Map>Stacked Profile** option will present a dialog allowing the selection of channels contained within the dataset to be displayed in a stacked profile display.

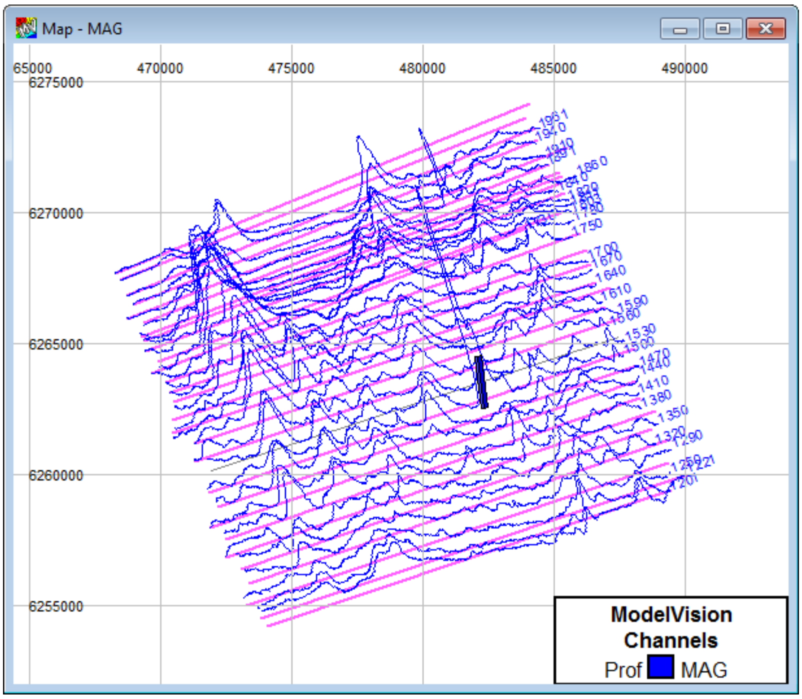


One or more channels can be selected for display. Highlight the MAG channel in the *Available* list on the left hand side of the dialog and click the **Select** button to move this channel to the *Selected* list on the right hand side.

If an existing contour or stacked profile window is open, a check box provides the option for the requested stacked profiles to be added to this window. For this example there are no other map windows open yet so it will create a new map window for the stacked profile.

Step 15

Once the channel(s) is selected, select the **OK** push button and a map of the dataset lines will be displayed in a new map window.



If the initial view of the map is not correctly positioned or scaled use the **Zoom In/Out**, **Pan** and **Fit** buttons of the Display Toolbar. By default, the viewing area of the stacked profile map and the scaling of the selected channel(s) are automatically computed.

These attributes can be reset after the stacked profile window is opened. The stacked profile map can be configured using the Map Configuration dialog, accessed from the pop-up menu option.

Configuring Map Displays

Step 16

Stacked profiles are an example of a map display. Other map displays include contours, images and grid profiles.

Common to all map displays is a **Map Configuration** dialog that is used to control the many display parameters. Once a map display has been created, other map presentation types such as contours, drillholes or backdrops can be added, deleted or configured. Each map display has its own configuration dialog, but access to these controls is the same for all map types. The stacked profile dialog is shown below.

Step 17

Position the cursor over the map window containing the stacked profile and click on the right mouse button to display a pop-up menu containing the **Configure Layers** option. Select this menu item and the **Map Layers** table will appear. Ensure that the *Stacked Profile Channels* object type is selected for the *MAG* channel; (this channel should be displayed in the table beneath the various Object Types) and right mouse click on the **Configure** option from the pop-up menu that appears.

Step 18

Adjust the *Z-Scale*, *Baseline Value* and *Shade Above* values as illustrated in the above dialog. Click on the **Colour** button for the *Shade Above* option and specify a *light blue* colour. Click **OK** on both dialogs to return to the map window containing the stacked profile and observe the changes made.

Note If you are using the original data used in the previous exercise for displaying a cross-section you will observe the modelled tabular body in the map window. This can be hidden from view by turning off the **Body** tick box in the *Visible* column of the **Map Layers** table.

Note

The Baselines for the survey will be highlighted in pink indicating that these lines are still active from the modelling performed in the previous exercise. These can be deselected from the **Select Lines** button in the **Model>Line Control** menu option or else select the Active Lines button from the Modelling Toolbar.

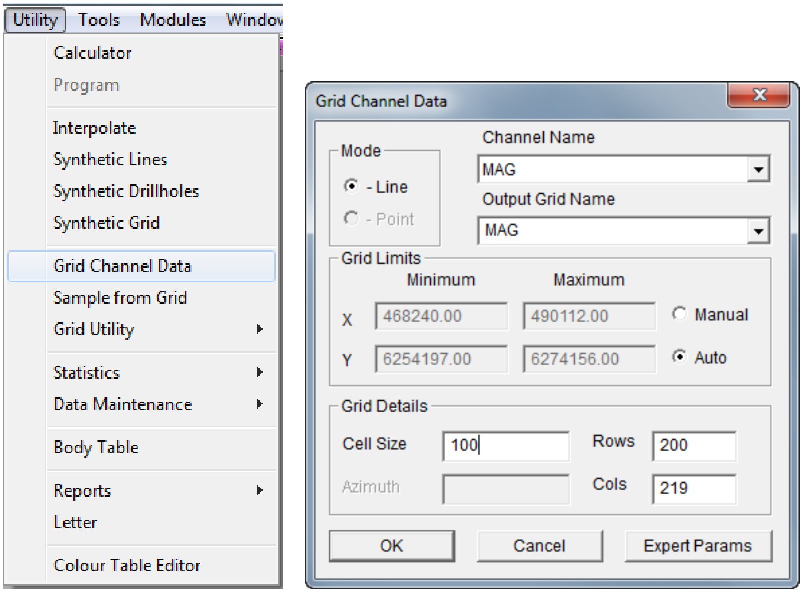
Contour Displays

A contoured display can be presented using the **View>Map>Contours** option. For this tutorial exercise, you can continue with the session already saved or clear all windows by selecting the Close button in the top right hand corner of each open window.

Contours are drawn from a grid stored in memory. The grids can be either imported or generated by ModelVision. Generated grid files can be derived from any loaded channel so both theoretical and measured data can be used. The procedure to be used for creating a grid within ModelVision is controlled by the **Utility>Grid Channel Data** menu option.

Step 19

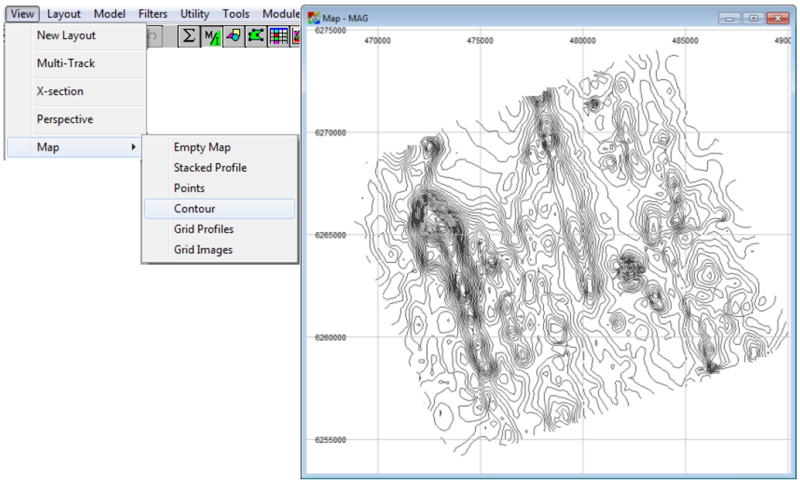
To grid the channel data from the line survey imported select the **Utility>Grid Channel Data** menu option to display the below dialog.



Select the *MAG* channel to grid and leave the default name as *MAG*. Round the Grid Cell Size to 100 and leave the **Expert Parameters** as their default. Click **OK** to compute the grid.

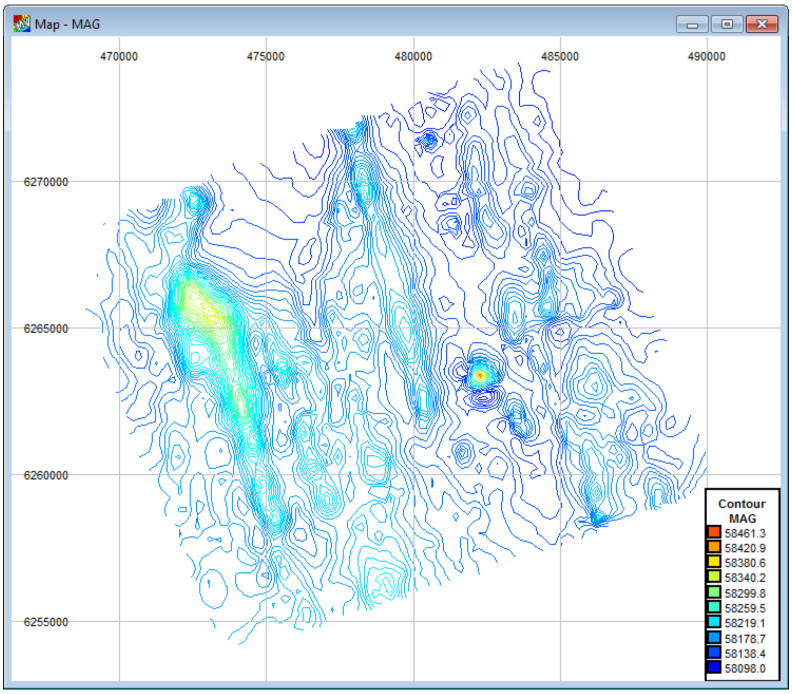
Step 20

Select the **View>Map>Contours** menu option and the grid selection dialog box will appear.



Step 21

When the When the *MAG* grid is selected and **OK** is pressed a contour map window will display as illustrated below.

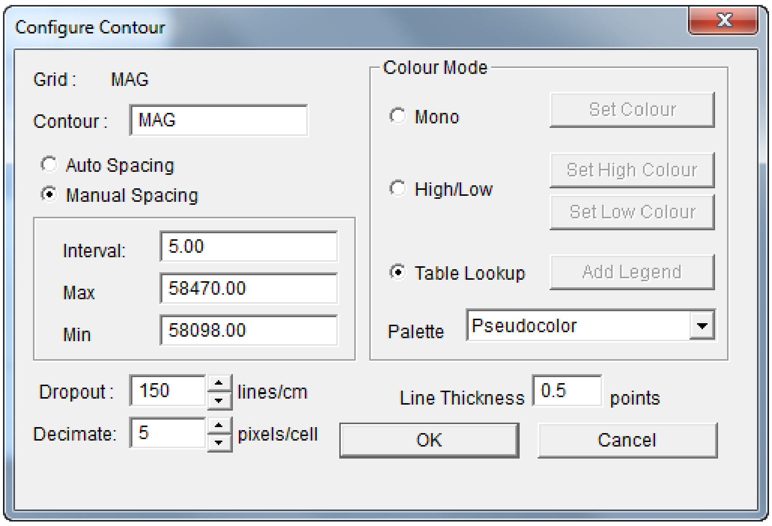


Step 22

The smoothness and presentation of the contours are a function of the grid that is used. Smaller mesh sized grids will produce smoother contours but take a longer time to compute and draw.

To configure the speed of drawing and the contours of a map press the right mouse button with the cursor over the contour window select the **Configure Layers** option from the pop-up menu that appears and display the **Map Layers** table.

View the **Contours** layer in the table and ensure the *MAG* grid is highlighted in the Name column box. Right mouse click on the Contours layer and choose the **Configure** option from the pop-up menu that appears to display controls for the presentation of contours. Colour, contour interval and the level of dropout of the contour display are all controlled by this dialog.



The dialog identifies the grid being used to generate the display. The contour interval to be used can be specified by the user (Manual Spacing) or automatically selected by ModelVision (Auto Spacing). When the manual mode for contour spacing is used a minimum and maximum value can be specified that will provide clipping of the contour level beyond these limits. Dropout controls the density of the contours drawn by defining the number of contours drawn per centimetre of display.

The Decimate option affects the speed with that a grid will be contoured. For example, a large dataset would take considerable time to contour and may have too high a resolution to be displayed in the window in any case. Because of this, ModelVision can be forced to skip every *n*th row or column for display and contour generation. This decimation process increases the speed of contouring enormously. Care should be exercised in setting the decimate value so that aliasing does not bias the contoured result.

Step 23

Save this session as a new file called `TUTE_NEW.SES` by selecting **the File>Save As...** menu option. Save this file to the same folder as the Project file, i.e. `..\TUTORIAL\TUTE1`.

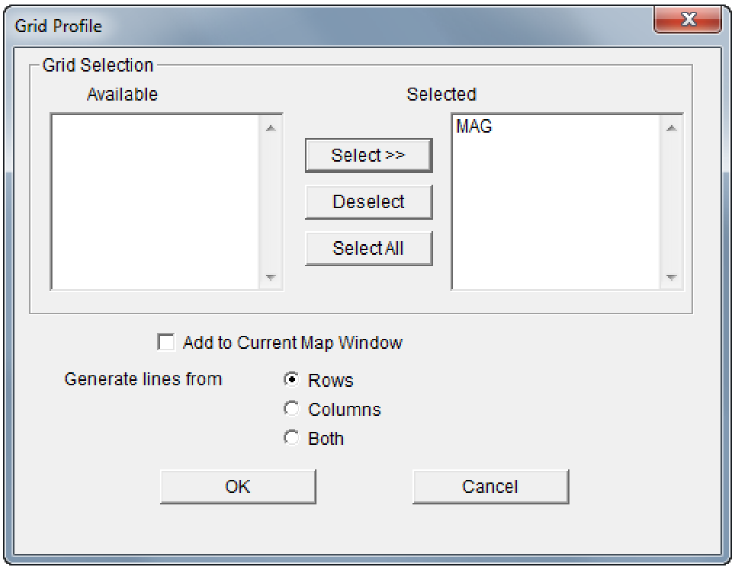
Grid Profile Displays

A Grid Profile display can be presented using the **View>Map>Grid Profiles** option. For this tutorial exercise, you can continue with the session already saved or clear all windows by selecting the **Close** button in the top right hand corner of each open window.

ModelVision can display a grid image of the rows or columns of an imported or generated grid by first of all gridding the data. This form of presentation is useful for modelling data from a grid since attempting to model by other grid displays (such as contours or imagery) is difficult. It is possible to control the number of rows and columns displayed thus making the presentation easier to visualize and faster to display. Control is also available to alter vertical scaling, display either rows, columns or both and the colour of the profiles.

Step 24

After selecting the **View>Map>Grid Profiles** option, choose one or more grids and whether rows, columns or both are to be used. The direction of the rows or columns will always match the orientation of the input grid in a vertical or horizontal sense.



Step 25

Display the **Grid Profiles** in row form initially. You may wish to extend this to both rows and columns to experiment with the display.

Note

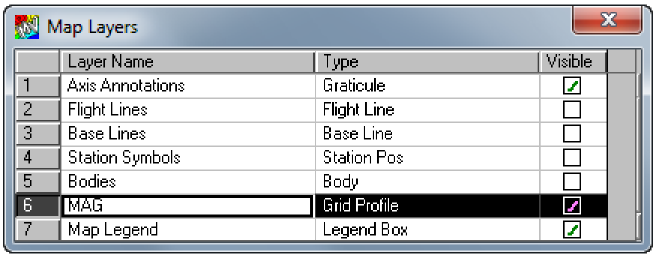
When first accessed, large grids will automatically be decimated, as presentation of every row or column in such cases can fill the display and be confusing in appearance. Decimation is the process of not displaying every row and column. This process speeds up the display operation, but can be overridden by configuring the display.

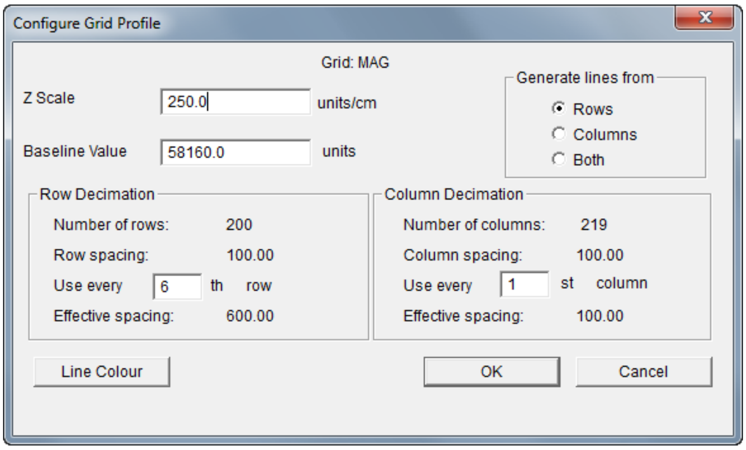
Step 26

Control of amplitude, decimation factor and colour of the display are accessed from the **Map Layers** table. The table is accessed by pressing the right button while the cursor is positioned in the Grid Profile window, and then selecting the **Configure Layers** option from the pop-up menu**.** Then click on the right mouse button while the cursor is positioned over the **Grid Profile** layer in the table and select **Configure.**

If both row and column decimation are used, then skipping in these two directions is implemented.

The vertical scaling (Z-Scale) and Baseline Value of the display are automatically computed using line data, if available, such that a stacked profile display will have the same vertical scaling as the grid profiles if they are displayed together.





Note The vertical scaling uses units/centimetre as its unit measurement. Computed values can be overwritten.

Modelling of grids generally requires compression to be turned on. The Model Compression factor should be selected to match the row or column decimation value. For example, if the grid profile decimation value is 2, use a Model Compression factor of 2, 4, 8, 16 etc.

Grid Image Displays

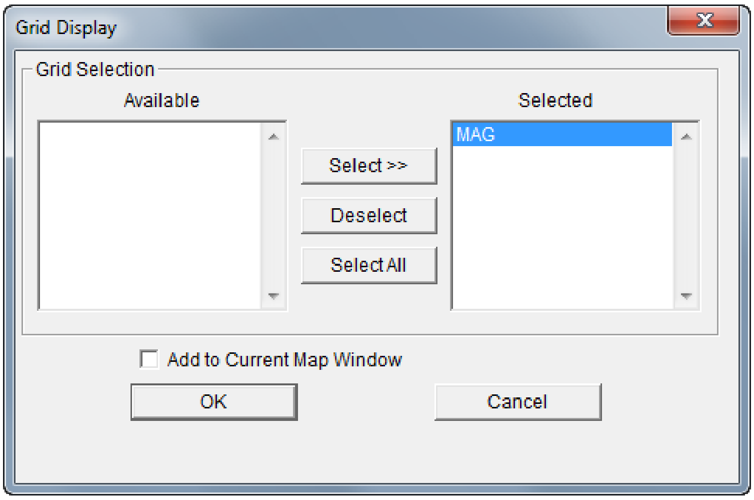
An image display can be presented using **the View>Map>Grid Images** menu option. For this tutorial exercise, you can continue with the data already imported (`TUTE.HDR`, `TUTE.LIN` data from the `/EXAMPLES` folder), or clear all windows by selecting the **Close** button in the top right hand corner of each open window.

Note If you are running ModelVision in Demonstration Mode, import `TUTE1_NEW.SES` using the **File>Open>Session** menu option.

Image displays are drawn from a grid stored in memory. The grids can be either imported or generated by ModelVision. Generated grid files can be derived from any loaded channel so both theoretical and measured data can be used.

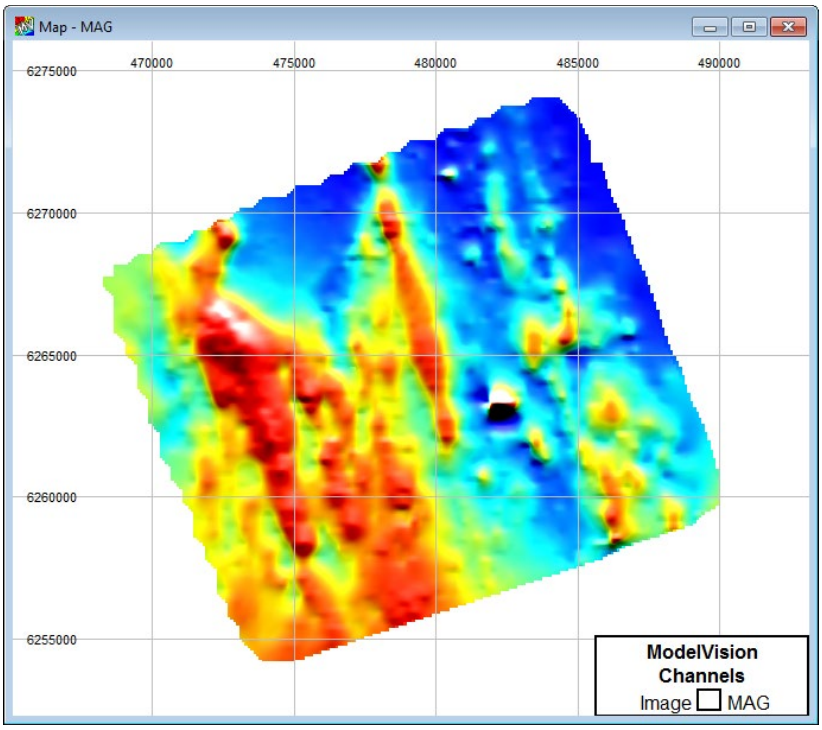
Step 27

When the **Map>Grid** option is selected, the grid selection dialog box appears.



Step 28

When the grid is selected and **OK** is pressed the image will display.



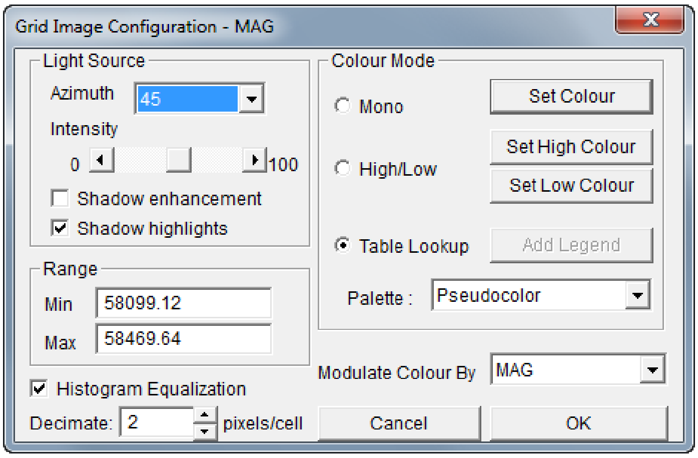
This form of presentation is useful to preview and obtain information that highlights anomaly highs and lows by their shaded representation. ModelVision cannot enhance imagery with the powerful options or functions available in software packages such as ER Mapper but it is capable of providing a fast previewing facility.

Note The appearance of a screen image is determined by the colour depth available with the computer hardware used. Displays with only 256 levels of colour will not produce high quality images. With colour levels greater than 256, the quality of the image will be significantly enhanced. On most computer video hardware, the colour depth can be increased but at the expense of reduced screen resolution.

Step 29

The smoothness and presentation of the image are a function of the grid that is used. Smaller meshed grids will produce smoother images however these will take a longer time to compute and draw.

To configure the appearance of the image display, press the right mouse button with the cursor over the image window to display the Map Layers table. Select the **Image** layer and ensure that the appropriate grid name appears in the list box. Double click on the Image layer in the layer table to view the Image Configuration dialog and to control the image illumination and colour rendering.



Data values stored in the grid are presented in the image display as a surface that is artificially illuminated by a computed light source. You can alter the location and the intensity of the light source by the controls contained in the dialog.

Note Even though a reduced colour depth image may be displayed on screen, ModelVision will use the full resolution and colour depth available when output to a printer or plotter.

The colour representation of the high and low areas of a displayed grid can also be modified by altering the colour look-up table. Three colour modes are available for the image display:

Mono

A single colour is used to present the image and shades of grey are then overprinted to provide gradient information. This is useful when you are interested in trend information and not amplitude. The colour used in this mode is definable by the **Set Colour** push button.

High/Low

A dual colour display where images above and below the average value of the grid are displayed in user definable colours.

Table Lookup

A number of external image palettes are available for setting and controlling the range of colours of the image representation. The order of colours in the colour table can be used to influence the perception of amplitude. For example, a table with colours ranging from blue to red can be used to great effect to display an image with low to high amplitudes. The palettes have a definition identical to that adopted by Leica's ER Mapper. The image colours are uniformly spread across the amplitude range of the grid data.

Pan and Zoom facilities from the Display Toolbar are available for the image map display. As for the other map window displays, contours, backdrop files, stacked profiles, flightlines, baselines etc. can be used as part of the image window display. In all cases these will plot over an image display.

Both observed and theoretical modelling responses can be presented in image form and updated as models and bodies are edited. To do this, ensure that the Grid Control option for modelling is set on and a grid has been created. Any alterations that are made to edited bodies will be reflected in both the body location or shape and the contoured display.

Note The speed of computation and updating of modelled grid output can be controlled by the **Model>Data Compression** option.

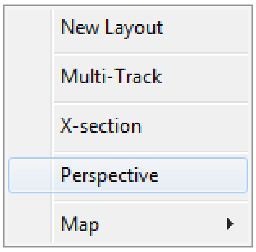
Perspective Display Window

The **Perspective** option will display a three-dimensional representation of a model, drillholes and surface location of data lines. The view can be rotated in space in a variety of planes so an optimal perspective of the model can be achieved. For this tutorial, you can directly access a perspective display by selecting the **File>Open>Session** option and selecting the file `TUTE1A.SES`.

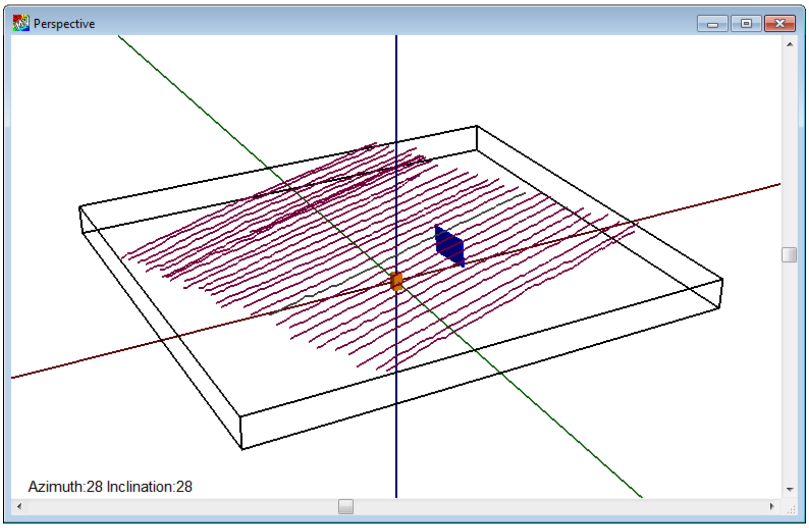
Step 30

This session contains a 2D map display of stacked profiles displaying a stacked profile data object of the MAG channel plus some simple tabular bodies modelled along line 1500.

Select the **View>Perspective** menu option. A display window will then present a perspective view of the model with each body drawn according to its default display mode.



This is typically similar to the **Solid** mode where each body will be filled with a different colour and be shaded in a style to suggest 'depth' and three-dimensionality. The bodies will be enclosed by a surrounding Wire Frame Box that will have its top surface at ground level and dimensions that relate to the size of the enclosed bodies. If at any stage the display of the perspective does not appear to enclose all the bodies, drillholes or data, select the **Fit** option from the Display Toolbar.



Step 31

Various methods are available to manipulate and move the Perspective view into the optimal viewing position. The user viewpoint is referenced to an origin that appears as a three co-ordinate axis labelled as x, y and z. The horizontal and vertical slider bars in the Perspective window will change the user's viewpoint of the display. The horizontal slider bar will rotate the view around a vertical axis. The vertical slider bar controls movement of the display around a horizontal axis. The viewing azimuth and inclination between the user's viewpoint and the model centre is listed in the window.

Step 32

Use the **3D Navigation** button on the toolbar to control the viewing position in a 3D display area. After selecting this button navigation in 3D is performed in 3 different methods:

- Clicking on the left mouse button will rotate the 3D area around a central axis position.
- Clicking on the right mouse button and moving the cursor around the 3D display will activate the zoom in/zoom out navigation options.
- Clicking on both the right and left mouse button together will activate a fly-through effect.

Note To move the position of the central axis point relative to the display area hold the **SHIFT** key down and click on the left mouse button while moving the cursor to either the top or bottom of the display area.

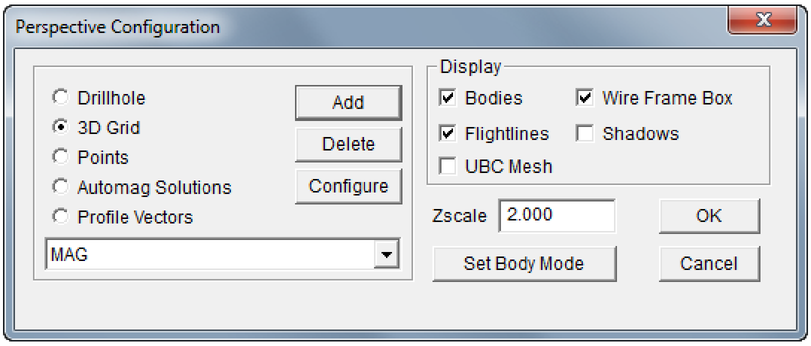
Three separate entities can be displayed with the Perspective view. These are:

- Models and individual bodies.
- Locations and data of loaded drillholes can be displayed. Colour modulated values or vectors can be displayed along the drillhole path.
- Survey lines are displayed on the surface. Active model lines are displayed in red.
- Grid surfaces with z scaling and colour modulation options.

Note The survey line is displayed on a zero datum and does not currently use the elevation channel.

Removal of hidden surfaces is a computationally intensive operation and ModelVision makes some approximations to provide rapid 3D visualization. In some circumstances the body ordering is not correctly visualized. If this occurs, try changing the viewpoint with the slider bars.

Control of the displays is provided by positioning the cursor in the Perspective window and then pressing the right mouse button to select the **Configure Layers** option from the pop-up menu. If you are in 3D Navigation mode at the time, hold down the **CTRL** key while clicking the right mouse button to display the pop-up menu.

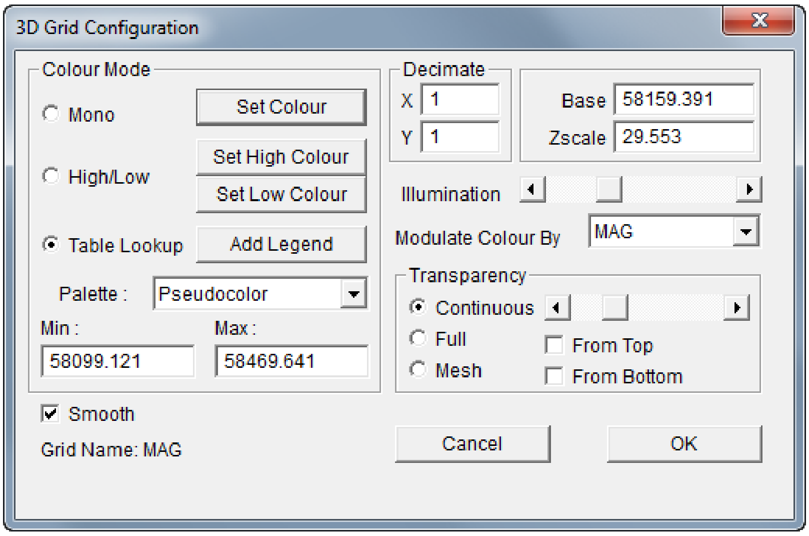


Step 33

A loaded grid can be displayed in 3D by selecting the **3D Grid** radio button in the **Perspective Configuration** dialog. The name of the loaded grid should be displayed in the drop down box in this dialog. If there is no grid name displayed then you will need to load a grid by selecting the **Add** button.

With the loaded grid name visible click on the **Configure** button and the **3D Grid Configuration** dialog will appear, allowing you to select the Table Lookup Palette of *Pseudocolor* from the drop down box, and adjust the Z-scale of the grid to be *5.0*. Also increase the transparency of the grid by moving the slide bar to the right for the **Continuous** Transparency option.

Click **OK** in both dialogs to return to the Perspective window and observe these changes.



Drillhole Displays

Downhole data obtained from drillholes can be viewed in a variety of displays, including map windows, perspectives and cross-sections. For this part of the tutorial save and close the session created so far (**File>Save**) and clear the memory using the **File>Close** menu option.

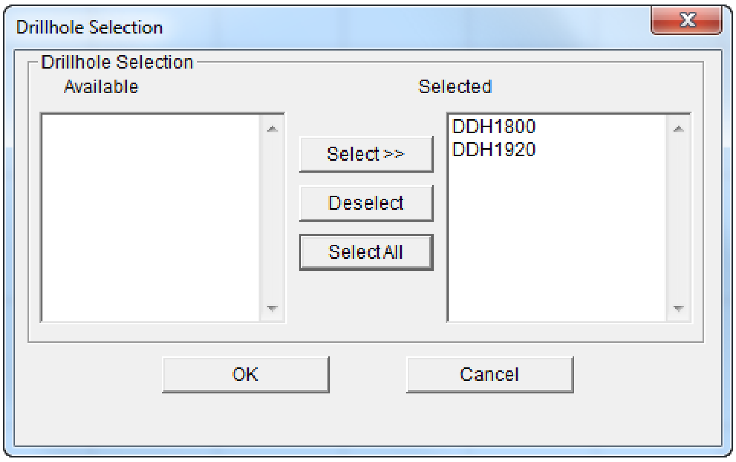
Step 34

Using the **File>Import>Drillhole Data>Simple XYZ (.LIN)** menu option import the file `DHOLES_EXAMPLE.LIN` file located in `..\EXAMPLES\DRILLHOLE`.

Once loaded, use the **Utility>Statistics>Drillhole Data** menu option to display information on the location of the drillholes and their downhole data values.

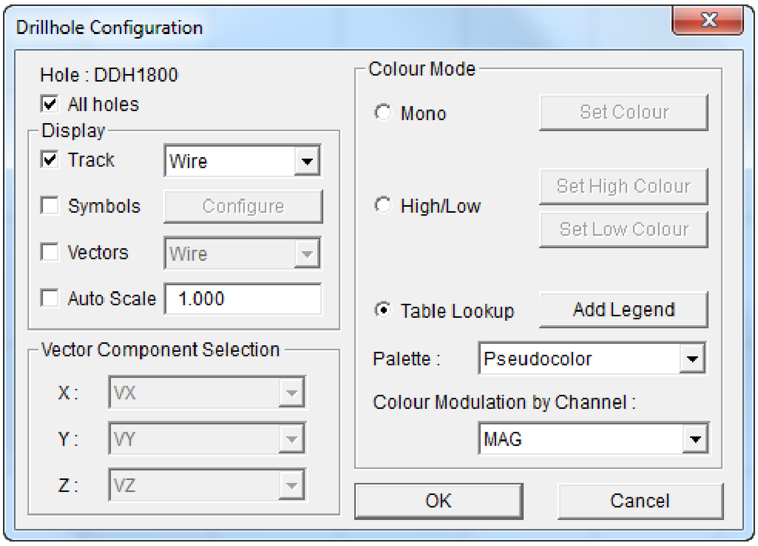
Step 35

Open a new map window by selecting the **View>Map>Empty Map** menu option. Click the right mouse button while the cursor is positioned in the map window and select Configure Layers to view the **Map Layers** table. Click the right mouse button anywhere in the table and select the **Add>Drillhole** option to select the two available drillholes (DDH1800 & DDH1920). Click **OK** to return to the map window and Map Layers table.



Step 36

Control of the drillhole display is made available by selecting the **Configure** option in the pop-up menu when the Drillhole layer is selected with the right mouse button. This will display the **Drillhole Configuration** dialog as illustrated below:



Note The same configuration options for drillholes are available in the **X-section** and **Perspective** configuration dialogs.

This drillhole configuration controls the display, colour, scaling and channels to be used for vector displays. Display controls available include:

Track

This option controls whether a trace of the drillhole is shown. A drillhole trace is presented as a line that joins the hole locations as defined by the X, Y and depth channels in the loaded data.

Symbol at each point

Displays a '+' symbol at each hole measurement location.

Vectors

Vectors are referred to in ModelVision by three components parallel to the X, Y and Z axes with default names of VX, VY and VZ. Scaling of the length of the vector is controlled by the Auto Scale option. Selection of the vector components is defined in the **Vector Component** Selection option. The individual vectors can be represented as lines or arrows radiating out from the drillhole trace.

Colour Mode

Three modes of representing the vector displays of drillhole data are available. These are:

- Single definable colour (Mono or colour)
- High/low dual colours
- Table lookup colour ranges

The colours used to display the vector amplitudes can also be modulated by a nominated channel. For example, if the three component vectors of downhole magnetics are VX, VY and VZ but the display is coloured according to the total magnetic vector, then the Compute operation could be used to calculate a new channel that represents the total vector amplitude √(VX2 + VY2 + VZ2). The new channel could then be used to control the colour of the vector display.

A colour legend can be displayed to indicate the vector colours used.

Step 37

For this example select the **Wire Track Display** type and allow for auto scaling by selecting the Auto Scale check box. For the Colour Mode select the **Table Lookup** radio button option and specify the *Pseudocolour* palette for colour modulating the drillholes by the *MAG* channel.

This concludes the introductory tutorial for ModelVision. If you would like to continue your education with using the software please refer to the additional tutorials in the Documentation folder.

Tutorial 2 – Display, Filtering and Controls

Aim

The aim of this tutorial is to demonstrate how data can be presented in various forms and to illustrate the ease with which various relationships between multi-channel information can be derived. An introduction to filtering and the calculator option are also provided.

Topics Covered

- Data Import
- Multi-Track display and control
- Convolution Filtering
- Calculator option

Intended User Level

Basic introductory and a refresher for experienced users.

Background Information

Contained in the data file for this tutorial is a single airborne profile extracted from a much larger survey flown in 1992 in northern New South Wales (Australia). This survey data has been designed with traverses that are oriented north to south. Within the data file are AMG coordinates, Fiducials, Magnetics and spectrometer channels of Potassium (K), Uranium (U) and Thorium (Th). The survey was over a granitic margin with the intention of determining the shape of the granite edges. Poor mapping in rough terrain and extensive overburden masked much of the granite boundaries. The target being sought in the area are tin skarns that are known to exist locally.

Tutorial Data

For this tutorial you will need to Import a file called `TUTE2.TK`. This single line of data has been extracted from a north-south oriented airborne survey and is a sequential line file containing 1999 data records. The data file is written in TOOLKIT multi-column ASCII format and is required to be read using the Import TK Option. See **Tutorial 1** that describes the steps taken to import a dataset if you have difficulty in initially reading the data file.

Tutorial Steps

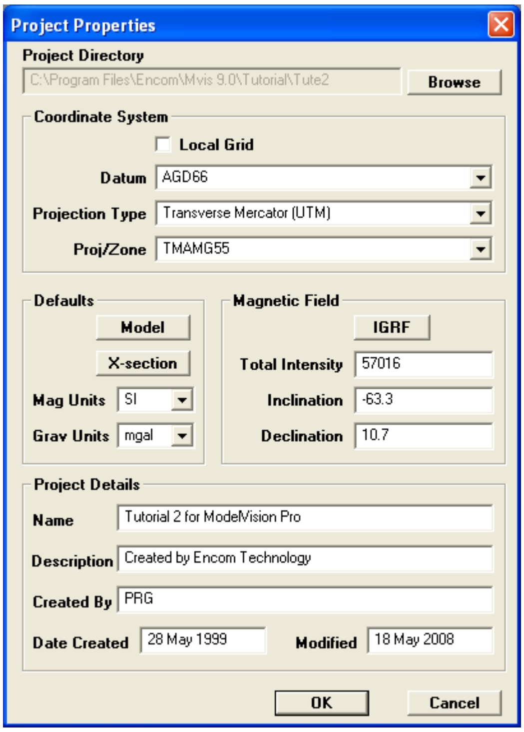
The following tasks should be undertaken to familiarize yourself with an analysis of typical airborne data in ModelVision:

Step 1

Initially, a project is required to define the various properties of the data. ModelVision uses projects to manage various analysis exercises and a `MVPROJ.INI` records your entries for later use. Binary session files (.SES) can also be used to store data, models and displays if required.

Select the **File>New Project** option and enter the information as required. For this area and projection information includes:

Datum:	Australian Geodetic Datum 1966 (AGD66)
Projection:	Transverse Mercator
Zone:	AMG Zone 55 (TMAMG55)



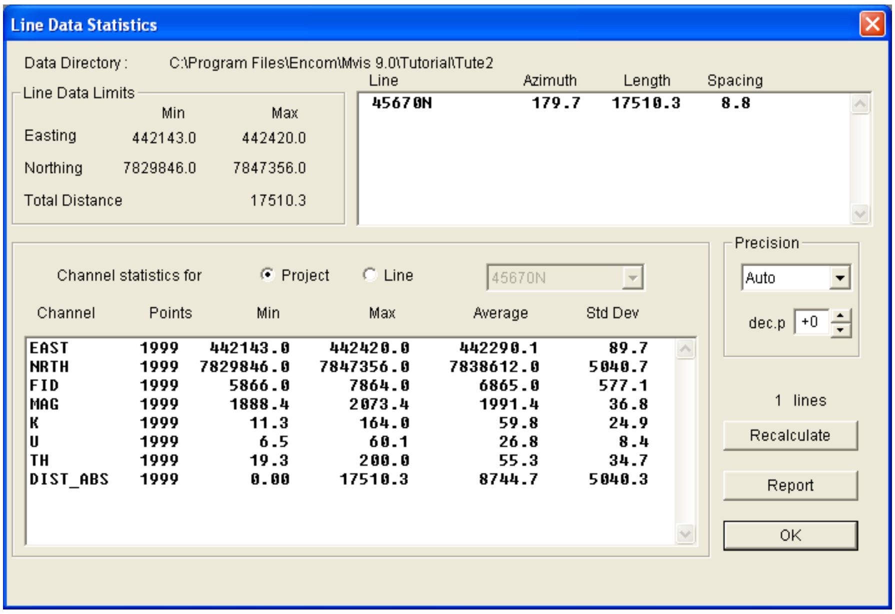
You can use the IGRF button to accurately position the survey (147° 6.33'E, 51° 31.75'S) and compute the Earth's magnetic field parameters. Defining the Earth's magnetic field at this stage ensures that any magnetic modelling performed later in the ModelVision session is correctly calculated.

Modify the Project Details as required. When the necessary entries are made, press the **OK** button.

Step 2

Using the **File>Import>Profiles>Toolkit Format** (.TK) option import the data file called `TUTE2.TK` located in `..\TUTORIAL\TUTE2`.

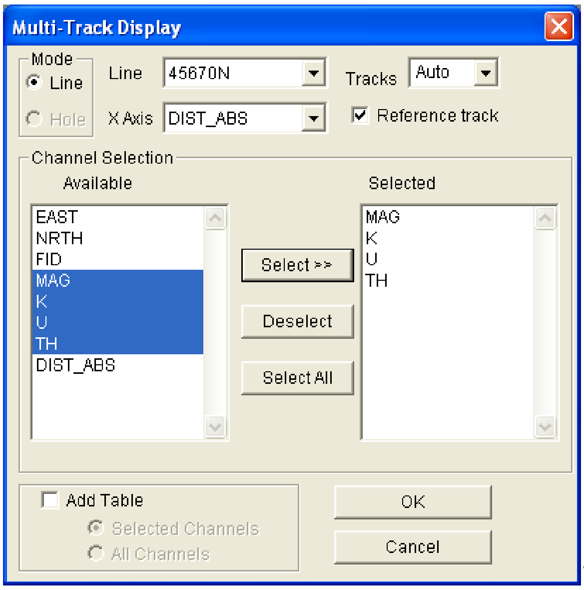
Then examine the data using the **Utility>Statistics>Line Data** menu option to determine line extent, channels ranges etc.



Note The distance (DIST_ABS) column has been computed automatically from the northing and easting data as distance calculated from the first data point location of a line to the relevant record location of each successive point. The results in the DIST_ABS channel are therefore a true distance from the line start and not a series of cumulative reading distances. This channel can be used as the horizontal scale for plotting and display purposes.

Step 3

Display the magnetics and radiometrics using the **View>Multi-Track** menu option to check the integrity of the data. The dialog provides a list of the channels to select. Choose the magnetics (MAG) and spectrometry (K, U and TH) channels. The **Add** reference track is not ticked by default. For this exercise, select this check box and then choose the **OK** button.

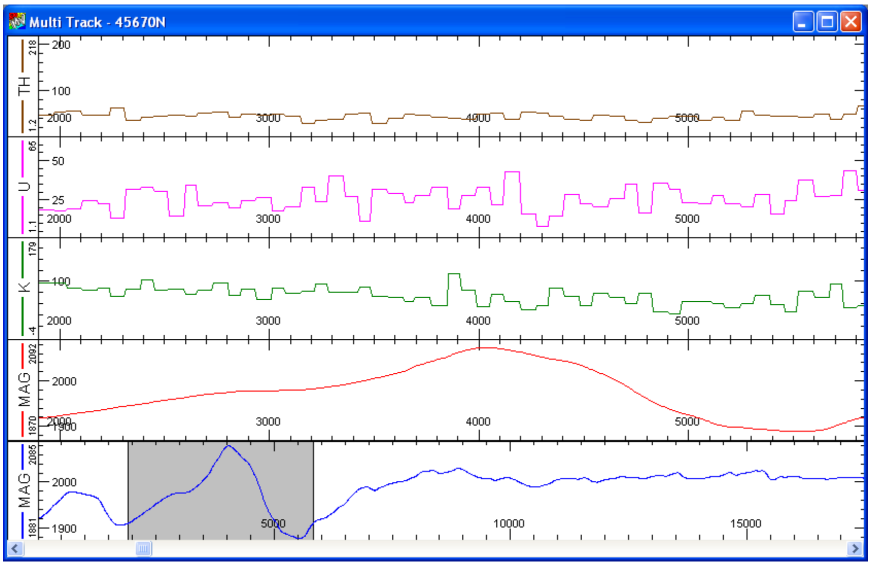


The **Reference** track will display the first channel of the selection dialog in the Multi-Track display as a ‘greyed’ track at the base of the window. When released, all but the Reference Track will have zoomed to the selected portion of data. The scroll bar at the base of the window can then be used to progressively move across the data display. When the channels are selected the profile will be displayed.

Note An option available in the Multi-Track dialog enabled the top portion of the window to display a spreadsheet of all the channel values displayed. By double clicking on any cell value in the spreadsheet it is possible to edit the values. Additionally, you can select a row number (left column) and the cursor will move to that location in the graph if it is in range of the current zoom. The converse is not true. That is a cursor selection in the graph does not move to the matching record in the table.

Step 4

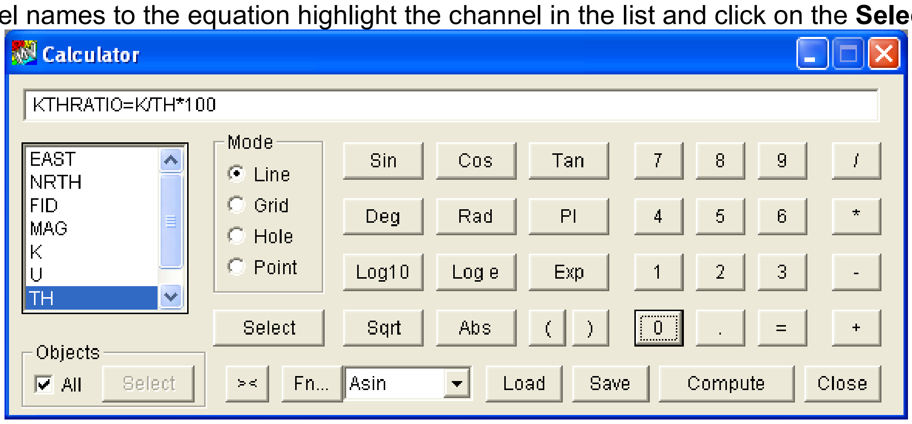
Return the track to its full size by positioning the cursor over one of the tracks and clicking the right mouse button to display the pop-up menu options. Select the **Fit Track>Horizontal** option to adjust the appearance of the track.



Step 5

It is possible to analyse data and to add tracks to the multi-track display. For example if you wanted to examine the relationship between two data channels (such as a spectrometry ratio) this can be done and then displayed in the same window as the other selected channels.

Calculate the K/Th ratio (use the **Utility>Calculator** menu option) and simply enter the desired formula as illustrated below. Ensure that the **Line** mode is selected and to add the Channel names to the equation highlight the channel in the list and click on the **Select**



button.

To create a new data channel of the ratio, press **Compute** after designing the formula.

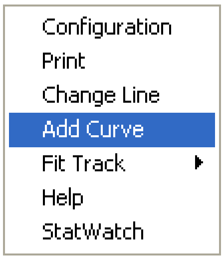
Note In the example shown, a new channel (KTHRATIO) will be created when the **Compute** button is pressed. Additional new channels can be created as required. This channel will be added to the memory for the loaded data and can be viewed in the Line Data Statistics (**Utility>Statistics>Line Data** menu option).

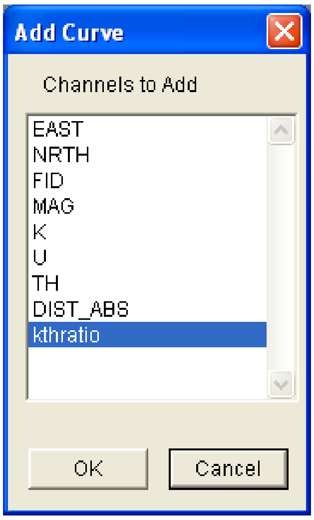
Note Certain restrictions apply to the use of the Calculator. For example, channel names cannot start with a number and there are reserved words such as SIN, TAN, TANH etc.

Step 6

To examine the shape of the new resultant channel, it can be added to the display in the Multi-Track. Bring the cursor to any track and right click. A pop-up menu as shown will be presented. Select and press the **Add Curve** button.

A new dialog will enable you to select the new channel.





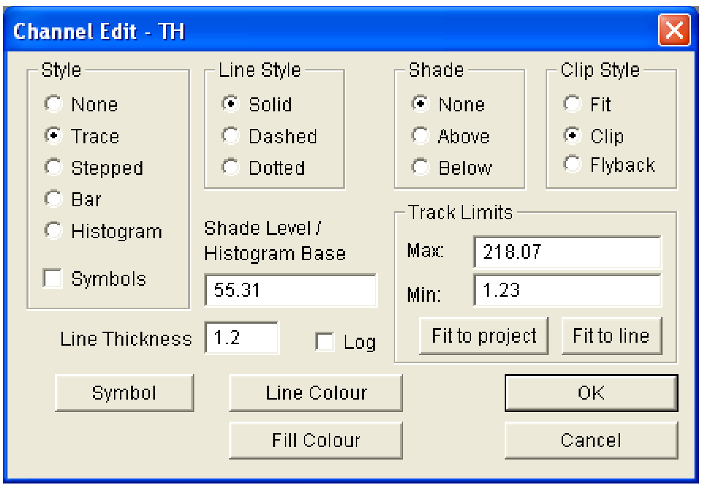
Select the *kthratio* track and press **OK**. The new channel is then added to the track initially selected.

Note In the example shown above, for multi-line datasets, the **Change Line** option is very useful to interactively examine a range of lines as required.

Step 7

If you wish you can reposition any of the data tracks by placing the cursor in the left hand header box, pressing the left mouse button and 'dragging-and-dropping' the channel to a new location.

Controlling the display attributes of any of the displays of the various channels is achieved by double left clicking in the channel header box at the left end of the Multi-Track display.



Step 8

Filtering is a powerful geophysical tool for extracting information from data. ModelVision provides two main filter techniques - Convolution and Fast Fourier Transform (FFT) filters. Filtering potential field data in particular is useful for:

- Extracting frequency information specifically from a particular set of depth sources.
- As a means of extracting noise from data.
- As a technique for highlighting special features or repairing problem data.

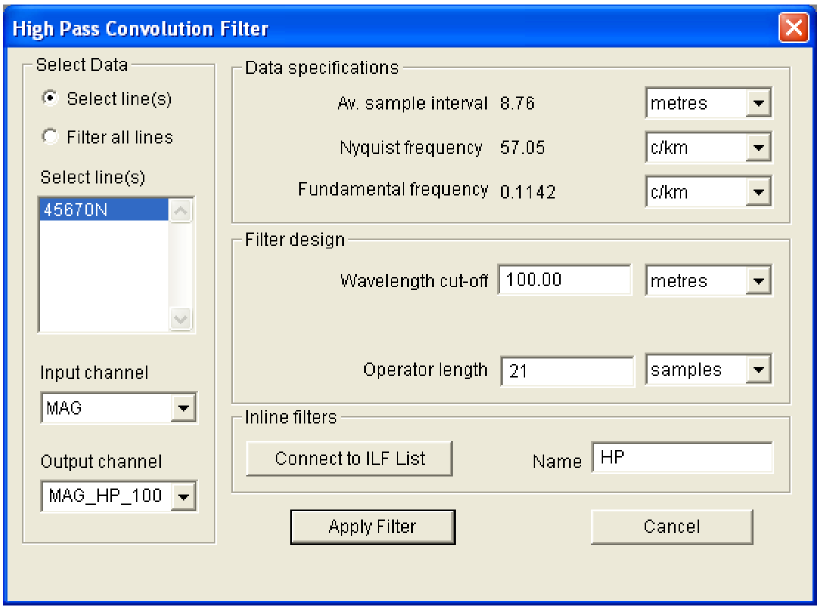
In ModelVision, filtering is usually first applied on an individual line basis to design a filter, and then the option is available to apply the same filter over a range of lines or all data in memory.

Use a series of high pass convolution filters to isolate the near surface response contained in the magnetic data. The filter options are accessed from the Filter menu.

Tip: The spacing for this data is about 10-11 metres collected at an altitude of about 80 metres above ground so use a range of filters of about 75,100 and 250 metres with filter operator of about 15, 21 and 51 respectively.

Note The best method of determining the optimal number of samples in an operator is to approximately calculate twice the longest wavelength that you are trying to remove from the data. For example, in the above design of a 100 metre wavelength high pass filter on 10 metre sampled data, the optimal operator width would be:

2 x 100 / 10 + 1 = 20 or 21 samples



Successively create the three filter outputs.

Warning: Each time a new filter is created, you must **rename** the output channel name to something appropriate. If this is not done, the filter output will overwrite your previously created results. This is done intentionally since in some instances, it is a feature to have the Multi-track display update automatically as a revised filter design is calculated.

Step 9

Display the filtered output of each filter using the **Multi-Track** view option. Make sure that the *Add Table* check box is turned off for this new Multi-Track window..

Try printing the generated display to a file. To do this, use the **File>Print Setup** menu item and then output the file using the **Files>Print** option. Note the scale and height controls that are available in the Print dialog.

Tutorial 3 – Data Analysis and Introductory Modelling

Aim

The aim of this tutorial is to extract magnetic noise from surficial sources and then determine a model that may be adopted as a drill target for further exploration. The exercise is designed to introduce complex filtering techniques and use the reduced data to develop basic magnetic models.

Topics Covered

- Data Import
- Multi-Track display and control
- Convolution Filtering
- Cross-section creation
- Introductory modelling using simple models.
- Introduction to inversion.

Intended User Level

Introductory and experienced refresher tutorial.

Background Information

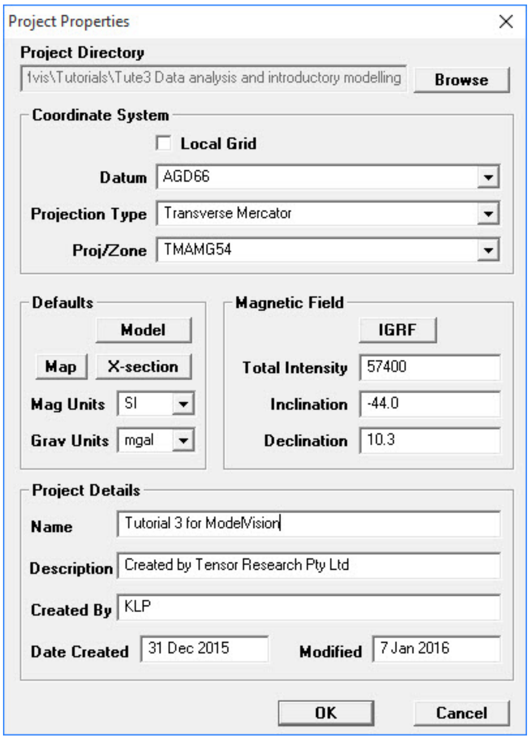
The data contained in the file used for this tutorial is a ground magnetic traverse in an area near Bourke NSW (Australia). The line crosses an area that contains severe magnetic noise produced by maghemite concentrated in paleo drainage channels. High copper and lead geochemistry from samples collected in the area of 1550-2050E make it necessary to interpret this noisy data.

Tutorial Data

Initially, a project is required to define the various properties of the data. ModelVision Pro uses projects to manage various analysis exercises and a `MVPROJ.INI` records your entries for later use. Binary session files (.SES) can also be used to store data, models and displays if required.

Select the **File>New Project** option and enter the information as required. For this area and projection information includes:

Datum:	Australian Geodetic Datum 1966 (AGD66)
Projection:	Transverse Mercator
Zone:	AMG Zone 54 (TMAMG54)



Note that the entries for the Earth's magnetic field are set to be:

Field Intensity:	57400
Inclination:	-44 degrees
Declination:	10.3 degrees East

Note The format of the file is a Geosoft single line DAT file. Use a text editor or alternatively use the DOS Type command to display the data. The data will appear similar to below:

Geosoft Single Line File - Demonstration for ModelVision Pro

Line:1110

X	Y	MAG
6323282.000	2190063.250	57242.500
6323284.500	2190072.750	57243.801
6323287.500	2190082.500	57242.602
6323290.500	2190092.000	57241.102
6323293.000	2190101.750	57239.500
6323296.000	2190111.250	57238.500
6323299.000	2190120.750	57236.801
6323301.500	2190130.500	57235.602
6323304.500	2190140.000	57234.199
6323307.500	2190149.500	57235.500
6323310.000	2190159.250	57234.500
6323313.000	2190168.750	57233.801
6323316.000	2190178.250	57234.602
6323318.500	2190188.000	57234.102
6323321.500	2190197.500	57237.500
6323324.500	2190207.000	57236.500

Tutorial Steps

The following tasks should be undertaken:

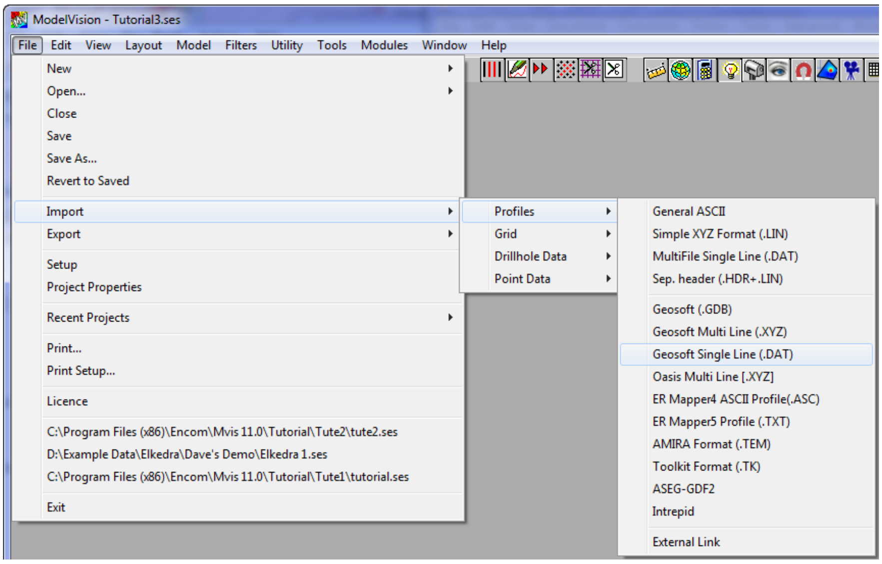
Step 1

Use the **File>Import>Profile>Geosoft Single Line (.DAT)** option to open the data file called `TUTE3.DAT` located in `..\MODELVISION_TUTORIALS\TUTORIAL3`.

or if in Demonstration mode, **File>Open Session** and browse for `TUTE3.SES`.

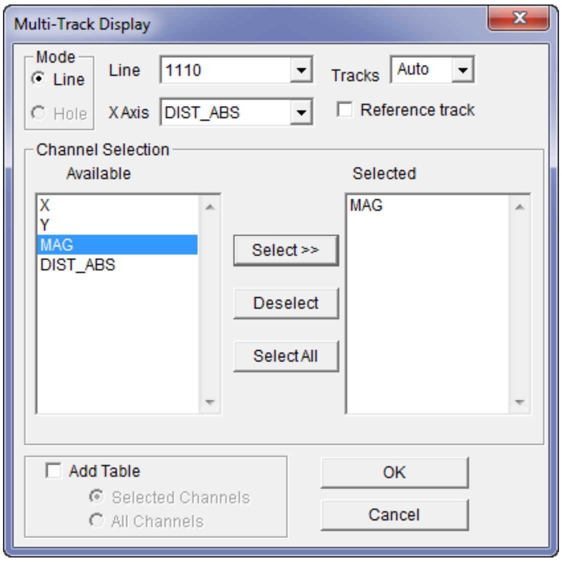
If you have difficulty reading the data file initially, see Tutorial 1 that describes the steps taken to import a dataset.

The data is a single line of magnetic readings that has been extracted from a ground magnetometer survey in the Bourke area of NSW. Only easting, northing and magnetometer data channels are present. The magnetometer data has been diurnally corrected.

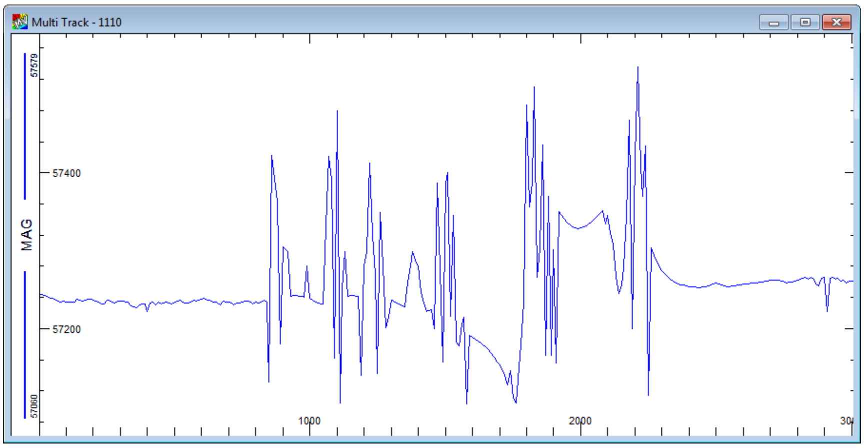


Step 2

Use the **View>Multi-Track** menu option to display the magnetics against the distance (DIST_ABS) channel and pay special attention to the wavelengths of the surficial noise evident in the data.



Note In the profile selection that the Reference Track or data table have not been selected for display.



Step 3

Design some long wavelength filters to obtain a variety of outputs that can be used for further analysis. The simplest method of doing this is by using the Low Pass convolution filter option. Try wavelength cut-offs of 50, 100, 250 and 500 metres. You will note that the data has been acquired with a 5 metre station interval. Therefore for this data, filter operators would optimally be 21, 41, 101 and 201 samples respectively.

Note Take care to assign unique names for each filter result.

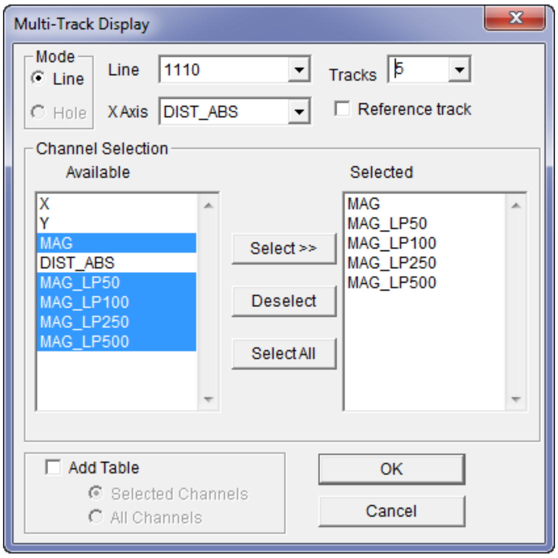
Note The best method of determining the optimal number of samples in an operator is to approximately calculate twice the longest wavelength that you are trying to remove from the data. For example, in the above design of a 100 metre wavelength low pass filter on 5 metre sampled data, the optimal operator width would be:

2 x 100 / 5 +1 = 41 samples

Filter lengths are always an odd number of samples.

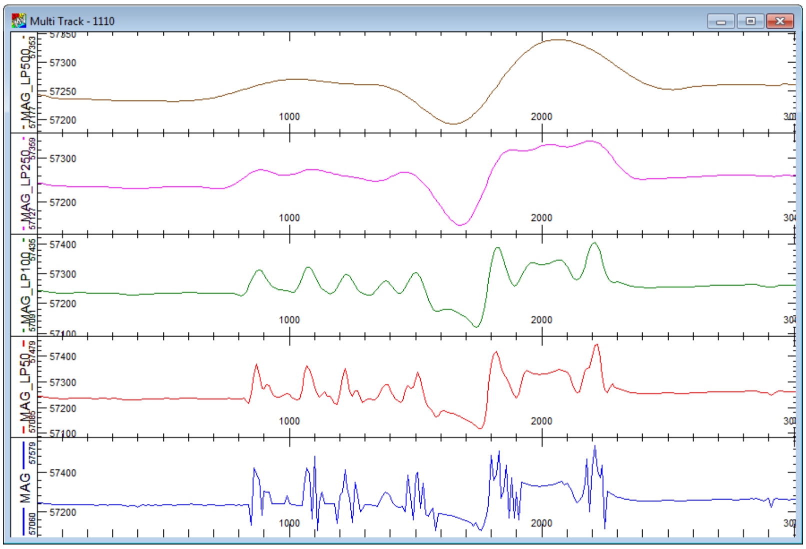
Step 4

Display the generated filter channels using Multi-Track display mode. A Multi-Track display of the various filter designs should enable selection of the best filter.

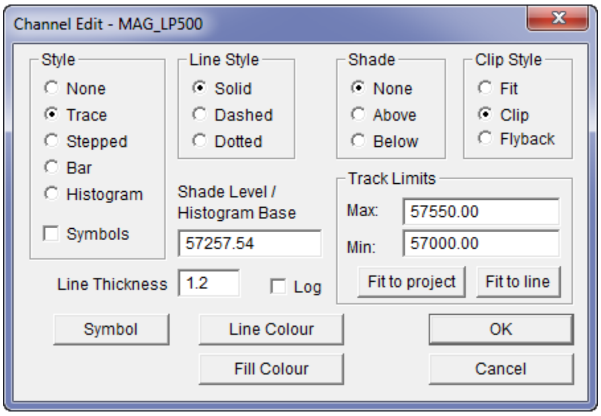


Note

You will need to define the number of tracks required and not use the Automatic option to create the tracks.



Adjust the vertical scalings for each track so that all vertical scales are the same (Max: 57550; Min: 57000). This is done by double clicking the left mouse button while the cursor is in the appropriate Track Label (at left end of Multi-Track). Each line needs to be adjusted individually.



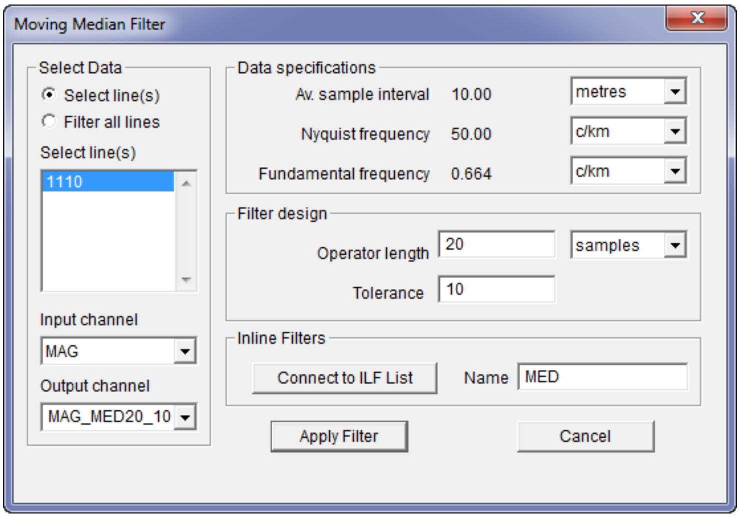
Step 5

You may wish to also design a series of high pass filters to isolate the near surface response. Once defined, this 'noise' can then be subtracted from the original data to result in a long wavelength response as determined above. It is recommended that frequency domain filters be tried for the extraction of the high frequency response. This will familiarize you with both filtering approaches and enable a comparison of results to be made.

Step 6

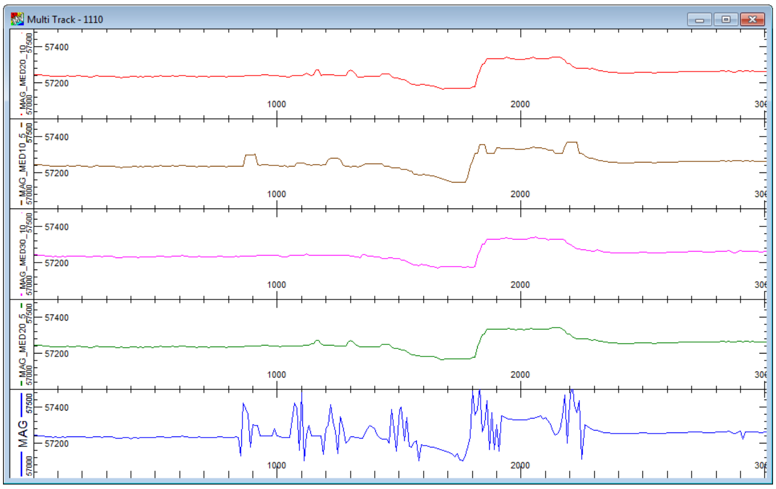
An alternative filter that may be of use is the spike rejection **Median Filter**. This filter requires a spike rejection threshold amplitude and operator window length to define the wavelength sensitivity. The filter incrementally moves along the selected line(s) and compares the middle sample amplitude of the window with the difference between the median value of the window and the threshold. If the absolute value is greater than the threshold, the median is substituted for the value.

For your test line, test a range of threshold values and compare the results. An example of filter parameters set in the Moving Median Filter dialog is shown below.



Create a number of Median Filter results (naming the Output Channels uniquely and display in a new Multi-Track display. Again ensure all tracks have the same vertical scaling to compare results accurately.

Another filter type that you may consider is an Upward Continuation. We will investigate this as an aid to modelling in a later stage of the Tutorial.

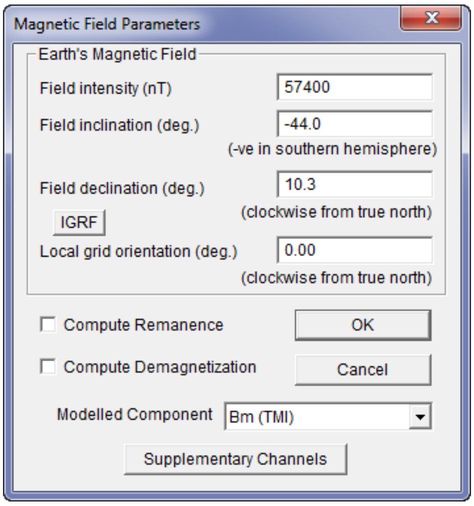


Step 7

Once a bedrock response with minimal surface magnetic noise has been generated, then a model can be developed to simulate a target for drilling. Initially it is important to check the magnetic field specification for the survey region:

Earths Total Field	57400 nT
Declination	-44 degrees (southern hemisphere)
Inclination	10.3 degrees East

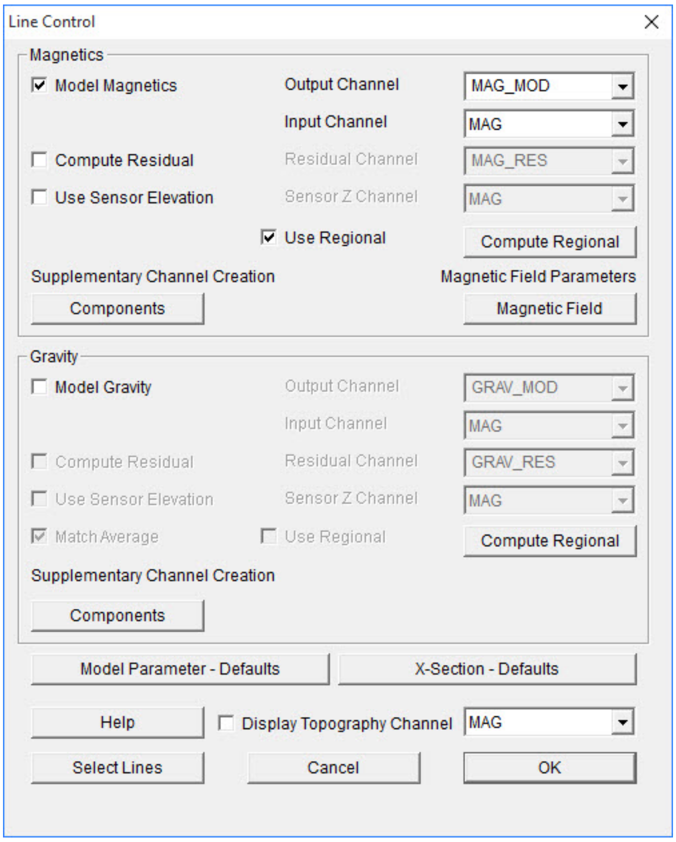
These parameters are available from the **Model>Magnetic Field** or the **File>Project Properties** option.



Step 8

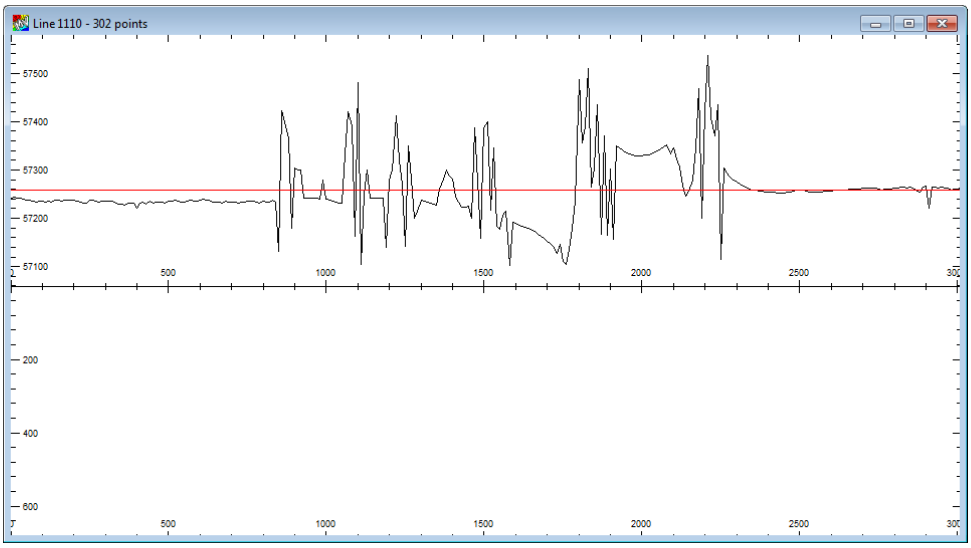
To model the filtered response, select the appropriate channel (MAG) as input in the **Model>Line Control** menu option dialog. Click **OK** to close this dialog.

Although any of the filter outputs could be used as the input for modelling, it would be difficult to estimate the effect a long wavelength filter would have on a derived depth for bodies we create in our model. Consequently, during the modelling process, an Upward Continuation filter (where the depth effect is known) will be adopted.



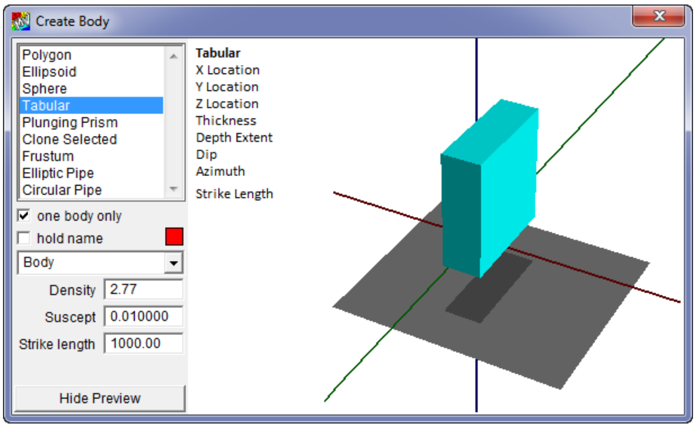
Step 9

Modelling will be performed using the **View>X-Section** menu option. This display will present the selected data channel, a computation output channel and a section below the data tracks for creating models.



Step 10

Create a Tabular shaped body using the **Create Body** option on the toolbar or the **Model>Body Operations>Create Body** menu option.



After selecting the body type from the Create Body dialog, position the cursor in the X-section area and click the left mouse button or click and drag out an enclosing rectangle. For this exercise select the simplest body type - a **Tabular** body. To generate a theoretical response after the body is positioned, select the **Compute** button on the toolbar

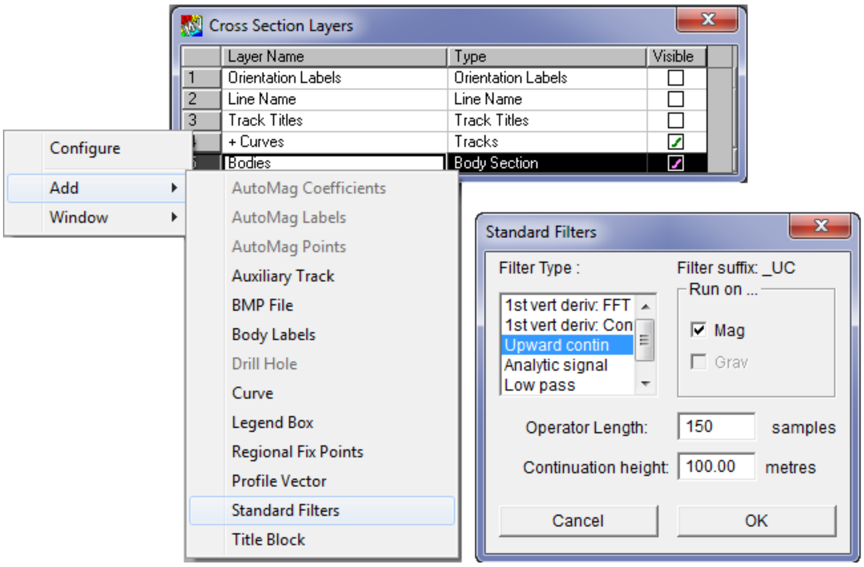
Use the **Reshape** button or icon once the body(s) has been created to position and orient it.

You will need more than one body to simulate the response. With the degree of noise present in this data it is impractical to model in its current form.

Step 11

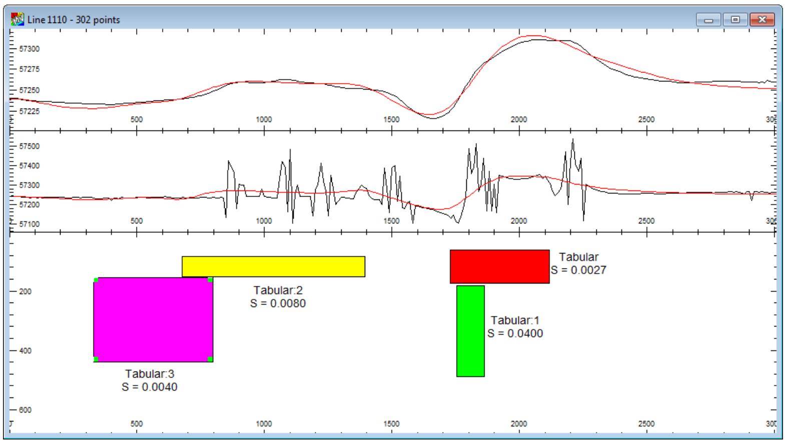
In-line filters can enhance the precision of forward modelling and enable modelling even in these difficult circumstances. In-line filters can be used to provide greater sensitivity between the observed and calculated magnetic responses while modelling. Since this test line was initially prone to surface magnetic noise, it is important that we know what effect the filter is having; we can use an **Upward Continuation In-Line** filter. To display a track of the in-line filter curve you can add it to a X-Section display. Click the right mouse button to select **Configure Layers** from the pop-up menu and display the **Cross-section Layers table**.

Then click the right mouse button on any layer in the layer table and select **Add>Standard Filters** from the pop-up menu that appears.



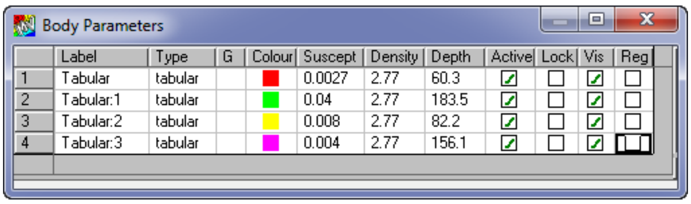
Note The In-Line filter operates on the original MAG channel and not the derived Median or Long Wavelength filters. If desired you may wish to use the Median spiked filter data. This could be easily done using the **Utility>Calculator** option and copying the Median filter output to the MAG channel. As a precautionary step, you may also wish to rename the original MAG channel to another name.

Select an optimal operator length and continuation height (say 100 metres). When **OK** is selected two new curves will be added to a new track above the X-Section display. These two curves will be the Upward Continuation of the computed and observed trace data.



Step 12

Use the **Create Body** toolbar button to create four *Tabular* bodies similar to that displayed above with similar body properties as outlined in the Body Table (**Model>Body Operations>Body Table** menu) dialog. To edit the individual body properties double left mouse click on each body in turn to display the body property dialog or in the Body Table itself.

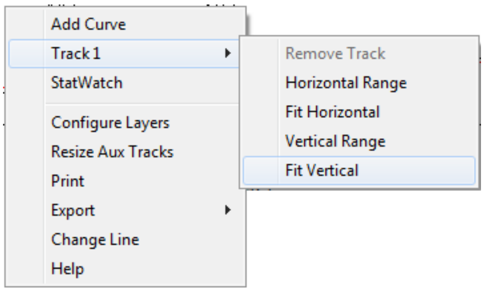


Body Parameters

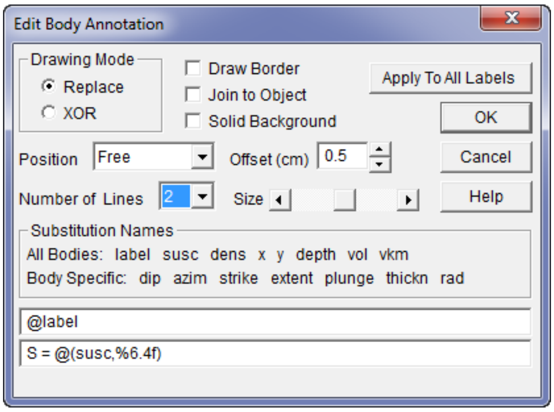
	Label	Type	G	Colour	Suscept	Density	Depth	Active	Lock	Vis	Reg
1	Tabular	tabular		red	0.0027	2.77	60.3	☑	☐	☑	☐
2	Tabular:1	tabular		green	0.04	2.77	183.5	☑	☐	☑	☐
3	Tabular:2	tabular		yellow	0.008	2.77	82.2	☑	☐	☑	☐
4	Tabular:3	tabular		magenta	0.004	2.77	156.1	☑	☐	☑	☐

Note

You may need to Fit and adjust the vertical size of the tracks of the display to balance the presentation. To fit a track either horizontally or vertically click the right mouse button in either track displayed in the x-section and select the **Fit Track>Horizontal** or **Vertical** pop-up menu options.



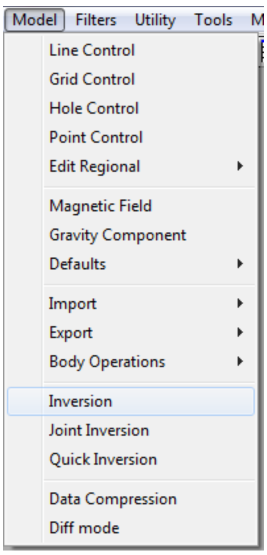
To apply a label to each body double click the right mouse button on the x-section view and select **Configure Layers** from the pop-up menu that appears. Then right mouse click on any row in the Cross-section Layers table that appears and select **Add>Body** labels. To edit the label position the cursor over the label and double click on the left mouse button. This will display the **Edit Body Annotation** dialog. Deselect the *Draw Border* and *Join to Object* options and adjust the *Number of Lines* to **2.** This will remove the display of the density property value.



Step 13

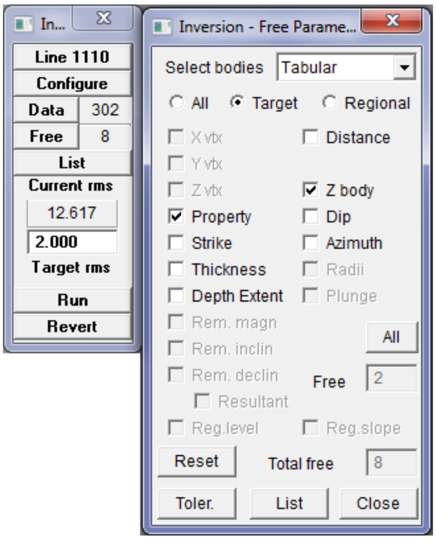
When a reasonable forward model is finalized, attempt to refine it with inversion. Use only a single free parameter initially and apply this to only one body at a time.

To begin the inversion process display the **Inversion** dialog by selecting the Inversion option from the **Model** menu.

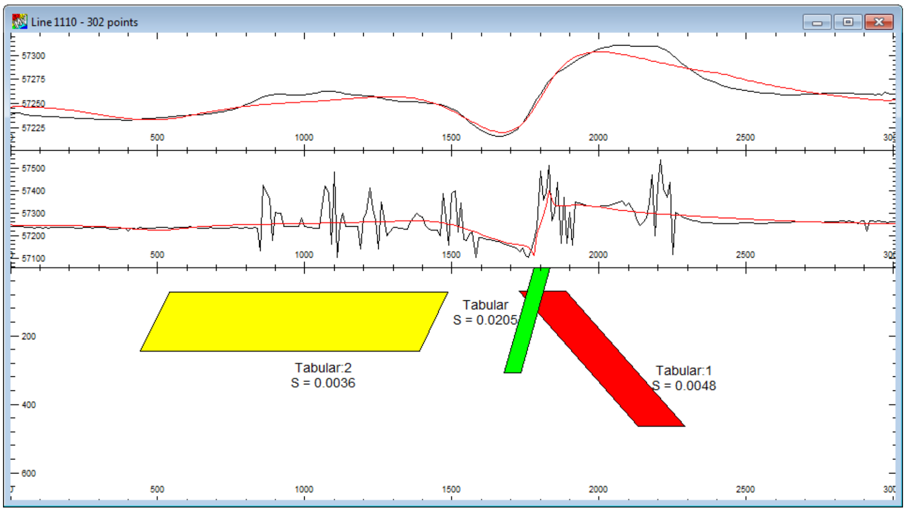


Use the **Free** selection to display the Inversion – Free Paramaters toolbar which controls which parameters to invert on.

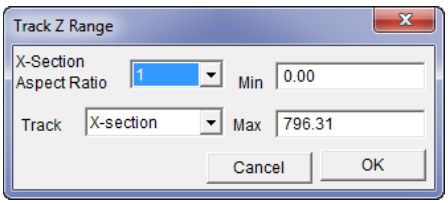
Note Use the ability of selecting a body (by placing the cursor over a body, and left mouse clicking) to nominate it for inverting. The name of this body will appear at the top of the Inversion Free Parameters dialog. You can then cycle through the various bodies without having inversion operate on all bodies at once.



Note Only permit parameters that are pertinent to a single line of data to be free in this case. Since only a single line of data is available, it is inappropriate to allow the inversion to model parameters such as the Y body location, or the Azimuth. These parameters require additional line data on either side of the bodies to permit their estimation.



As the fit and model parameters are improved, free additional parameters and then move to select additional bodies to improve their fit. At the end of the modelling exercise, would you be confident to drill this target? Where would you drill? Create a cross-section with 1:1 aspect print ratio to scale the depth and horizontal position for the target body.



Access the **Track Z Range** configuration dialog by clicking the right mouse button on the **+Curves** layer in the **Cross Section Layers** table and selecting **Configure** from the pop-up menu. Use the **X-Section Aspect Ratio** set to 1:1 to ensure a vertical to horizontal presentation aspect ratio that permits scaling directly from a section print.

Step 14

Try printing the generated display to a file. To do this, use the Print Setup option and then output the file using the Print option.

Note Scale and height controls are available in the Print dialog.

Tutorial 4 – Multi-line Displays

Aim

The aim of this tutorial is to introduce the concepts of multi-line displays, grids and modelling. Regional and residual data are also used to examine modelling procedures.

The modelling exercise is designed to:

- Examine and assess the data initially
- Estimate the regional field within the survey area.
- Produce simple models to simulate the observed magnetic and gravity responses. When a satisfactory forward model has been developed, examine inversion to refine the results
- Propose a drill site for testing the developed model of the magnetic and gravity source which is coincident with the geochemically anomalous body in the south-west corner of the survey.
- Prepare a Layout window presentation of your interpretation.

Intended User Level

Medium and experienced refresher tutorial.

Background Information

Geologically, the area contains two discreet anomalies adjacent to a NNW-SSE trending dyke. An anomalous body in the south has high copper-gold geochemistry draining south and a drill target on this feature is required. The area surrounding the individual bodies contains metasediments with low magnetic and gravity response. A regional trend is noted to pass through the area. Note that certain cultural features have forced the survey lines to deviate from being straight traverses. This can be determined by examining the data point locations (or flightlines). It should be recognized that analysis of data profiles such as these, is usually augmented by geological knowledge, to some degree. All knowledge of a prospect should be considered when developing models and assessing survey data.

The data file is an ASCII sequential line file containing four semi-parallel data lines with 63 points on each line. Contained in the file are AMG coordinates, diurnally corrected magnetics and Bouguer gravity values. The Bouguer density used in the reduction of these values was 2.67 g/cm^3.

Tutorial Data

The contents of file `4LINES.XYZ` are to be used for this tutorial. The data has been extracted from a ground survey where both magnetic and gravity data has been acquired over a group of four parallel traverse lines. Note that if you are using this Tutorial while operating ModelVision in Demonstration mode, load the data from `TUTE4.SES` (using the **File>Open>Project** option).

Tutorial Steps

The following tasks should be undertaken with this data:

Step 1

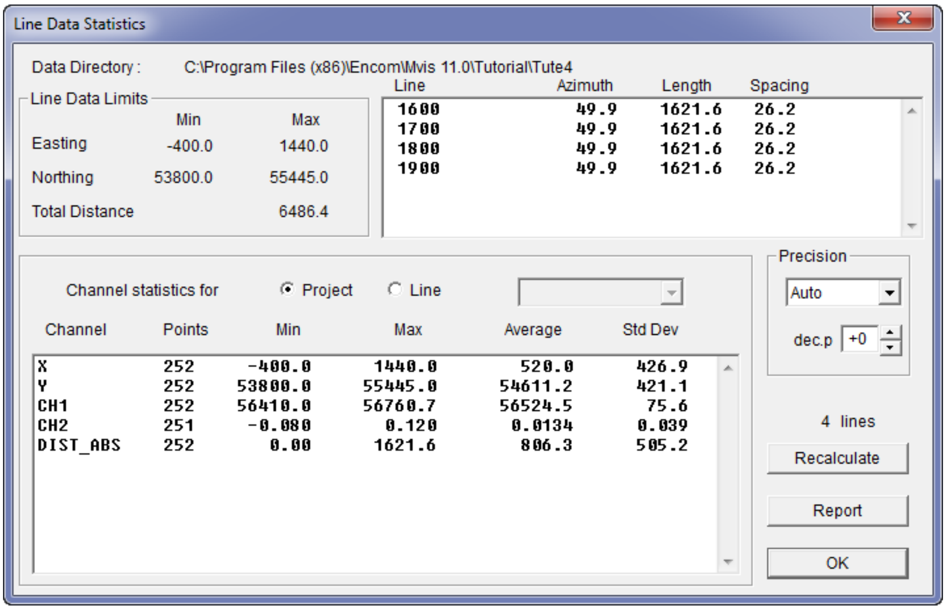
Open the project file in the `\MODELVISION_TUTORIALS\TUT4` folder and then select the **File>Import>Profiles>Geosoft Multi Line (XYZ)** menu option to load the data file called `4LINES.XYZ`. Note that the file extension is XYZ (this indicates the data type is Geosoft multi-line. Use a text editor or alternatively use the DOS Type command to examine the file prior to importing this. A portion of the data for the file appears as below:

Line 1600

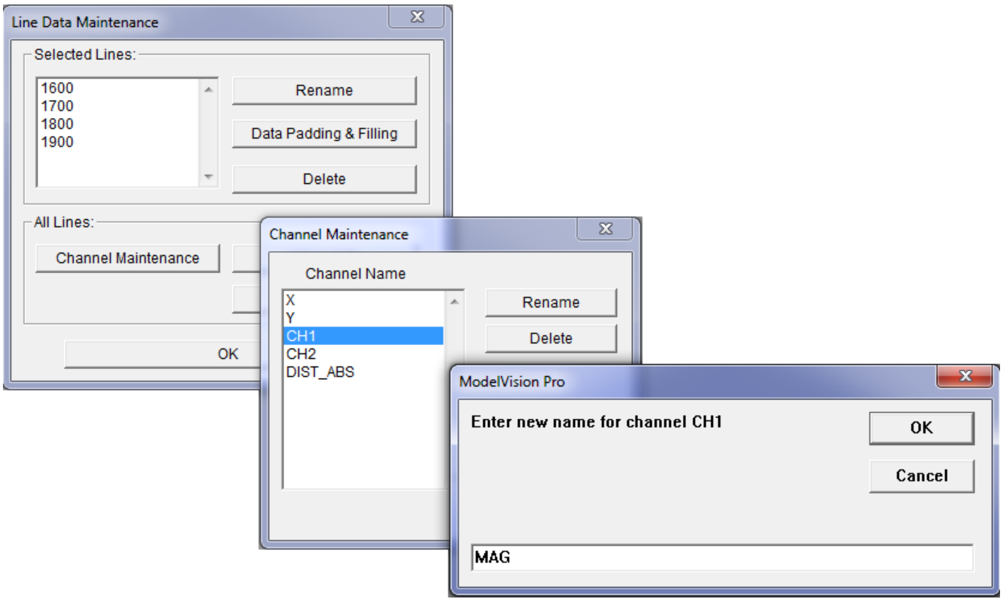
\ (EAST)	(NORTH)	(MAG)	(GRAV)
200	53800	56481	-0.01
220	53810	56479.5	0
240	53820	56478.4	-0.003
260	53830	56477.5	-0.001
280	53840	56477	0
300	53850	56476.8	0
320	53860	56476.7	-0.001
340	53870	56476.7	-0.003
360	53880	56476.7	-0.005
380	53890	56476.7	-0.008
400	53900	56476.4	-0.011
420	53909.2	56476	-0.014
440	53917.6	56475.4	-0.017
460	53926.4	56474.5	-0.02
480	53936.8	56473.3	-0.022
500	53950	56471.9	-0.023
520	53968.2	56470	-0.023
540	53989.6	56467.8	-0.022
560	54010	56465.5	-0.021
580	54030	56463.3	-0.019
600	54050	56461.3	-0.016

Step 2

Once the data has been imported examine the extent and range of the data using the **Utility>Statistics>Line Data** menu option. Note that the Z-Channels are labelled as CH1 and CH2 (Magnetics and Gravity). This is because the Geosoft multi-line format does not label the channels in the file in other than in a comments line (indicated by the backslash in the first column).



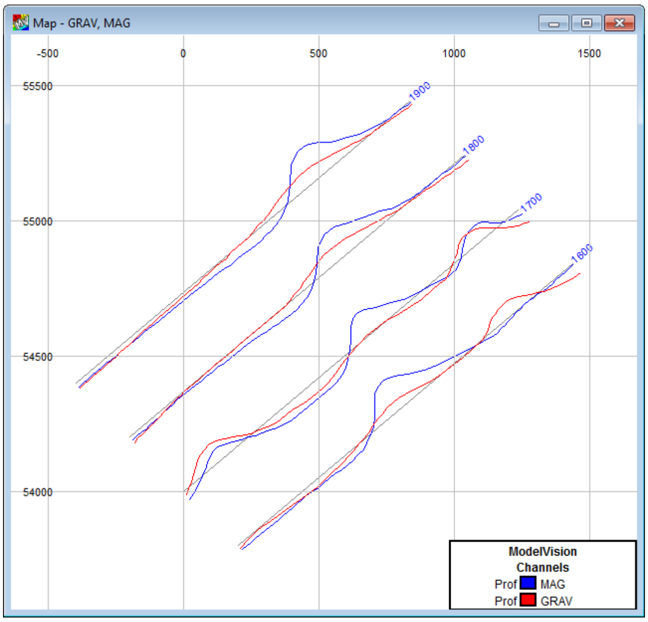
As it is more convenient to have channels with appropriate channel names, use the **Utility>Data Maintenance>Line** menu option to rename the channels as *MAG* and *GRAV*. Click on the **Channel Maintenance** button in the **Line Data Maintenance** dialog to view the channel names. Then highlight the channel to be renamed and select the **Rename** button to enter the new name.



Step 3

Display the *MAG* and *GRAV* data channels as Stacked Profiles using the **View>Map>Stacked Profile** menu option.

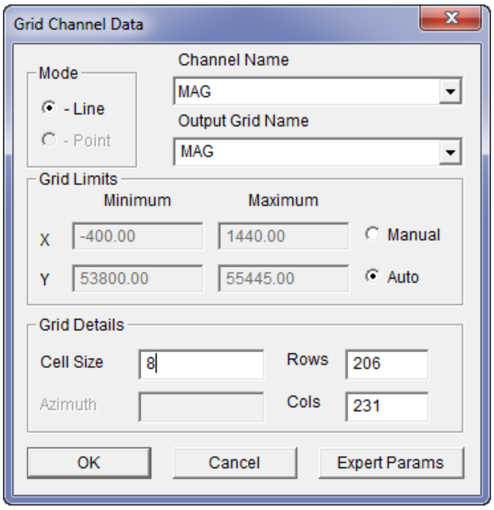
To adjust the vertical scaling of these channels make the map display window active and click the **Layer Table** button in the main toolbar to view the **Map Layers** table. Then click with the right mouse button the *Stacked Profile Channels* layer in the table and click on the **Configure** option from the drop down list beneath the object types. The Scaling can then be adjusted from the pop-up menu that appears. A vertical scaling of **200 units/cm** for the MAG and **0.2 units/cm** for the GRAV will highlight the magnetic and gravity profiles on the display as shown below.



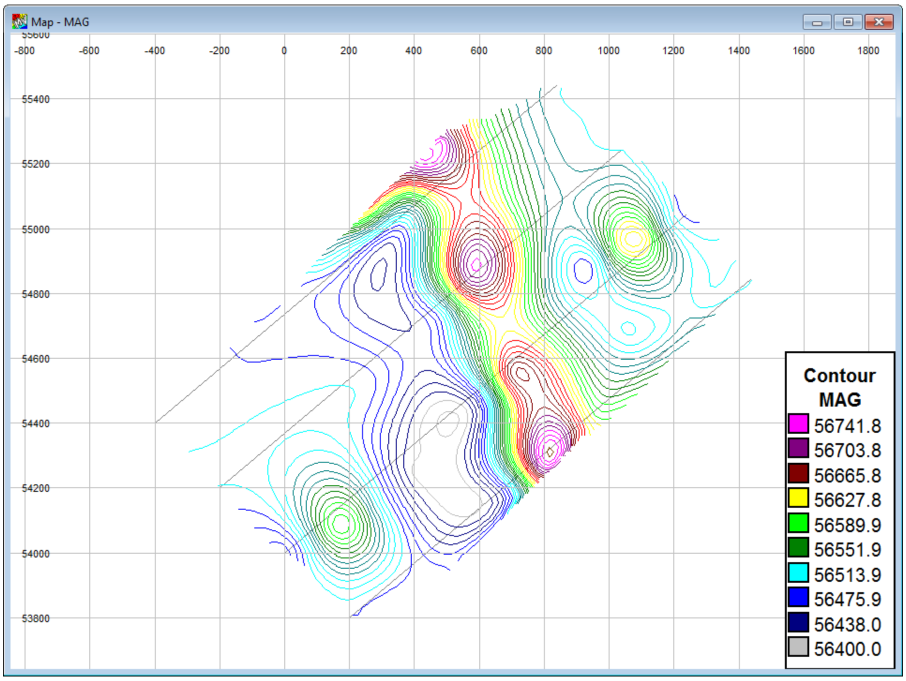
Note The position of the anomalies and also note the position of the traverses and their orientation (turn on the FlightLine layer in the Map Layers table to reveal the track of the traverses).

Step 4

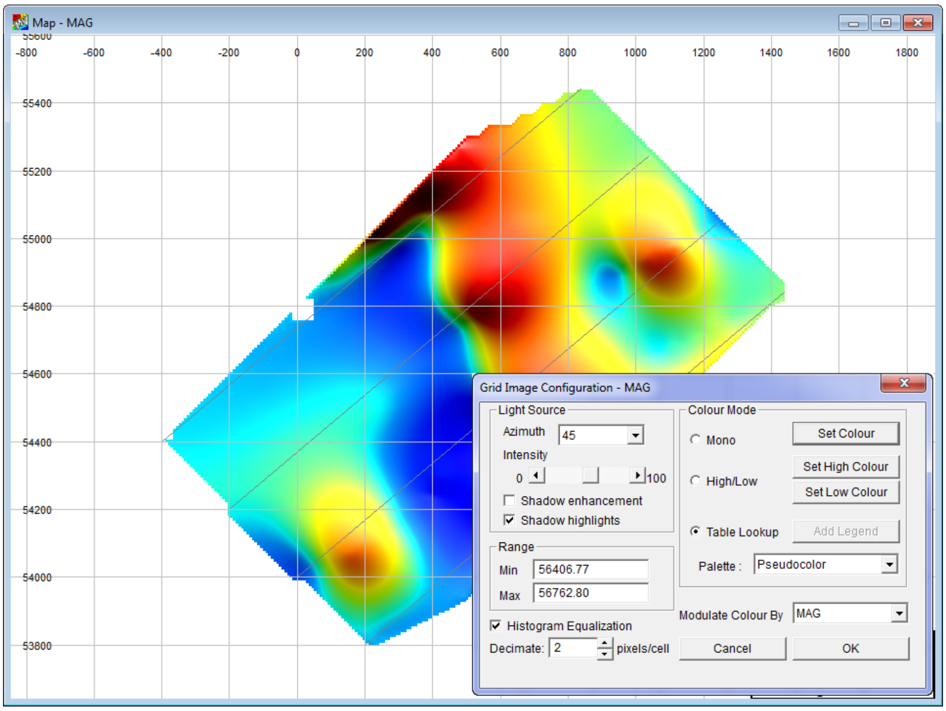
The data can also be presented as either (or both) contours or an image. To do this a grid must be initially created using the **Grid Channel Data** option in the **Utility** menu. Nominate an appropriate grid cell size and create a magnetics and gravity grid. For additional information on gridding, review the Display Tutorial (Tutorial 5).



Display the grid as a contour map by selecting the **View>Map>Contour** menu option. Display the **Map Layers** table (right click on the map and select the **Configure Layers** pop-up menu option) and with the *Contours* object type selected with the right mouse button click on the **Configure** option in the pop-up menu that appears to display the **Configure Contour** dialog. Apply a **Table Lookup** *Colour Mode* with *Palette* of **Solid Contour** and click on the **Add Legend** button to display a legend for these colours based on the MAG data values.



A display of the contours indicates a strong anomaly in the centre of the area, trending north-westerly. Other associated anomalies in the northeast and southwest are also noted.Display the same grid as an image (**View>Map>Grid Images**) and vary the display attributes by altering the decimation, sun angle, colour and data range.



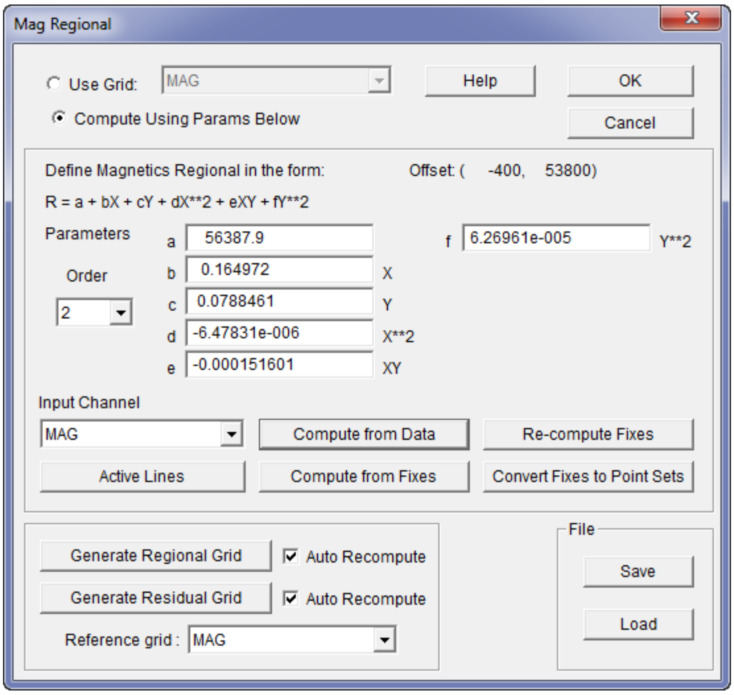
Imaging the grid is a fast method of assessing the data and developing an initial starting point for on-going interpretation.

Note On some computers with video display characteristics which are only capable of presenting 256 colours, the imagery may not be ideal. This is due to the fact that of a 256 colour capability, only 16 true colours (at pixel resolution) are capable of being displayed – the other 250 colours are dithered which is not useable by image presentation. The only method of improving this is to increase the number of available colours in the hardware.

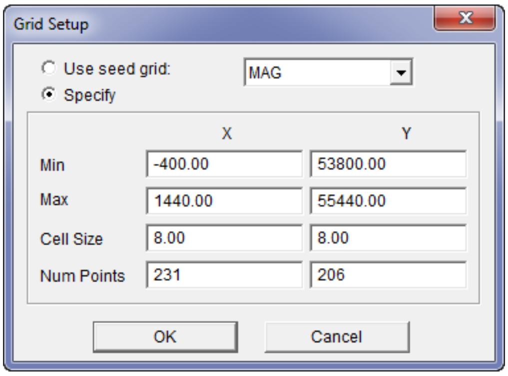
Step 5

From either the contour or image displays, it is apparent that a regional effect is present in the data. To assess the regional effect, a regional surface can be generated and then, if necessary, extracted from the line or grid data to improve the data for modelling.

Select the **Model>Edit Regional>Magnetics** option to create a magnetic regional surface. It is possible to create a regional surface over only selected lines (see the **Active Lines** button). In general practise, you would assign only those lines which are representative of the background regional effects and not influenced by local anomalism when choosing the Active Regional lines.

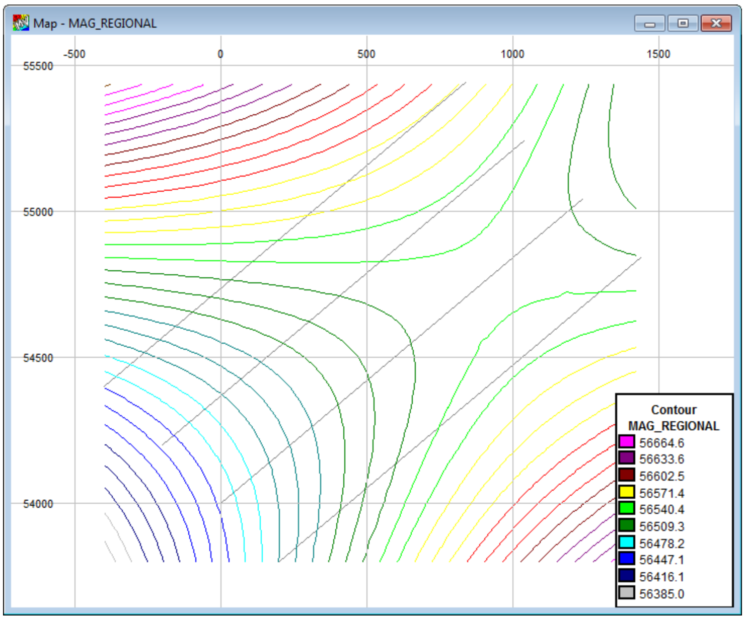


Note You must define a polynomial order to apply to the selected data when creating a regional surface. Generally you will need to use the lowest order polynomial which suits the 'geological' regional. After defining the lines to operate on, and the polynomial order, select the **Compute from Data** button to create a regional. Note that the polynomial parameters computed are displayed and these define the surface created. If you wish the surface to be also represented as a grid, select the **Generate Regional Grid** button.



Note To this point the tutorial steps have concentrated on analysing the magnetic data and only minimal processing has been undertaken on the gravity data. Similar display and regional analysis steps to the gravity data could be undertaken. Duplicate menu items are provided in the various ModelVision menu options and dialogs to manipulate the gravity data.

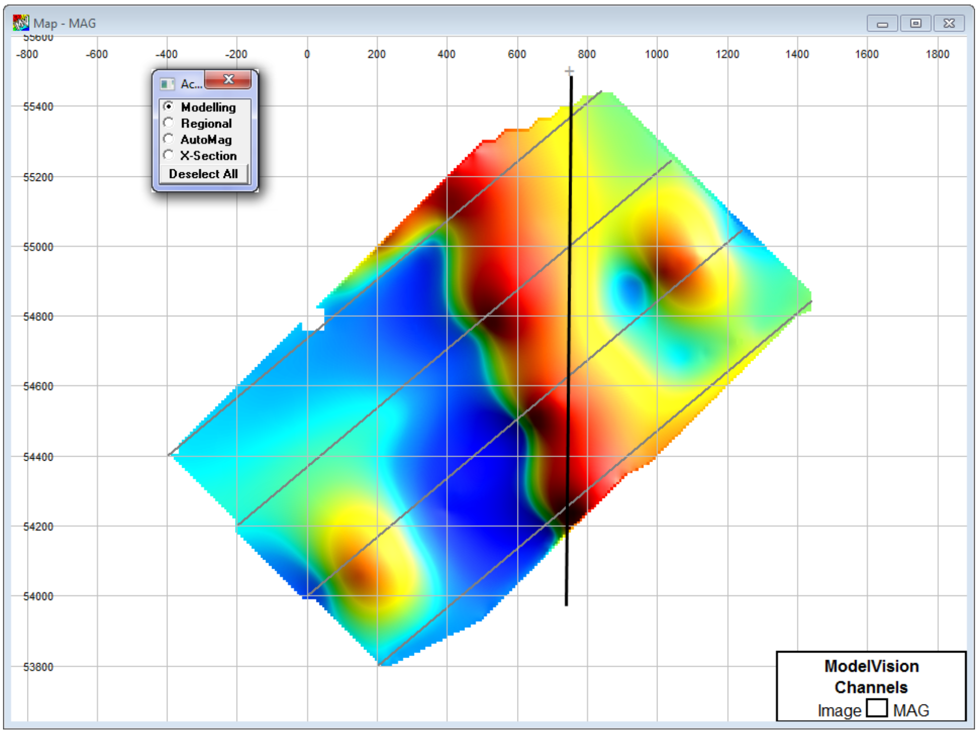
Display the regional grid as a contour or image. You will be able to tell from the display whether the regional grid is representative of the regional effect.



Step 6

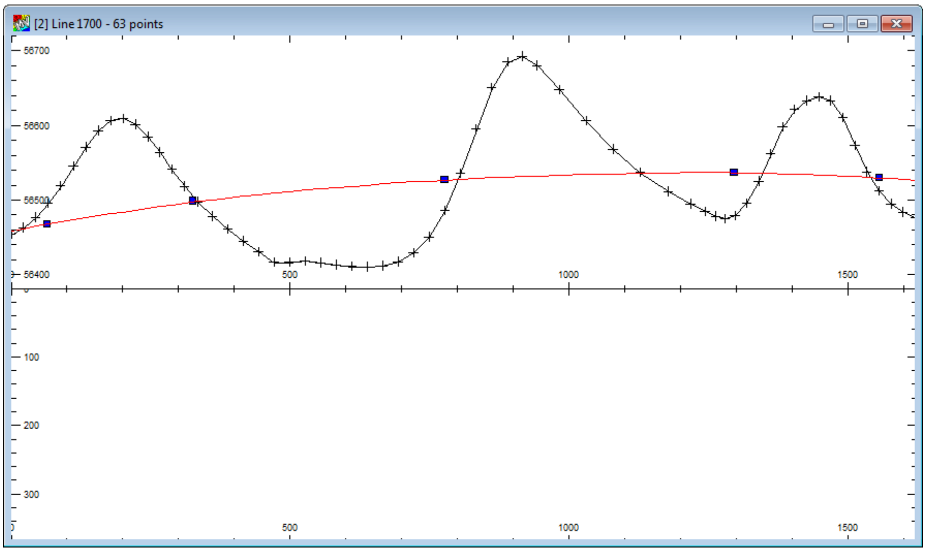
In preparation for modelling, use the **Line Control** option in the **Model** menu to control the channels to be modelled (MAG and GRAV and regional channel - MAG_REGIONAL). Enable all of the lines available to be modelled (**Select Lines** button) and turn on modelling for both Magnetics and Gravity.

Making lines active for modelling can be done either from the **Select Lines** list of the **Model>Line Control** option, or by using the **Active Line** toolbar button and dragging the cursor (now appearing as a cross-hair) over the lines to be made active.



Step 7

Use the **View > X-Section** option to display the observed data with the regional enabled and the computed response (which initially will overly the regional curve) for **Line 1700**.



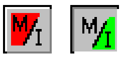
Step 8

In all magnetic modelling, it is important to use the correct value of the Earth's magnetic field within the area being analyzed. In the area studied here, the following field parameters should be used:

Earth's Total Field:	56500 nT
Inclination:	-56 degrees
Declination:	5.4 degrees

The setting of these parameters is undertaken from the **Model>Magnetic Field** menu option or else in the Project Properties dialog (**File>Project Properties**).

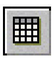
Step 9



Enable the Immediate Computer (green) mode button and create some Tabular bodies which will simulate a measured response. Refer to Tutorial 3 for additional information on creating bodies. In the example provided here, the modelling has been undertaken on **Line 1700N** since in the south-western portion of this line an anomaly exists.

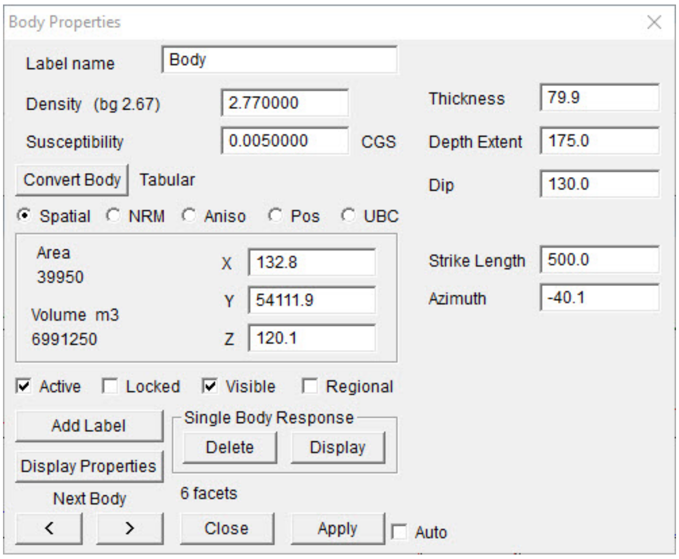
Step 10

Investigate the use of Single Body Responses, and In-Line Filters. The Single Body Response option displays the theoretical responses of an individual body.

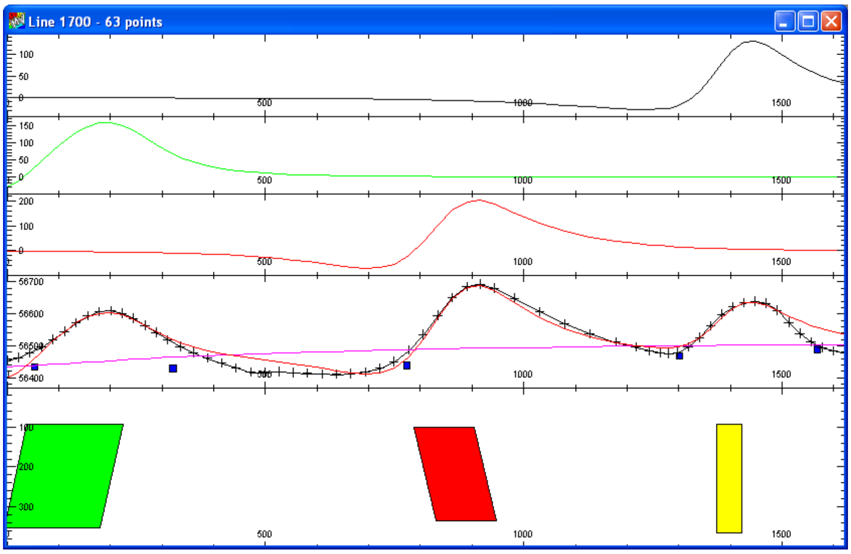


Firstly, access the Cross **Section Layers** table by clicking on the Layer Table button in the main menu and click on any layer in the table with the right mouse button to select Window>**Single Body Response** track.

In the X-Section window position the cursor over the first body. Double click the left mouse button and display the **Body Properties** dialog. Select the **Display** button for the *Single Body Response* group item and when OK is clicked to close the dialog the cross-section re-displays with an additional track containing only that body's theoretical response.



Repeat this for the other two bodies being modelled and the cross-section display should look like the image below:



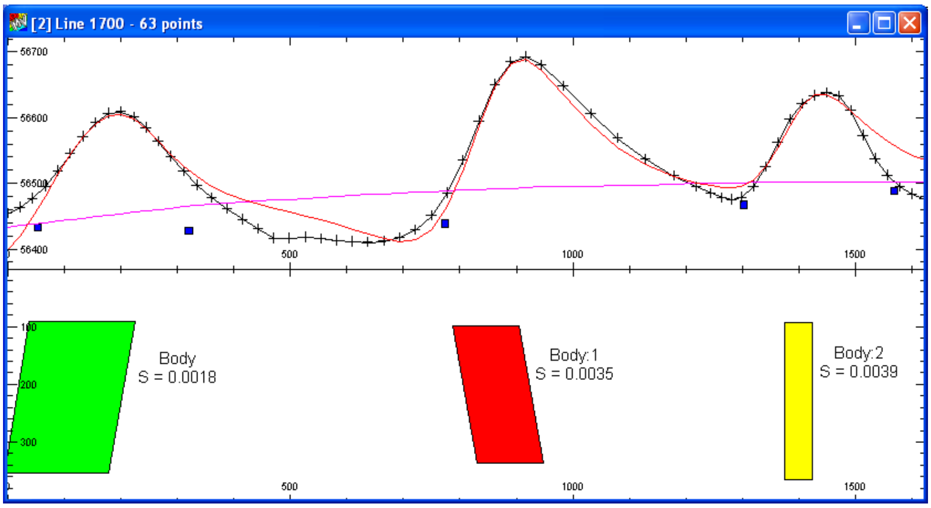
Step 11

To refine and complete the modelling, implement inversion and carefully select parameters to be free starting with only the Property and Z value (depth to the top of the body). Modify these parameters and develop a feeling for the sensitivities of varying the values. Use tolerances if required to limit the range of inversion solutions.

Refer to Tutorial 3 for an introduction to inversion.

Step 12

Progress through the lines sequentially until optimal fits for each traverse are obtained.



Another technique worth investigating is to create a grid of the residual and set the **Grid Control** option (in the **Model** menu) to enable re-computation of grids after editing of the model is performed. This technique requires that the residual contours are minimized so the differences between the computed and observed responses approach zero. If the time to compute after each body edit becomes too long, try using the **Data Compression** option in the **Model** menu.

Step 13

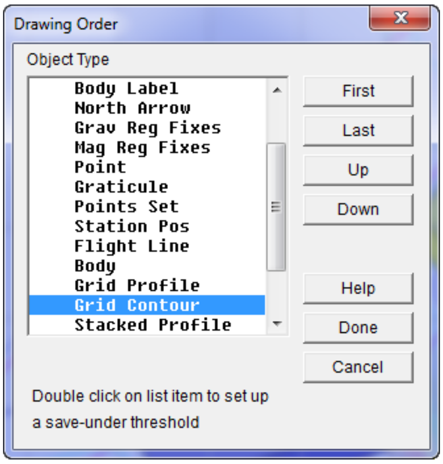
When satisfied with the obtained model, estimate a drill site for the south-western anomaly. Note that an optimal location is approximately 670E on line 1700 N, angled to the SW at approximately 60 degrees.

You can use the X-Section Aspect Ratio setting of 1 to provide a section which has the same vertical to horizontal scaling.

Step 14

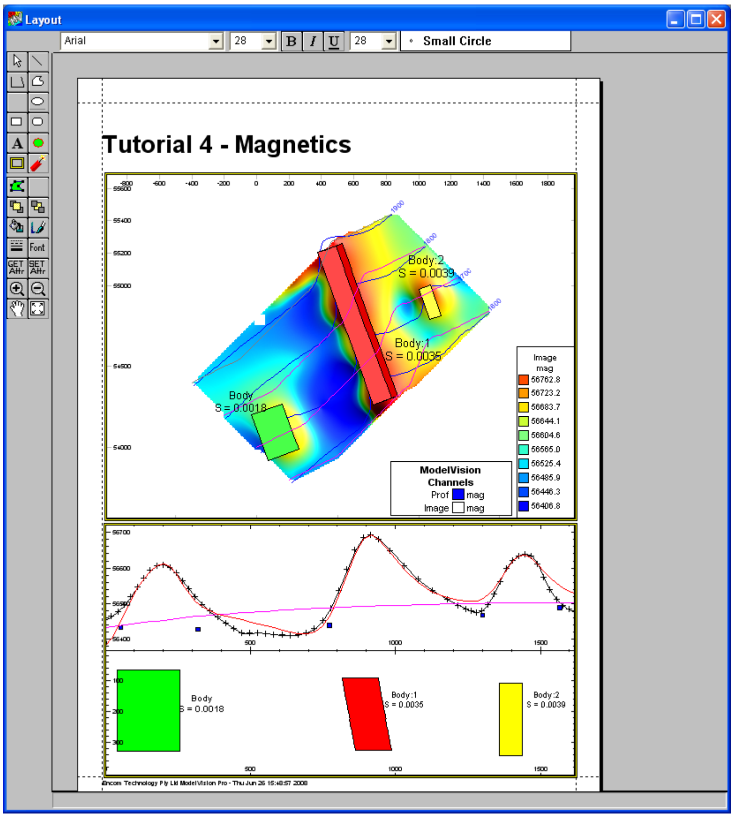
Generate a Layout window (**View>New Layout**) in which a map and section plus explanatory notes can be laid out for a presentation.

To change the order of the data objects in the map window use the **Drawing Order** dialog accessed from the **Map Layers** table by selecting the **Drawing Order** option from the **Windows** pop-up menu option, visible when any of the layers in the table are selected with the right mouse button. For example, to display the Stacked Profiles on top of the modelled bodies highlight the Stacked Profiles object in the list of the Drawing Order dialog and click the **Up** or **Down** buttons reposition the object in the drawing order.



If a printer is available produce a printout of the finished product.

Note ModelVision is not compatible with some print drivers. If a warning message appears informing you that the printer set in **File>Page Setup** or **Print** menu option is not compatible you are given the option to print to a free to download and use PDF writer software program called, PDF995.



Tutorial 5 – Gridding, Lines, Points and Displays

Aim

The aim of this tutorial is to introduce grid manipulation and data extraction techniques. Point data which is randomly acquired is frequently available in regional areas. Such data can be effectively used to assist higher resolution data interpretation. This exercise illustrates some of the 'lateral' ways in which ModelVision can be used.

Intended User Level

Introductory, medium and experienced users.

Topics Covered

- Data Import
- Grid generation and grid import
- Calculator point interpolation
- Graphical point editing.

Tutorial Data

The contents of `TUTE5` files are used for this tutorial. The files include:

`TUTE5.XYZ`	Airborne magnetics data
`TUTE5`	ER Mapper grid of airborne data
`TUTE5.ERS`	ER Mapper grid header
`TUTE5.PTS`	Ground data points of gravity
`TUTE5_GR`	ER Mapper grid of gravity data
`TUTE5_GR.ERS`	ER Mapper gravity grid header

This exercise uses two grids (magnetic and gravity), in addition to magnetic line data to illustrate how co-sampled gravity data can be derived. It is rare that gravity data is available at the same sampling density as magnetics. More coarsely sampled gravity data may, however, be available from a local and regional gravity survey. This exercise illustrates a method whereby one dataset can be used to extract data from another.

The survey area involves a semi-regional gravity dataset plus a detailed airborne magnetics survey.

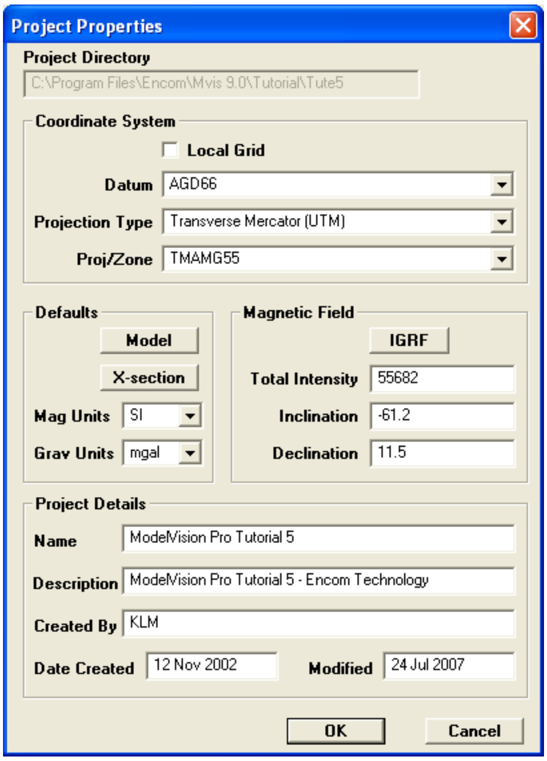
Tutorial Steps

The following tasks should be undertaken with this data:

Step 1

The basic dataset used in this tutorial is `TUTE5.XYZ`. Note that if you are in **Demonstration** mode, the `TUTE5.SES` file can be used instead of importing the necessary data and grid files. Loading the session file will also create the necessary project information.

If you wish to import the ASCII dataset (`TUTE5.XYZ`) you will initially need create a project or at least load the already existing project. To create a new Project file, select the **File>New>Project** menu option and after selecting the Browse button to navigate to the `TUTE5` folder enter the following information:



For importing the `TUTE5.XYZ` dataset, the file is a multi-line Geosoft dataset of airborne magnetics with easting, northing and magnetic data. Load the `TUTE5.XYZ` aeromagnetics line data file using the **File>Import>Profiles>Geosoft Multi-Line (XYZ)** option. The data being imported appears as below if viewed in a text editor program (e.g. Notepad):

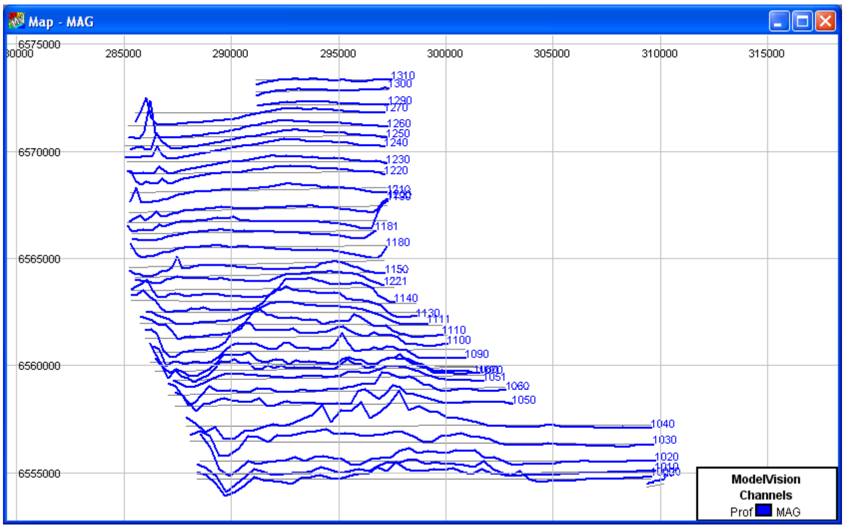
Line 1000

288399.375	6554725.5	58058.242
288841.438	6554725.5	58017.32
289283.5	6554725.5	57880.383
289725.563	6554725.5	57724.328
290167.625	6554725.5	57774.047
290609.688	6554725.5	57864.094
291051.75	6554725.5	57984.188
291493.813	6554725.5	57935.434
291935.875	6554725.5	57921.984
292377.938	6554725.5	57915.672
292820	6554725.5	57879.086
293262.063	6554725.5	57981.961
293704.125	6554725.5	57989.727

294146.188	6554725.5	58015.801
294588.25	6554725.5	57976.426
295030.313	6554725.5	57959.805
295472.375	6554725.5	58004.629
295914.438	6554725.5	58025.348
296356.5	6554725.5	58023.938
296798.563	6554725.5	58120.633

Use the **Utility>Data Maintenance**>**Line** facility to rename the magnetics channel (Z_CHAN) to MAG or another meaningful channel name. This requirement exists because the Geosoft file has no channel name header information.

Display the data in a map view as Stacked Profiles using the **View>Map** menu option.

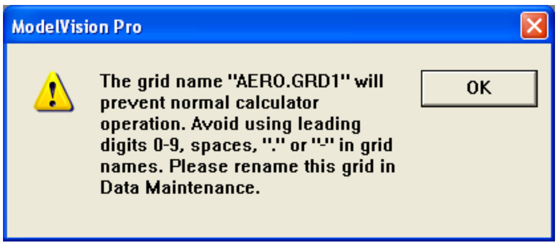


Note In the statistics for this tutorial, the magnetics have been de-sampled by interpolation to reduce the original data volume.

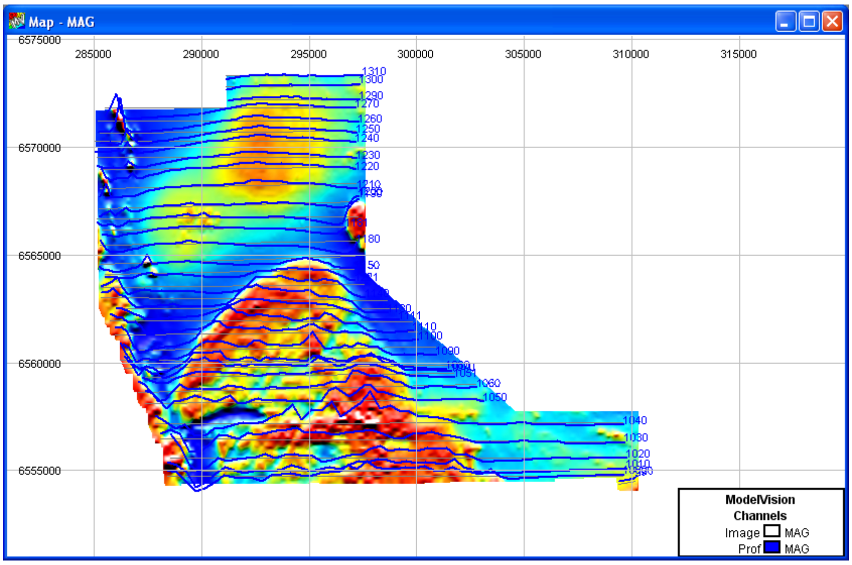
Step 2

Grids can be generated in ModelVision, or imported from other sources. In this case, a grid of aeromagnetics data is contained in the `..\TUTORIAL\TUTE5` folder with header file `TUTE5.ERS` (an ER Mapper file format).

Import the grid using the **File>Import>Grid>ER Mapper** option. Take note of the message that appears regarding the name of this grid. The message appears because nothing meaningful has been included in the description of the ER Mapper header (which is often the case).



You will need to rename the grid for use in the calculator function using the **Utility>Data Maintenance>Grid** menu option. Display the grid as contours or as an image over the stacked profiles (using the **View>Map>Contour** or in the **Map Layers** table for the map window click the right mouse layer on any layer and choose the **Add>Grid Contour** or **Add>Grid Image** button to load the newly named *MAG* grid).



Step 3

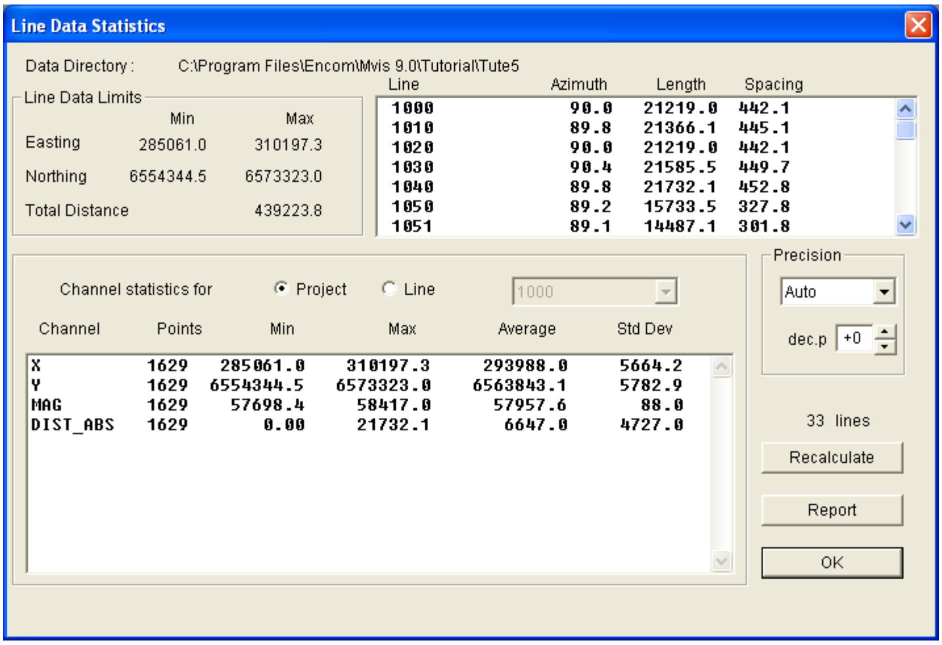
Load the gravity data points from the file `TUTE5.PTS` using the **File>Import>Point Data>Simple XYZ (.PTS)** – standard menu option. Use an editor such as Notepad or Wordpad to examine this file before importing.

Note If using ModelVision in DEMONSTRATION mode, the gravity data points have been included in the loaded `TUTE5.SES` session file.

The gravity data is similar to that below:

POINT	X	Y	Elev	ObsGrav	FreeAir	Boug2.67
1000	290317.813	6567573.000	287.12	9793745.22	29.27	-291.94
1000	285590.000	6590143.000	225.25	9793607.75	11.51	-240.49
1000	285317.688	6583827.000	242.32	9793674.09	29.44	-241.65
1000	289011.313	6562047.000	262.74	9793678.59	8.42	-285.52
1000	287788.688	6555632.500	312.42	9793608.15	-3.48	-352.99
1000	288908.688	6596862.000	320.04	9793491.49	-37.59	-395.63
1000	290597.313	6602272.000	330.10	9793354.62	-107.01	-476.30
1000	291605.000	6606471.500	376.73	9793208.64	-62.78	-484.24
1000	290722.375	6611583.500	431.90	9793254.97	181.96	-301.22
1000	289273.375	6617146.000	680.92	9792786.62	490.13	-271.63
1000	288612.625	6574692.000	646.48	9792832.84	442.14	-281.10
1000	340526.375	6615288.000	644.65	9792760.40	384.15	-337.04
1000	339921.000	6613413.500	351.74	9793290.48	126.69	-266.81
1000	337533.188	6614768.500	366.37	9793261.87	68.35	-341.52
1000	336478.188	6613494.500	427.63	9793227.25	167.20	-311.20
1000	351114.813	6609526.500	357.84	9793556.12	-38.64	-438.97
1000	348149.125	6610414.500	319.03	9793811.86	86.40	-270.51
1000	353535.125	6605875.000	312.31	9793813.76	79.57	-269.82
1000	357612.813	6603559.000	302.99	9793829.17	78.22	-260.74
1000	335965.188	6603374.000	297.73	9793827.17	72.00	-261.08

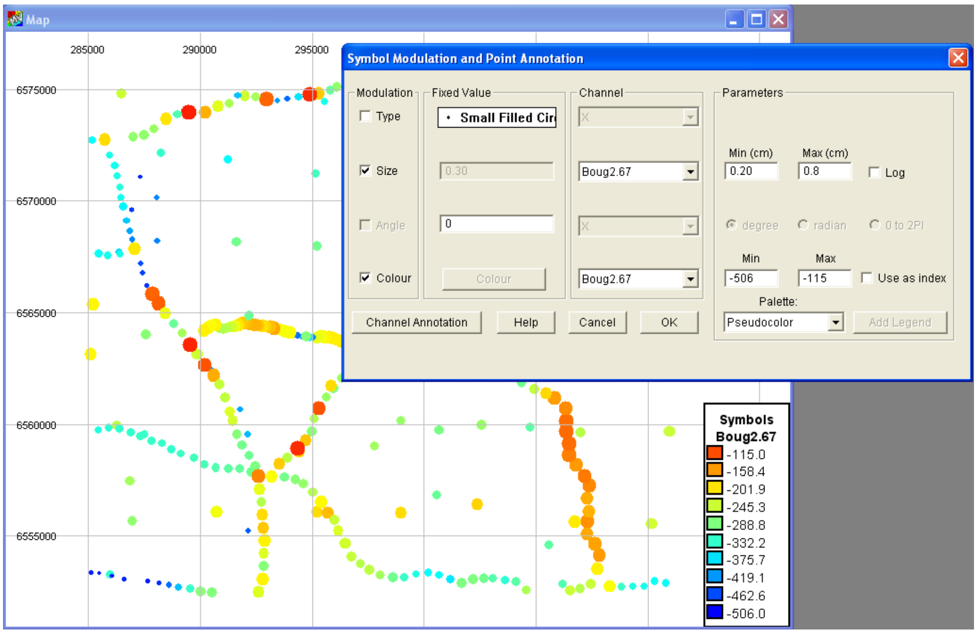
After importing, examine the statistics using **Utility>Statistics>Points Data** menu option.



Step 4

You may wish to display the data at the imported points with the airborne data grid to determine the two data coverages. However in this tutorial the points will be displayed in a separate map window. Select **View>Map>Points** and in the **Points Selection** dialog that appears move the only available points group in the *Available* list to the *Selected* list and click **OK**.

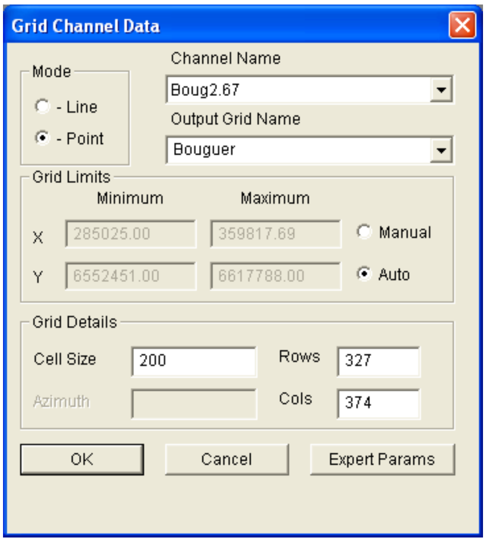
Configure the point data by modulating the colour and size in relation to the Bouguer 2.67 data values. Use the **Points Set** layer in the **Map Layers** table to alter the points appearance according to the below image.



You will see after configuring the points appropriately, the locations of the regional gravity stations.

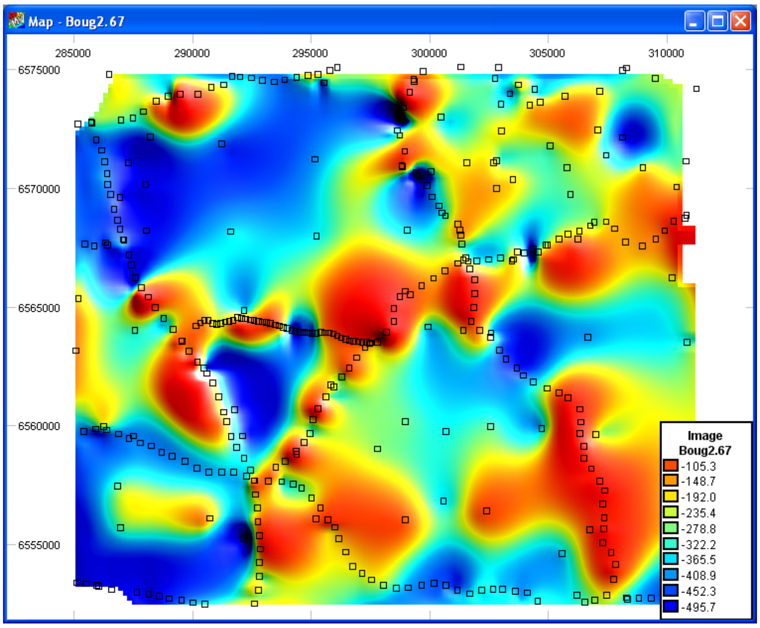
Step 5

To compare the gravity data with the magnetics in grid image form, create a grid from the gravity data points using the **Utility>Grid Channel Data** option. Grid the gravity points *Bouguer* data (Channel Boug267) using a 200 metre sampling interval.



Note To grid the points data, you will need to select the Point mode option in the top left corner of the **Grid Channel Data** dialog as illustrated above.

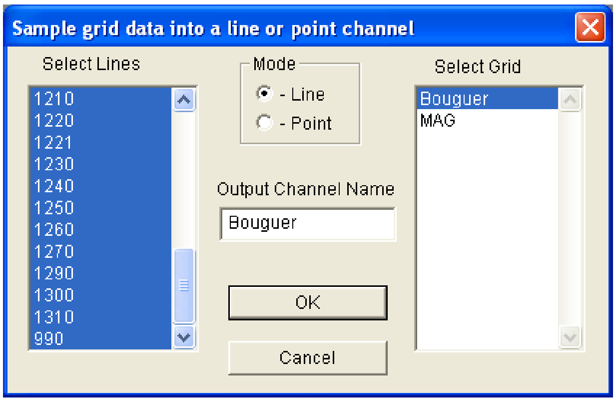
Display the newly created grid image in a different display window by using the **View>Map>Grid Image** menu option. Add the Bouguer gravity points (refer to **Step 4** above) as constant point size and constant square symbol colour.



Step 6

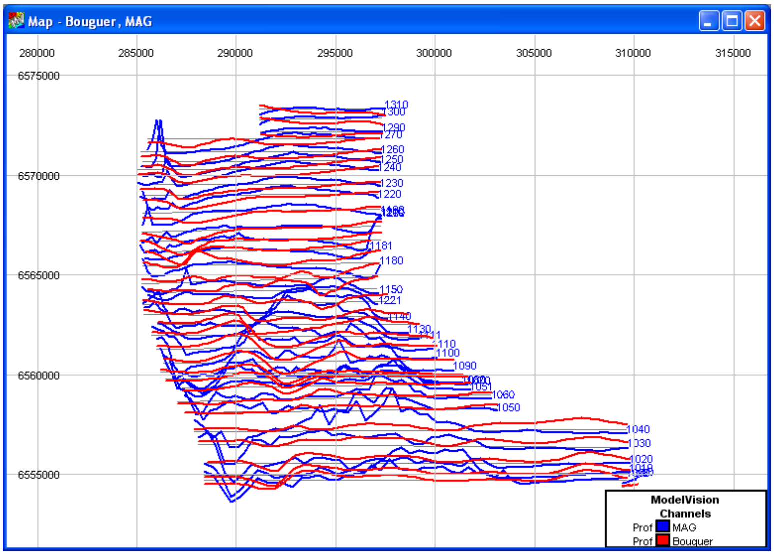
You can co-sample the reworked gravity data to the same locations as the aeromagnetic data and hence creating a gravity channel in the aeromagnetic by the following method.

Access the **Utility>Sample From Grid** menu option. Nominate the lines onto which the grid data is to be interpolated, specify the grid and output channel name and click the **OK** button.



Redisplay the stacked profiles with both the magnetics and grid-interpolated gravity data (select **View>Stacked Profile** menu option). For modelling or regional purposes, you may want to rename the BOUG2.67 gravity channel to GRAV (using the **Utility>Data Maintenance>Line>Channel Maintenance** option.

Note This approach could also be used to interpolate elevation, or any other data type from a grid to individual readings along a line.



Tutorial 6 – Regionals and Advanced Modelling

Aim

The aim of this tutorial is to illustrate the various presentation formats available and to introduce the concepts of multi-line displays, grids and modelling.

Geologically, the area contains a complex area with a number of steeply dipping interbedded volcanics trending north-south (to NNE-SSW). In some instances, the units are faulted typically with NE-SW trending structures. Granitic intrusions are evident within the data and these are typified by low internal magnetic responses. Surrounding these intrusives are magnetic aureoles and demagnetizing affects associated with intersected volcanics.

The aim of modelling and the analysis here is to identify and investigate anomalous features within the data, identify structures and provide mapping information. In particular, anomalies lying adjacent to the granitic margins are of interest since these are known sites of tin deposits associated with heavy metal mineralization.

Intended User Level

Medium and experienced users. Advanced modelling.

Topics Covered

- Data Import
- Grid generation and grid import
- Display formats
- Regional grid generation
- Multi line modelling
- Introduction to advanced modelling

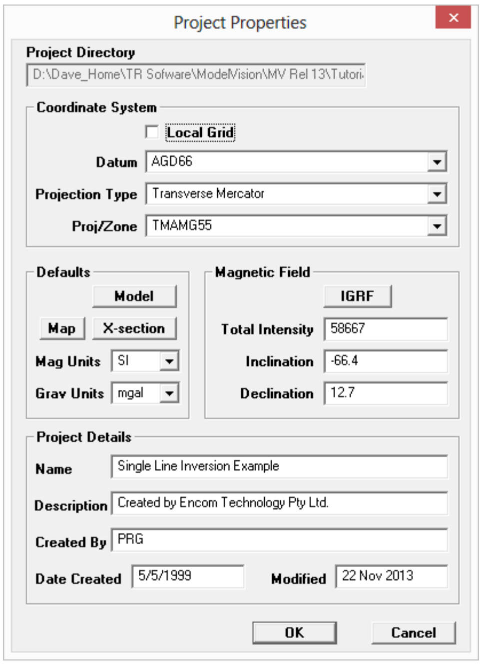
Note In DEMO mode you cannot import your own data. You can, however, load one of the supplied binary session files which contain data. For access to the tutorial you are required to load a session file.

Setting up the Tutorial

Initially, ModelVision requires a project to define location, magnetic properties and a description of the work. ModelVision stores project information in a file (`MVPROJ.INI`) which resides in the project folder. Before any data can be imported or a session with ModelVision is commenced, a project must be created.

Select the **File>New>Project** option and enter the information as required. Initially, select the **Browse** button and navigate to `.\PROGRAM FILES\ENCOM\MVIS 14.0\TUTORIAL\TUTE6`.

It is important at the commencement of a project that you know the location of the data to be used in the session. You can enter the project datum and projection information plus you can supply the location to derive the local Earth's magnetic field using the **IGRF** tool as shown in the image below.



Tutorial Background

The data file for this tutorial was acquired from a small airborne survey flown in 1989 in south western New South Wales (near West Wyalong). This survey data has been acquired with traverses that are oriented northeast to southwest with line spacing of 1200 metres (only every fourth line has been retained for this tutorial). Within the data file are Australian Metric Grid (AMG) coordinates (using AMG Zone 54), Fiducials, Magnetics, Altimeter and the spectrometer channel for Potassium (K40). The survey was over relatively flat grazing and intensely cultivated agricultural areas. Known granites, volcanics and metamorpohosed units predominate in the area with some minor basic intrusives and dyking. Extensive structural deformation and faulting exist in the survey coverage.

Tutorial Steps

The following tasks should be undertaken for this tutorial:

Step 1

Load the data with the **File>Import>Profiles>Sep. Header (HDR & LIN)** option. The data is located in `.\PROGRAM FILES\ENCOM\MVIS 14.0\TUTORIAL\TUTE6`.

The import data file is called `TUTE6.LIN` but the Separate Header option uses the `TUTE.HDR` file. This import file format illustrates how large multi-column data files can be loaded without extensive editing of the data file. The header and data file appear as below:

Header File

LINE X Y FID MAG K40 ALT

Data File

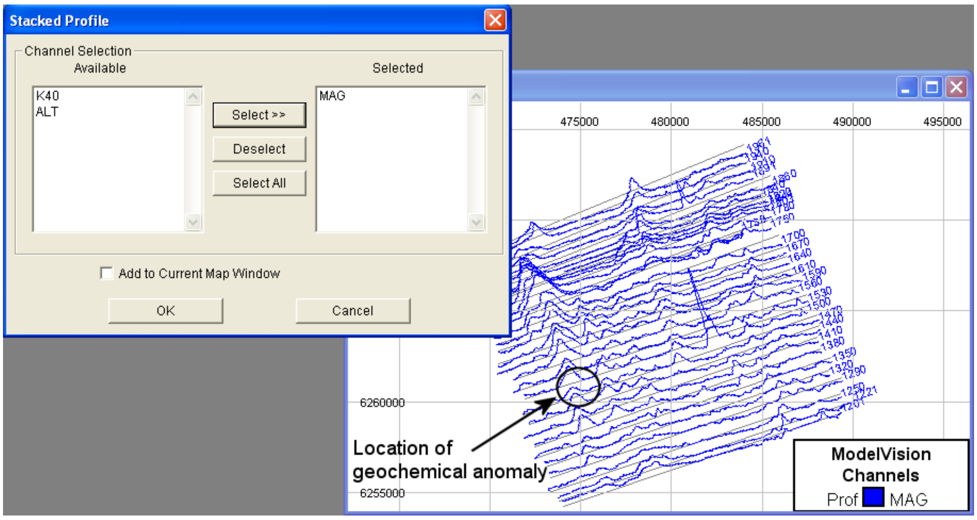
1201	489435	6259404	18612	58158.1	91	292
1201	489415	6259397	18613	58157.8	81	292
1201	489396	6259390	18614	58158.4	72	292
1201	489377	6259384	18615	58159.2	70	293
1201	489358	6259377	18616	58159.3	68	295
1201	489338	6259371	18617	58159.7	74	295
1201	489319	6259364	18618	58158.5	80	296
1201	489300	6259357	18619	58157.8	77	296

1201	489280	6259351	18620	58156.4	74	297
1201	489261	6259344	18621	58155.4	70	296
1201	489242	6259337	18622	58153.9	66	296
1201	489222	6259331	18623	58153.2	67	297

After loading the data file, examine the data ranges and statistics of the channels when the statistics dialog opens at the end of the import. You can analyse the statistics of any individual line by double clicking on the nominated line.

Step 2

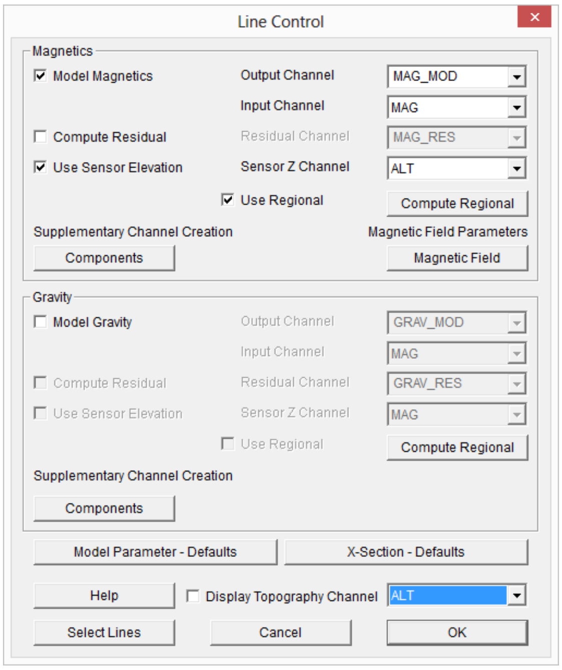
ModelVision then opens a dialog for selection of a channel for a stacked profile map display. Use the **View>Map>Stacked Profiles** option and select the *MAG* data channel. Once a map is presented you can adjust the vertical scaling to highlight features in low gradient areas. This is done by selecting the **Layer Table** button in the main toolbar to display the **Map Layers** table and then click on the **Stacked Profile** layer in the table with the right mouse button to select the **Configure** pop-up menu option. Edit the vertical scaling and highlight features accordingly.



From regional geochemical sampling, the location of an area with anomalously high tin and base metal associations is known (see above). The anomaly correlates with a magnetic trend produced by volcanics and was covered by the airborne survey along line 1500.

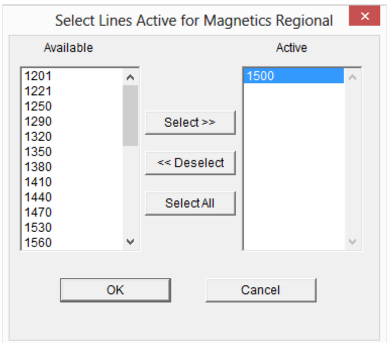
Step 3

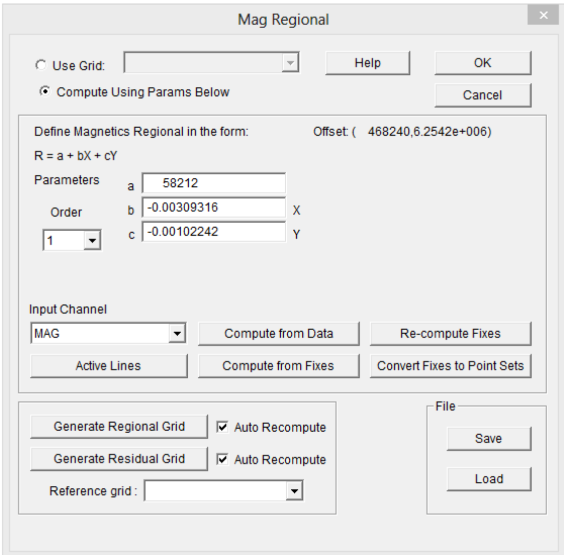
To prepare line 1500 and the anomaly for modelling, it is necessary to setup the fields in the **Model>Line Control** dialog. Assign the field *Mag* to the **Input Channel** and use the altimeter channel *ALT* for the **Sensor Z channel**. When you use the altimeter channel, the model depths will be in metres below the ground surface rather than absolute elevation above sea level.



There is a broad regional background trend superimposed on the data so we will need to remove this from the data with the regional modelling tools. In other circumstances, a two-dimensional regional surface can be created and used for multi-line modelling.

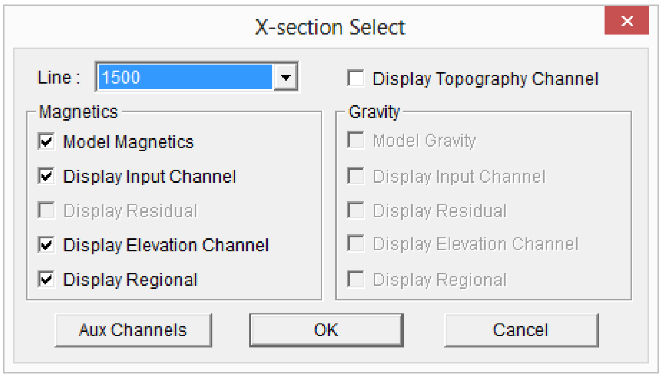
Check the **Use Regional** checkbox and then activate the **Compute Regional** button. Using the **Active Lines** button, nominate line **1500**. Once selected, press **OK** and click the **Compute from Data** button using a polynomial order of 1.



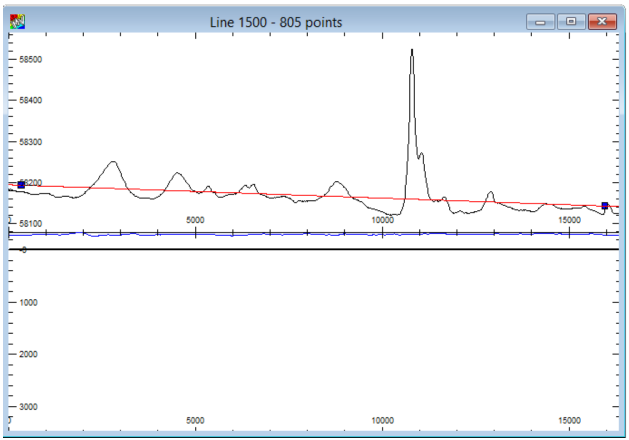


Step 4

With the regional created, you can now display line 1500 with a track beneath to be used as a cross-section below the flight line (use the **View>X-Section** option). The displayed dialog allows you to select line 1500 from a pull-down list. Note also that the dialog enables magnetic modelling on the line and uses the regional in the computation of magnetic body responses.



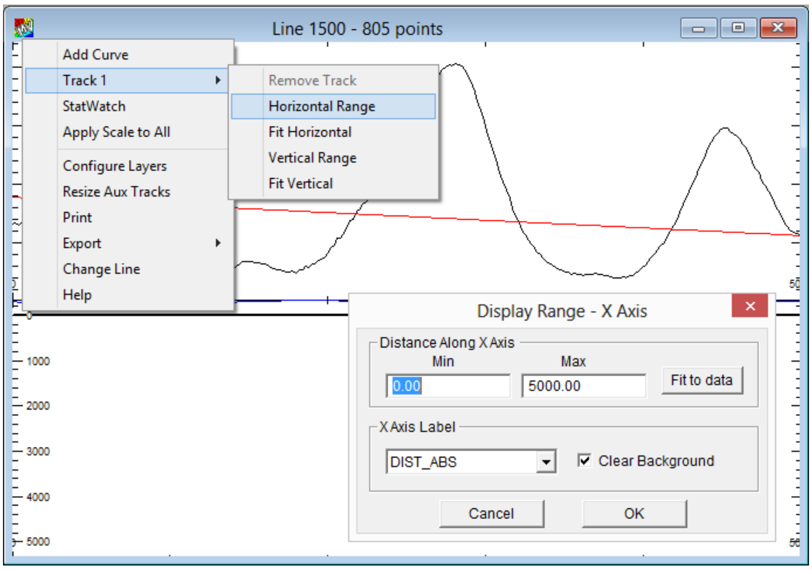
Once specified, click the **OK** button. Line 1500 is displayed in a profile window with a cross-sectional area beneath. This area is used to create and edit the magnetic model. The model response is superimposed on the regional trace (indicated by three regional 'handles').



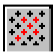
The location of the magnetic anomaly is at the left margin of the profile. In preparation for modelling, you need to zoom into the anomaly. You also need to instruct ModelVision that only this anomaly is to be modelled and not the whole profile.

Step 5

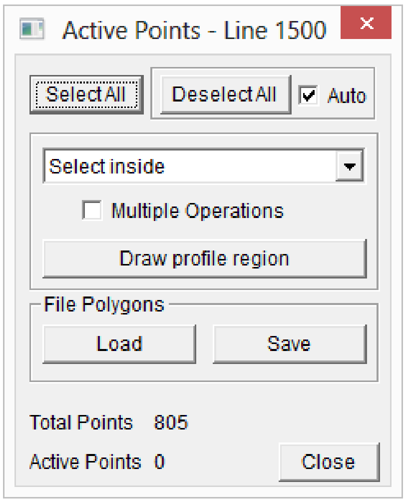
You can zoom into the target anomaly by using the zoom button, but in this case we will set the Zoom range manually. Use the right mouse click to access the **Configuration>Track1>Horizontal Range** dialog. Specify the Min and Max range of the **Distance Along** Profile to be between 0.0 and 5000.0.



You may wish to adjust the Z range also to use more of the space provided. To adjust the amplitude of the profile right mouse click on the top half of the x-section window and for **Track 1>** select **Fit Vertical**.



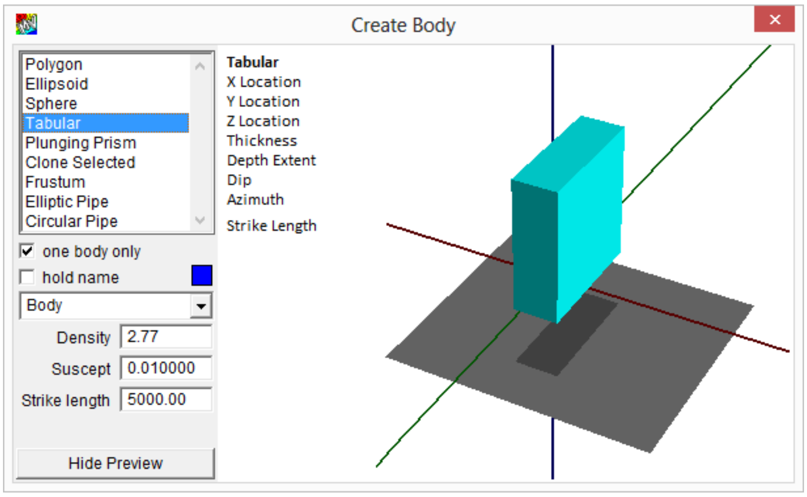
Make the target anomaly active for modelling and inversion by using the **Active Points** button. Select the **Draw Profile Region** button and position the cursor in the profile window at the start of the area to be modelled. Click the left mouse button and drag an area along the profile for modelling. Ensure the complete anomaly is defined with any side lobes that may affect the source modelling.



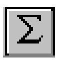
After the bounding lines of the active region have been defined, release the mouse button and you will notice the model response curve changes colour.

Step 6

To create a model and compute a magnetic response, select the **Create Body** button. As the geological source is not defined, the simplest body type should be used initially. From the Create Body dialog, select the **Tabular** body.

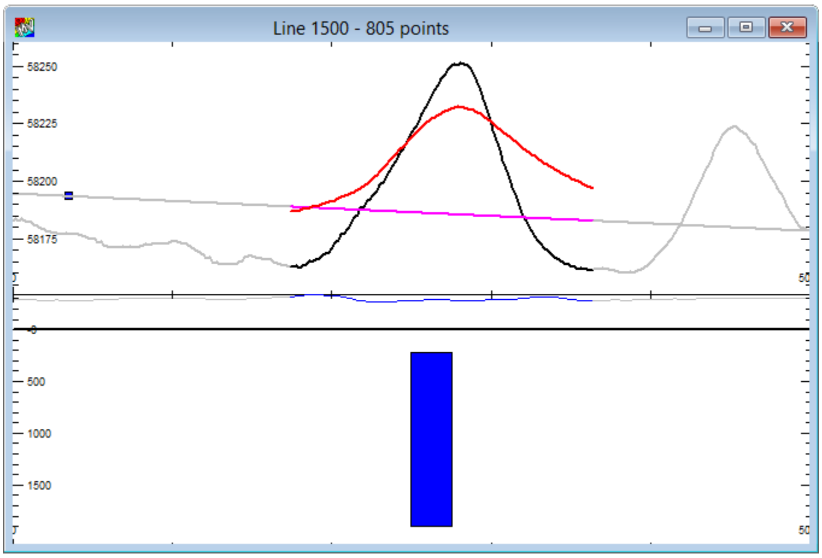


Note that the dialog displays the current default properties that will be assigned to the body and a preview of the body style. Once the tabular body has been selected, position the cursor in the profile cross-section, click the left mouse button and drag a rectangle that will form the body outline. When the button is released, the body is created.

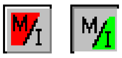


You can display the magnetic response due to the created tabular body by clicking on the **Compute** button.

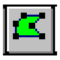
Only the response along the nominated active points of line 1500 is computed as no other readings in the dataset have been selected as active. To adjust the display depth for the x-section window right mouse click in the bottom half of the display and select the **Track 0> Vertical Range** pop-up menu option that appears. Adjust the maximum Z Range to 2000m.



Step 7

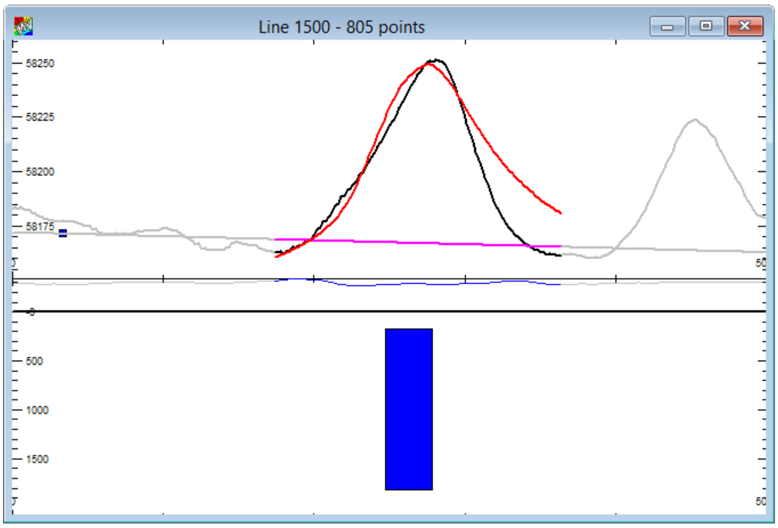


Initially, the body location and orientation is unlikely to be correct for a match between the computed magnetic response and the observed data. You can edit the body, magnetic susceptibility and regional until you get an approximate match between the model response and the original data.. After each edit you can force ModelVision to update the computed response by toggling the Manual/Immediate mode of computation.

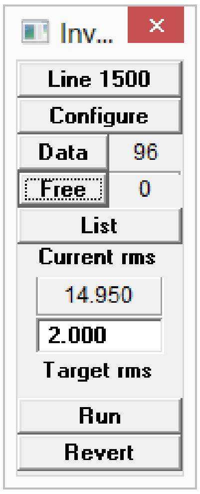


The tabular body can be positioned by selecting with the cursor and locating as appropriate. Its width can also be modified interactively by selecting a corner handle and dragging. You need to have ModelVision in the Pointer mode for these operations (select the Point icon or button). By using the **Reshape** button you can edit the dip of the body.

Modify the location and orientation until there is a close match to the anomaly shape. You will also need to adjust the magnetic susceptibility to obtain an amplitude match. Note that there is a level shift between the observed and theoretical traces. You can select the left had regional handle and drag it down so that the shift is minimised. The fastest methods of removing the level difference and improving the model fit is to use inversion.



Step 8



The use of inversion in ModelVision is a powerful tool for rapidly refining models.

To initiate inversion, select the **Tools>Inversion** option (or click the right mouse button when the cursor is in the ModelVision screen window). The Inversion toolbar is displayed and can be positioned on the screen.

Note that you are using standard inversion rather than join inversion for this tutorial.

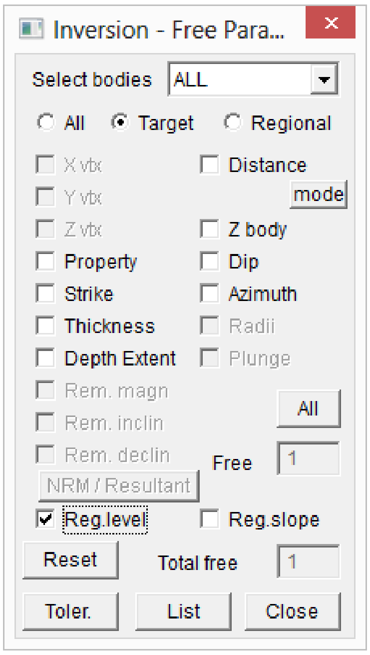
The tabular body type is defined by a set of parameters, each of which can be freed during inversion. The more parameters that are set free, the easier it is to get a match with the field data. This is not generally the best approach because the smallest number of free parameters provides the optimum approach. This process lets you evaluate the maximum geological information that you can infer from the magnetic anomaly.

The logical order for constraining a tabular body magnetic inversion is as follows:

DC regional
Position (distance) or X, Y
Magnetic susceptibility
Depth
Thickness
Dip
Regional gradient
Depth extent.

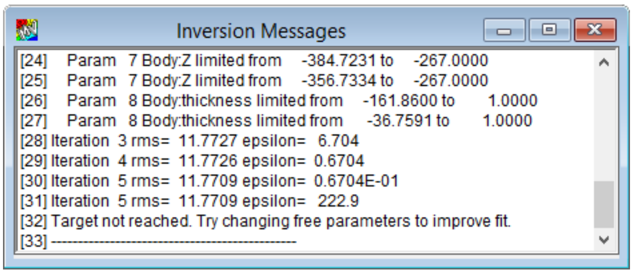
The azimuth and strike length are normally determined by inspection of the map. Note also that magnetic susceptibility and thickness trade off against each other during inversion and unless the body thickness is greater than its depth, it is not possible to uniquely resolve either property.

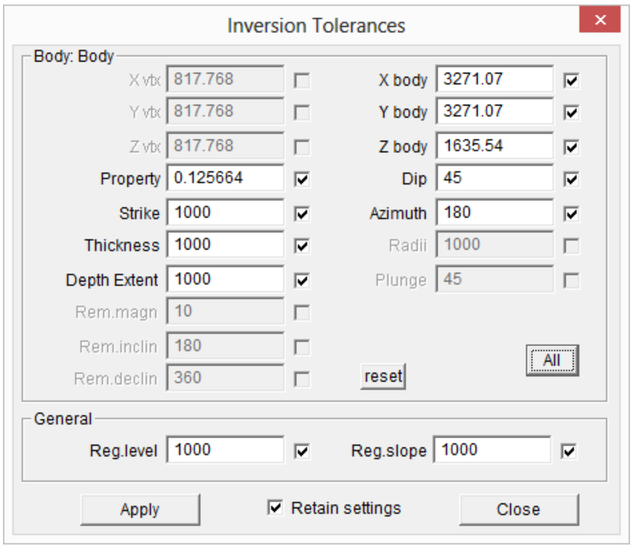
Select the **Free** button to open the Free Parameters dialog and set the **Regional Level** check box to on and then **Run** the inversion. You will see a minor shift in the regional trace.



Note that the **Select Bodies** item at the top of the **Free Parameters** dialog is set to **ALL**. In this case, only one body exists but in more complex cases where additional bodies may be present, you can individually select the body and its free parameters. Set the Distance (position along the section), Z (depth) and Property (susceptibility) free and run the inversion. There will still be a significant mismatch in the model and field data.

Now free Thickness and run inversion again. If the Inversion Messages dialog indicates that thickness is being constrained too tightly, then you can use the Toler. Dialog to increase the constraints.

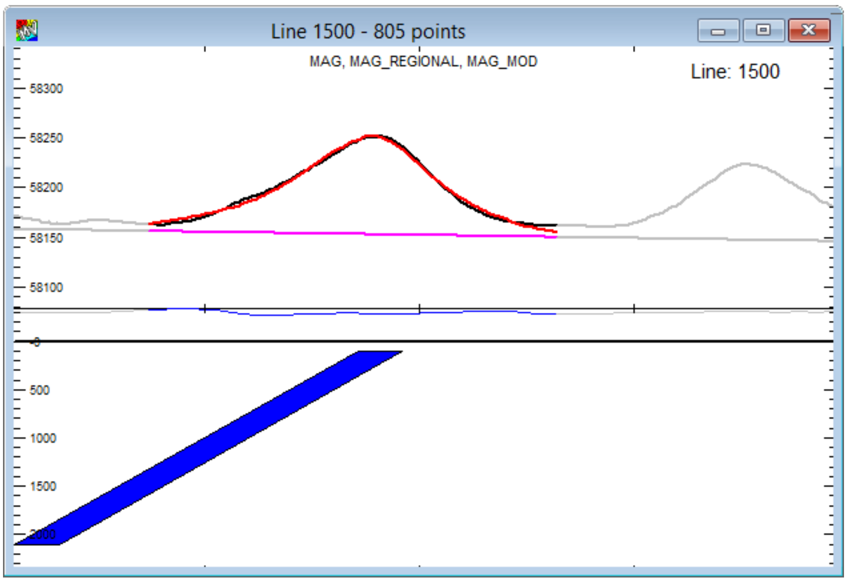




Step 9

You may need to perform a few **Runs** of the inversion to optimally fit the theoretical and observed response curves. If the appropriate parameters are freed sequentially, a good fit between the curves can be achieved quickly. This approach can save considerable time in evaluating even complex anomalies.

The final match between the modelled and observed magnetic responses is shown below.



Tutorial 7 – Modelling, Regionals and AutoMag

Aim

The aim of this tutorial is to introduce the depth-to-basement option known as AutoMag. The ease of use and rapid interpretation is illustrated by the fast analysis of a relatively large dataset.

Note This tutorial uses a ModelVision option called AutoMag. Unless you have access to this option, you will not be able proceed with this tutorial. It is possible to access the option and data if ModelVision operates in Demonstration Mode.

If you have a licensed version of ModelVision which is UNLICENSED to operate AutoMag, the option will not operate unless a valid demonstration session file is accessed (for this tutorial, `TUTE7.SES`). This provides you with the ability to experiment and evaluate AutoMag, but only on supplied, demonstration datasets.

Intended User Level

Medium and experienced users. Advanced options.

Topics Covered

- Data Import
- Noise removal
- Display formats
- Regional grid generation
- AutoMag

Background Information

The dataset to be used is from a segment of a larger airborne survey acquired in 1993 from Western Australia. There was considerable noise in the magnetic channels caused by the low altitude of acquisition (50 metres AGL) and surface maghemite known to be present in this area. The data has therefore been upward continued 50 metres to smooth the magnetic response. It is important to take note of this operation, since depths derived from the AutoMag option will need to take this additional 50 metre height offset into account.

Tutorial Data

Initially, a project is required to define the various properties of the data. ModelVision uses projects to manage various analysis exercises and a `MVPROJ.INI` records your entries for later use. Binary session files (.SES) can also be used to store data, models and displays if required.

The contents of file `TUTE7.LIN` and its associated files are used in this tutorial and can be accessed from `.\PROGRAM FILES\ENCOM\MVIS 14.0\TUTORIALS\TUTE7`.

Note that if ModelVision is being used in Demonstration mode, you can load the data from the `TUTE7.SES` file using the **File>Open>Session** menu option.

This tutorial presents an overview of the ModelVision option, AutoMag. It provides an understanding of the principles involved in using AutoMag for a rapid depth-to-basement analysis of data.

AutoMag Background

Depth to magnetic source calculations have occupied geophysicists for months and years depending upon the scale and objectives of their interpretational project. Manual techniques such as the Straight Slope method and Peter's Length have been used for depth estimation for many decades. These procedures are mechanical, repetitive and time consuming but require a skilled geophysicist to reject inappropriate magnetic anomalies.

Manual methods are still used today because they produce consistent results with judicious rejection of spurious anomalies. Computer procedures for automatic depth interpretation have become increasingly popular to supplement manual procedures. One such procedure that has become popular over the last five years is the Euler 2D method. These methods work well as a first pass assessment of the magnetic source distribution, but lack the critical input of the interpreter in rejecting spurious solutions. Invariably there are too many solutions and the relationship between the magnetic anomaly and an individual solution is unclear.

Manual methods are time consuming but produce geologically constrained solutions while fully automated methods lack the critical intervention of the geophysical interpreter. AutoMag bridges this gap by providing a consistent automated process that reduces the manual intervention and provides the interpreter with the ability to quickly test any solution against forward model solutions. Interpreted depths can be plotted in cross-section, map view or exported to an ASCII file for use in another application. Depths from the AutoMag solutions can be gridded to produce a contoured map of 'depth to basement'.

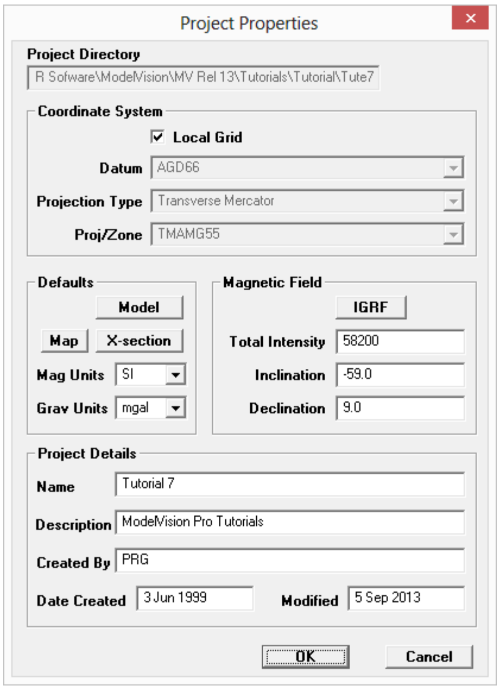
Tutorial Steps

The following tasks should be undertaken for this tutorial:

Step 1

Unless loading the `TUTE7.SES` session file, create a project for the tutorial (using **File>New>Project**). The information required for the project is:

Datum	AGD66
Projection	TMAMG55
Magnetic Field	58200 nT
Field Inclination -	59 degrees
Field Declination	9 degrees



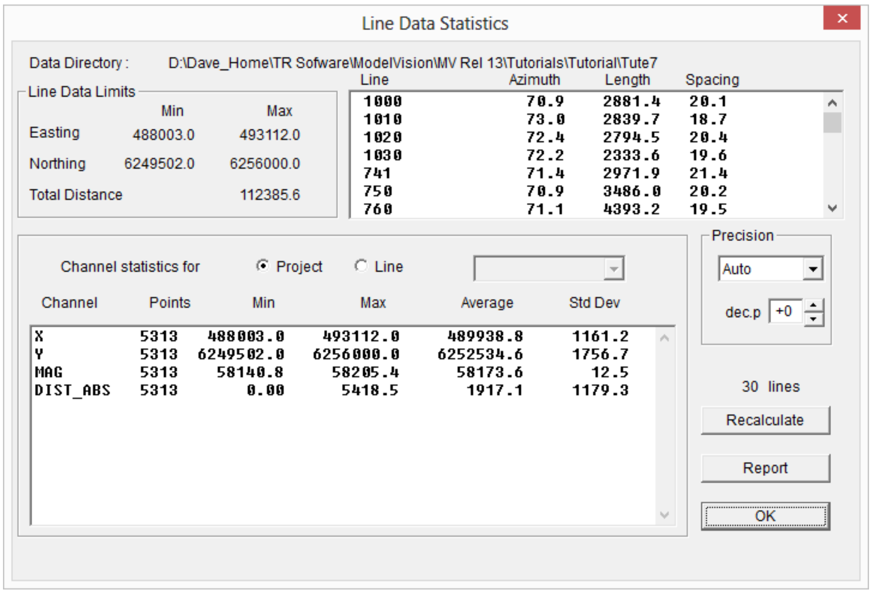
Refer to tutorials 3,4 or 6 for additional information on creating a ModelVision project.

Load the supplied LIN format file (`TUTE7.LIN`) using the **File>Import>Profiles>Simple XYZ format (.LIN)** format menu option (or the `TUTE7.SES` session file if operating ModelVision in Demonstration mode).

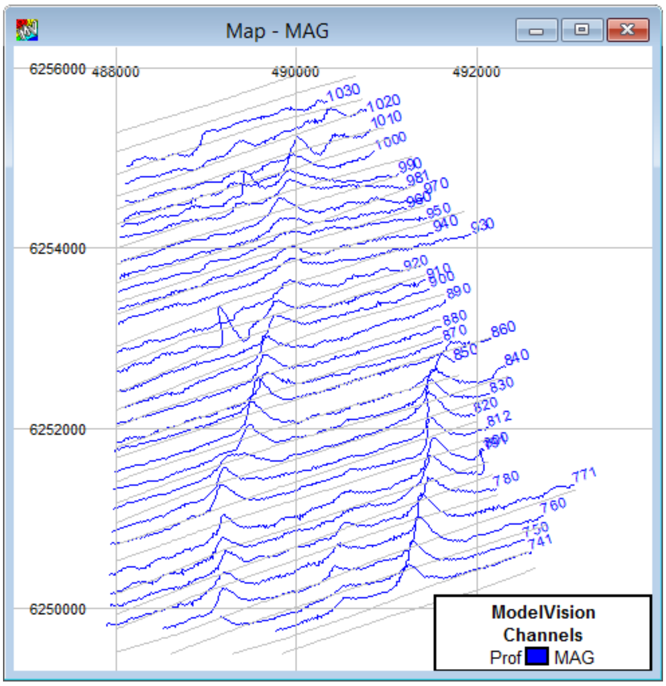
The data contained in this file has a header and format similar to that below:

LINE	X	Y	MAG
1000	488008.000	6254576.000	58157.699
1000	488029.000	6254583.000	58158.398
1000	488050.000	6254590.000	58158.199
1000	488071.000	6254597.000	58158.602
1000	488092.000	6254604.000	58158.699
1000	488113.000	6254611.000	58159.398
1000	488134.000	6254618.000	58158.801
1000	488155.000	6254625.000	58159.898
1000	488176.000	6254632.000	58158.398
1000	488197.000	6254639.000	58159.102
1000	488218.000	6254646.000	58158.602
1000	488239.000	6254653.000	58158.301
1000	488260.000	6254660.000	58157.801
1000	488280.000	6254667.000	58157.102
1000	488301.000	6254674.000	58157.000
1000	488322.000	6254681.000	58157.102

The line statistics menu will open automatically and you can see that a large number of data lines are present and that the total survey distance covered by the data was 112 kilometres with an average sampling of 20 metres between readings.

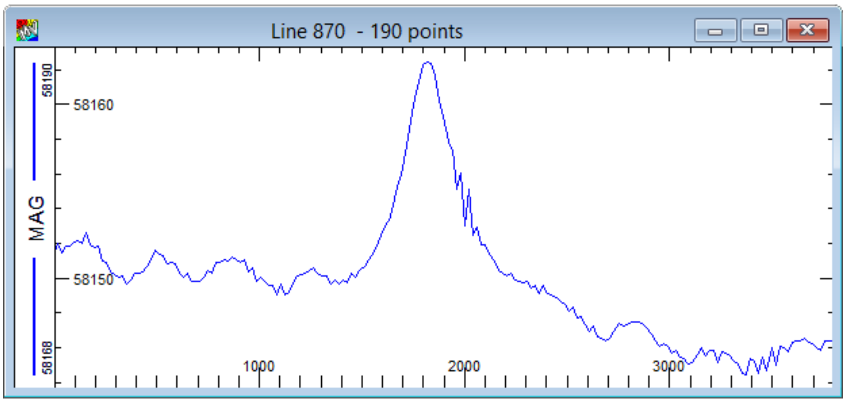


When you close the dialog, you will see an option to display a stacked profile map.

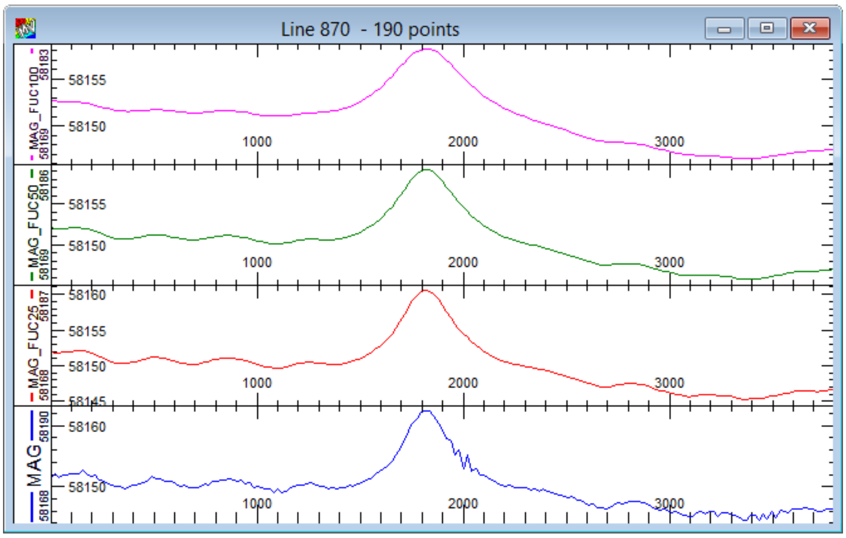


Step 2

Create a Multi-Track of the magnetics channel (use the **View>Multi-Track** option). (For information on Multi-Track displays, refer to *Tutorial 2*.) If you zoom in you will notice that a high degree of noise is contained in the data. The noise is derived from a combination of near-surface geological noise as well as instrumental noise. (The figures below are for line 870.)



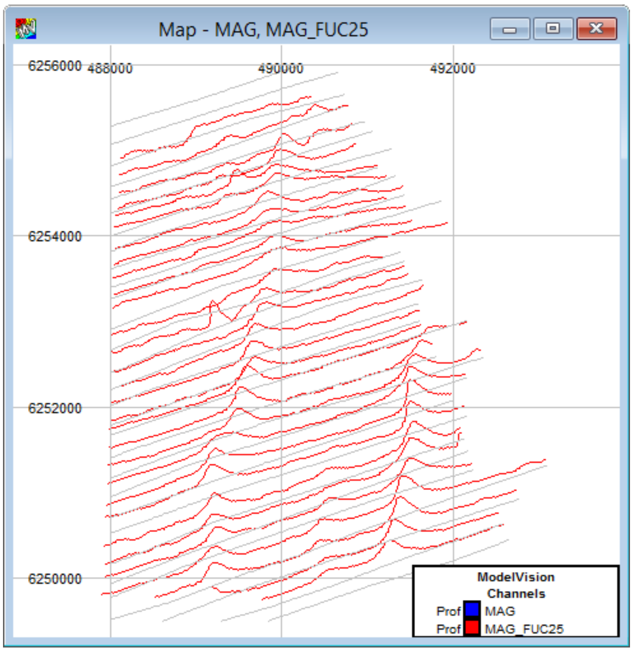
Prior to the use of AutoMag we will reduce the impact of some of the high frequency noise. A simple but effective way of doing this is to upward continue the data. The upward continuation height can be subtracted from the depths. Create a FFT filter of say **25**, **50** and **100** metre upward continuations and apply it to all lines Name the output channels as *MAG_FUC25*, *MAG_FUC50* etc. Compare these in a Multi-Track display for Line 870.



From this analysis, noise is barely noticeable with an upward continuation height of 25 metres and negligible for 50 and 100 metres. Select the 25 metre upward continuation height.

Step 3

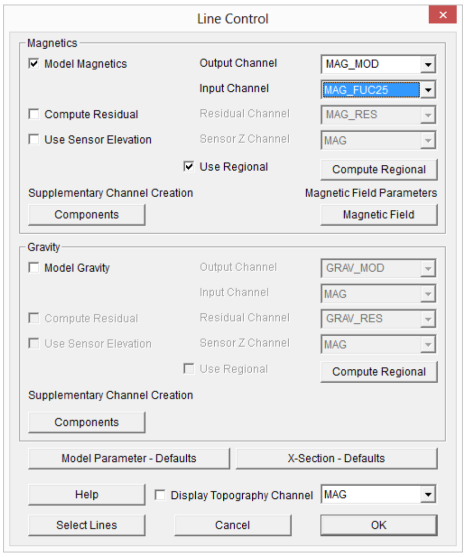
Review the data by displaying a stacked profile map of the *MAG_FUC25* channel.



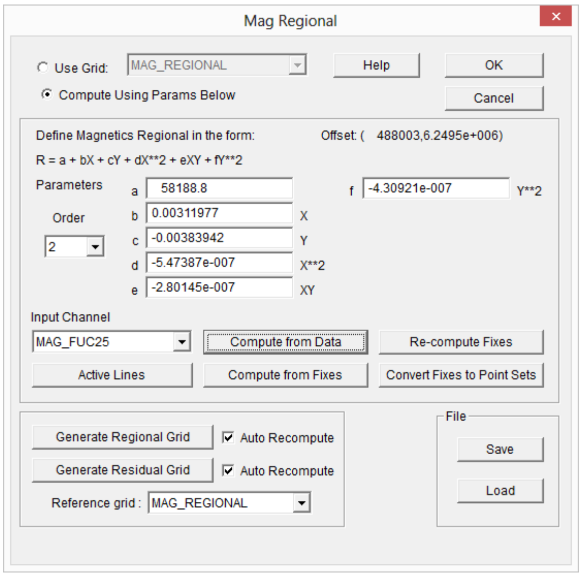
Another method of reviewing the data is by a contour or image display. You can import the grid (TUTE7.ERS) which is supplied in ER Mapper format, or you can grid the data yourself. Review *Tutorial 3* for an overview on gridding and image/contour displays.

Step 4

From the gridded data you will note that a southwest to northeast gradient is present in the data. This will need to be taken into account for modelling and AutoMag. To create a Regional channel, first make sure that the *MAG_FUC25* channel is chosen as the *Input Channel* in the **Model>Line Control** menu option.

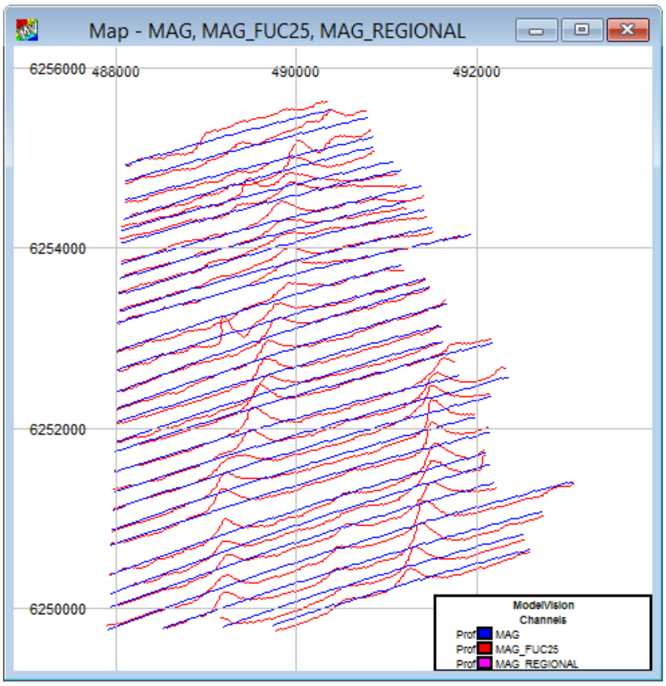


Select the **Model>Edit Regional>Magnetics** option. From experimentation a 2nd or 3rd order polynomial seems to be most appropriate to represent the original field for this data.



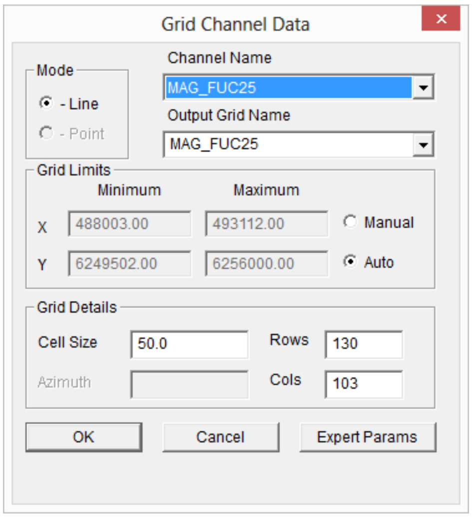
You will need to set all of the lines available to be Active (use the **Active Lines** button) to enable all data to be used for the computation of the Regional surface. When selected, compute the Regional with the **Compute From Data** button using a 2nd order polynomial on the *MAG_FUC25* channel.

You can add the regional (*MAG_REGIONAL*) as a new stacked profile on the existing stacked profile map. Change the colour to blue to see the contrast between the original data and computed regional.

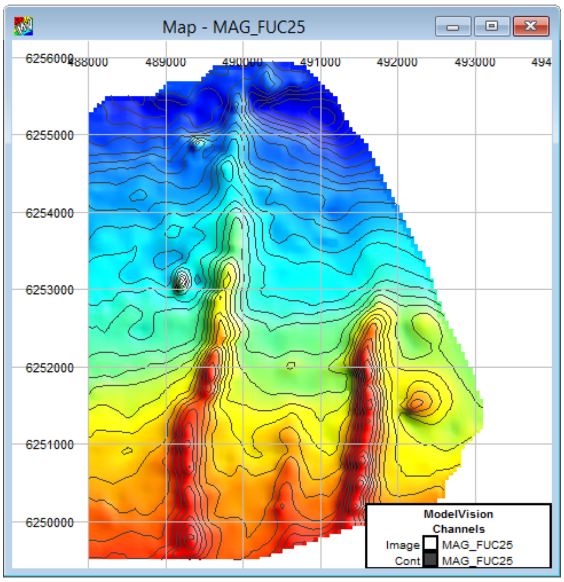


Step 5

Grid the data at 50m (**Utility>Grid Channel Data**) and use the **Expert Params** button to set the iterations to 1000.



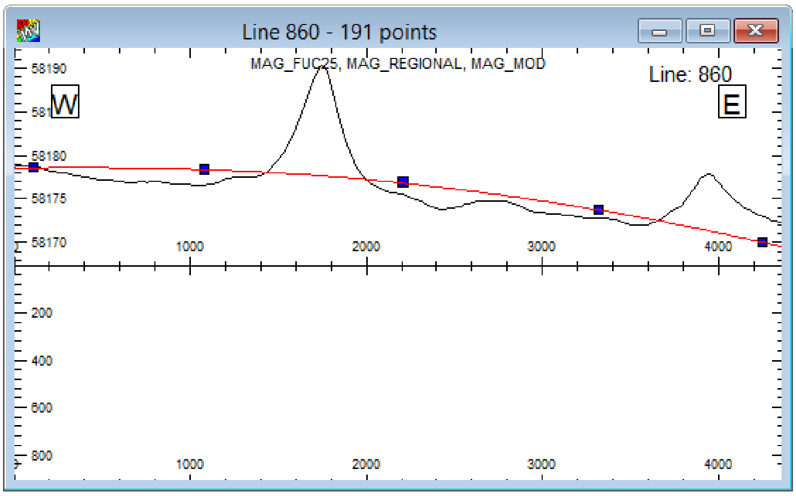
Display the grid with a contour overlay.



Note that the central and southern portion of the data is dominated by two elongate anomalies (from dyke sources?) and a third source/structure lies between the two. The initial phase of operating with AutoMag is to select a line which is representative of the geological regime to be interpreted. This line will be used to ‘train’ the AutoMag parameters. When depth solutions are satisfactory, these parameters can then be applied to all the other required lines. **Line 860** can be selected as appropriate for our purpose.

Step 6

Use the **View>X-Section** option to present the line cross-section. When this line is selected it will automatically become active for modelling. Turn on the *Configure Layers* menu using the *right-click* pull down menu and turn on *Orientation Labels*, *Line Name* and *Track Titles*.

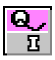


Usually this profile form of display is used for model creation. With AutoMag it is used to display and checking the various AutoMag solutions and for selection of the various AutoMag parameters.

Step 7 Optional Tuning with Quick Invert

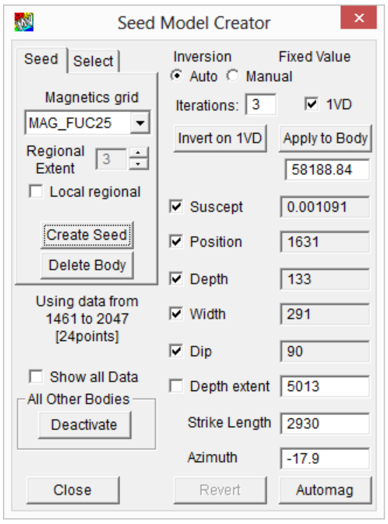
One of the more difficult parts of using AutoMag is the choosing of best tuning parameters. To help with this, it is possible to use QuickInvert on a representative anomaly and have the appropriate parameters automatically taken into AutoMag. You can go straight to Step 9 if you want to skip the QuickInvert process.

Before stating AutoMag, choose a representative anomaly such as the one at station 1750 on Line 860 and have it displayed in the X-Section window. Make sure this window is active. Choose the menu **Model>Quick Inversion** or click on the Quick Invert icon. This tutorial is

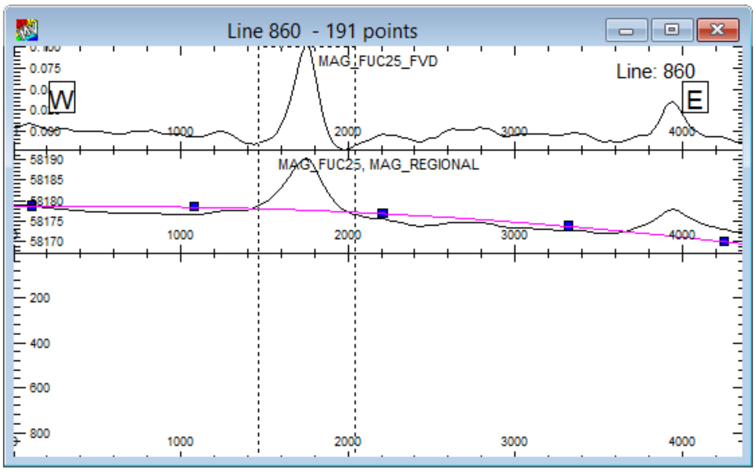


not focused on inversion, so use this segment of the tutorial to learn how you can use QuickInvert to set the starting parameters. See Tutorial 8 for more detail on Inversion.

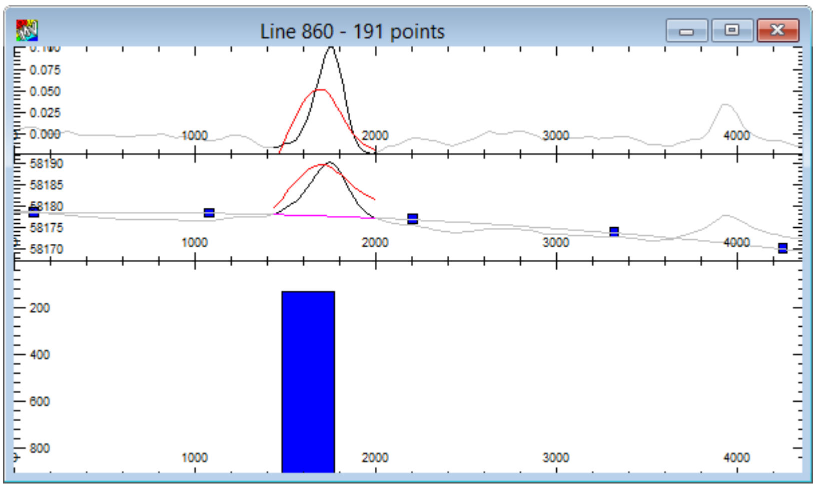
In the Quick Inversion dialog box, select the **Seed** tab and make sure the upward continued grid is selected (*MAG_FUC25*) and the local regional is unchecked. Check the 1VD checkbox and a first vertical derivate curve will appear in a new track above the total magnetic field track.



Click **Create Seed**. You will be asked to drag out a region in the cross section window. Click **OK** and drag out a region which is approximately the width between the low on each side of the first vertical derivative curve.

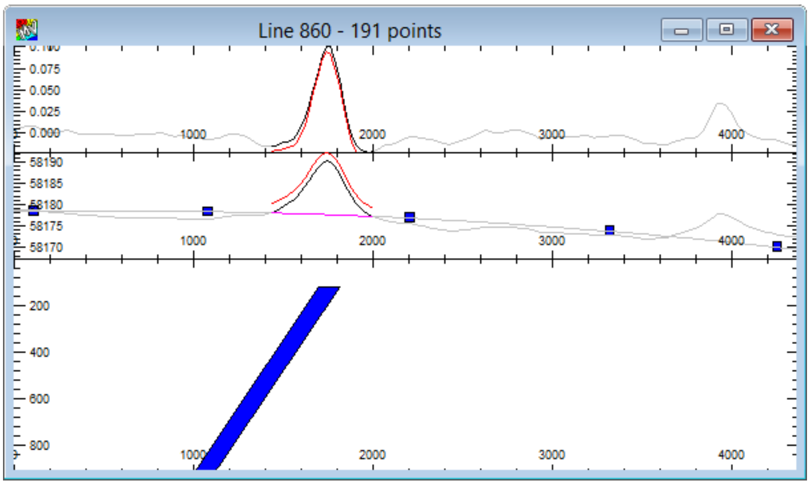


A seed body will be created and displayed in the section.



The seed body is roughly in the correct location and has a similar anomaly amplitude to the field data. If an error message is generated and a poor fit, then drag the body into approximately the right location prior to starting inversion.

Check options *Suscept, Position, Depth, Width* and *Dip*. Uncheck *Depth extent* and enter 2000. Enter 1000 for *Strike extent* and click **Apply to Body**. You are now ready to invert. This will perform 3 Iterations (as specified in the iterations box) and get close to a solution which matches the 1VD curve. Invert several times until you get no further improvement.



If the match is not good, click on **Delete Body** and then start this step again.

Finally, click the **AutoMag** button. This will not only start AutoMag but also copy the depth and width values into the AutoMag Run dialog.

Step 8

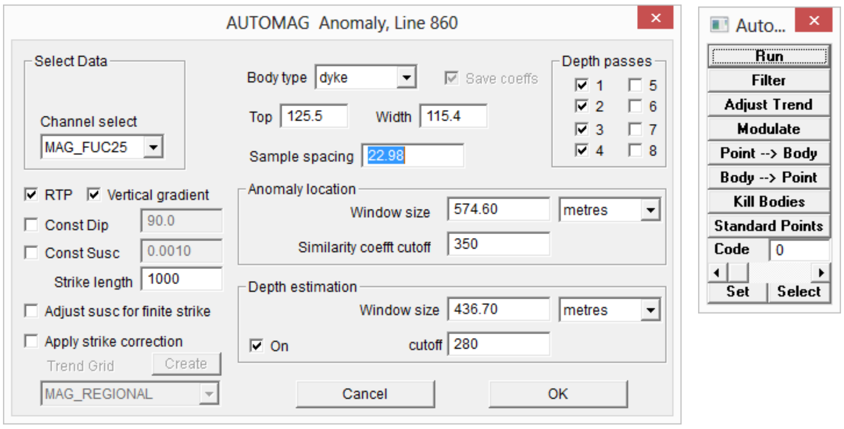
If you have existing model bodies, you should save them to a model file **Model>Export>MV Pro Format**. If you use the **Kill Bodies** on the AutoMag toolbar, you will delete all existing bodies as well as AutoMag created bodies.

Step 9

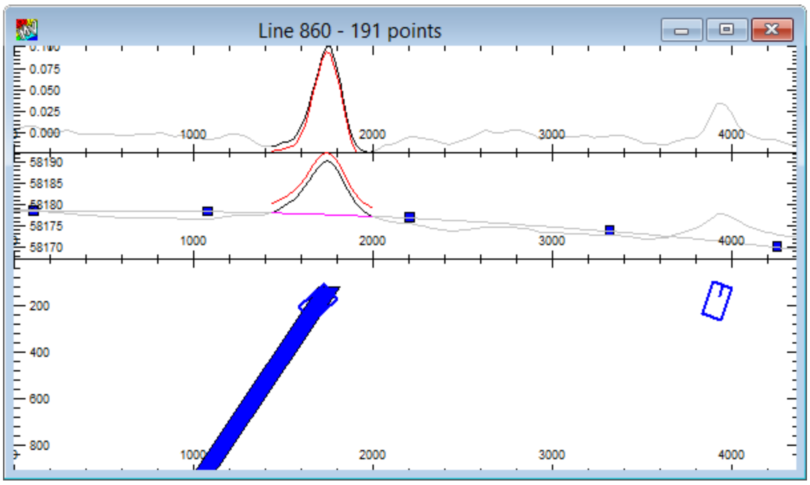
If you have not done the Quick Inversion step, start AutoMag by selecting the menu option, **Modules>AutoMag**.

Warning If AutoMag will not operate, you will need to load the data from the `TUTE7.SES` session file. Refer to the Warning at the start of this Tutorial.

You will notice the AutoMag toolbar displays (see image below). With the line displayed, select the **Run** option and the AutoMag control dialog displays as illustrated below.



The survey line separation for this survey is approximately 200 metres. The default for AutoMag strike lengths are set to 1,000 metres. Alter this to 400-500 metres. Change the dyke depth (top) to 125 and width to 110 which are the approximate values derived from the Quick Invert process. If you did not do this then using the defaults of 100 would suffice for now. Select OK and run AutoMag on this line only. In this example, the AutoMag solution closely matches the trial anomaly and a second solution is shown on the right hand end of the section.



AutoMag derives depths using bodies of infinite strike length. A specified strike length is only for convenience in giving that value to bodies created from the solutions.

After you have completed the AutoMag tuning, activated all data points for the line using the **Active Points** toolbar button followed by the *Select All* in the **Active Points** dialog.

Parameter Setting

Points to note in setting the AutoMag parameters are:

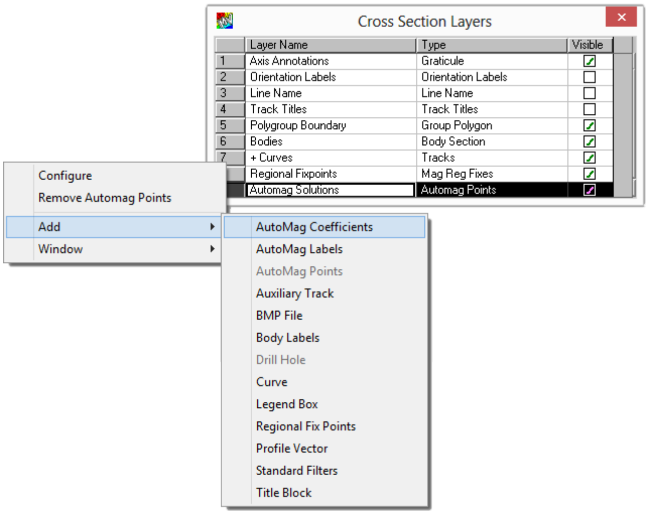
- The Sample Spacing is reported by ModelVision automatically.
- The number of Depth Passes is usually set at 3-4 since this provides for a reasonable range of operator width in the solution computation.
- The best guide to setting the Window size is to examine the anomaly widths of the profile data being interpreted. The ideal Window Size will span the minimum and maximum amplitude points of an anomaly. This window setting is best addressed for central depth passes. Note also that if you alter the Window Size to be measured in Samples rather than metres, the best starting point will be approximately 25. This value is derived from the Sample Spacing.
- The Similarity Coefficient Cutoff is the threshold value below which AutoMag will attempt to isolate the anomaly. If the similarity coefficient is below the specified value, AutoMag searches for a local similarity coefficient minimum. AutoMag will then use this location as the centre of the window for performing a detailed analysis of the data within the window.
- Initially do not be concerned by the Depth Estimation parameters. Refer to Single Anomaly Tutorial in the User Guide for more detail on manual setting the tuning parameters..

Step 10

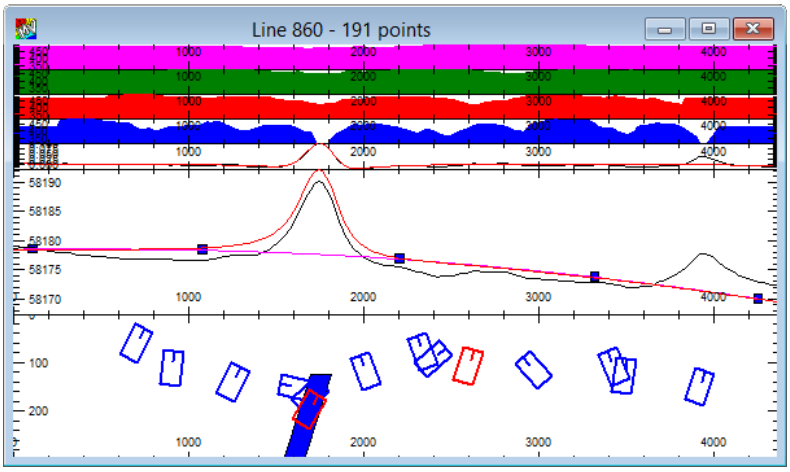
In this step we will look at the display of similarity coefficient and dynamic filtering of the AutoMag solutions to remove lower quality solutions. Since the initial setting derived from the QuickInvert settings were excellent, we need to change the default similarity coefficient threshold settings to demonstrate the filter behaviour. Set the similarity cutoff defaults of 450 and 400 rather than the 350 and 280 default values. The higher thresholds locate more minima as shown in the cross-section below.

When the various AutoMag parameters have been entered, select **OK** and the AutoMag module will execute. A speed bar will be presented and after completion will report on a number of solutions. When **OK** is selected, the solutions derived will be displayed on the cross-section.

Display the Similarity Coefficients by displaying the **Cross Section Layers** table and selecting the **Add>AutoMag Coefficients** from the pop-up menu that appears when any layer is clicked on with the right mouse button.



On pressing **OK**, tracks appear at the top of the data to indicate the similarity coefficients for the various window computation passes.

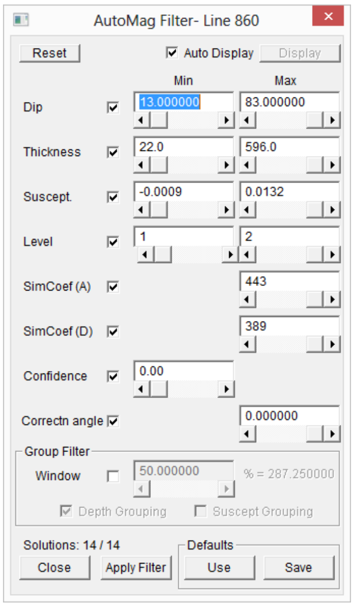


In the event that too few solutions have been computed, position the cursor beneath the minima of anomalies on which you feel a solution should have been computed. Read the cutoff coefficient value (this is displayed in the status bar at the base of the ModelVision screen). To adjust the number of solutions, revise the **coefficient cutoffs** in the **AutoMag Run** dialog. Increasing the similarity cutoff will allow more solutions to pass. Reducing the level will reduce the number of solutions.

When a new similarity value has been entered, re-run AutoMag to create a new set of solutions. It is probably better to choose a higher similarity cutoff and generate more solutions (as shown) and then use the Filter (see below) to quickly determine an appropriate cutoff value.

Step 11

Each solution has physical properties assigned to it based on the chosen model (in this case a dyke). If the similarity coefficient cutoff is chosen too high, solutions will be determined for parts of the curve which might be noise or other artefacts. In addition, some solutions may be generated where a dyke model is inappropriate. To get rid of unwanted solutions, you can use the **Filter** option on the AutoMag tool bar.



Having the *Auto Display* option checked allows changes to parameters to be displayed in real time. The visibility of solutions is turned on and off as solutions fall within or outside the filter parameters.

Clicking *Reset* sets all filter parameters back to their initial values.

Dip
For a dyke model, dips shallower than 20-30 are probably inappropriate and are probably more appropriate to edge solutions. You can view which solutions are shallow by entering say 20 in the Max side and seeing which ones are left. To remove these, enter 20 in the Min and make the Max 90. In this case enter 30 in the Min side.

Thickness
Extra wide dykes are probably inappropriate so you may wish to reduce the Max width. In this case, remove widths greater than 1000m by entering 1000 in the Max.

Suscept
You sometimes get negative susceptibilities. These are often the result of picking the low between two closely spaced anomalies. You can check which solutions have negative susceptibilities by entering zero in the Max side. In this case we will remove all solutions with negative susceptibilities, so enter zero in the Min.

Level
This is the similarity coefficient pass number. You can filter out solutions for pass numbers that you do not want. In this case we will leave this as is.

SimCoeff(A) and SimCoeff(D)
Use the slider to reduce the similarity coefficients until you have only a couple solutions. This is a tuning exercise so you should choose the largest similarity coefficients that still give reasonable solutions. In this case, you will need to find your own optimum but err on the side of making it greater than smaller. When you apply this slide, think about what is geological reasonable.

For now, leave the *Confidence* and *Correction Angle* unchecked.

Confidence and Correction angle
These sliders are only used when you have a strike angle correction grid which is not the case in this tutorial. See the AutoMag documentation for information on how to create and use the strike angle correction method.

Group Filter
This filter detects near coincident solutions which may occur on the same pass or multiple depth passes. A window size is selected automatically by AutoMag and you use the slider to change the effective size of the window as a percentage of the initial size. As soon as you turn on this filter, some solutions will disappear. Any solutions found within the window group are analysed for the best similarity coefficient and if very close, the shallower solution will be used. The secondary susceptibility filter is used to reject extremes

Experiment with each filter on an individual basis to see how it behaves before combining them in a geologically sensible manner. The logical order for application of the filters is as follows:

Dip (>30)
Susceptibility (>0)
Simcoef (D) (<300 generally)
Window

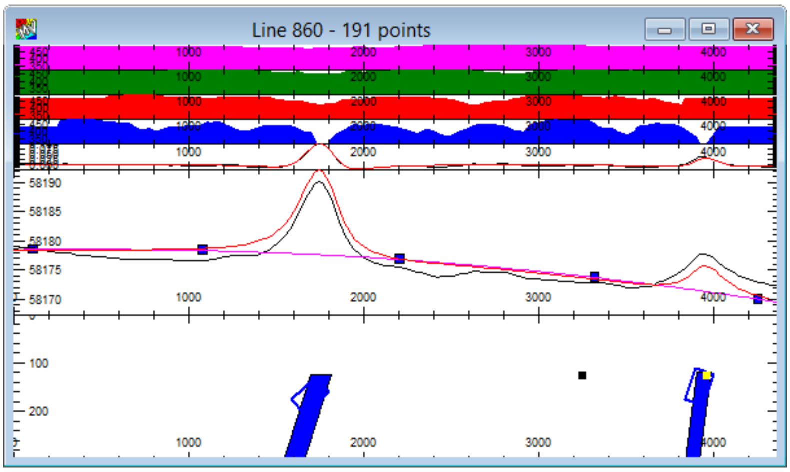
Note After you have the filter parameters set as you wish, you must then click **Apply Filter** for the actual filter to be applied. If you close the filter dialog before Apply then all solutions will be retained.

If you wish, you can save the filter parameters by clicking **Save** and later bring them back using **Restore**.

Step 12

Not all solutions derived will be satisfactory. Solutions contain information on the physical properties of the body type requested - in this case a dyke. Hence, thickness, dip, susceptibility etc will have been computed for each solution.

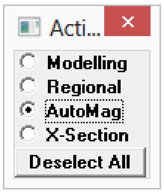
To test if a solution is viable, select the solution box with a mouse click and then convert it into a body by pressing the *Space Bar*. You can also do this with the *Point → Body* toolbar button. The solution will be converted into a dyke (tabular body) with computed properties. After this has been done for a selected number of bodies, the **Compute** toolbar button can be used to forward model the response and check against the observed data (unless the **Immediate Compute** button is already active).



In most instances the solution is likely to be reasonable and so it can be kept. In a few cases the solution will be unreasonable and can be rejected. To do this simply select and **Delete**.

Step 13

Once you are satisfied that the AutoMag parameters derived from the analysis for this line are satisfactory, you can apply these same parameters to an AutoMag computation of the entire dataset.



Open a map window and display a contour map of the filtered grid plus the flight lines. When you *Run* AutoMag from the toolbar, you will be able to select all of the lines from the *Selected Lines* button. You can activate a subset of lines using the AutoMag **Active Lines** toolbar option.

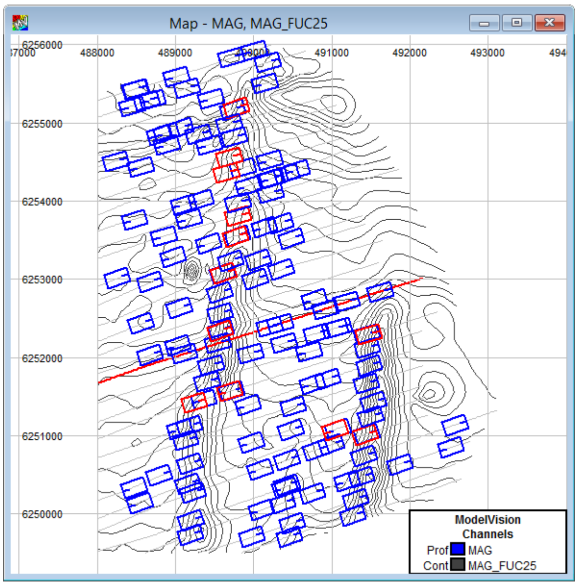
Place the cursor at the top of the display and with a left mouse click, drag and draw the ‘rubber-band’ over all the data lines to be made active for AutoMag computation.

After the Active-Line ‘rubber band’ has crossed the various lines, they will display as a thick line indicating they are now active for AutoMag operation.

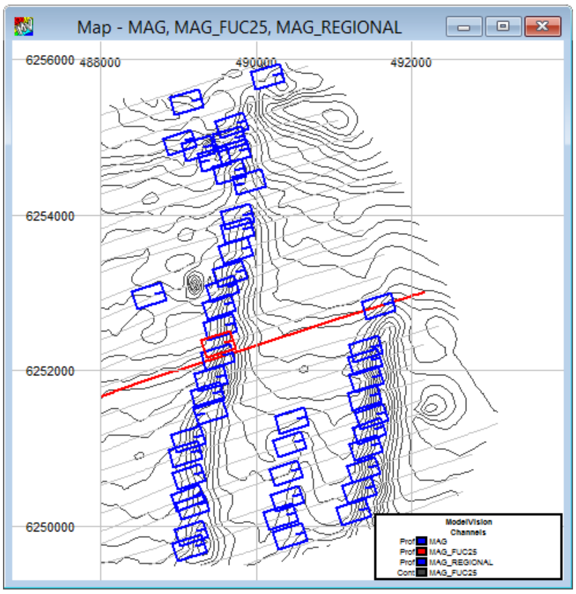
Step 14

Again select the Run option from the AutoMag toolbar. Leave the parameters as they have been set from the ‘training’ line. Run the AutoMag option by selecting **OK**. AutoMag will now operate sequentially on all the selected active lines. This may take some time but is relatively fast.

When the computation is completed, you will see the derived solutions displayed in the contour window. From comparison with stacked profiles or field contours you can determine from which feature of the field variation each solution has been derived. Solutions can be selected and deleted in map views as well as section displays.



You should again run the **Filter** to remove poor solutions.

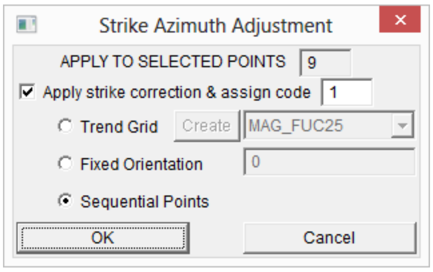


After the filter, some of the computed solutions may require editing or deletion. This can be done either in the Map or in X-Section windows by selecting the solutions and pressing the Delete key.

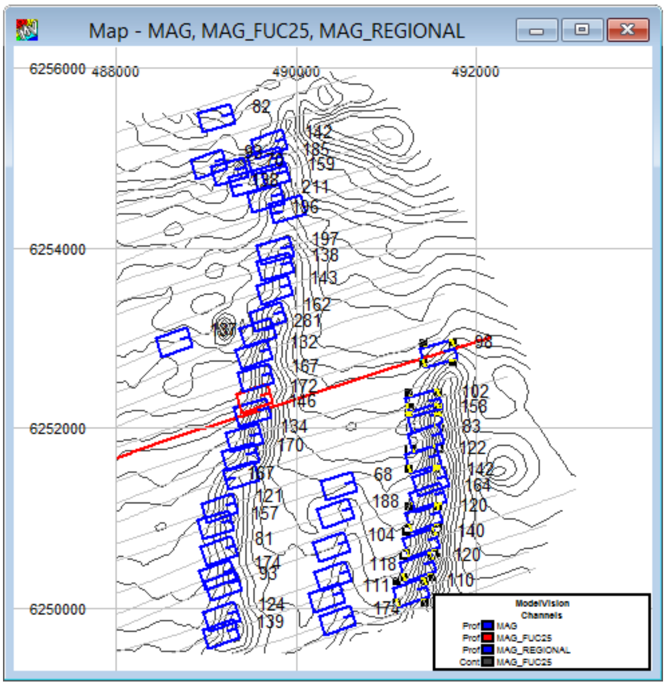
Step 15

Depth estimates derived from line data are based on the assumption that the target geology is trending perpendicular to the flight lines. If a profile crosses a feature obliquely, the magnetic anomaly is stretched which increases the apparent depth of the target. AutoMag provides manual and automated methods for strike correction.

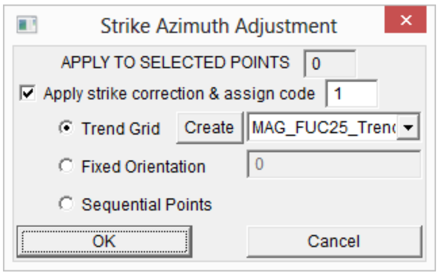
To perform strike correction manually, solutions must be selected sequentially along strike. The **SHIFT** key is depressed while the solutions are selected. When you finish, select *Adjust Trend* from the AutoMag toolbar and then the Sequential Points checkbox.



This will immediately adjust the depth values of the AutoMag points. To see the values in map view, use the Channel Annotation button which is access from the *Map Layer>AutoMag Solutions* layer. The example below shows annotated depths with the right hand trend has been manually depth corrected.



To perform the automatic strike correction, a trend grid must first be created. Click **Adjust Trend** on the AutoMag toolbar and select the Trend Grid option.



Click on the **Create** button and select the MAG_FUC25 grid used for the trend analysis. You can also run the trend analysis from the AutoMag Run dialog and the menu Utility>Grid Utility>Trend. In the latter case you will need to manually import the trend grid.

The GridUtility application will open the trend grid function and the selected grid. Click **Save As…** and give the multi-channel grid the following name: MAG_FUC25_Trend.ers. The two grids MAG_FUC25_Trend and MAG_FUC25_Confidence will open automatically in their own map windows.

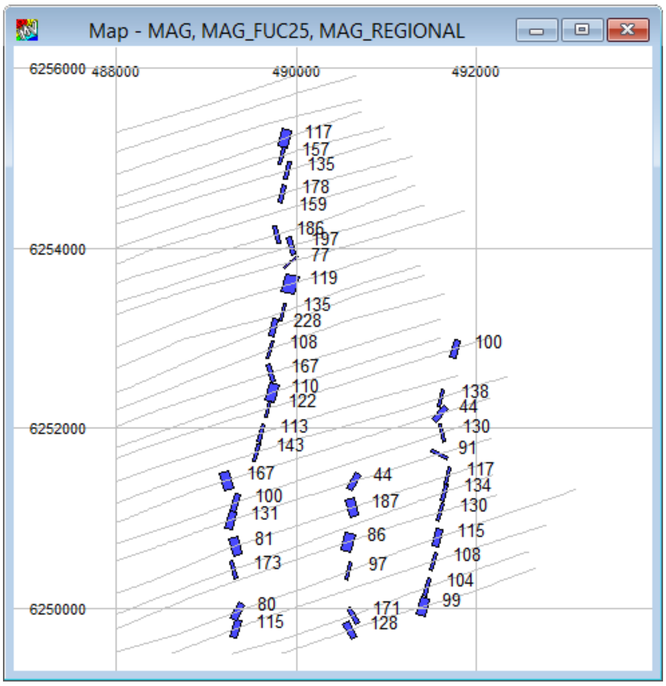
You will need to close the **Adjust Trend** dialog and reopen it to refresh the list of grids. Once the trend and confidence grids have been created, you can use them each time you run the AutoMag dialog by turning on the *Apply strike correction* checkbox. Make sure you select the *MAG_FUC25_Trend* grid from the pull-down list.

You can clean up poor solutions where the trend azimuth derived from the grid was poor. If the angular correction is significant (>30 degrees), then the depths will appear shallower than the immediate neighbours. You can delete these solutions, or convert them to bodies and use inversion to refine the depths.

Step 16

Select all solutions in map view and use **Pt->Body** to generate model bodies from the AutoMag solutions. The map below shows the location of the tabular bodies derived from the AutoMag solutions with corrected depth below sensor values (DEPTHC). If sensor elevation data is included

The bodies create are best viewed in Perspective view to look for anomalous or erroneous bodies.



You can correct the depths for flying height and upward continuation so that you have depth below ground.

The table below lists the fields available from each AutoMag point. Note that depth is positive down and changes meaning according to the information provided in the Model>Line Control dialog. If sensor elevation (SENSOR) and DTM (TOPO) is provided as input channels for modelling, then this information is used to compute depths relative to sea level.

Note also that where **C** is appended to the channel name, then it means that it has been azimuth corrected. When **BG** is include in the name, this means below ground depth. This information takes into account both the sensor elevation and dtm for computing the local depth below terrain. If the sensor elevation is provided as an altimeter value, then the dtm is assumed to be zero and the depth below ground is computed by subtraction of the sensor height from the computed depth.

Name	**Description**
X	X value (usually Easting)
Y	Y value (usually Northing)
BTYPE	Body type (1=dyke)
DEPTHBGC	Dyke depth below ground - Corrected
ELEVC	Elevation of dyke - Corrected
DEPTHC	Depth to top of dyke (from datum) - Corrected
SUSC	Susceptibility
DIPAZIMC	Dip Azimuth of dyke - Corrected
DIPC	Dip of dyke - Corrected
HALFWC	Dyke half width - Corrected
TRENDCONF	Trend grid confidence value
CORRANG	Trend grid correction angle
DEPTHBG	Depth below ground
ELEV	Elevation of the dyke
DEPTH	Depth to top of dyke (from datum)
SUSA	Magnetic susceptibility
DIPAZIM	Dip Azimuth of dyke
DIPA	Dip of dyke
DPTHXTNT	Depth extent of dyke
STRIKELEN	Strike length of dyke

HALFW	Half width of dyke
TOPO	Topographic elevation
SENSOR	Sensor Elevation
SCOEFD	Similarity coefficient for the Depth determination
SCOEFA	Similarity coefficient for the Anomaly location
CODE	Solution Quality Code Value
LEVEL	Depth pass number
AZIMUTH	Dip Azimuth of dyke

Tutorial 8 – 2D and 3D Modelling and Inversion

Aim

The aim of this tutorial is to introduce:

- Synthetic survey line data simulation and survey design
- Inversion in two and three dimensions

Intended User Level

Medium and experienced users. Advanced options for inversion.

Topics Covered

- Grid Import
- Line creation
- Display formats
- Modelling
- Two dimensional inversion
- Three dimensional inversion

Background Information

An airborne geophysics survey was undertaken in the Esperance area of Western Australia in search of evidence of mineral sands known to exist in strandlines in the region. In many cases the depth to basement is important in this exploration process. To search for the low amplitude magnetic responses associated with mineral sand accumulation, it is necessary to remove the effects of basement anomalies. Continued exploration for mineral sands in the region would require access to the magnetic line data. This is not the intent of the tutorial, but to define the basement features only.

The only data available to carry out the basement analysis is the airborne magnetic grid data. In this case it is necessary to create a series of synthetic survey lines which will be used for modelling and inversion.

Tutorial Data

File `TUTE8.GRD` (`..\TUTORIALS\TUTE8`) contains a small segment of a larger grid extracted from an aeromagnetic survey undertaken in southern Western Australia, near the town of Esperance. The grid is in Geosoft format and defines a linear magnetic high that is broken into a series of magnetic anomalies. The intention of this tutorial is to interpret the varying depths of magnetic sources along this trend and in so doing, introduce some of the concepts associated with inversion.

Tutorial Steps

The following steps should be undertaken for this tutorial:

Step 1

Initially, a project is required to define the various properties of the data. ModelVision uses projects to manage various analysis exercises and a `MVPROJ.INI` records your entries for later use. Binary session files (.SES) can also be used to store data, models and displays if required.

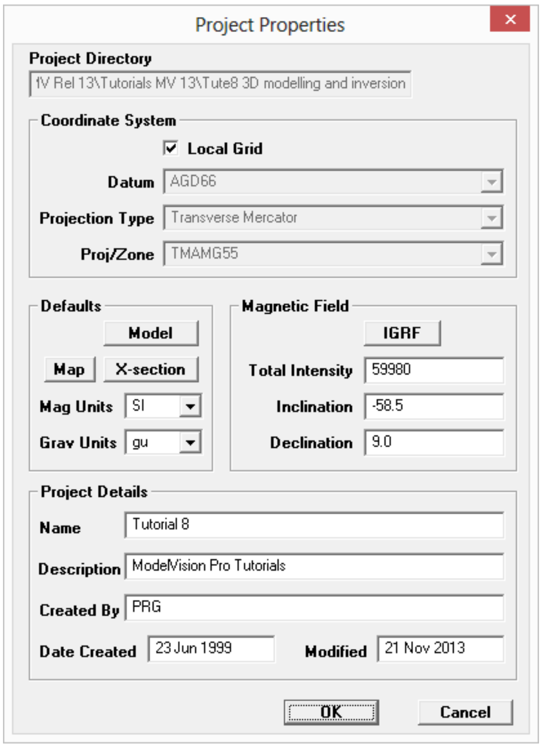
Select the **File>New>Project** option and enter the information as required. For this area and projection information includes:

Datum: Australian Geodetic Datum 1966 (AGD66)
Projection: Transverse Mercator
Zone: AMG Zone 49 (TMAMG49)

Note that the entries for the Earth's magnetic field are set to be:

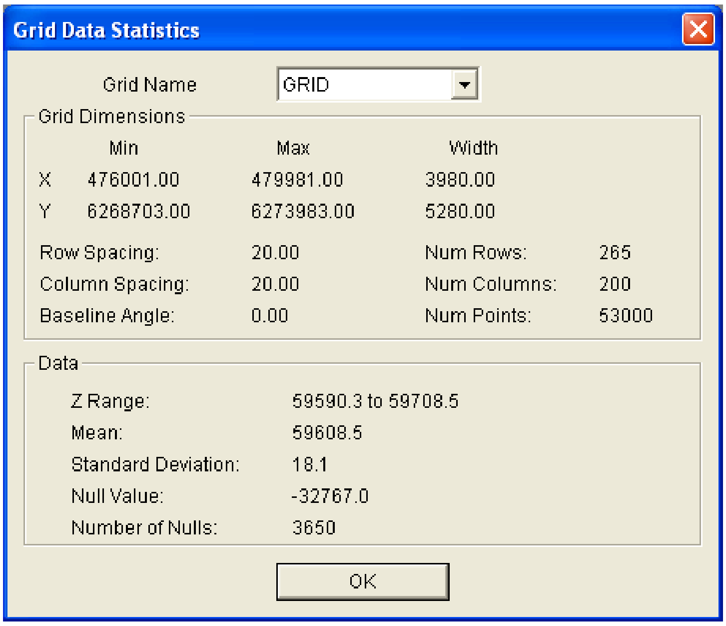
Field Intensity : 59980
Inclination: -58.5 degrees
Declination: 9 degrees East

The project properties dialog should look like this:



Load the available Geosoft format grid (`TUTE8.GRD`) using the **File>Import>Grid>Geosoft** option (or the `TUTE8.SES` session file if operating ModelVision in Demonstration mode). On import and image of the grid will be generated automatically.

From reviewing the **Utility>Statistics>Grid Data** option, note that the grid has mesh size of 20 metres and the data contained in the grid is for the total magnetic field.

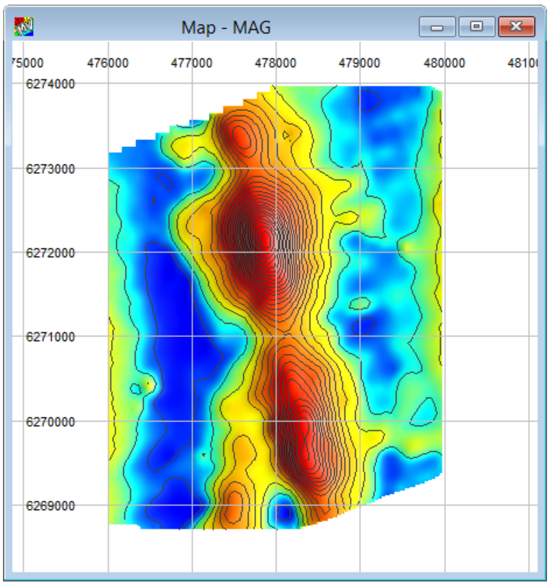


Step 2

In the Grid Data Statistics dialog the imported grid will be called *GRID*. Access the **Utility>Data Maintenance>Grid** menu option and rename it to *MAG* by clicking on the **Rename** button and entering the new name. Click **OK** to exit this dialog.

Step 3

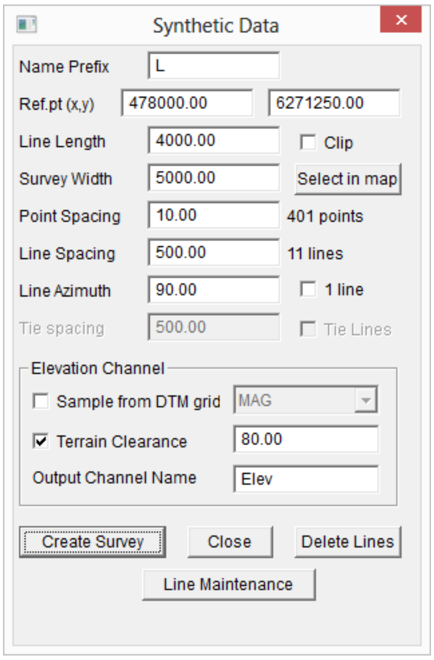
Add a contour display of the imported grid data and set the contour interval to 5, the image illumination light source azimuth to 90 and the image decimation to 3. Note that the grid defines the north-west trend well and is broken into four discrete anomalies. For this exercise, we will assume the magnetic feature defines a series of discrete magnetic sources along a common trend.



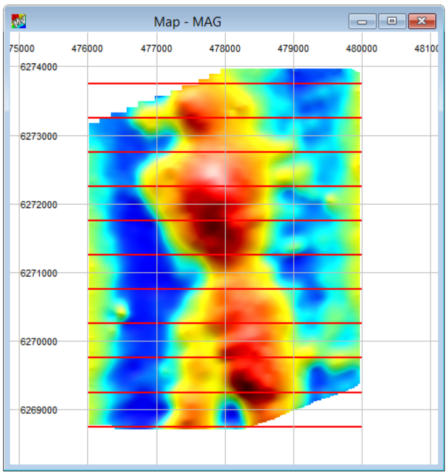
Step 4

We will use the **Utility>Synthetic Lines** menu option to create the synthetic survey. Note also that you can generate arbitrary lines from the grid using the **Traverse** button.

Open the **Utility>Synthetic Lines** dialog. Note in the entries that the survey centre (Ref. pt) is defined and then the line width and survey width (or height) define the survey dimensions. Make sure that the Clip check box is off for this exercise so that you can set the line length and survey height (Width). If **Clip** is on, then the lines will automatically be truncated at the boundary of the displayed grid. Set the parameters as shown in the dialog below.



Use the **Create Survey** to generate the initial set of survey lines. This will create a set of flight lines with an elevation channel with constant terrain clearance of 80m.

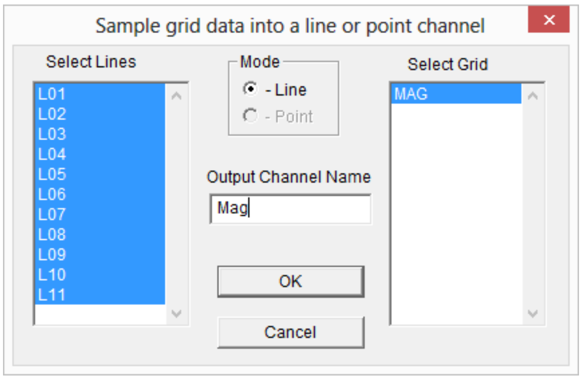


If you are not happy with the location of the lines, then change the change the dialog parameters, delete the current set of lines and then run **Create Survey** again. If you forget to delete the previous lines, they become permanent and you must use the **Line Maintenance** option to delete them.

Step 5

You can sample the grid data onto the synthetic lines using the menu function **Utility>Sample from Grid** or the **Utility>Calculator** in grid interpolation mode:

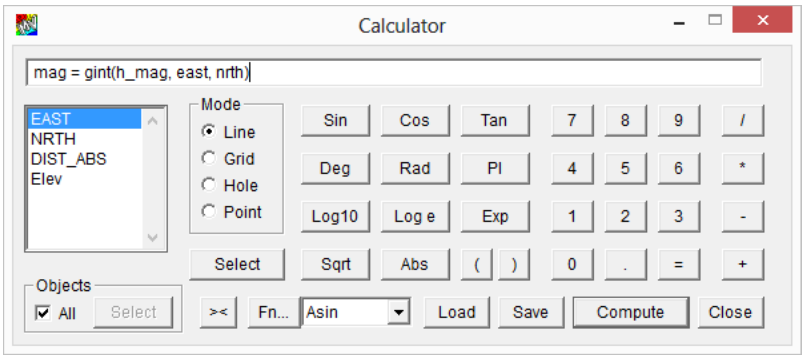
a) Utility>Sample from Grid
Select the synthetic lines on the left and the nominated grid (MAG) on the right. Enter the channel name that will be exported as the new field on each line.



b) Utility>Calculator using the GINT function
Open the **Utility>Calculator** option, check Line mode and enter the following formula:

mag = gint(h_mag, east, nrth)

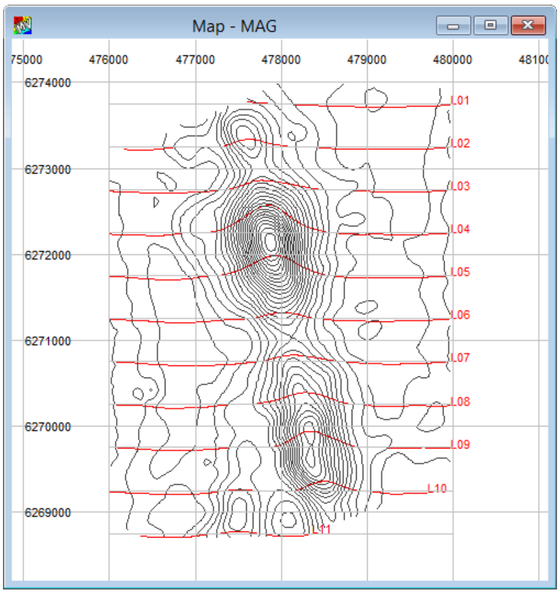
Note that the formula is not case sensitive.



In this case, the derived magnetic data values are interpolated from the '**mag'** input grid (indicated by the h_grid syntax convention) and at the locations defined by the **EAST** and **NRTH** channels.

Step 6

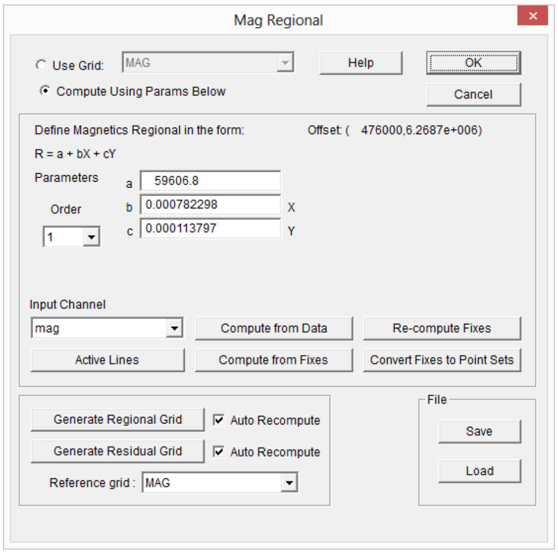
As each data reading on the synthetic lines now has a data value, they can be displayed as well as modelled. Add a stacked profile map to the window, but turn off the image display to make it easier to see.



You can alter the default vertical scaling, colour and line thickness of the stacked profile from the map configuration menu.

Step 7

We will use a regional magnetic field to remove the background prior to modelling. Open the **Model>Edit Regional>Magnetics** option, select a first order polynomial and make all lines active (from the **Active Lines** button within this dialog). Press the **Compute From Data** button and a regional surface will be created.

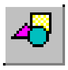


Note that you should always use the lowest possible order that removes only the long wavelength features that are not part of the geological problem your are trying to solve.

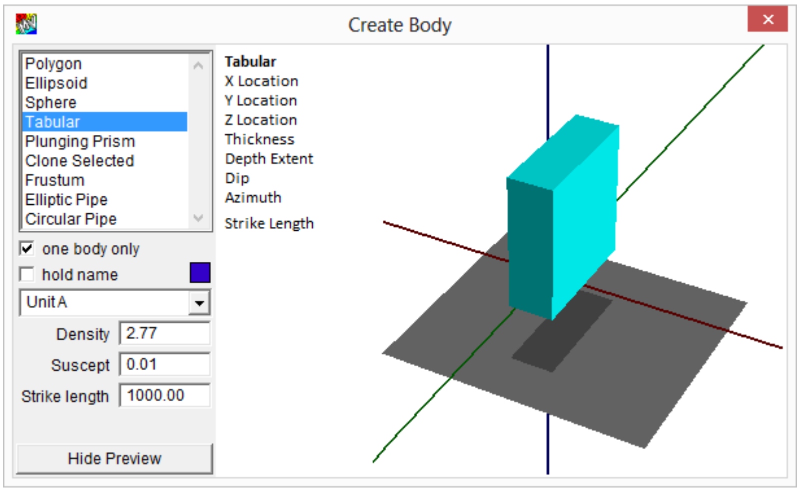
Step 8

Select the **Model>Line Control** menu option and ensure that the *MAG* field is selected for the **Input Channel,** the **Use Regional** check box is selected, the **Use Sensor Elevation** check box is selected and *Elev* is selected as sensor elevation channel. Use **Select Lines** to active all lines in the dataset for modelling.

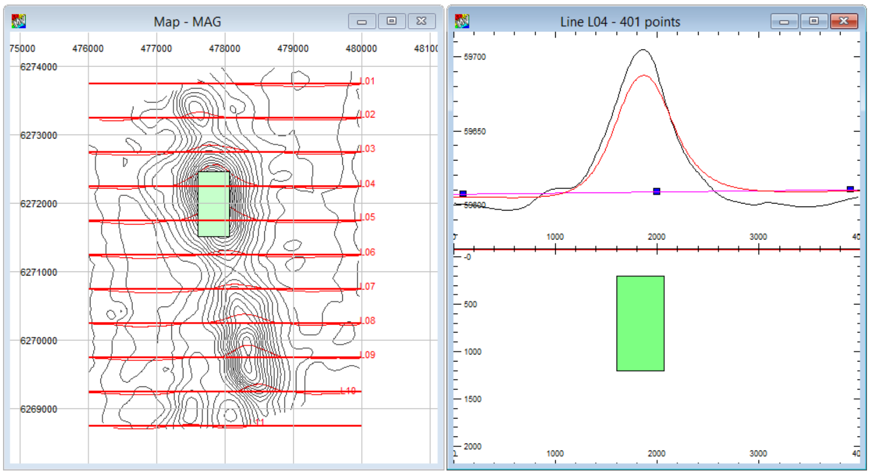
Step 9



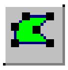
Select and display the 4th data line (L04) using the **View>X-Section** menu option. Note that the observed data (created from the MAG grid), the regional trace and the model response (conforming to the regional line) and are all displayed. In this cross section create a tabular body using the **Create Body** button on the toolbar or the **Model>Body Operations>Create Body** menu option.



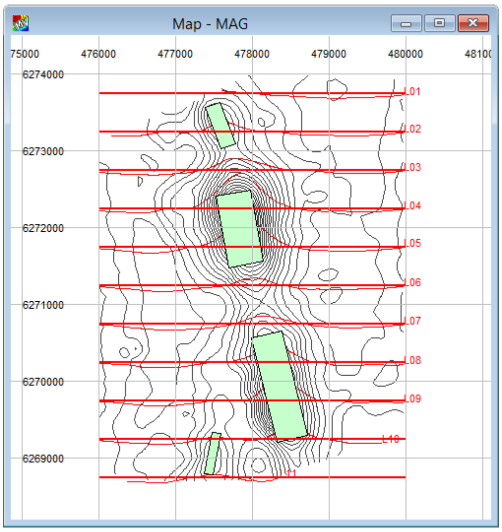
As you create the body in the cross section, you will note it also is displayed in the contour map.



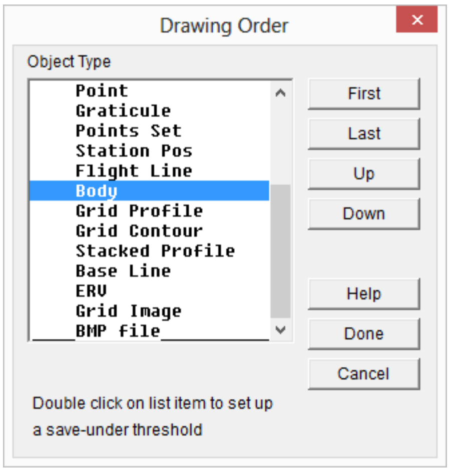
In a map window you may need to alter the strike length (700m), azimuth (350 or -10 deg) and position slightly to straddle the top two or three data traverses as we will use this body to model these lines concurrently. To perform this action double click on the left mouse button while the cursor is positioned over the body to view the **Body Properties** dialog. Adjust the magnetic susceptibility, depth and position until you get an approximate data match as shown above. In this example the regional is too high and the width of the body and depth are only approximate. Inversion will allow us to get much closer.



Continue adding tabular bodies in the map window along the main anomalies. We only need to use tabular for this study to estimate the depth to basement. In the map window, use the **Reshape** toolbar button to interactively change the azimuth and strike length of each tabular body.

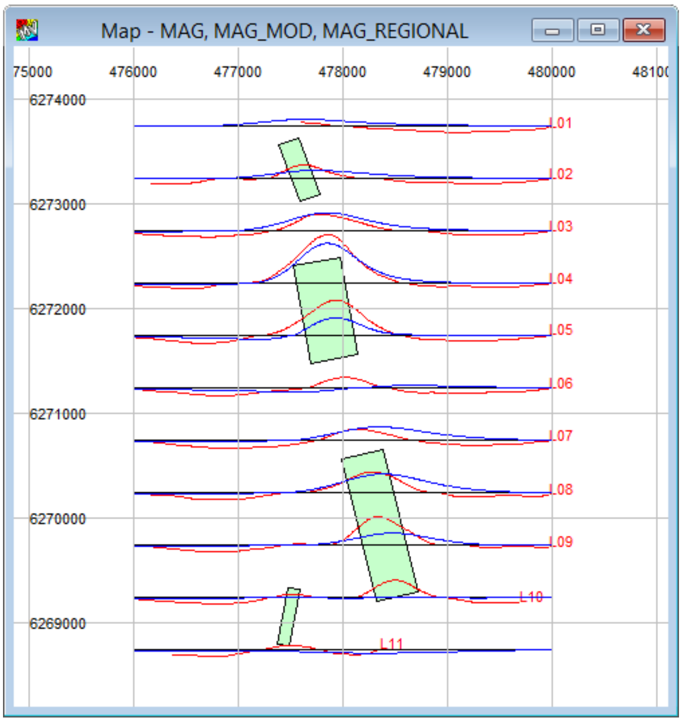


Change the scale of the stacked profile of the *Mag* channel in the map configuration layer control to ensure there is sufficient dynamic range to see minor changes in the data. The bodies obscure the stacked profile and you can change this by changing the body display mode to *wire frame* or change the body order from the map window configuration layer control. You access this control from a right click and selection of **Window>Drawing Order** menu option. Select *Body* and move it down the list until it sits below the *Stacked Profile* Object Type.



Step 10

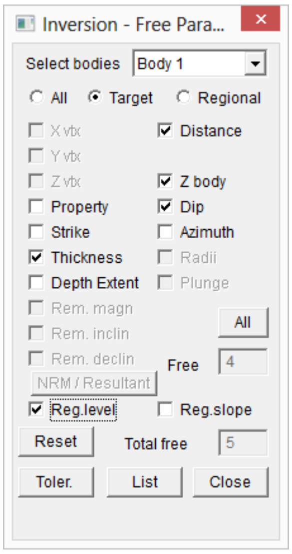
We can now add the regional magnetic field and forward model response for all flight lines to the existing map display. Earlier we activated all the synthetic flight lines for modelling, so press the compute button to make sure the model response and regional are up to date and then add the regional field (MAG_REGIONAL) and magnetic model field (MAG_MOD) to the map view. Change the colour of the regional to black and the model field to blue. Also, turn of the *Contour, Flight Lines and Base Lines* layers to decrease clutter.



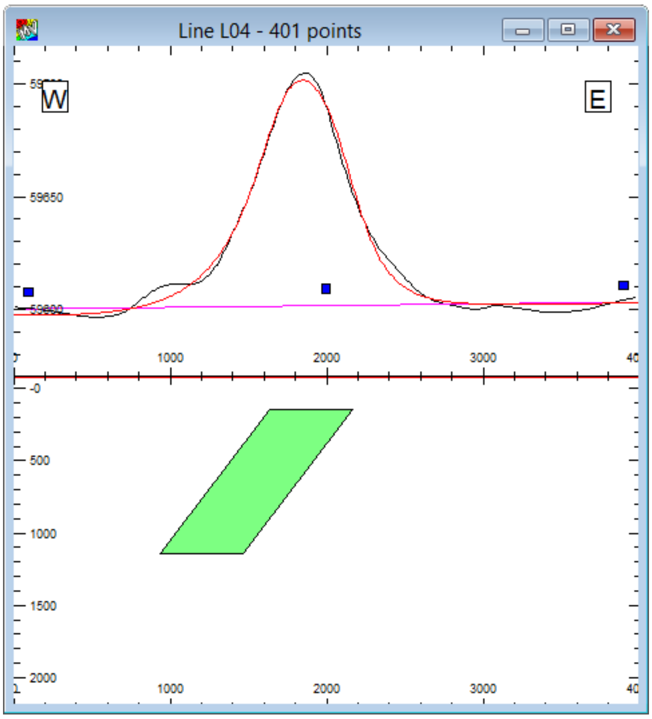
Step 11

We will start the inversion process for the single body in Line L04 before moving on to full map inversion. This helps us become comfortable with the depth, properties and position of the basement target. First select the section for Line 4 and then open the inversion toolbar **Tools>Inversion** menu item or right click in the background workspace. This will display the Inversion toolbar which contains the controls for the inversion.

Select the **Free** button which opens the Free Parameters dialog. In the cross section view select the active body to make sure that it appears in the **Select bodies** field in the Free Parameters dialog. All bodies are selected by default and if you start inversion on the single line when all bodies are selected, the behaviour of the individual bodies that are not near the line will behave poorly during inversion. We will address the other bodies during full map inversion.



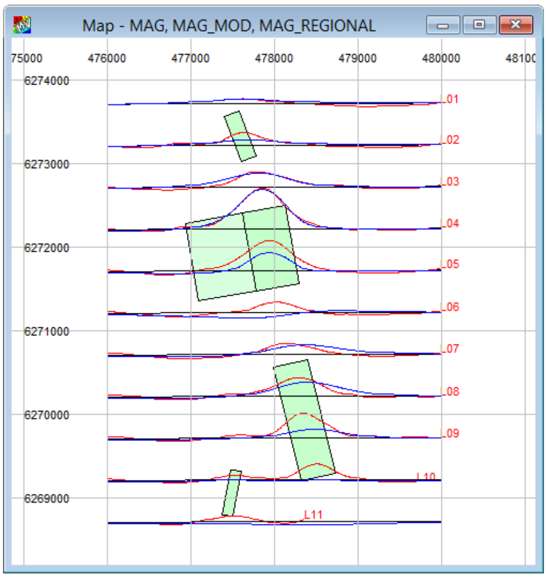
Next set the **Distance, Z body (depth), Dip, Thickness** and **Reg level** free. If you see X, Y parameters instead of Distance, then you must have had the Map Window active when you opened the inversion toolbar. Set the data decimation for inversion to 2 in the data control dialog accessed from the **Data** button on the Inversion dialog. Run the inversion and you should end up with a result similar to that shown in the figure below.



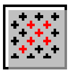
Note that the regional has shifted down from the regional fix points that were calculated at the start. These can be reset in the regional dialog, but we will do that later in the map inversion. The initial inversion gives a depth of around 150m with a dip to the west. Local deviations from the model on the flanks indicate the existence of shallower magnetic sources. These could be associated with the mineral sands targets that we are trying to enhance.

Step 12

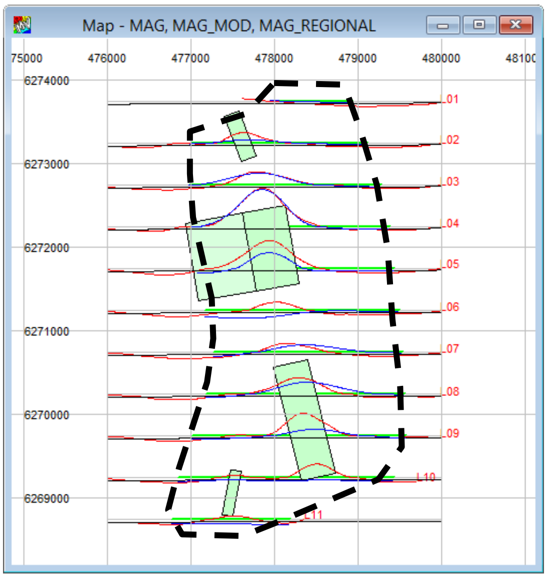
Now it is time to do a full 3D inversion from the map where we will use the stacked profiles to help guide the process. Make the map window the active window and it should look something like the following figure. Make sure that the contour and image layers are turned off.



Note that the Mag channel in red does not cover the full extent of the lines because null values were assigned where the grid data was null. This fact is important because there can be no null value in the data used by inversion.



For this reason, we want to select only non-null data points for inversion and also, only the data points that are immediately affected by the four active bodies. Ensure that the map window is active and then select the **Active Points** toolbar button. Select the **Deselect All** button first followed by the **Draw map polygon** button and then draw a selection polygon around the data points in a shape similar to that shown in the next figure. Make sure there are no null data values in the selection.

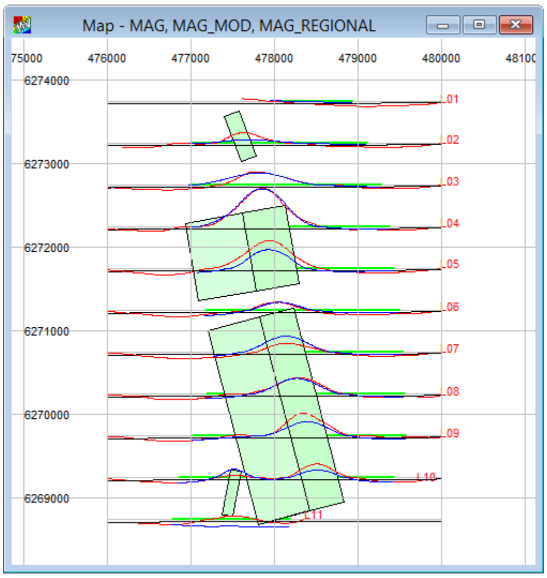


Step 13

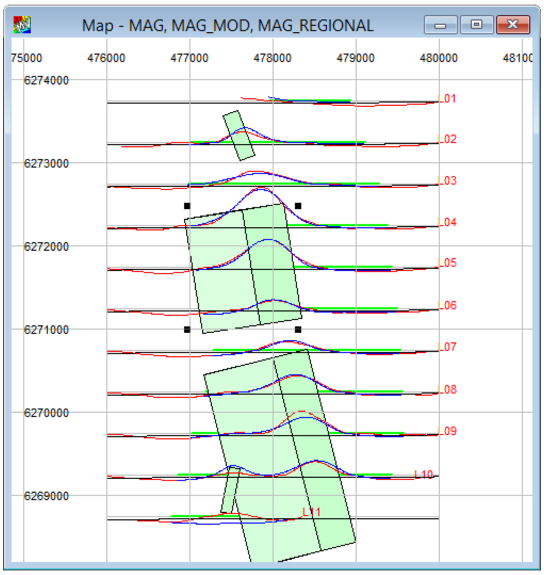
Set the depth of the small bodies to the north and south to 150m to be consistent with the initial inversion results. Adjust the susceptibility if needed.

Set the data decimation to 5 to keep the number of data points used in the inversion to around 500.

Focus on the second large body under lines 8 and 9 bcause the fit for the body covered by line 4 is OK as a starting point. Progressively free up the **Property**, **Azimuth**, **Z**, **X** and **Y**, **Dip** and **Strike**. You can use the **Revert** button at any stage where you feel the inversion has gone in the wrong direction. You can adjust the **Reg. level** at the end of this process after turning off all the free parameters.

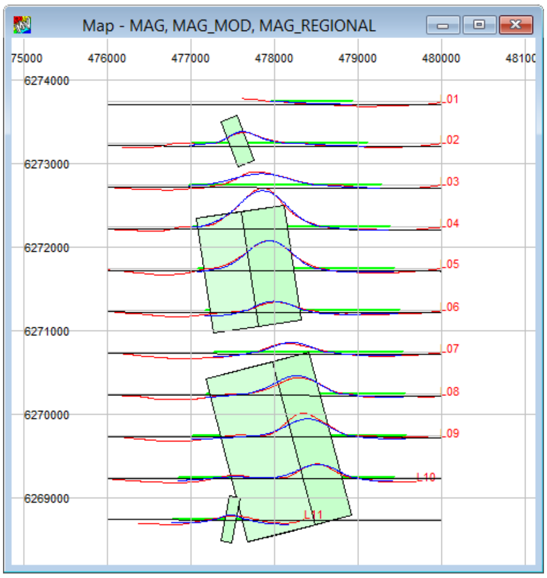


Now select both large **bodies and free up their Thickness, Azimuth, Z, X and Y, Dip and Strike** and you should end up with a solution similar to that shown below**.**



Now disable all free parameters for all bodies and enable the regional level and slope. Run the inversion once and you should see an improvement in the RMS.

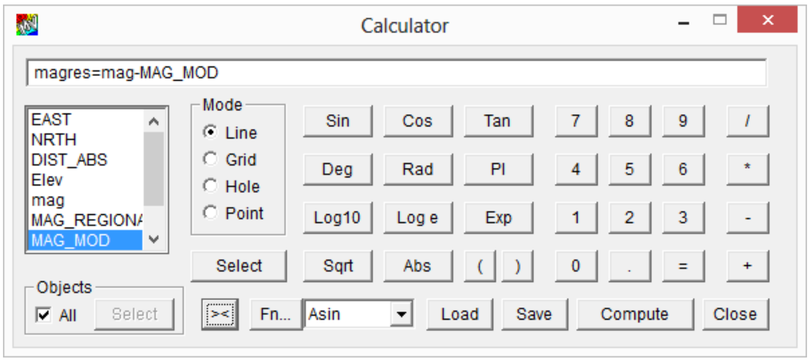
Since the two small bodies have only a minor influence, we can now free up their position, depth and, property. Once you have reasonable matches for these, disable their free parameters and tidy up the two large bodies by free position, depth, dip, azimuth, strike and thickness. You should end up with a solution similar to the figure below.



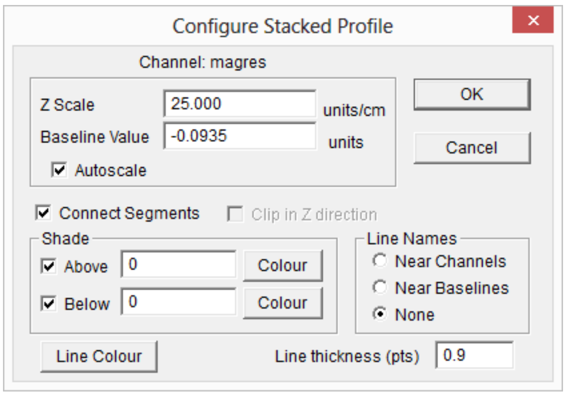
Step 14

Now that we have a model result for the basement geology, we can move on to look at the residual response of shallower magnetic sources. Note that the model response has only been compute within the area of the active points polygon so we must activate all the points on all the lines and run the forward model calculation. Use the Active Points dialog and choose Select All which will activate all data points.

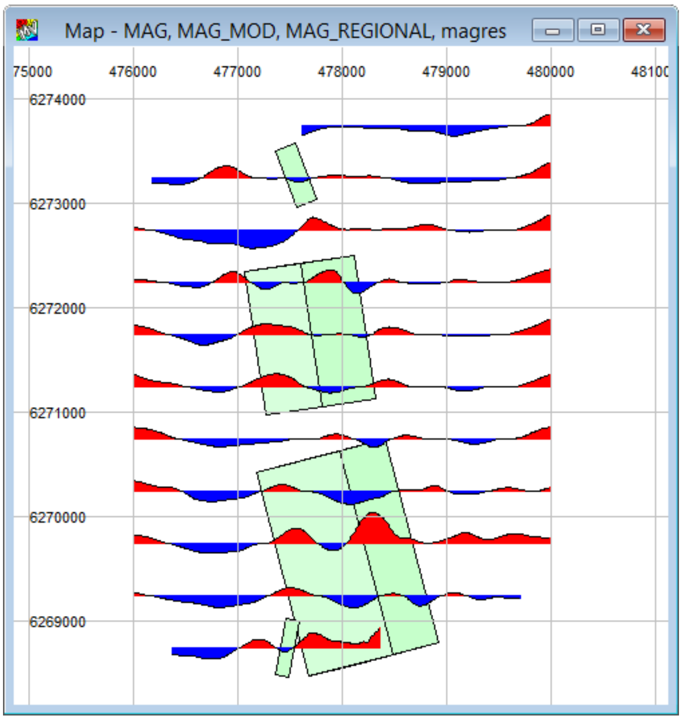
Compute the residual difference between the *mag* field data and *MAG_MOD* model response using the Calculator in Line mode.



In the map window, turn off the stacked profile maps and display the new *magres* field. Set the vertical scale to 25, the line colour to black. Turn on shade **Above** and **Below** and set the base level to zero with colours of Red and Blue respectively.

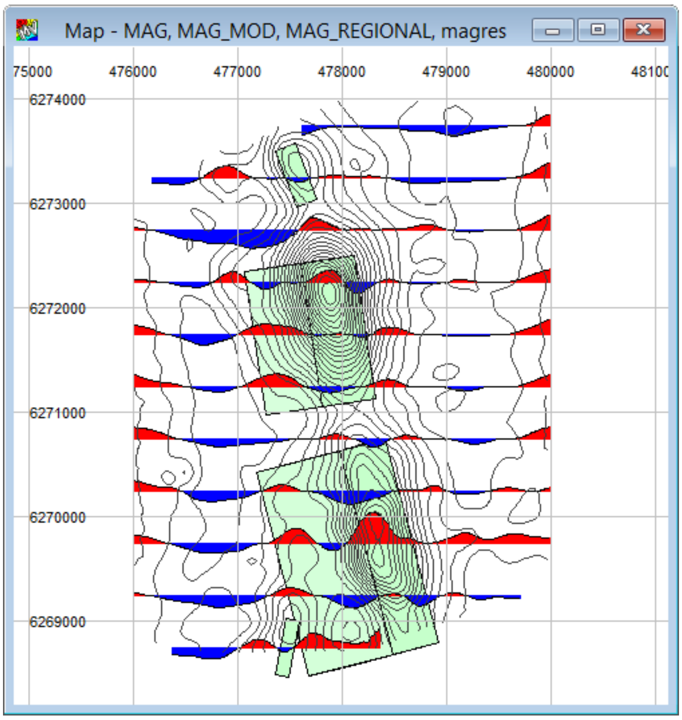


The colour shaded residual highlights both the positive and negative residual trends.



There are small residual difference that result from imperfect modelling, but the knowledge that comes from the modelling work can be use to better understand the character of the residuals and potentially identify and further model small residual anomalies that could be associated with near surface mineral sand deposits.

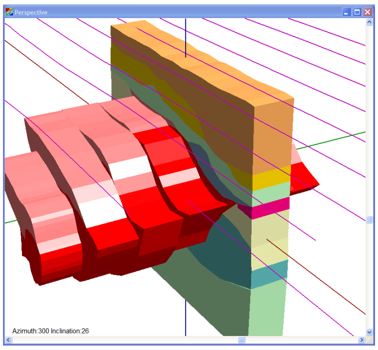
You can add back the contour layer to see how the residuals relate to the main trends.



Tutorial 9 – Modelling PolyGroup Bodies

Aim

This tutorial will teach you how to use the polygroup to build a complete geological section by drawing the model over the tops of a geologist's cross-section. This is the building block for modelling regional geological sections, depth converted seismic sections, full mine models or complex geology associated with an individual magnetic anomaly.



Intended User Level

This tool will be used by all levels as it forms an important part of ModelVision's tool set. Since the tool produces full geological sections it is also important for those wishing to produce reports and publication graphics.

Topics Covered

- Loading the session file
- Checking the project settings
- Checking the modelling settings
- Adding a geological section bitmap to a model section
- Creating your first polygon in the polygroup
- Attaching additional polygons to the group
- Assignment of densities

Background Information

The gravity data used in this exercise is taken from a larger geophysical study of the San Nicolas deposit. The San Nicolas volcanogenic massive sulphide deposit was discovered in 1997 near Zacatecas, Mexico by Teck Corporation. Geophysical and drilling data from the deposit have been used previously by Encom for the evaluation of the UBC GRAV3D voxel inversion using constraints derived from ModelVision. (Pratt, D & Witherly, K 1993, "Integration of polyhedral and voxel-based inversion strategies", GAC-MAC SEG May, 2003, Vancouver, extended abstract 4 pp.)

Tutorial Data

This tutorial is designed to demonstrate the method for building complex cross-sections using a bitmap backdrop. A session file `TUTORIAL 9 SAN NICOLAS POLYGROUP.SES` is provided with the appropriate settings for the regional, model channels and project properties.

Tutorial Steps

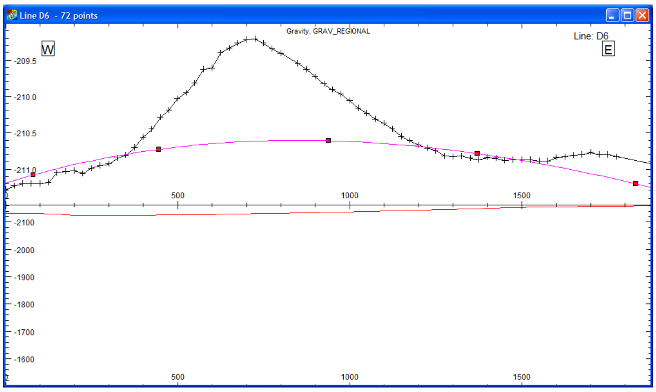
The tutorial will take you through the process of loading a geological section as a bitmap and then work through the steps required to populate the complete geological section.

Loading the session file

Step 1

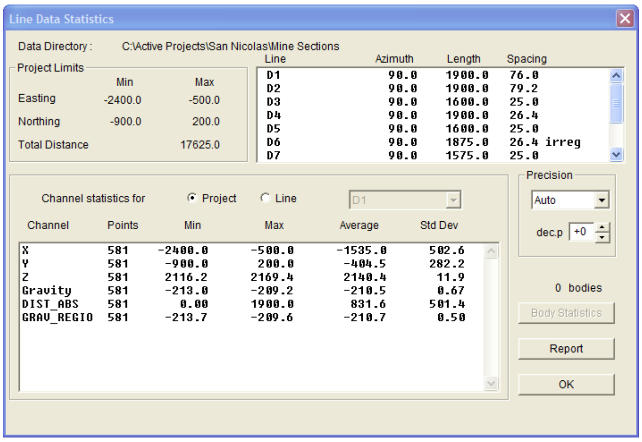
Load the session file from `\PROGRAM FILES\ENCOM\MVIS 14.0\TUTORIAL\TUTE 9\TUTORIAL 9\TUTORIAL 9 SAN NICOLAS POLYGROUP.SES`.

You should see a cross-section view for line D6. The curves show the gravity, regional gravity and terrain.



Cross section for line D6

Check the data channels using the menu function **Utility>Statistics>Line Data**. A gravity data channel (gravity) and gravity regional channel (GRAV_REGIONAL) have already been loaded. A 3rd order polynomial has been computed from all available lines in the dataset.

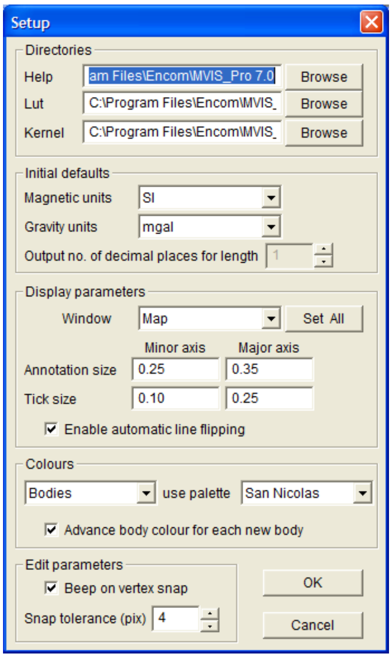


Statistical information for the San Nicolas gravity data

Checking the Project Settings

Step 2

Access the setup dialog from the **File>Setup** menu and check that the colour palette for Bodies is set to San Nicolas. This activates a colour lookup table that matches the colours and formation names associated with the geological section used in this tutorial.



Setup dialog showing the San Nicolas colour palette

Note If the San Nicolas Tutorial lookup table is not listed in the Use Palette drop down list of this dialog then this file will need to be copied into the appropriate folder using the following instructions:

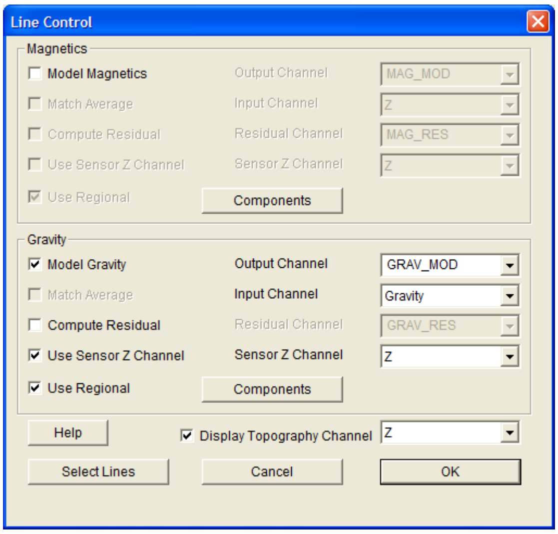
Navigate to `.\PROGRAM FILES\ENCOM\MVIS 11.0\TUTORIAL\TUTE9` and copy the `SAN NICOLAS TUTE.LUT` file to `.\PROGRAM FILES\ENCOM\MVIS\LUT` folder.

Note The snap tolerance shown in the above dialog is set to 4 pixels. ModelVision applies a proximity test to see if you are near an existing vertex on another body and if it finds one, ModelVision will automatically attach to the detected polygon.

Checking the Modelling Settings

Step 3

Check the line modelling settings from the menu **Model>Line Control** to ensure that all channels are appropriately selected as shown in the dialog below.

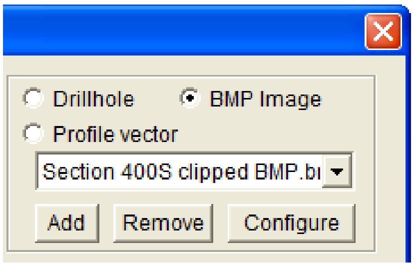


Line Control dialog for the polygroup tutorial session

Adding a Geological Section Bitmap to a Model Section

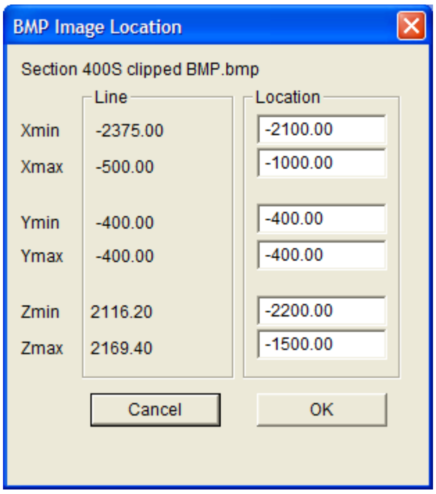
This tutorial uses a geocoded bitmap as a backdrop to guide the construction of a polygroup model. This is not necessary for creation of polygroup models, but it is a useful illustration to show how you can create complex models from control information.

You can attach the geological section bitmap from the cross-section dialog. Select the **Layer Table** button in the main toolbar to display the **Cross Section Layers** table and click on any of the layers with the right mouse button to select the **Add>BMP File** option from the pop-up menu that appears.



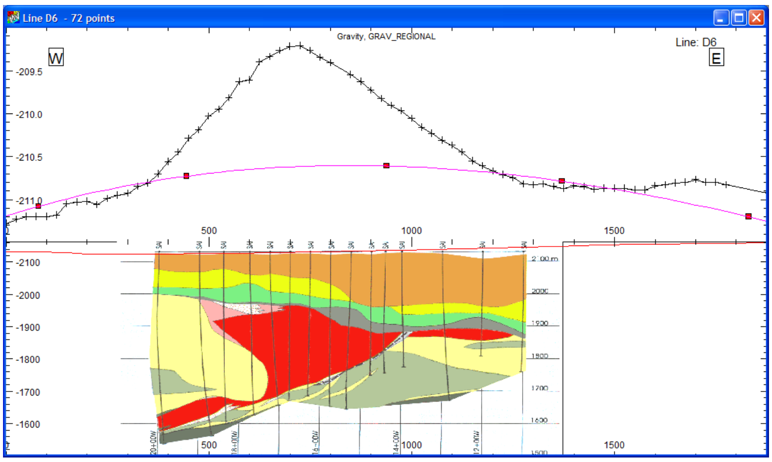
Use the file chooser to locate the BMP file in `\PROGRAM FILES\ENCOM\MVIS 11.0\TUTORIAL\POLYGROUP\SECTION 400S CLIPPED BMP.BMP`.

You will then need to enter the x,y,z coordinates for the image corners.



Dialog for geocoding of the geological section bitmap

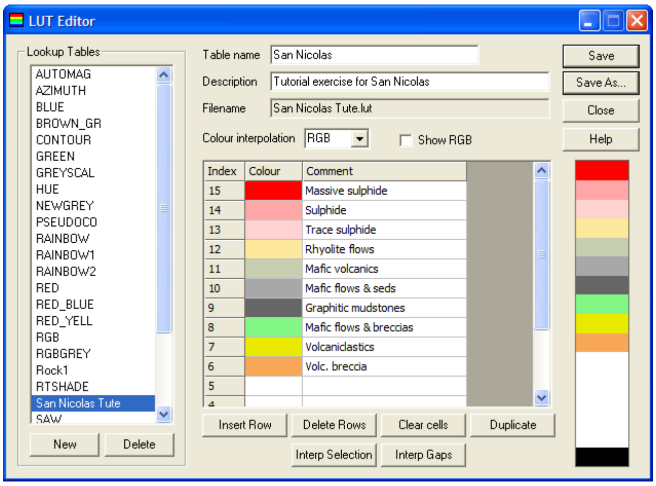
You should now see the geological section appear in the lower section.



Cross-section view with georeferenced bitmap of the geological cross-section

Colour Coded Lithology Table

A colour lookup table called San Nicolas Tute was created specifically to mimic the colours in the geological section. Each colour is assigned a formation name.

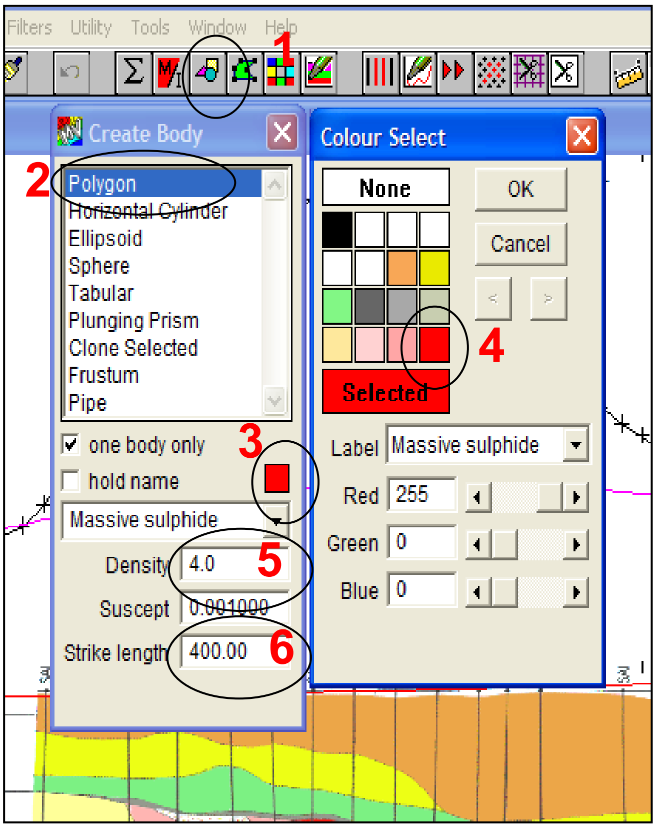


San Nicolas Tute colour lookup table in the LUT Editor

This look-up table file is provided for you in the \TUTORIAL\TUTE9 folder.

Creating your first Polygon in the Polygroup

The first polygon in a polygroup is a standard isolated polygon and you will later attach new polygons to start construction of the section. The figure below illustrates the first set of steps required to create the polygon.

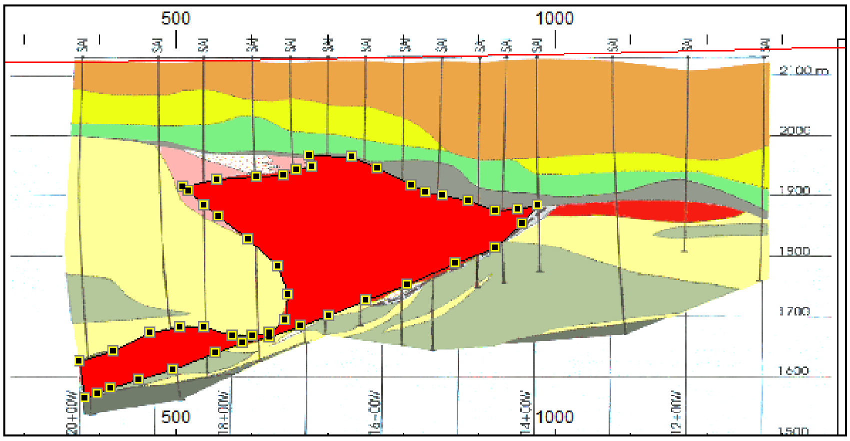


Steps required to create the first polygon in the cross-section

Select the toolbar button Create Body (1) to open the dialog, select Polygon (2) then select the colour square to access the colour/lithology selector (3). Choose the red colour for the Massive sulphide (4) and this lithology will be assigned to the polygon that you draw.

Alternatively, Step 4 can be replaced by selecting from the pull-down list by the name of the formation. Next enter the density (5) and strike length (6) for the polygon. Note that each subsequent polygon in the polygroup will have the same strike length. Also, if you change the strike length of any polygon, all will be changed at the same time.

Trace the path of the red area (Massive sulphide) and close the polygon by selecting the point where you started. The figure below shows the shape of the polygon with each point highlighted. This is achieved by selecting the Body Edit mode toolbar button and then the polygon.



A polygon is drawn around the Massive sulphide formation outline

Attaching additional Polygons to the Group

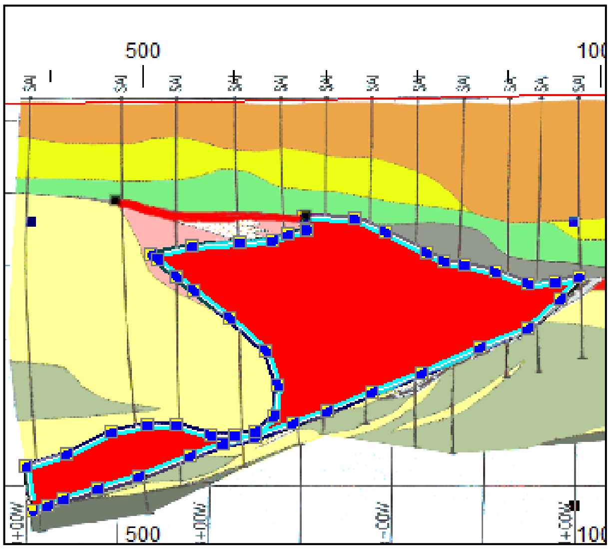
A polygroup is formed by attaching a new polygon to an existing polygon. Two or more connected polygons are required to form a polygroup. Individual polygons within the group can be selected, edited, detached and deleted.

The polygroup has many of the advantages of individual polygons, but share the attributes of strike length, group position and group azimuth. If you change any of these parameters on a single polygon, the polygroup will change.

We recommend that you don't change the azimuth as this may make it difficult to append additional polygons as the polygroup is nominally attached to a line of section.

The process begins by starting the new polygon at a point that is not on the connecting polygon. That is, you will start drawing at a point beyond the first polygon and then later connect to the target polygon. Although starting from a point on the polygon may work, it is not recommended as the attachment is not always reliable.

Select the lithology style called "Sulphide" which has a pink colour and start drawing from the upper left extremity of the zone as shown in the figure below. Trace the upper margin until you connect with the first polygon. You must hear an audible "click" from the computer to know that the point is properly selected. If you don't hear the sound, then move back and over the target vertex until you hear the click.

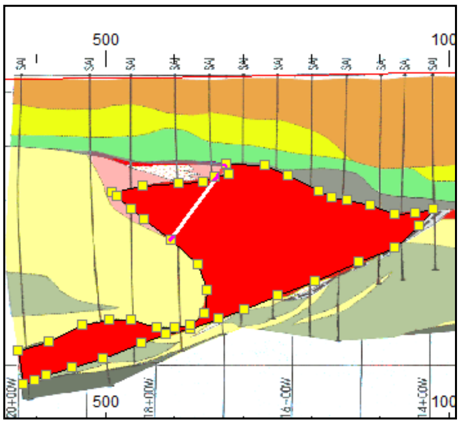


Trace the top of the Sulphide zone (pink) until connecting to the first polygon where an audible click will be heard

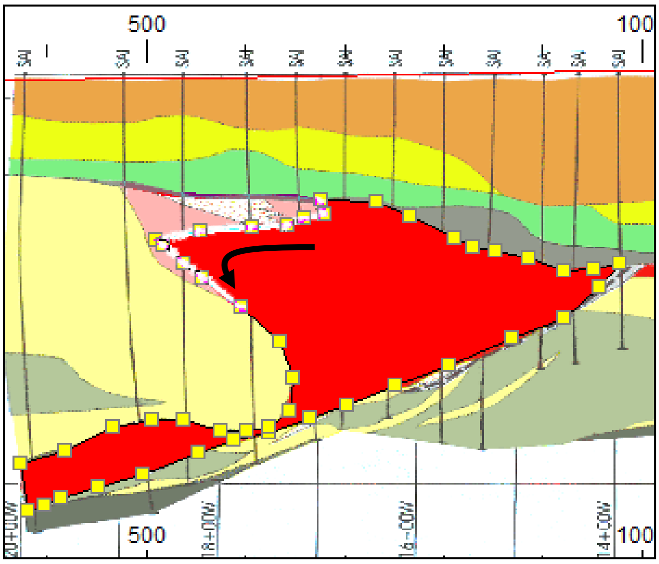
Now that you have "connected", ModelVision knows that you are building a polygroup as opposed to a single polygon. If you accidentally connect to the wrong point, you can undo the point with a right mouse click and connect somewhere else. You can always disconnect an individual polygon from the group via the property page for the body.

Instead of clicking on each point in the joined polygon, you can shortcut the process by connecting to a distant point along the edge as shown in the next figure.

Once selected, ModelVision autotraces the edge of the polygon to the selected point.

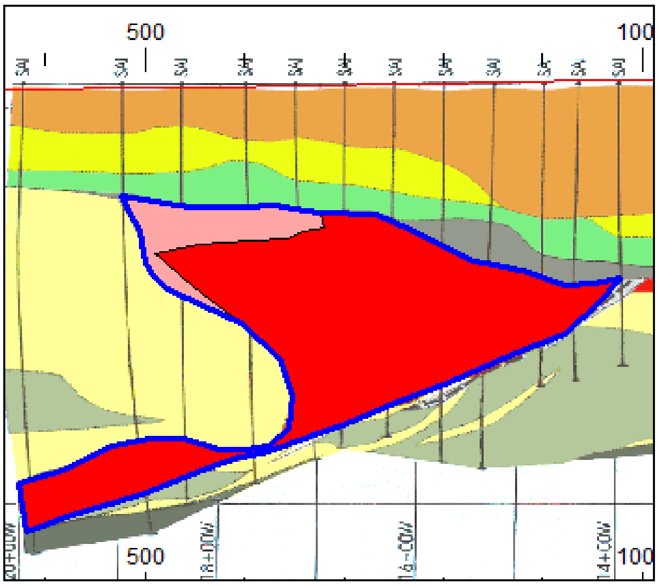


Shortcut selection across the connecting polygon



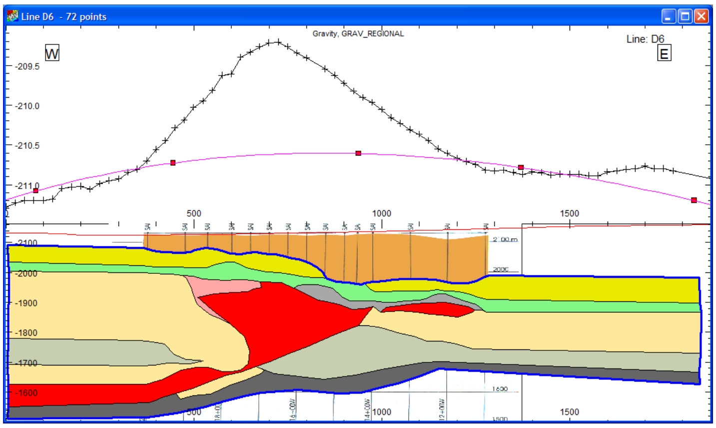
ModelVision completes the job by auto-tracking around the connecting polygon to the selected vertex

You can now complete the drawing of the "Sulphide" zone by tracing the pink zone back to the starting point as shown below. The "Sulphide" zone is filled with the pink colour and the outer border of the two polygons is traced by a thick blue line to define the outer limits of the polygroup.



Completion of the Sulphide zone and blue outline of the polygroup

You can complete the whole geological section in a few moments as shown below.

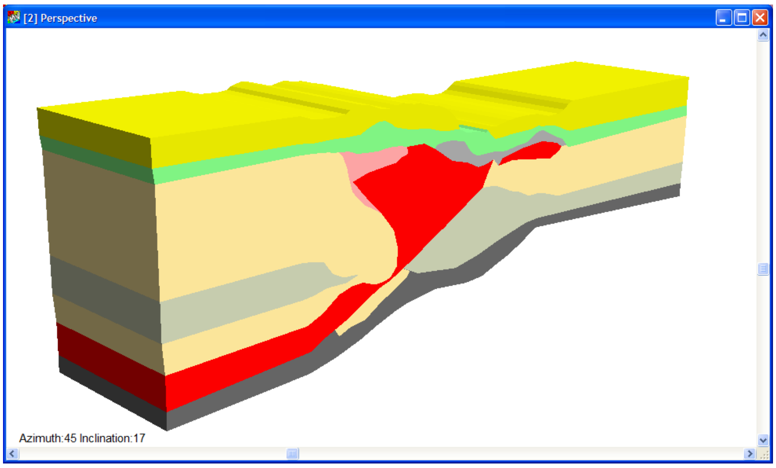


Partially completed and extended polygroup section

The uppermost formation has been left out to illustrate the shape of the polygroup.

Note The outer formations have been extended laterally to avoid sharp edge effects from the arbitrary termination of the section at the end of the geological section.

You can check the shape of the section in the 3D perspective view as it is updated automatically as the model is drawn.



3D perspective view of the polygroup model

Assignment of Densities

You can assign the densities as you create each polygon or edit them in the body table as seen in the figure below.

Body Parameters

	Label	Type	G	Colour	Suscept	Density	Depth	Active
1	Massive sulphide	polygon	1		0.0010	4.0	-1766.9	
2	Sulphide	polygon	1		0.0010	3.7	-1911.0	
3	Rhyolite flows	polygon	1		0.0010	2.65	-1809.4	
4	Mafic volcanics	polygon	1		0.0010	2.75	-1733.8	
5	Mafic volcanics:1	polygon	1		0.0010	2.75	-1733.8	
6	Rhyolite flows:1	polygon	1		0.0010	2.65	-1816.5	
7	Rhyolite flows:2	polygon	1		0.0010	2.65	-1632.3	
8	Graphitic mudstones	polygon	1		0.0010	2.55	-1608.6	
9	Massive sulphide:1	polygon	1		0.0010	4.0	-1877.9	
10	Mafic flows & seds	polygon	1		0.0010	2.80	-1913.3	
11	Mafic flows & breccias	polygon	1		0.0010	2.70	-1953.5	
12	Volcaniclastics	polygon	1		0.0010	2.40	-1946.4	
13	Volcaniclastics:1	polygon	1		0.0010	2.40	-2036.1	

Body parameter table for the completed polygroup body

The Body Parameter table lists the group that each polygon belongs to and in the above table, they are all set to 1.

Note Some formation names have a number following them to indicate a subsequent occurrence of the same formation, while providing a unique name for each polygon.

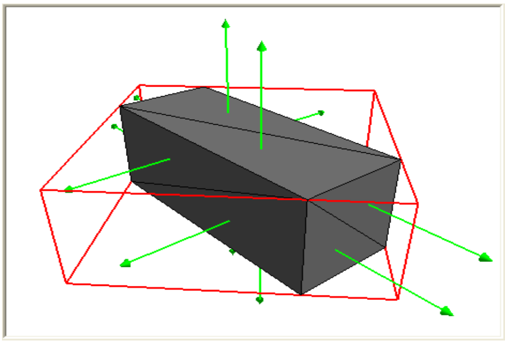
You can now experiment with inversion of the polygroup by allowing some of the densities to vary. The densities will change to get the best match with the gravity along the section. If you are confident in the assignment of densities, then you can use the residual gravity response to look for unexplained components of the geological section, particularly additional massive sulphides.

Note The process can be repeated on adjacent cross-sections to build full 3D models of the sub-surface. The sections must be parallel and touch exactly midway between the sections.

Tutorial 10 – Integrity Check of Imported Models

Aim

This tutorial will teach you how to use the Topology Checker plugin tool to check the integrity of models imported from 3D DXF files and other file formats.



Intended User Level

This tool will be used by all levels as it forms an important part of ModelVision's tool set. Since the tool produces full geological sections it is also important for those wishing to produce reports and publication graphics.

Topics Covered

- Loading the session file
- Checking the project settings
- Checking the modelling settings
- Importing a 3D DXF of a simple model
- Checking the vertices of model faces using the Topology Checker
- Exporting the corrected model as a .TKM file

Background Information

Previously it was essential that the software which created a DXF file used a consistent method of ordering the vertices of every face and that all surfaces were closed. This is a requirement of ModelVision and any gaps or incorrectly defined faces in a model will lead to errors in the model computations.

The topology checker collates all faces and organizes them into closed surfaces reporting any that break the rules. If the problem is with badly ordered faces then it corrects these automatically and allows the user to export the intact surfaces as a new DXF file or as individual polyhedron bodies in a TKM file that will be automatically loaded into ModelVision.

If there are holes in the surface of the body then it does not fix them for you. It will allow you to export the valid closed bodies and reject the invalid ones. It gives the user the means of tracking down the problem in the surface but the user must relate this back to his DXF file and fix it. The fix should involve going back to the program which wrote the DXF file rather than correcting the file.

The topology checker has its own 3D visualizer with many features to identify and examine anything from the entire model right down to individual facets and their coordinates.

Tutorial Data

This tutorial is designed to demonstrate the method for using the Topology Checker to determine the integrity of an imported 3D DXF model. A session file called `TUTORIAL 10_TOPOLOGYCHECKER.SES` is provided with the appropriate settings for the regional, model channels and project properties.

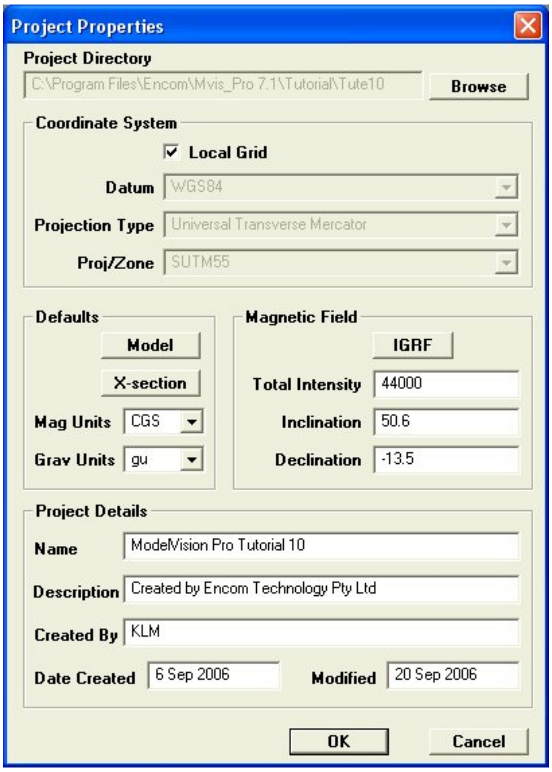
Tutorial Steps

The tutorial will take you through the process of importing a tabular model as a 3D DXF and using the Topology Checker utility to check the model for badly ordered faces or holes in the surfaces.

Step 1

Load the project file from `.\PROGRAM FILES\ENCOM\MVIS 11.0\TUTORIAL\TUTE 10`.

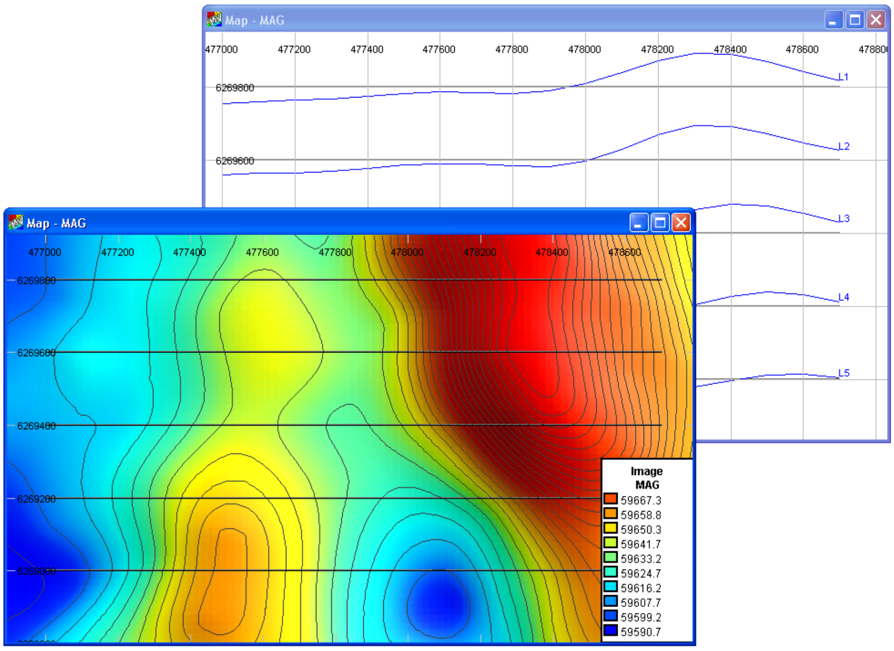
The modelling parameters should be similar to below:



Load the session file from the same project folder, called:

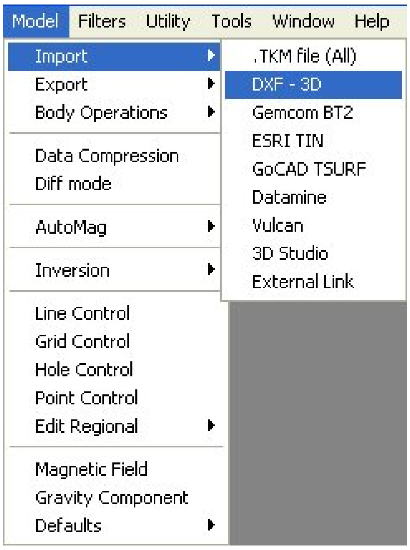
Tutorial 10_TopologyChecker.ses

You should see a 2D grid image map display, showing the magnetics of the dataset.

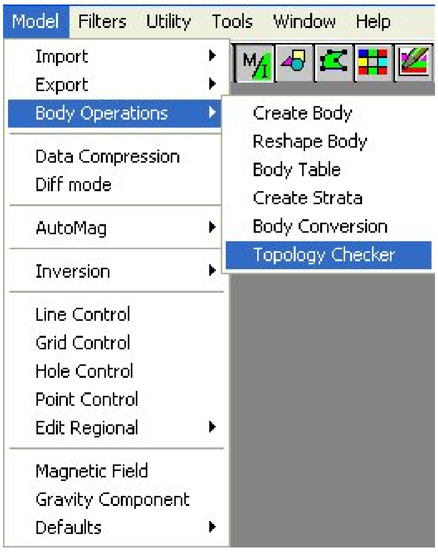


Step 2

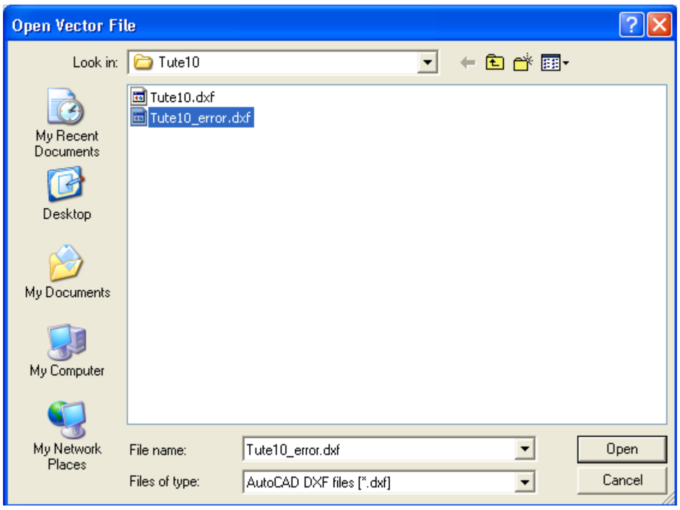
The Topology Checker utility can be accessed from two menu locations in ModelVision. The most obvious place to access this and to import a model is the **Model>Import** menu option. Selecting any of the file formats excluding the native ModelVision format (.TKM file) displayed in this menu option will activate the Topology Checker.



The other location to initiate the Topology Checker is the **Model>Body Operations** menu option.



Initiate the Topology Checker using one of these two menu options and browse for the x-section_error.dxf file located in `.\PROGRAM FILES\ENCOM\MVIS 11.0\TUTORIAL\TUTE10` folder.

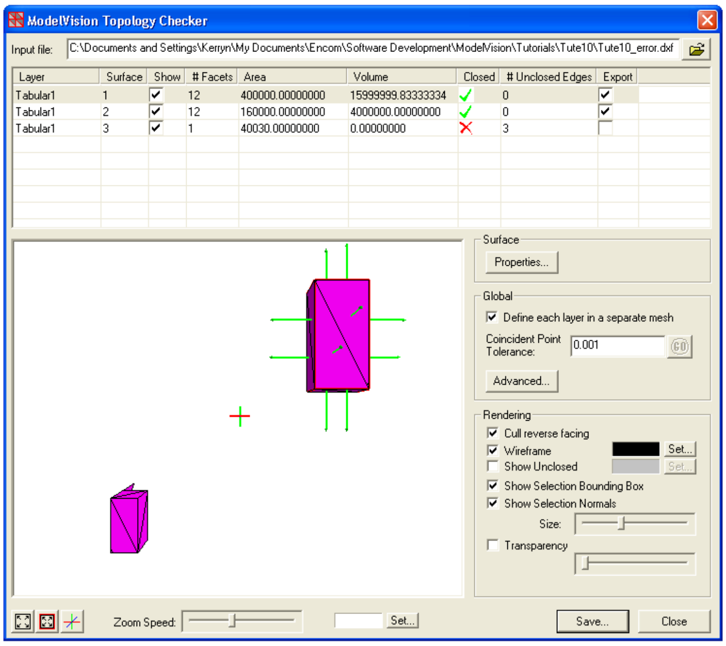


The Topology Checker interface will display with the model DXF file loaded as illustrated below:

Step 3

The Topology Checker interface consists of three display areas:

1. 3D View
2. Input File and List View
3. Surface and Rendering Properties

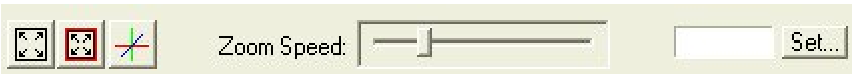


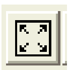
The first and most obvious is the 3D display window where the imported model is displayed in an area where 3D navigation is possible. Similar to the Perspective view in ModelVision the left and right mouse buttons allow navigation in the space. The left mouse button activates the horizontal and vertical rotation of the model and the right mouse button activates the zoom in/zoom out action of the model.

Experiment with the 3D navigation using the left and right mouse buttons to view the model at different orientations.

Step 4

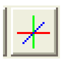
The toolbar beneath the 3D display allows some simple properties of the 3D view to be edited.



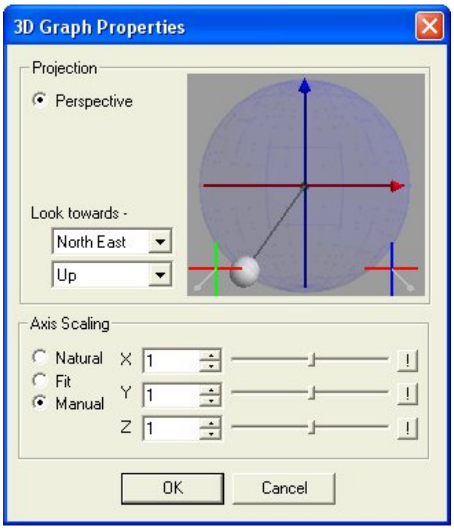


The **Fit view to window** button adjusts the view so that the entire model can be seen. Select this button to reset the view for all bodies in the model.

The Fit view to bounding box of selected surface button fits the view so that the selected surface is the focus of the view. Select this button to reset the 3D view to focus on the selected surface from the List View.



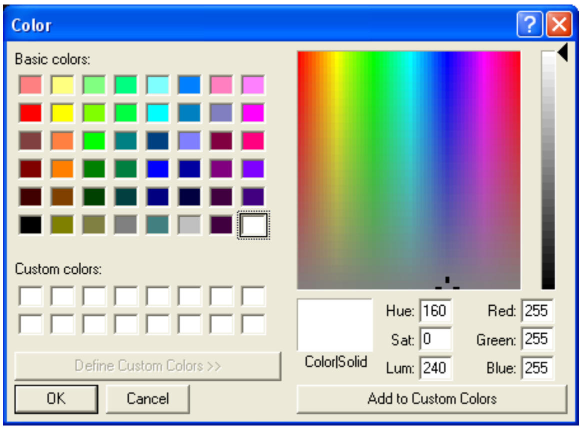
The advanced settings for the axes scale button allows the user to apply individual scales to the X, Y and Z axes, as well as defining the position where the user wants the view to look at. Experiment with this setting by adjusting the Look towards angles and Scaling options. Click on OK after each adjustment to observe the change in the 3D view.



The **Zoom Speed** button changes the speed at which the user can zoom in/out to/from the model. This improves the handling of the model in 3D. Adjust the slide bar for this option and observe the changes in the zoom speed while navigating around in the 3D View.

The **Background Colour** button 

sets the colour of the background window for the 3D view. Depending on the colours used in the model being displayed you may wish to adjust this to **Black** for improved visualisation.



Step 5

The Input File button selects the model file to load into the Topology Checker. To open the Topology Checker a file has to already be chosen but once opened another file can be opened using this button.

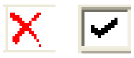
The List view with a (scroll bar option) displays all surfaces for the input model and their corresponding properties. Use the button on the right of the mouse to select the List View column headers to activate a context menu with various options on how to sort and select cells (i.e. ascending or descending).

Layer	Surface	Show	# Facets	Area	Volume	Closed	# Unclosed Edges	Export
Tabular1	1	✓	12	400000.00000000	15999999.83333334	✓	0	✓
Tabular1	2	✓	12	160000.00000000	4000000.00000000	✓	0	✓
Tabular1	3	✓	1	40030.00000000	0.00000000	✗	3	

A definition of the List View Columns is as follows:

Layer	The name of the layer in the input file of one is provided.
Surface	The surface number
Show	Select/Unselect the tick box to show/hide the surface
# Faces	The number of faces in the mesh for the surface
Area	The area of the surface
Volume	The voume of the surface
Closed	Green tick indicates that the surface is closed. A surface is closed if all edges of the triangles in the surfaces mesh are shared by another triangle in the mesh. Blue tick indicates the surface is closed but has a negative volume. Red cross indicates that the surface is not closed.
# Unclosed Edges	The number of edges that are not shared in the surface.
Export	Indicates whether the mesh for this surface should be added into the output file when saving.

Examine the list of model surfaces for the displayed model in the spreadsheet view of the Topology Checker.



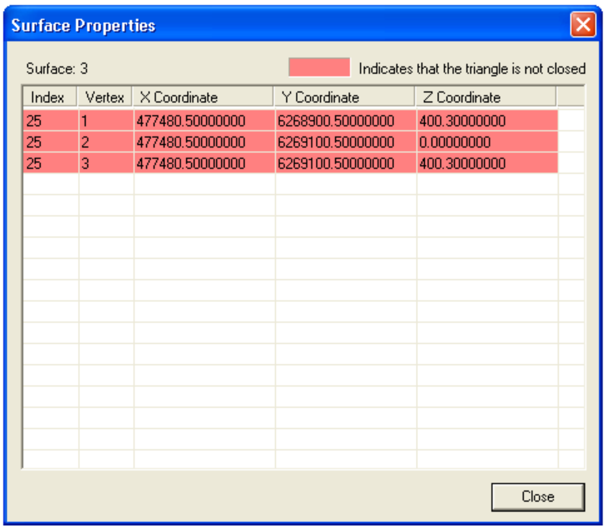
You will notice that the last layer of the imported model has a red cross assigned to the closed column and that the **Export** box has been deselected.

It is obvious while rotating around in the 3D view that the smaller polygon body has an unclosed edge on it.

Step 6

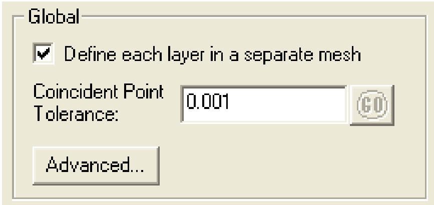
The **Properties** button displays a spreadsheet containing the X,Y, and Z coordinate for each vertex in each triangle for the surface selected in the List View. Triangles that are not closed, i.e. have an edge that does not join another triangle are displayed in a different colour.

Select the **Properties** button to open the “Surface Properties” dialog for the currently selected surface in the list view.



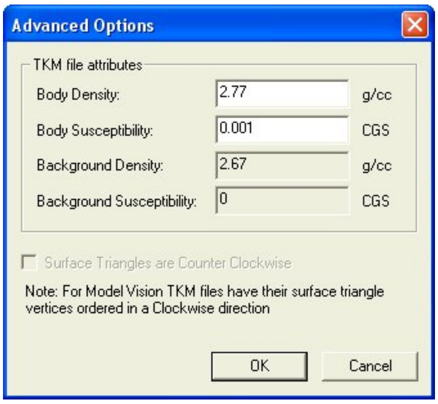
Step 7

In the Global options the **Define each layer in a separate mesh** check box will determine whether a defined layer in the input file should be defined in its own mesh or whether all layers should be defined in a single mesh. This is done on loading the input file.



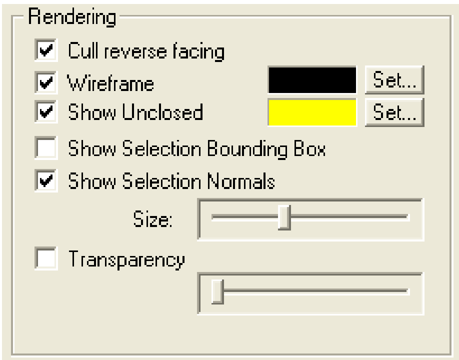
The **Coincident Point Tolerance** value displayed is used to improve the mesh by detecting coincident points and reassigning the triangles to use the first instance of any repeated point. Set this to -1 for the Topology Checker to automatically calculate this value.

Selecting the **Advanced…** button will open the “Advanced Options” dialog. This dialog allows the user to specify the ‘Body Density’ and ‘Body Susceptibility’ values for the model. These are used when saving the model to a .TKM file.



Set the Body Density to **2.77g/cc** and the Body Susceptibility to **0.001** cgs.

Step 8



In the Rendering options the **Cull reverse facing** check box will turn on/off the display of any back facing triangles.

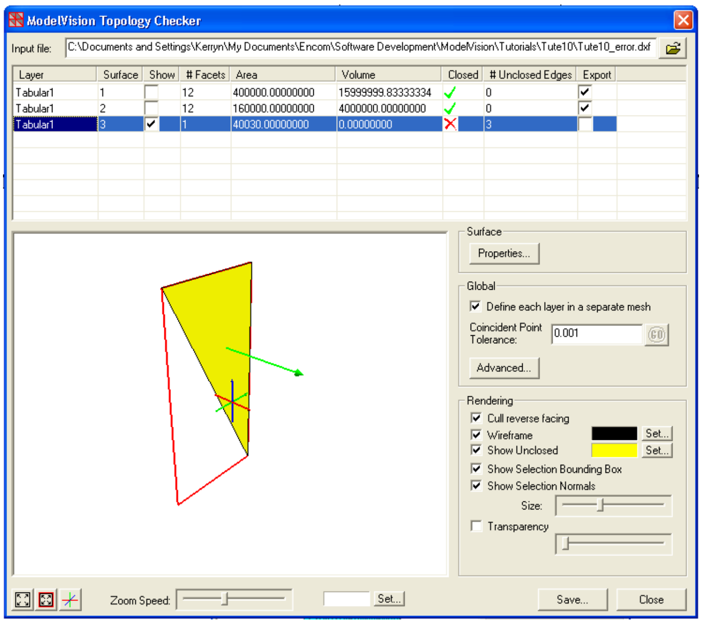
Toggle this option off and on again to observe this feature.

The Wireframe check box will turn on/off the display of the wireframe mesh for the model. The colour of the wireframe mesh can be chosen by using the appropriate ‘Set…’ button.

Turn this option on and set the colour of the wireframe mesh to Black.

Ensure that the **Show Unclosed** check box is turned on and select the **Set** button located next to it to yellow. When selected the **Show Unclosed** check box will override the colour of any triangle faces that are not closed in the model with the colour specified in the ‘Set…’ button.

Then turn off the **Show** check box in the spreadsheet view for the first two surfaces to visually inspect this layer in its entirety. A break in the surface has rendered an edge of the triangle in the mesh surface to not be shared by another triangle in the mesh and hence these are recognised as two separate surfaces with a computed area but no volume.

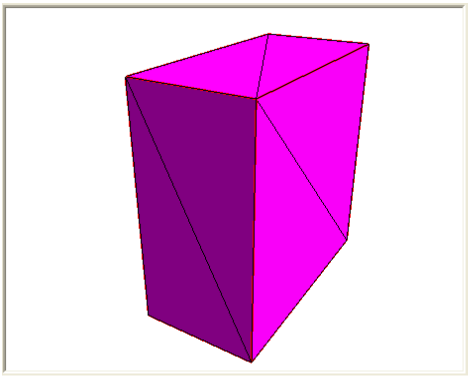


Step 9

The **Show Selection Bounding Box** check box will turn on/off the bounding box for the model surface. The red coloured bounding box is defined for the currently selected surface in the list view.

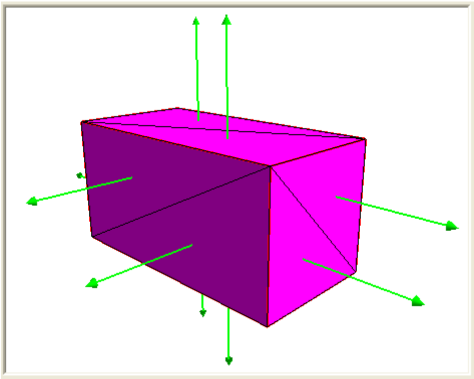
Select the problematic surface in the List View and then if not already done so, turn on the **Show Unclosed** check box from the Rendering options section of the dialog. Turn on the **Show Selection Bounding Box** check box and you will notice a red box displays around the surface highlighted in the List View.

Layer	Surface	Show	# Facets	Area	Volume	Closed	# Unclosed Edges	Export
Tabular1	1	✓	12	400000.00000000	15999999.83333334	✓	0	✓
Tabular1	2	✓	12	160000.00000000	4000000.00000000	✓	0	✓
Tabular1	3	✓	1	40030.00000000	0.00000000	✗	3	



Step 10

The **Show Selection Normals** check box will turn on/off the display of normals for the model. The normals are defined for the currently selected surface in the list view only. The size of the normals can be modified using the ‘size’ slider control. **Show Selection Normals** check box and select the first problem volcanic breccia layer.



Note

A normal is a line perpendicular to another. In this sense it is a direction or line perpendicular to a surface (a face or facet on the body).

Normals are important for two reasons both related to the direction of the normal which can be either into or out of the body. The direction is determined by the ordering of the vertices and ModelVision requires each face to have clockwise ordering when looking into the body from outside.

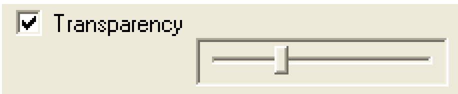
1. In the display the direction of the normal determines whether it is an outside or an inside face and it may be displayed differently or not at all. In ModelVision for instance back faces are not displayed.

2. The computation of the magnetic or gravity response is done for each face on a polyhedral body and the sign will be reversed if you change the direction of the vertices and hence the surface normal. Unless all faces are ordered consistently the response will be wrong.

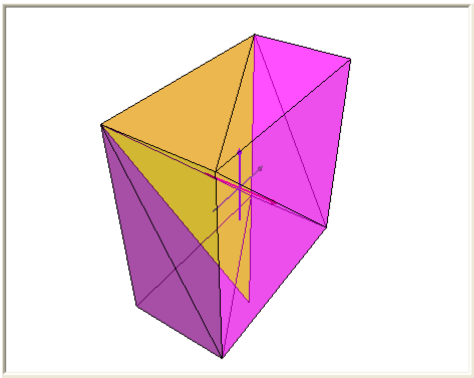
The display of the normals is useful in seeing incorrectly ordered faces.

Step 11

The **Transparency** check box will turn on/off transparency for the model. The degree of transparency can be modified using the appropriate slider control.



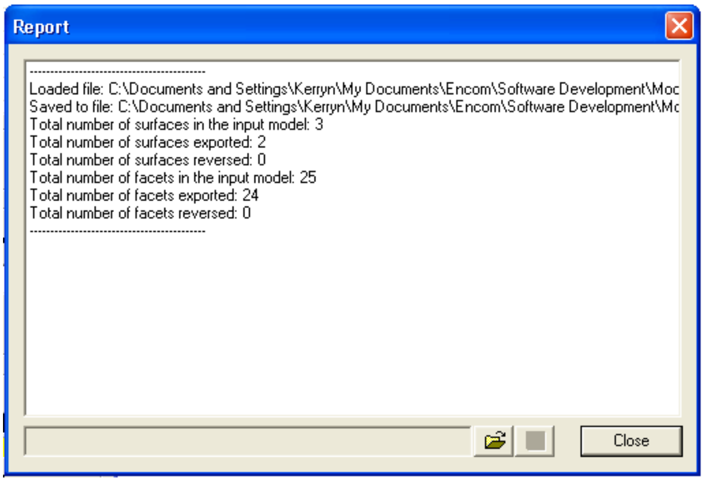
Adjust the **Transparency** slide bar to 30% and observe the changes in the model's appearance.



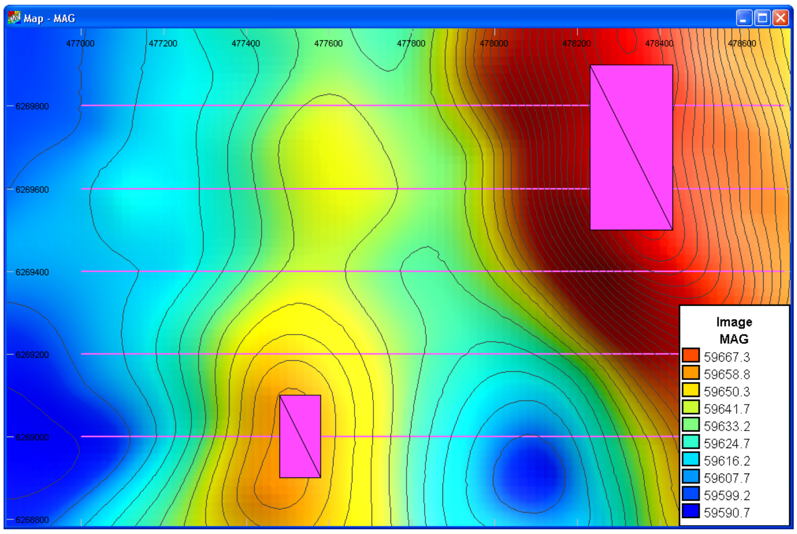
Step 12

Finally select the Save button and save the imported model in the project folder as a TKM file called `TUTE10.TKM`.

A report window will appear providing a summary of the imported model changes made while using the Topology Checker. This can be saved as a .TXT file for future inspection.



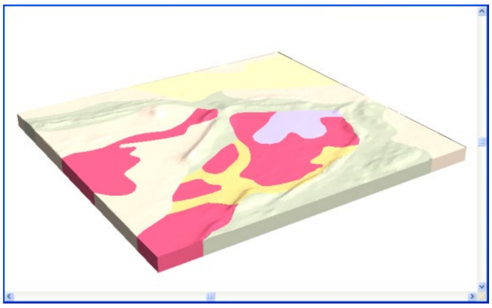
Select the **Close** button and the report dialog will disappear to reveal the map window showing the imported bodies.



Tutorial 11 – 2D Model Generator

Aim

This tutorial will teach you how to use the 3D Model Generator to create a closed 3D solid body with geophysical attributes from an imported vector file or digitized polygon extruded between a top and bottom surface.



Intended User Level

This tool will be used by advanced modellers and those wishing to extend their use of ModelVision by using the UBC Mesh Creator module. Since the tool produces fully closed 3D geological models it is also important for those wishing to produce reports and publication graphics.

Topics Covered

- Loading the session file
- Checking the project settings
- Assigning surfaces to 3D Model Generator
- Drawing a polygonal boundary on an elevation surface
- Importing a GIS vector file containing geological polygons
- Generating a 3D body
- Displaying the 3D model in Perspective view

Background Information

ModelVision will allow you to build complex 3D geological models from hand drawn polygons or imported GIS vector files using the 3D Model Generator and Extrusion Wizard. With this tool you can convert a two dimensional map polygonal interpretation into a meaningful three dimensional model with associated properties of elevation, volume, density and susceptibility.

High resolution terrain surfaces are compressed using a triangle reduction method developed by Encom that is designed to preserve the resolution of the terrain and reduce the numerical computation time for the models. This is an excellent tool for computation of terrain corrections.

Use the 3D Model Generator to create a ModelVision model from a freehand drawn polygon or an imported geological map in the form of a vector file (such as .SHP, or .TAB file format) containing polygons with attributes assigned to density and susceptibility. The polygons are then extruded vertically between two surfaces (typically an elevation surface and basement or base of weathering surface) to produce a closed 3D solid body.

The Extrusion Wizard allows you to refine your 3D model by applying other properties such as body name, colouring, azimuth and dip for the 3D body.

These models can be applied to problems such as terrain corrections in airborne gravity gradiometry, synthetic survey simulation in a known terrain, waste dump modelling in engineering geophysics, salt diapir modelling for petroleum or seeding of a UBC voxel inversion model.

Tutorial Data

This tutorial is designed to demonstrate the various methods for using the 3D Solid Generator and the related Extrusion Wizard plug-in tool to produce a 3D body with modelling properties such as density and susceptibility. A session file called *Tutorial 11_3DModelGenerator.ses* is provided with the appropriate settings for the grid surfaces, geological setting and project properties.

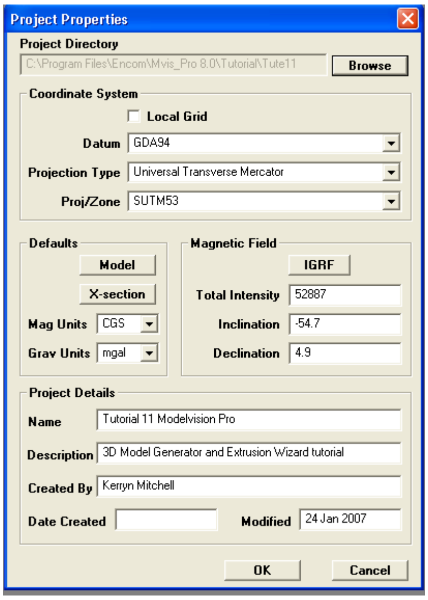
Tutorial Steps

The tutorial will take you through the process of importing a GIS vector file and drawing a polygonal boundary to create a 3D model using the 3D Model Generator and then using the advanced properties of the Extrusion Wizard plug-in.

Step 1

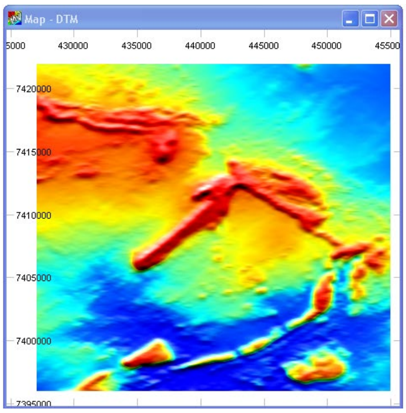
Load the project file from `.\PROGRAM FILES\ENCOM\MVIS 11.0\TUTORIAL\TUTE 11`.

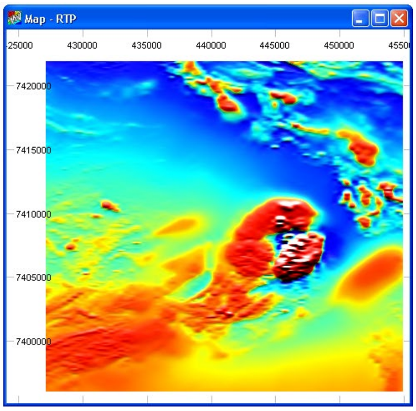
The coordinate and modelling parameters should be similar to below:



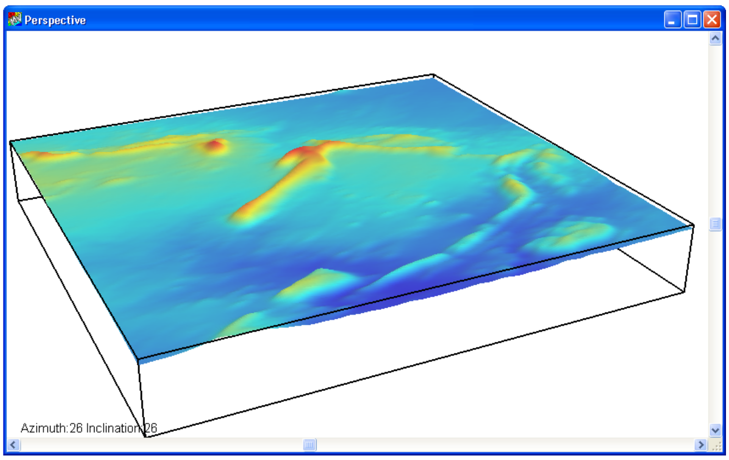
Load the session file from the same project folder, called `TUTORIAL 11_3DMODELGENERATOR.SES`.

You should see two map windows displaying an elevation (DTM) grid in one window and a corresponding RTP magnetics grid in the other.





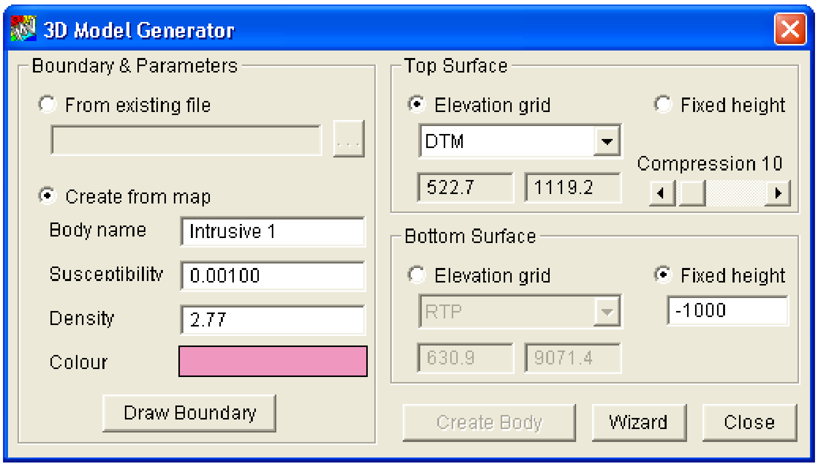
In a **Perspective** view you will also see a **3D Grid** surface of the DTM data vertically scaled by a factor of 2.



Creating a 3D Model by drawing a boundary on a grid image

Step 2

The **3D Model Generator** can be accessed from the toolbar of ModelVision. When this button is selected the 3D Model Generator dialog appears similar to that below:



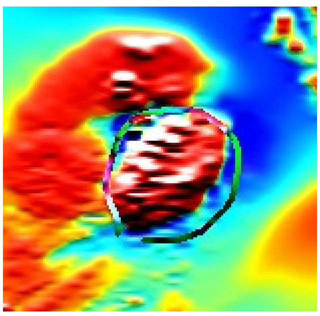
Ensure that the Boundary & Parameters option to **Create from map** is selected and enter a **Body name** of *Intrusive 1*. Also set a **Susceptibility** of *0.001* and **Density** of 2.77. Click on the coloured box and change the colour to *Pink*.

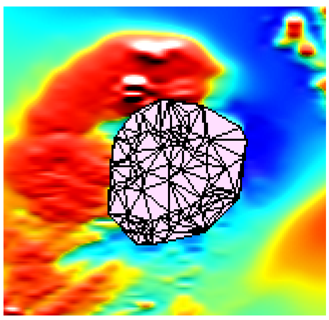
For the **Top Surface** specify an *Elevation grid* and select the loaded *DTM* grid surface from the drop down list. For the **Bottom Surface** specify a *Fixed height* of *-1000m*.

Step 3

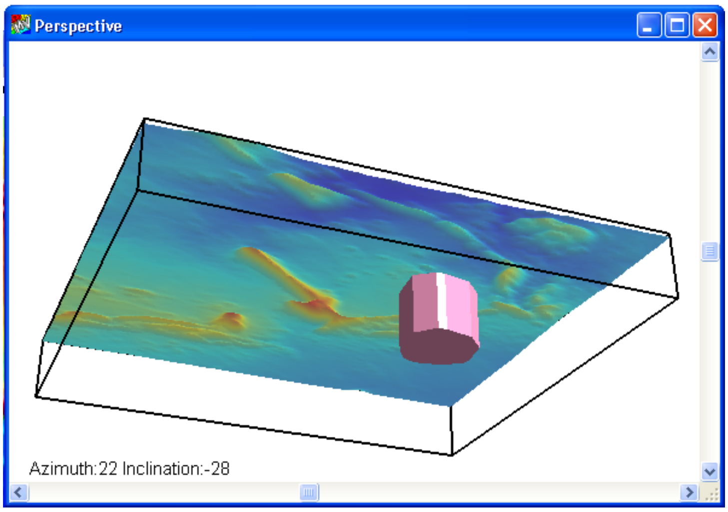
Highlight the map window containing the RTP grid image and then select the **Draw Boundary** button on the 3D Model Generator.

Move the cursor into the map window and the cursor will change appearance to a cross-hair. Draw a polygon by using either free hand (by clicking and holding down the left mouse button) or specifying vertex locations of the polygon around the magnetic anomaly as illustrated below. To close the polygon double left mouse click or create the last node in the same position as where you began the polygon.





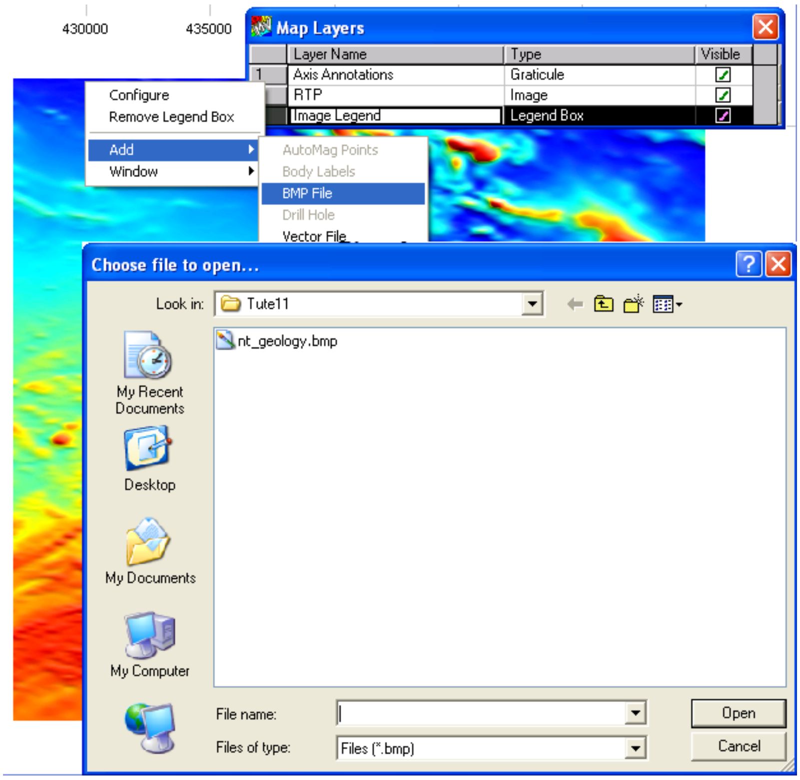
Once the outline has been drawn on the map select the Create Body button and the 3D model will appear in the map window as well as the Perspective view.



Creating a 3D Model by drawing a boundary on a BMP Backdrop image

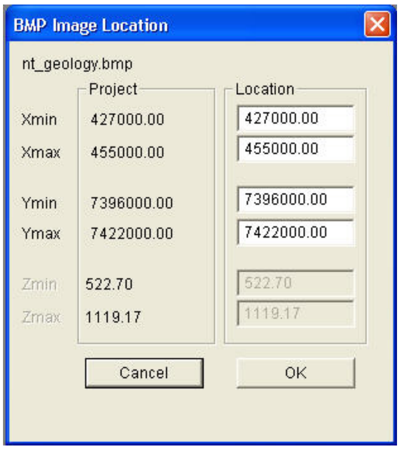
Step 4

Return to the map window containing the RTP grid image and select the **Layer Table** button in the main toolbar to display the **Map Layers** table. In the Project folder select the `NT_GEOLOGY.BMP` file and open this.



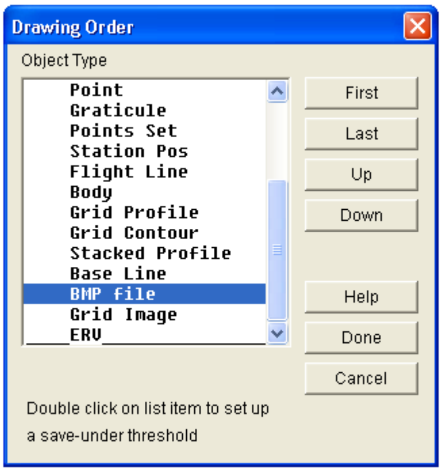
The BMP Image Location dialog will appear next. If the image has already been geo-located (e.g. from a .EGB file) the X and Y coordinates will be automatically loaded into this dialog. If the image is independent of any geo-location you will need to enter the X and Y minimum and maximum values into this dialog manually.

A .EGB file exists for this bitmap image so the coordinates will appear in the dialog automatically. Click OK to exit from this dialog and return to the Map Configuration dialog where you will also need to click OK.



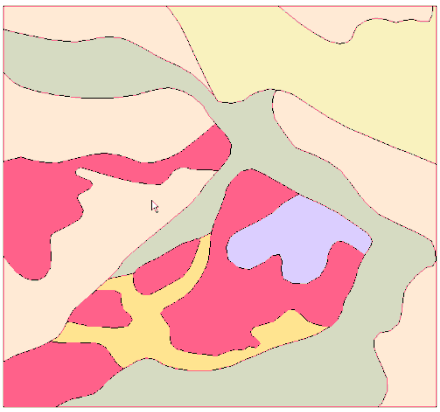
Step 5

ModelVision has a specific drawing order for most display options and in this case a grid image will always taken precedence over a bitmap backdrop image. Therefore the map window will still contain the RTP grid overlaying the bitmap image. To change this order click on any layer in the **Map Layers** table with the right mouse button and select the **Window>Drawing Order** pop-up menu option.



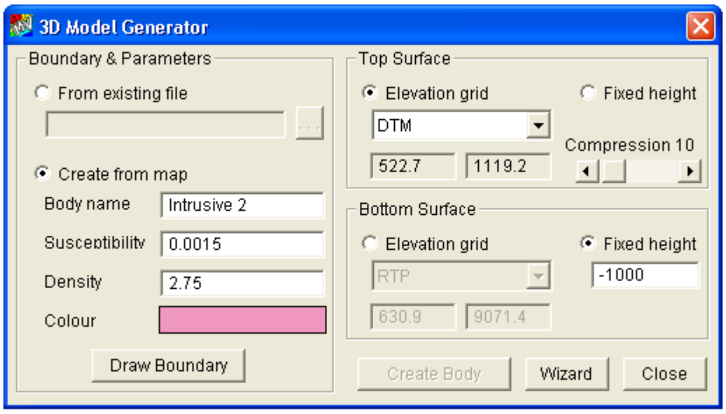
The list displayed in this dialog shows a hierarchy for drawing in the map window. Scroll down to the bottom of the list to find the *BMP file* object and highlight it. Select the Up button to move this object so that is sits above the *Grid Image* object type. Select the **Done** button to exit the Drawing Order dialog and to return to the map window.

A geology map bitmap image of the project area will be displayed over the top of the RTP grid image.



Step 6

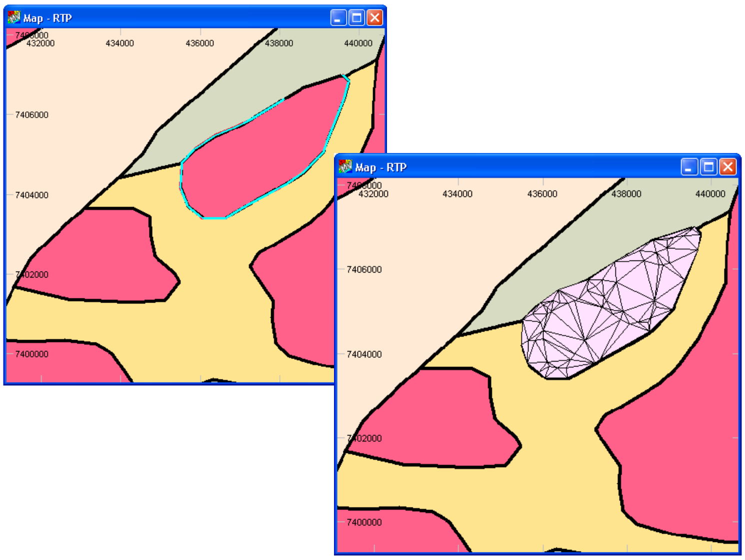
We can now use this image as a reference for drawing a polygon representing a geological boundary in the 3D Model Generator. Zoom in to the map window to view closer the two smaller pink polygons (representing granite intrusions) then select the 3D Model Generator button on the toolbar to display the dialog.



For the *Boundary and Parameters* option select the **Create from map** option and enter a **Body name** of *Intrusive 2*. Specify a **Susceptibility** value of *0.0015*, a **Density** value of *2.75* and **Colour** of *Pink*.

For the Top Surface specify the **Elevation grid** to be represented by the *DTM* grid and set the Bottom Surface to be at a **Fixed height** of *-1000m*.

Select the **Draw Boundary** button and digitise around the more easterly intrusion as illustrated below. Once completed select the **Create Body** button and Close the 3D Model Generator dialog to observe the new body displayed in the Map and Perspective views.



Creating a 3D Model from a Drawn Boundary using the Extrusion Wizard

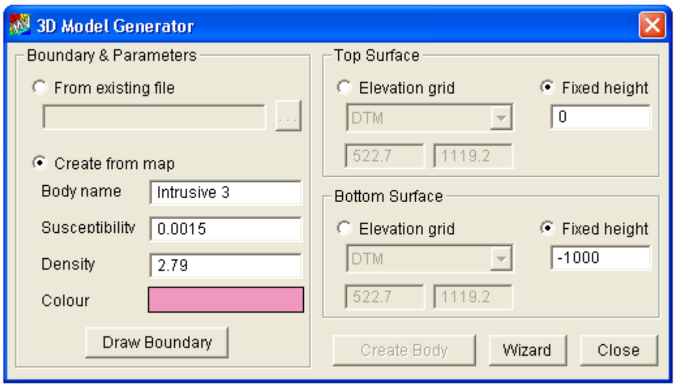
Step 7

To assign additional properties to the created 3D model the Extrusion Wizard lug-in utility can be accessed from the 3D Model Generator dialog after initially drawing a polygon from a map view.

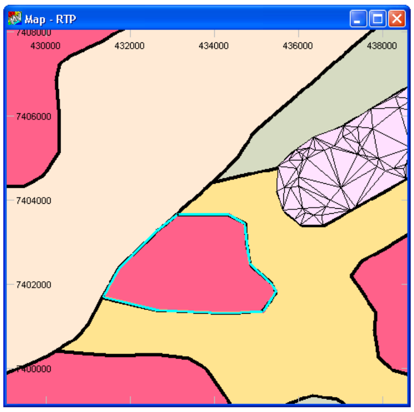
First of all centre the zoomed in geology map image on the granite intrusion (pink) polygon located south-west of the previously digitised polygon.

Open the **3D Model Generator** dialog and specify the **Create from map** option for creating the Boundary & Parameters of the model. Enter a **Body name** of *Intrusive 3*, **Susceptibility** value of *0.0015*, **Density** of *2.79* and a **Colour** of *Pink*.

This time leave the Top and Bottom Surface properties as the default values (i.e. Fixed height of 0 and -1000).

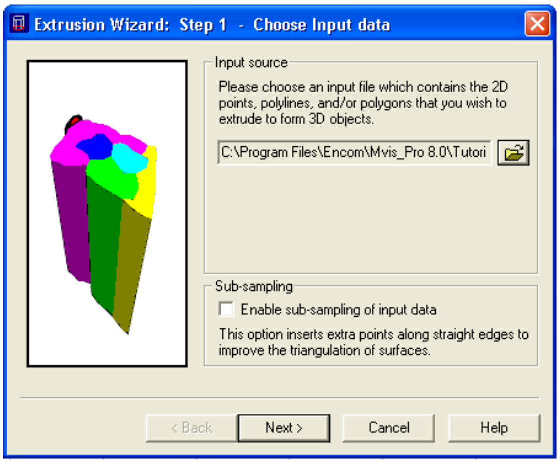


Select the **Draw Boundary** button and digitise around the centred granite intrusion as illustrated below.

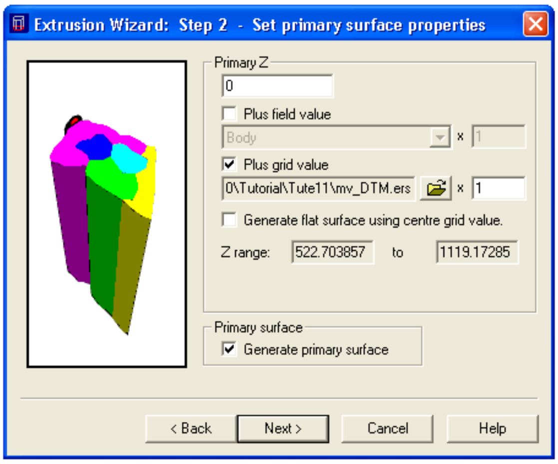


Step 8

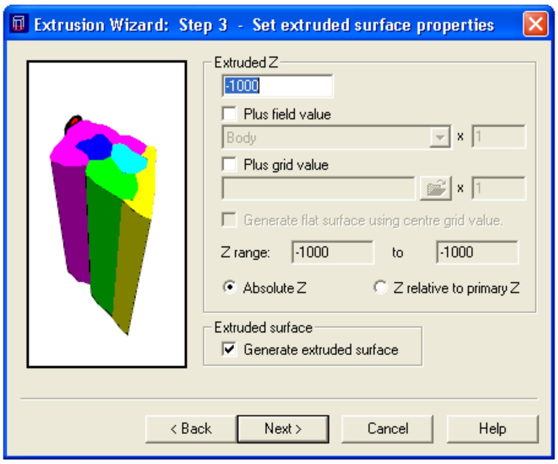
Once completed instead of creating a 3D model from this dialog select the **Wizard** button to open the **Extrusion Wizard** plug-in dialog. The first dialog of the wizard will appear similar to below showing the input file source of the polygon digitised on to the map window, a .TAB file called `MV_TMP`. Select the **Next** button to move to *Step 2* of the wizard.



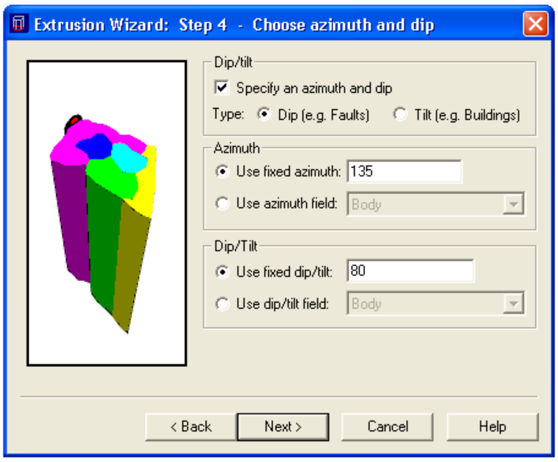
The **Step 2** dialog displays the **Set primary surface properties** as was shown in the 3D Model Generator dialog for the Top Surface. Set the **Primary Z** value as *0* and click on the **Plus grid value** and browse to the Project folder for `TUTE11` to select the `MV_DTM.ERS` grid file. The *Z range* of the loaded grid will be displayed beneath this entry. Click **Next** to proceed to *Step 3*.



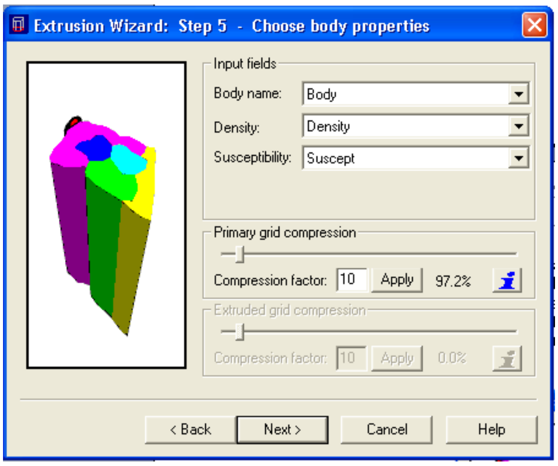
The **Step 3** dialog displays the **Set extruded surface properties** as was shown in the 3D Model Generator dialog for the Bottom Surface. The **Primary Z** box has *-1000* entered as was the value specified in the 3D Model Generator dialog. Click **Next** to proceed to *Step 4*.



The **Step 4** dialog of the Wizard allows the user to specify an **Azimuth** and **Dip** of the 3D model created. Select the box for **Specify an azimuth and dip** to place a tick here and in the **Azimuth** properties select the **Use fixed azimuth** option and enter a value of *135* degrees. In the **Dip/Tilt** properties select the **Use fixed dip/tilt** option and enter a value of *80* degrees. Click the **Next** button to proceed to *Step 5*.

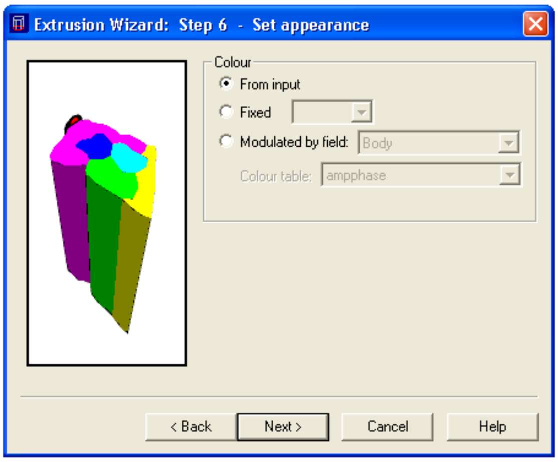


The **Step 5** dialog allows you to **Choose body properties** to be assigned, consisting of **Body name**, **Density** and **Susceptibility**. In this case the properties are taken from the *Boundary & Properties* section of the **3D Model Generator** dialog. When importing an existing GIS vector file (which we will do in the next exercise) data fields from the vector file can be assigned to these input fields. Leave these fields as the default setting and click **Next** button to proceed to *Step 6*.

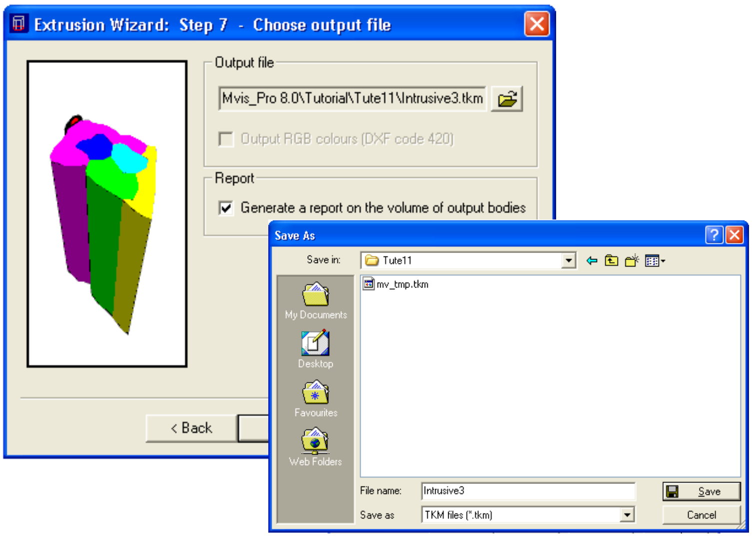


The **Step 6** dialog allows you to specify the colour appearance of the 3D model to be generated. The default option of **From input** will colour the model with the colour that was specified in the **3D Model Generator** dialog, i.e. Pink or in the case or an existing vector file that was imported the colour assigned in the vector file. The next option called **Fixed** allows you to override the input colour with a specific colour from the drop down list. The **Modulated by field** colour option allows the model to be colour modulated by a specific field from the input file, such a susceptibility using a specified **Colour table** from the drop down list.

For this exercise select the **From input** colour option and click the **Next** button to proceed to *Step 7*.

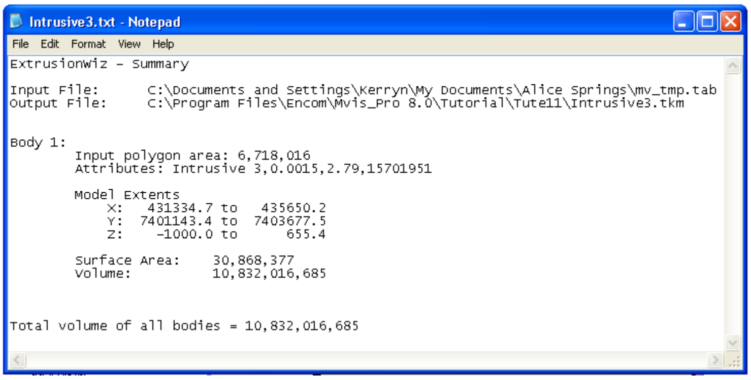


The **Step 7** dialog for the Extrusion Wizard is to **Choose an output file** (.TKM) for the 3D model. Select the browse button to specify the output file location and name and in the project folder for `TUTE11` call the file `INTRUSIVE3.TKM`. Select the **Save** button to return to the Step 7 dialog.

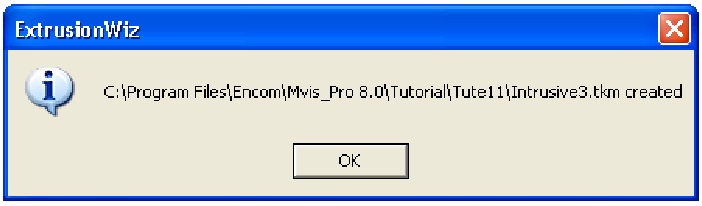


Note A DXF file format can also be specified here but if this format is chosen the model will not be automatically displayed in ModelVision and will need to be imported using the **Model>Import>DXF format** menu option.

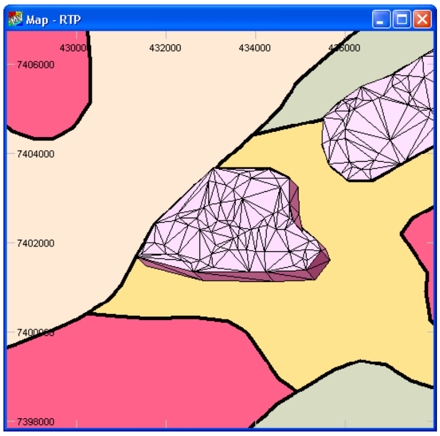
The option to **Generate a report on the volume of output bodies** exists for producing a .TXT file containing the calculated dimensions of the 3D model. An example of this is illustrated below.



Select this option and click on the **Finish** button to complete the 3D Model generation using the Extrusion Wizard. A message will appear similar to the one illustrated below.



Click **OK** and a new message will appear asking to **Append** or **Overwrite** the currently displayed models from the previous exercises. Select **Append** and the new 3D model of a dipping body will be displayed in the map and perspective view windows.



Save the current models in a new session by using the **File>Save As** menu option and call it `TUTORIAL11_3DMODELGENERATOR_A.SES`.

Exit from this session by selecting the **File>Close** menu option.

Creating a 3D Model from a Vector File using the Extrusion Wizard

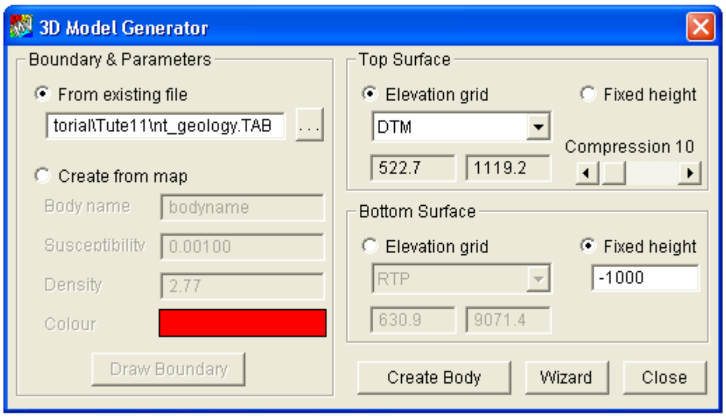
When working with a series of complex polygons as found in a geology map, rather than drawing a boundary around each rock unit one at a time as instructed in the previous exercises it is more beneficial to import an existing GIS vector file containing these polygons and already assigned geophysical properties.

Step 9

From the same `TUTE11` project folder open the original session file called `TUTORIAL11_3DMODELGENERATOR.SES`.

Open the 3D Model Generator tool from the ModelVision toolbar button and for Boundary & Parameters select the **From existing file** option. Select the **Browse** button for this option and navigate to the `TUTE11` project folder and load the vector file called `NT_GEOLOGY.TAB`.

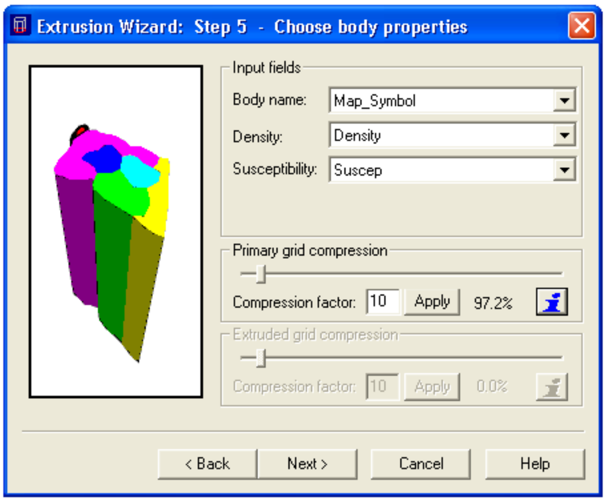
For the Top Surface specify an **Elevation grid** of the loaded DTM grid file and for the Bottom Surface specify a **Fixed height** of -1000m.



Step 10

Select the **Wizard** button to execute the Extrusion Wizard plug-in utility. A boundary is not required to be drawn in this exercise because the polygons contained within the loaded vector file provide the boundaries instead.

Navigate through **Steps 1-4** of the Extrusion Wizard and accepting the default parameters assigned by the 3D Model Generator for each of these steps until you reach **Step 5 – Choose body properties**. In this dialog the properties for the bodies to be created are supplied by the fields within the input vector file. The Wizard has automatically detected the Density and Susceptibility fields but the **Body Name** field needs to be manually assigned.

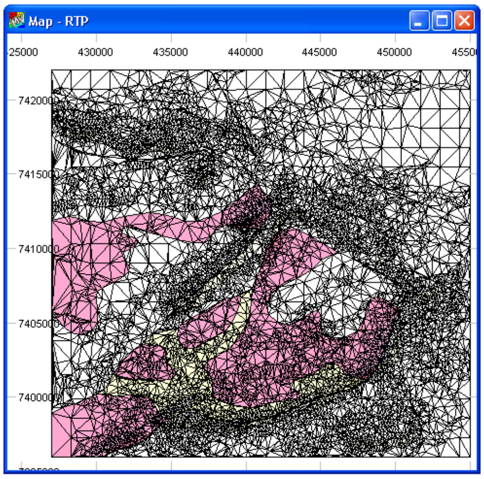


Select the down arrow of the drop-down list for **Body name** and select the field called *Map_Symbol* that is connected to the vector file.

Proceed to **Step 6 – Set appearance** by selecting the Next button and leave the default Colour option of **From input**. Click the **Next** button to proceed to **Step 7 – Choose output file** and check that the file destination is for the project folder and that the name of the output TKM file is called `NT_GEOLOGY.TKM`.

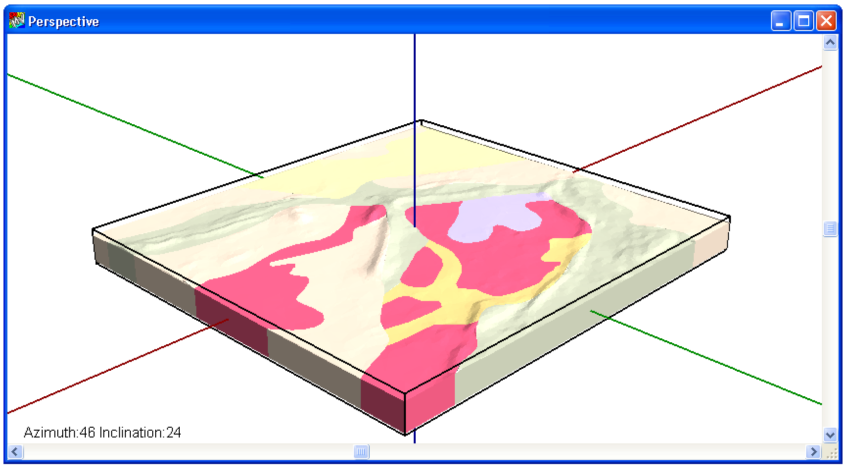
Select the **Finish** button to exist from the Extrusion Wizard and select **OK** on the message window that appears notifying you of the successful creation of the TKM file.

A 3D model of the entire geology map of the project area is added to the map window with colour modulations matching that of the input rock units.



Step 11

Select the **View>Perspective** menu option to display the model in 3D.



Step 12

Finally view a list of the bodies for this model by selecting the **Body Table** button on the toolbar. The body name for each polygon extruded will be displayed in the Label column and the Susceptibility, Density and Depth values will vary according to the input data from the vector file and the DTM grid file.

Body Parameters

	Label	Type	G	Colour	Suscept	Density	Depth	Active	Lock	Vis
1	Sandstone	polyhedron			0.012	2.60	119.7	☑	☐	☑
2	Granite	polyhedron			0.025	2.65	62.8	☑	☐	☑
3	Mafic Intrusives	polyhedron			0.031	2.70	76.2	☑	☐	☑
4	Metamorphics	polyhedron			0.028	2.68	49.7	☑	☐	☑
5	Granite:1	polyhedron			0.025	2.65	-58.3	☑	☐	☑
6	Quartzite	polyhedron			0.013	2.62	-40.8	☑	☐	☑
7	Granite:2	polyhedron			0.025	2.65	24.5	☑	☐	☑
8	Gneiss	polyhedron			0.0014	2.61	64.9	☑	☐	☑
9	Metamorphics:1	polyhedron			0.028	2.68	-55.3	☑	☐	☑

Tutorial 12 – Joint Inversion

Aim

The aim of this tutorial is to demonstrate the use of joint inversion on conventional cross-wing total field gradiometer survey data. The technique is easily extended to other data such as full tensor gravity gradiometry and full tensor squid magnetometers.

Intended User Level

Experienced users. Advanced options for inversion.

Topics Covered

- Data import
- Data visualisation
- Calculation of x,y horizontal gradients
- Modelling setup
- Select data for joint inversion
- Build a starting dyke model
- Run a 3D Joint Inversion on a single line data segment

Background Information

This survey was flown with a helicopter horizontal gradient system with a ground clearance of 20m to 25m. The area has shallow dipping dykes that can impact upon mining operations at depth and we need to estimate the dip of the dyke with as much precision as possible from a single line of data. The dykes are believed to be normally magnetized.

The survey data contains levelled cross-wing (lateral) gradient (dTdc) and total field (TMI) data, radar altimeter(RadAlt), absolute flying elevation (Zgps) and ground elevation (DTM).

Tutorial Data

The flight line data is stored in the file `TUTE12 GRADIENT DATA.GDB` and an ER Mapper format grid of the total magnetic intensity is called `TUTE 12 TMI.ERS`.

Tutorial Steps

The following steps should be undertaken for this tutorial:

Step 1: Data import

Initially, a project is required to define the various properties of the data. ModelVision uses projects to manage various analysis exercises and a `MVPROJ.INI` records your entries for later use. Binary session files (.SES) can also be used to store data, models and displays if required.

Select the **File>New>Project** option and enter the information as required. For this area select local datum

Note that the entries for the Earth’s magnetic field are set to be:

Field Intensity :	29132
Inclination:	-62 degrees
Declination:	-15 degrees

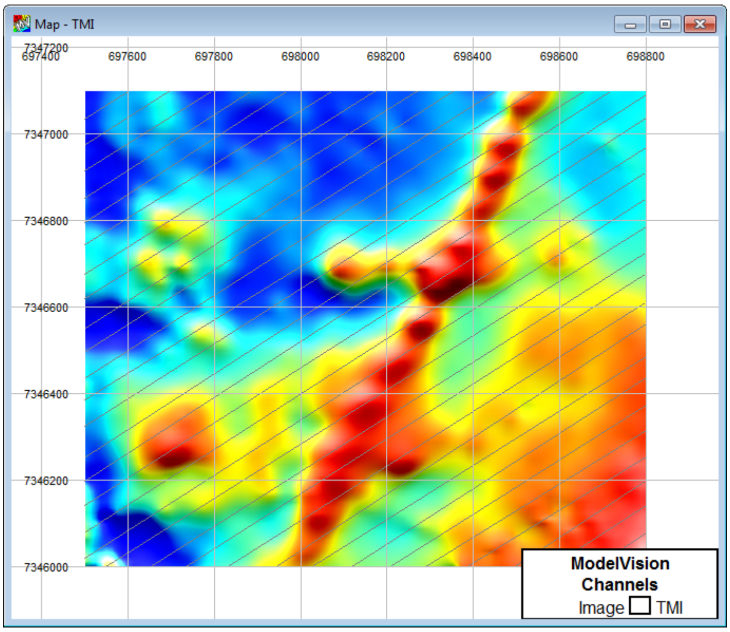
Load the ER Mapper format grid (`TUTE 12 TMI.ERS`) using the **File>Import>Grid>ER Mapper** option (or the `CROSS-WING GRADIOMETER TUTE 12 START.SES` session file if operating ModelVision in Demonstration mode).

Load the flight line data from the Geosoft database file (`TUTE12 GRADIENT DATA.GDB`) using the **File>Import>Profiles>Geosoft(.GDB)** option.

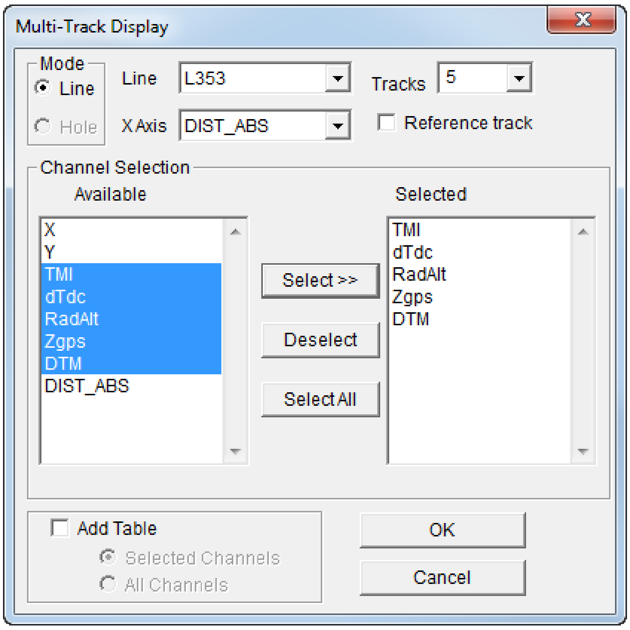
Step 2: Data Visualisation

Display the TMI grid and flight lines using the **View>Map** option.

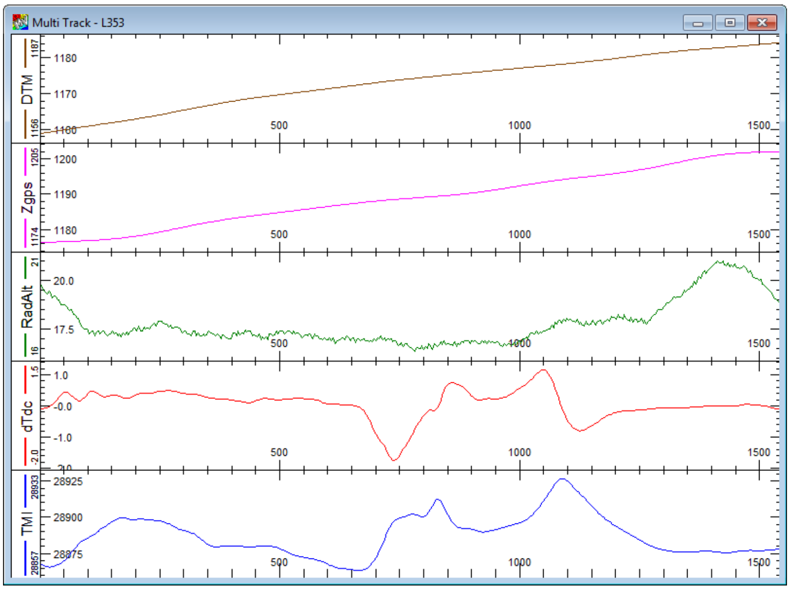
The dyke that we will model is associated with the dominant linear magnetic anomaly trending in a north-north-east sense.



To preview the line data using the multi-track display, use the **View>Multi-Track** option, select line L353 and the TMI, dTdc (cross-wing gradient), RadAlt (radar altimeter), Zgps (magnetic sensor elevation) and DTM (ground elevation below sensor position). Make sure that you allow for 5 tracks so that each data channel has a separate track in the display.



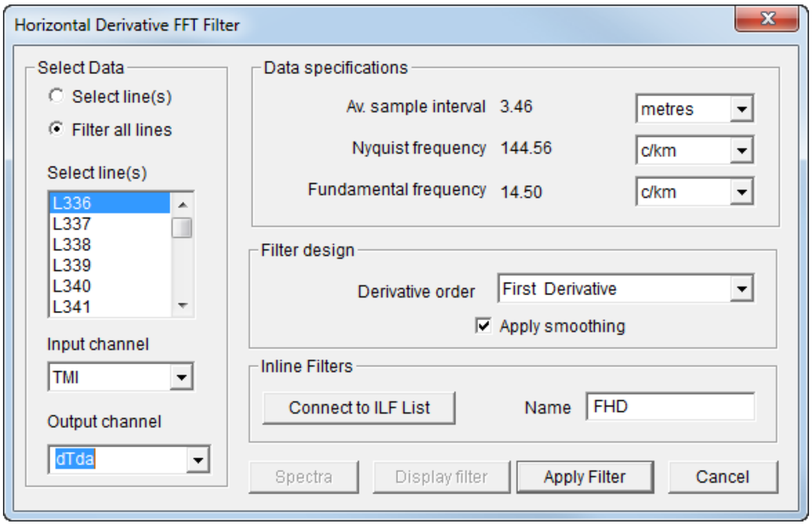
The multi-track display will appear as shown below and note the relationship between the total magnetic intensity channel TMI and the cross-wing gradient (dTdc).



Step 3: Calculation of dT/dx, dT/dy Horizontal Gradients

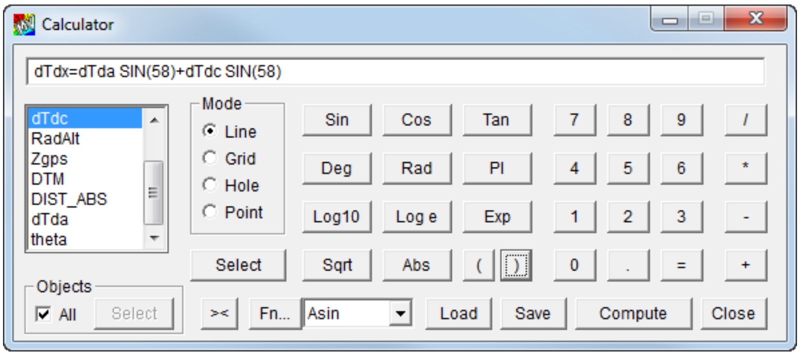
Before we can use the data for inversion, we need to convert the gradient data into east and west gradients (dT/dx and dT/dy), where T is the total magnetic intensity. Calculation of these parameters requires a vector transformation of the cross wing gradient (dTdc) and the along line horizontal gradient (dTda).

The along line gradient is calculated by computing the first horizontal derivative along the line using the **Filters>FFT filters>Horizontal derivative**.

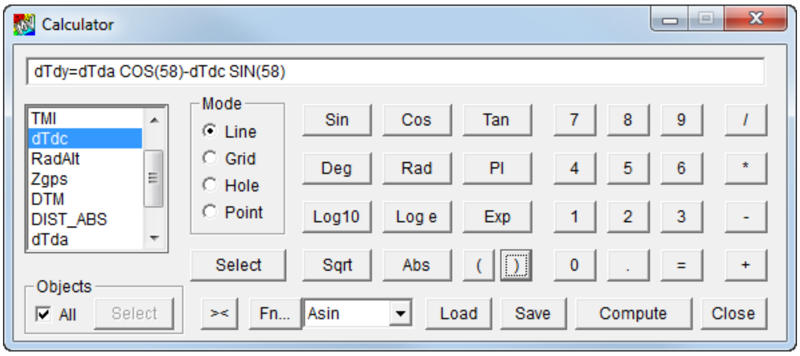


You convert the cross-line and along line gradients to east and west gradients through a simple vector transformation.

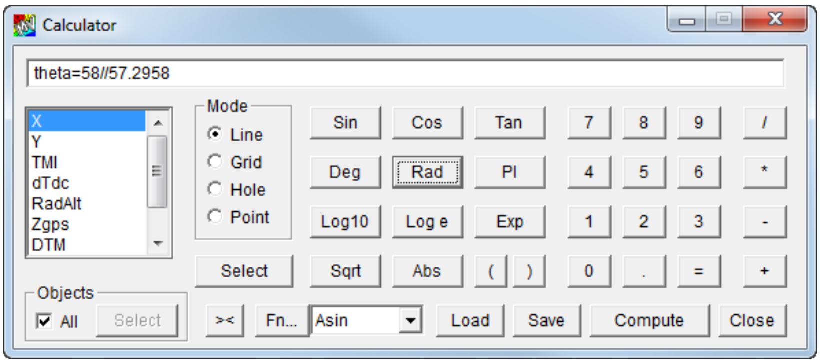
dT/dx = dT/da Sin(Θ) + dT/dc Sin(Θ)



dT/dy = dT/da Cos(Θ) – dT/dc Sin(Θ)

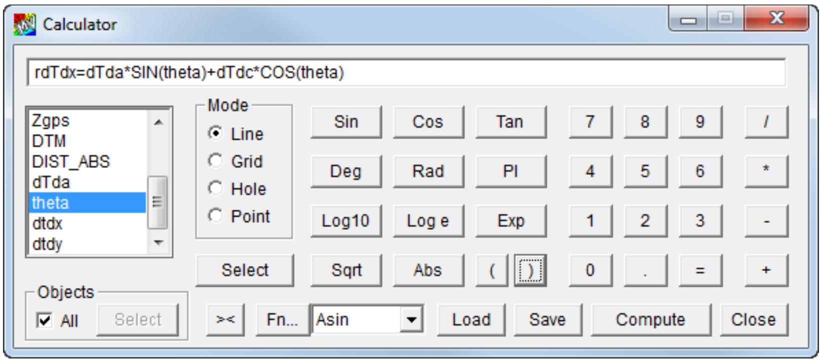


In this tutorial, dT/da equates to channel dTda and dT/dc to dTdc and "Θ" is the azimuth or the survey lines relative to north (58 degrees). Create a new channel with the line calculator by assigning theta = 58 / 57.2958 (radians).

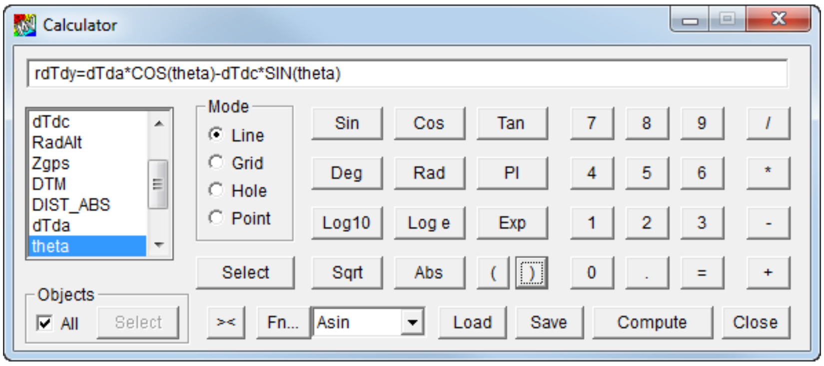


Now calculate the east and west horizontal gradients using the formulae below.

rdTdx=dTda*sin(theta)+dTdc*cos(theta)



rdTdy=dTda*cos(theta)-dTdc*sin(theta)



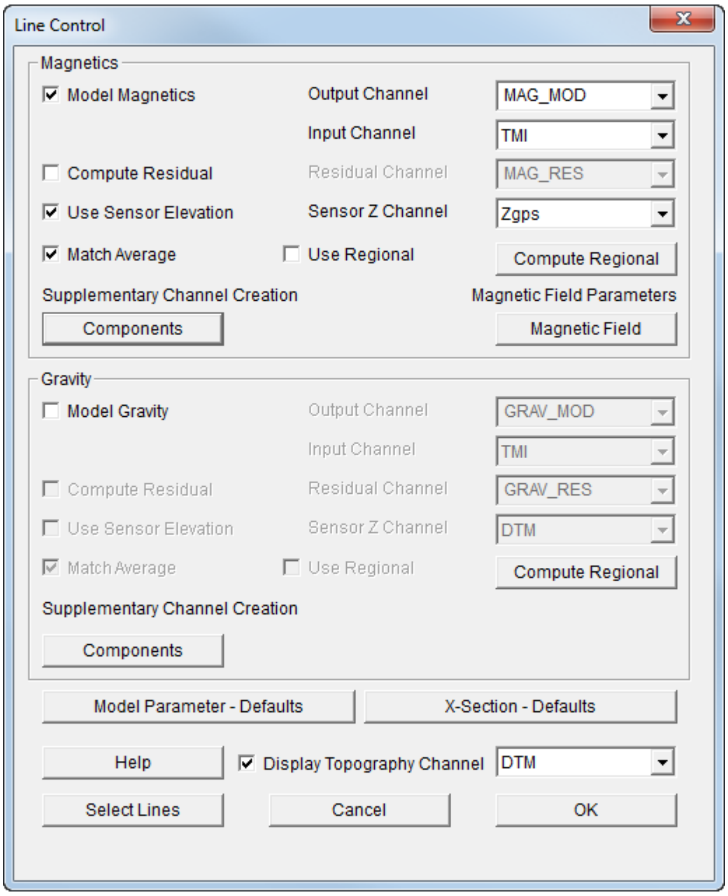
The “r” prefix is used to distinguish the measured data from the model channel names which are automatically assigned to dTdx and dTdy.

Warning

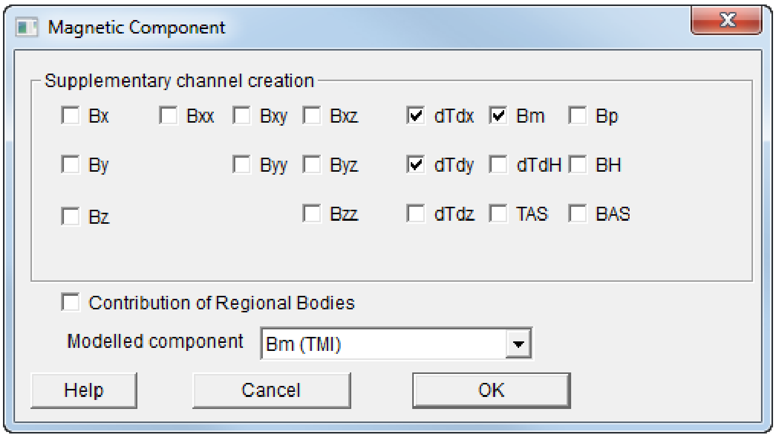
The calculation of the horizontal in line gradient dT/da using the FFT filter assume that the lines are all ordered in the same direction. In this case, the data is ordered from west to east. ModelVision automatically orients the line data sequence to the same direction at the time of loading, even when the flight lines are stored in the Geosoft database with alternating directions. Do not attempt to use the FFT horizontal gradient filter unless it operates in the same geographic direction for each line. The dT/dc data supplied by the contractor had already been aligned in the same sense. If you are uncertain about the correct orientation, then it will be obvious that the model profiles that you create are different and guide you in selecting the correct process.

Step 4: Modelling Setup

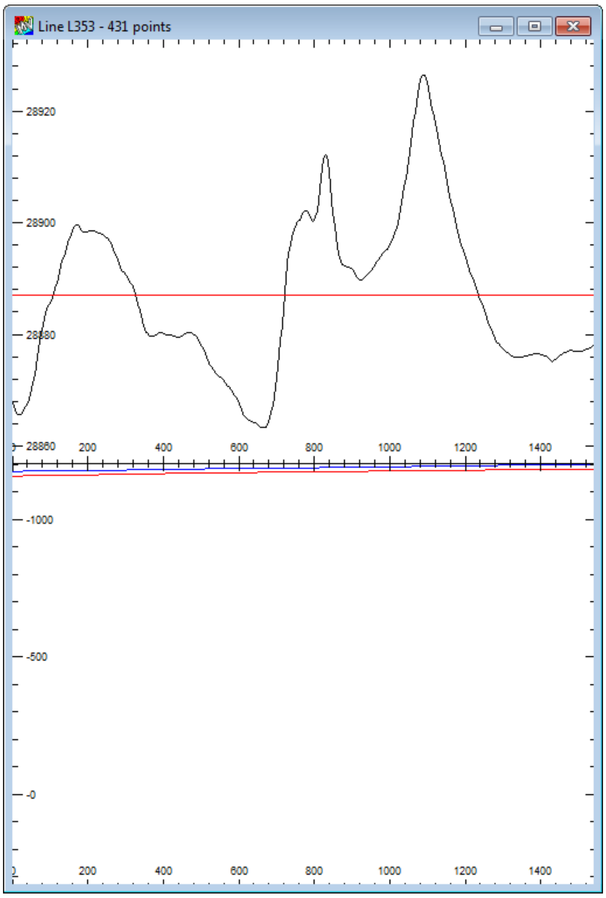
In the **Model>Line control** dialog select model magnetics, select the Input Channel as “TMI”. Select the **Components** button and activate the dTdx, dTdy and Bm supplementary channels and select Bm as the modelled component. Use Zgps as the sensor elevation channel and ensure the "DTM" channel is specified for the "Display Topography Channel" option.



Note that the TMI channel Bm will be used for display only and not inversion.

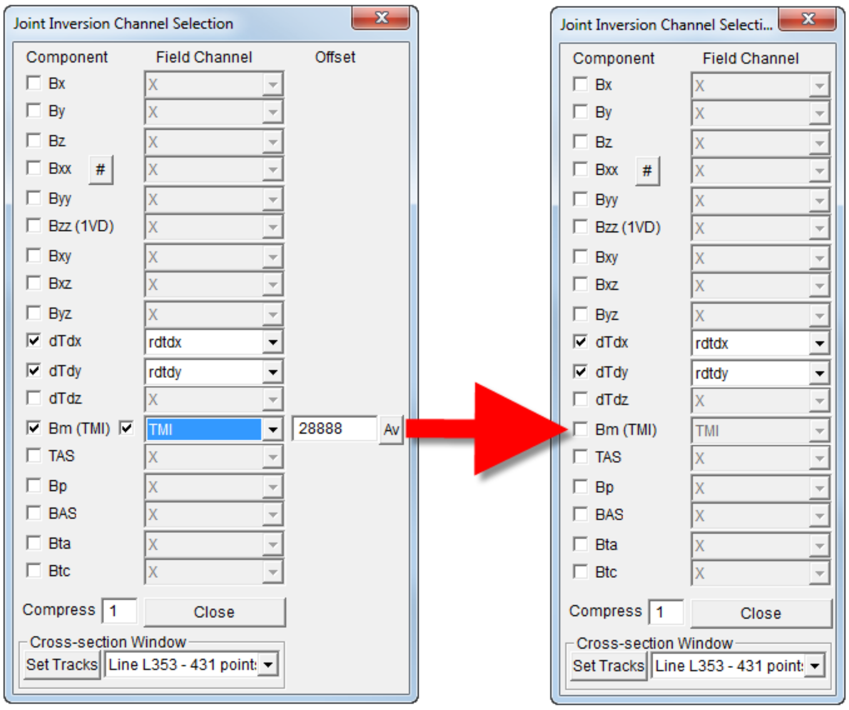


Open a cross-section window for line L353 using the **View>X-section** menu option which will open a standard view with the TMI data above the cross-section. Change the aspect ratio by making the frame narrower as shown below and this will accommodate space for the two horizontal gradient components dTdx and dTdy.

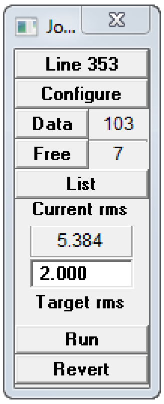


Make sure that you have the cross-section window selected.

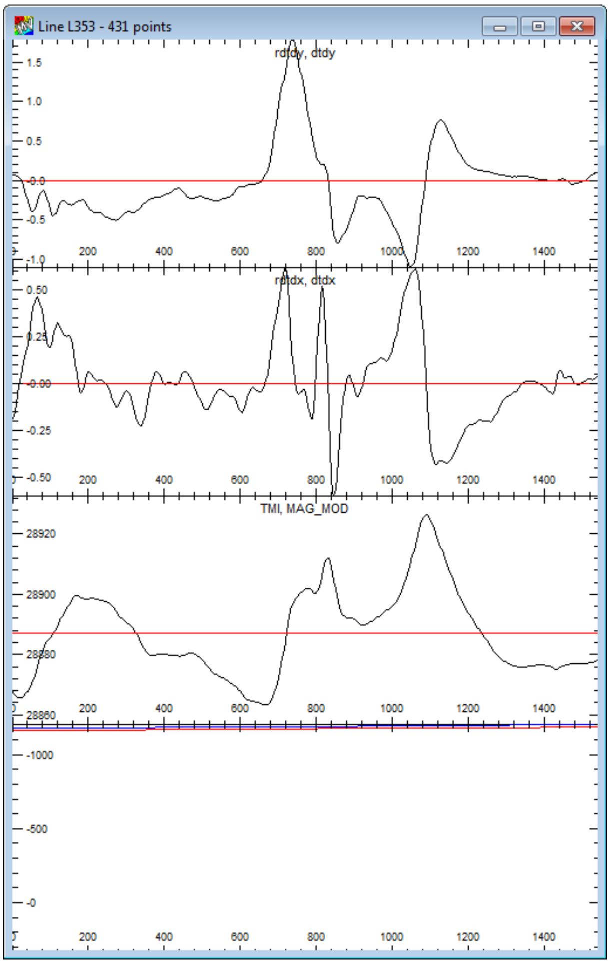
From the **Tools** menu, select the **Joint Inversion-Magnetic** option and a tall thin dialog will appear, showing all available model components. Select dTdx and dTdy, and select the tick box located to the right of the Bm (TMI) component, which will display the DC Offset option. Select the TMI channel from the channel list and enter the value 28888.0. This is for display purposes only and will not be inverted, therefore once the offset is specified deselect the tick box located to the left of the Bm (TMI) component.



To display the inversion channels in the section view, select the Set Tracks button after Line 353 is selected. This will automatically add two auxiliary tracks and the inversion channels above the TMI track. Both the recorded data and model channels will be added to each track. Since no modelling has yet been performed, the model channels will be set to the background value. In the case of the gradient channels, this is zero and 28888 for the TMI channel. Press **Close** and the Joint Inversion Channel Selection dialog will close and the Joint Inversion toolbar will open.

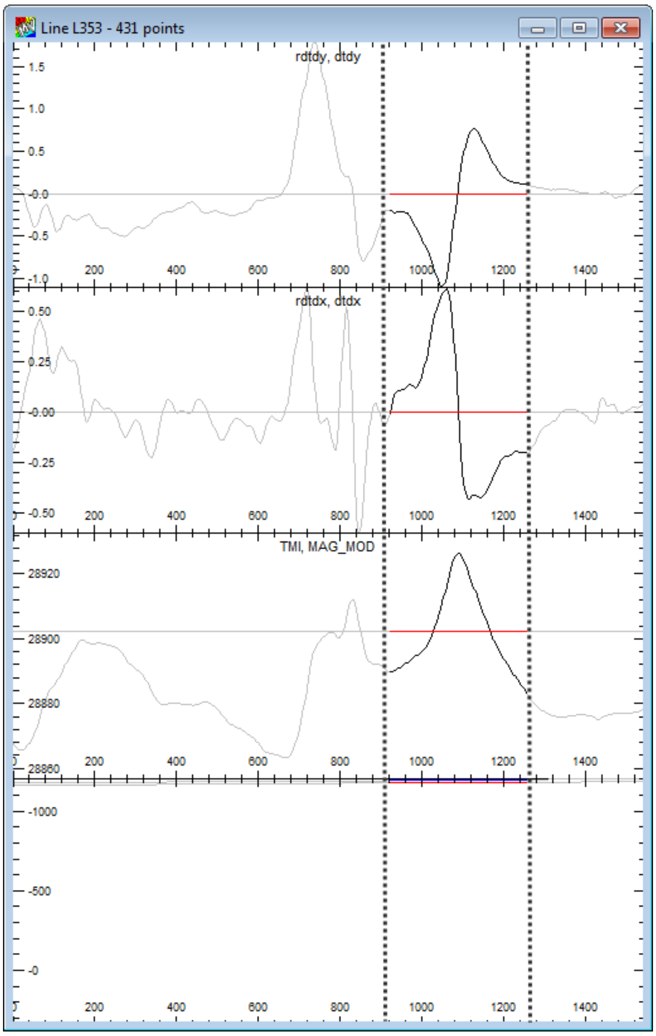


Step 5: Select Data for Joint Inversion



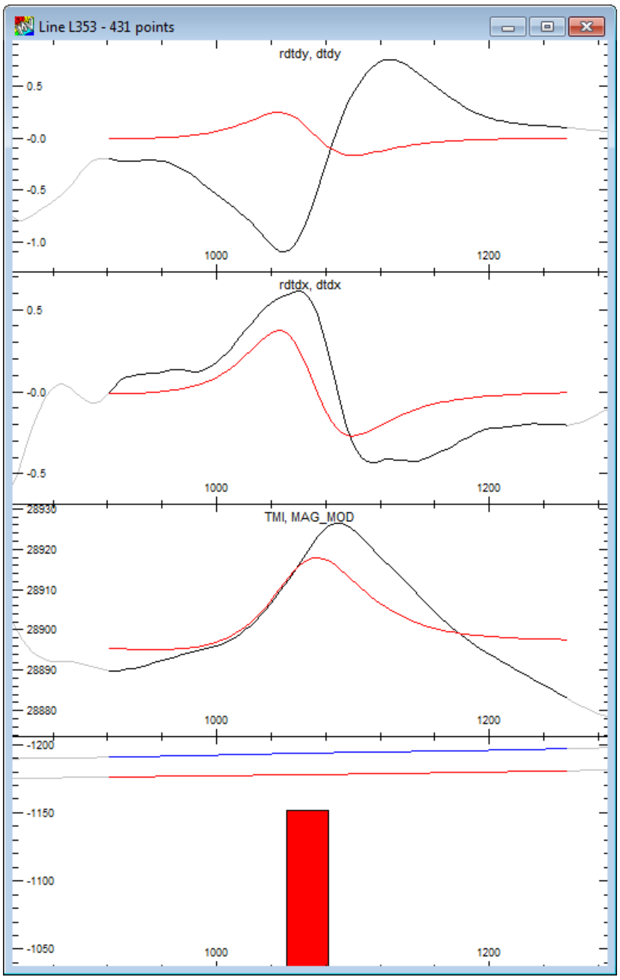
In the cross-section view zoom into the area between the dashed lines which contains the magnetic anomaly for the dipping dyke. Select the **Active Points** toolbar button and drag out a section of the profile that selects the clean segment of the gradient anomaly.

See the example below for setting up your own view.

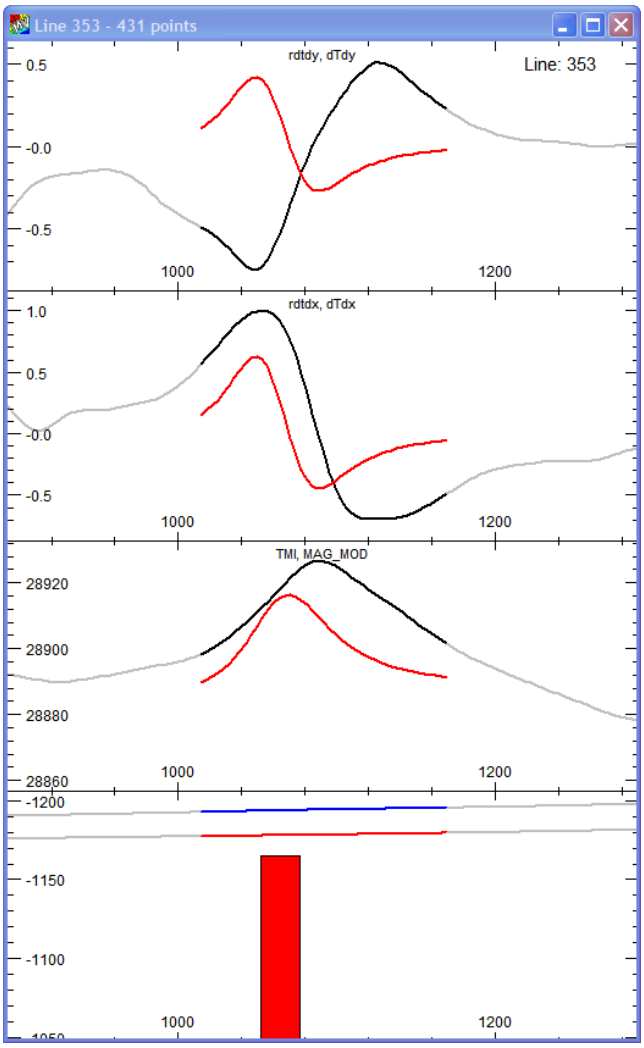


Step 6 Build a Starting Dyke Model

Set the vertical aspect ratio for the section to 1.0 so that you can see the flight path and ground surface clearly separated. Insert a tabular body into the section as shown below with a depth extent of 500 m and thickness of 25 m. The map view clearly shows that the starting model azimuth is grossly different from the true direction of the dyke.

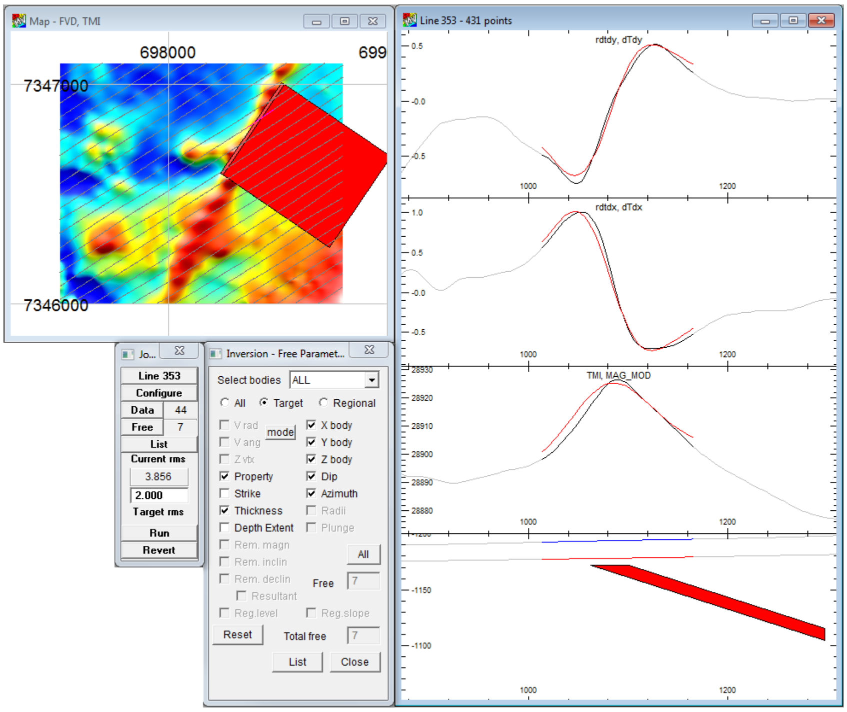


The initial model is fitted approximately using manual procedures where you can see that there is a big mismatch in the dTdy channel. It is this mismatch that contains the essential information that allows joint inversion to detect the rotation of the dyke.



Step 7: Run a 3D Joint Inversion on a Single Data Segment

Open the Free Parameter dialog from the Inversion toolbar and allow X body, Y body, Z body, Dip, Azimuth, Property and Thickness to float. Note that you cannot set bounds on parameters in Joint Inversion, they are either fixed or unbounded. Run the inversion until you are happy that no further convergence is possible and you should end up with a result that looks similar to that shown below.



Note that there is a slight phase shift in the peak of the TMI model match. In part this may be due to a slight regional tilt or an uncorrected lag in the survey processing.

If for some reason your results don't match those shown here, then open the session file `CROSS-WING GRADIOMETER TUTE 12 FINISH.SES` to check correct data channels and settings.

Tutorial 13 – Magnetic Anisotropy

Aim

This tutorial is designed to show you how to use magnetic anisotropy measurements in a forward modelling exercise and learn how it can impact on the interpretation of dip and remanence.

We note that magnetic anisotropy can only be used in forward modelling and is not available as an inversion parameter.

Intended User Level

Experienced users. Advanced options for modelling and inversion.

Topics Covered

- Computing and initial model with an effective magnetic susceptibility.
- Inverting on data where anisotropy is present.
- Entering anisotropic magnetic susceptibility measurements.
- Visualising the maximum, intermediate and minimum susceptibility axes.
- Computing the forward model with magnetic anisotropy.
- Inversion on resultant magnetization.
- Equivalence of magnetic anisotropy and magnetic remanence.

Background Information

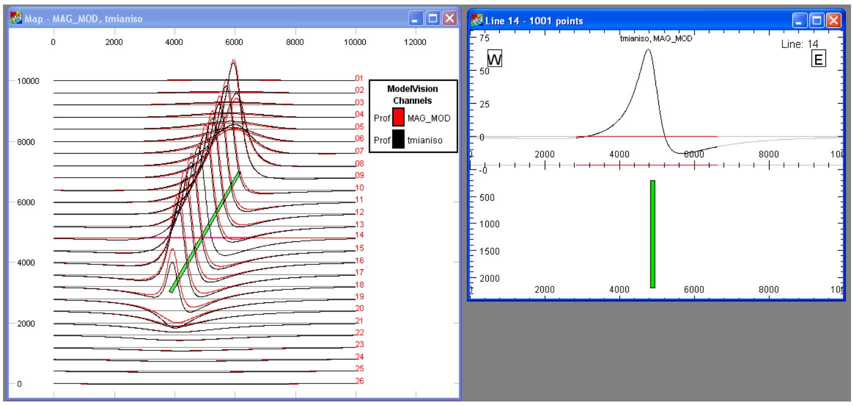
The data for this exercise is created from a synthetic model that is designed to simulate a linear magnetic horizon similar to a banded iron formation. The anisotropy measurements used in the tutorial are taken from laboratory measurements of a banded iron formation, but they are not aligned with the model tabular body. The following table has an example taken from a laboratory report that shows the declination, inclination and magnetic susceptibilities for the major, intermediate and minimum susceptibility axes.

Sample #	Depth (m)	Ellipsoid Axes	Dec	Inc	k x 10^{-6}
		Max	256	19	35146
PS631	121.8	Int	148	40	33954
		Min	5	43	10649

Be careful in drawing any general conclusions with this example as the high amplitude major and intermediate axes would generally be aligned with the plane of the formation sheet. That is not true for the model used in this example.

Tutorial Data

You will use the session file `TUTE 13 ANISOTROPY.SES` for learning about magnetic anisotropy. Load the session file and you will be presented with a map view of the data and model responses along with a central cross section that you will use for the modelling process.



Starting session for the anisotropy tutorial.

The original anisotropic magnetic model values are shown in black and the model response in red.

Note that only the central Line 14 is active during the process and this is set to zero for the initial run. The surrounding lines have been pre-computed for the starting model where the magnetic susceptibility has been assigned to the equivalent, non-anisotropic value (effective susceptibility).

Tutorial Steps

Step 1: Enter the Effective Magnetic Susceptibility

The effective magnetic susceptibility (keff) that would be computed from the following relationship:

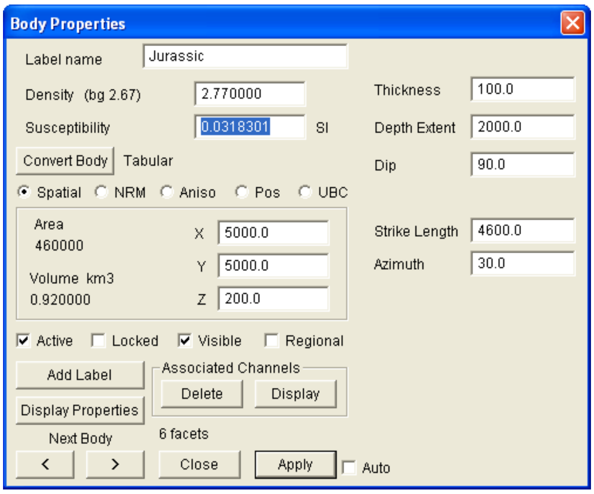
keff = |**K.F**| / |**F**|

where,

K is the anisotropic magnetic susceptibility tensor

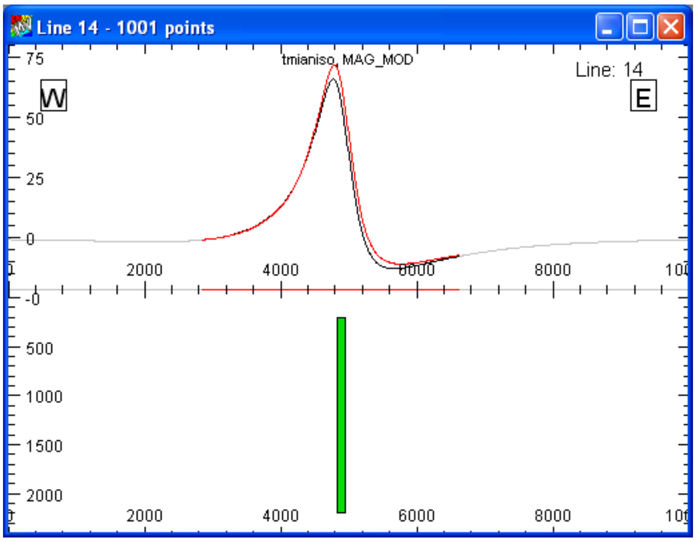
F is the geomagnetic field vector.

Enter the effective susceptibility value of 0.0318301 into the susceptibility field.



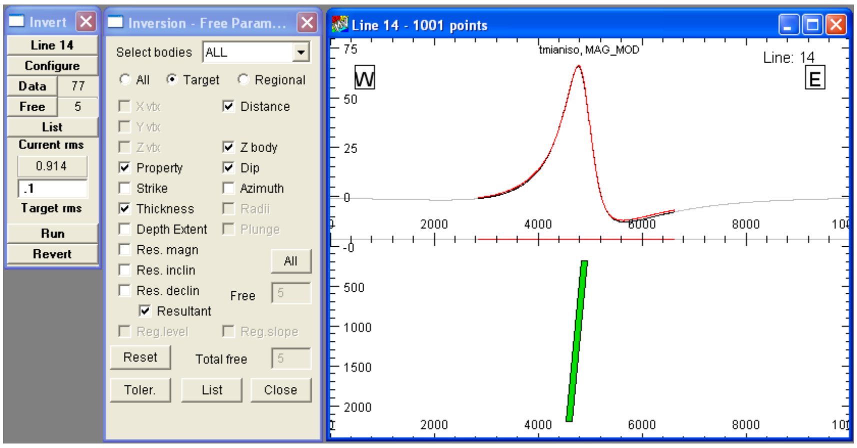
Enter the effective magnetic susceptibility. [image An-2.png]

Now compute the forward model response and you should see the response as shown below. Note that anisotropy mode has not been turned on at this time.



Model response(red) for the effective susceptibility curve and anisotropic total magnetic field response (black) computed using the values from the laboratory.

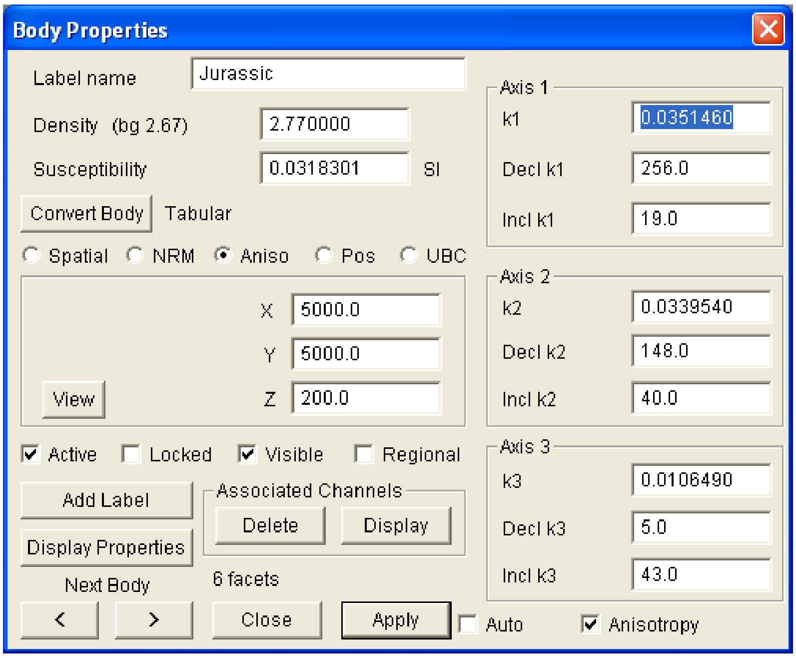
There is a difference between the two curves that cannot be explained by the same geometry. You can use inversion to try and match the shape by varying the position (distance), depth (Z), dip and property, but you will not get an exact match as shown below.



Inversion results for show a close but imperfect match

A dip of approximately 8 degrees is required to improve the curve match.

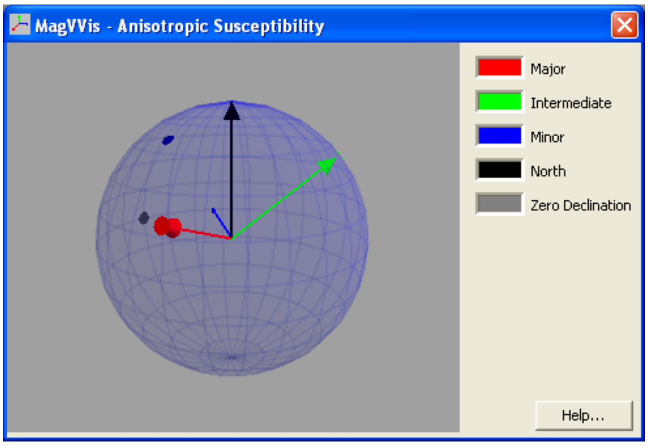
Now it is time to activate the anisotropy computation by selecting the **Aniso** radio button, activate anisotropy with the **Anisotropy** check box at the bottom of the anisotropy panel and enter the values from the laboratory table as shown in the dialog below. Note that the three susceptibility axes are orthogonal and if you make a mistake, the program will flag the problem with a pop-up dialog and suggest which axis has a problem. This is a handy feature as it is almost impossible to tell from the numbers that you type that you have made a mistake.



Enter the magnetic anisotropy tensor values as shown for the major (Axis 1), intermediate (Axis 2) and minor (Axis 3) axes after checking the ***Anisotropy*** *checkbox.*

Note also that the magnetic anisotropy for this laboratory measurement is very high at 3.3, yet the difference between the models with and without anisotropy is relatively small. The magnitude of the difference will vary with the orientation of the anisotropy axes.

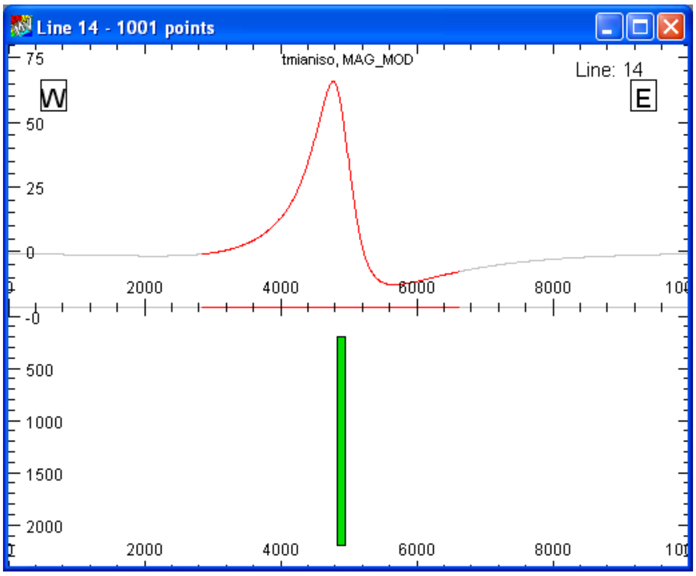
Use the View button to check the orientation and amplitude relationships of your anisotropy tensor. It can also be used to help diagnose problems with your data entry.



Use the ***View*** *button to display a 3D view of the anisotropy tensor.*

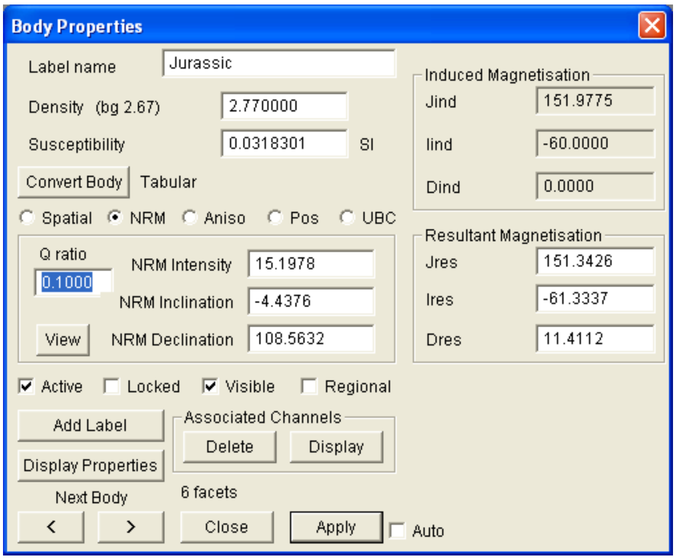
Click with the left mouse button on the 3D sphere and then click a second time to activate the 3D movement. You can rotate the sphere to any view point to obtain the perspective that you need. Note that the three axes are shown in red, green and blue vectors, north as the black vector and zero declination as a grey dot on the equator.

Now compute the forward model response after you close the property dialog and you should see a perfect profile match between the original data (black) and the newly compute anisotropy response in red.



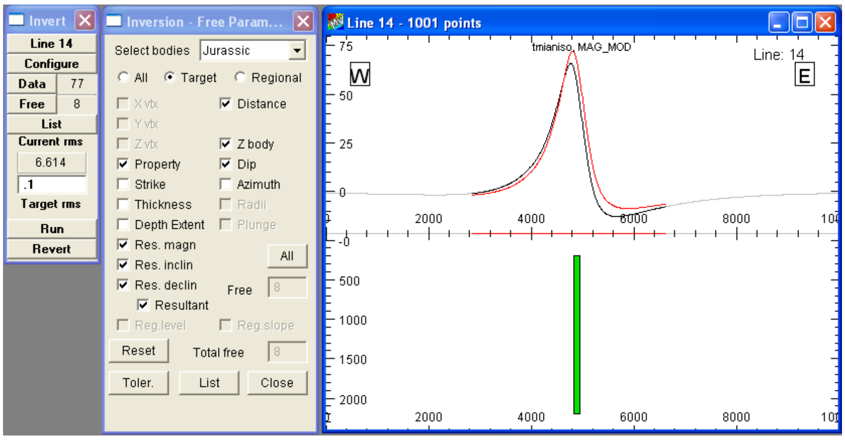
Model response(red) for the anisotropic magnetic susceptibility model curve and the original total magnetic field response (black) computed using the values from the laboratory.

Anisotropic magnetic susceptibility has a similar effect to magnetic remanence because it changes the resultant magnetization direction inside the target. Thus it is appropriate to look at this equivalent effect by using inversion to determine the resultant magnetization direction. First turn off the magnetic anisotropy calculation by selecting the **Aniso** radio button and then uncheck the **Anisotropy** check box at the bottom of the anisotropy panel. Select the **NRM** radio button and enter a value of 0.1 for the Koenigsberger ratio Q.



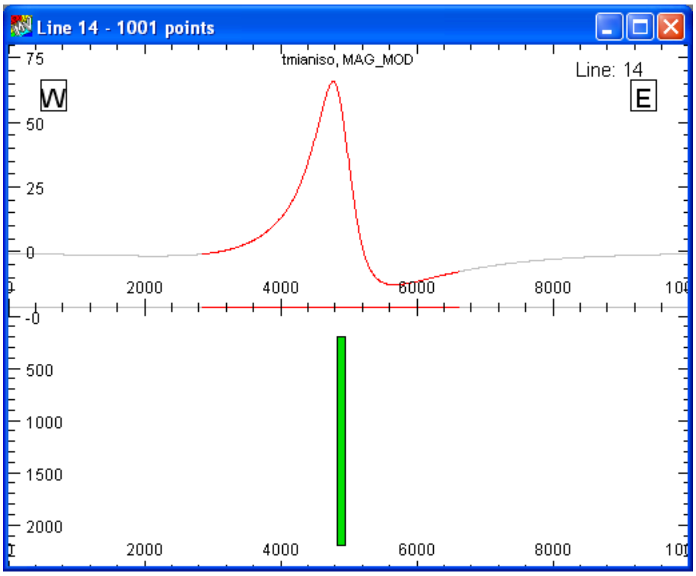
Set the Koenigsberger ratio Q to 0.1 prior to inversion.

Now open the inversion toolbar and set the **Target rms** to 0.1. Next select the **Free** button on the toolbar and check the **Distance**, **Z**, **Dip**, **Property**, **Res magn**, **Res. Inclin**, and **Res. declin** boxes:



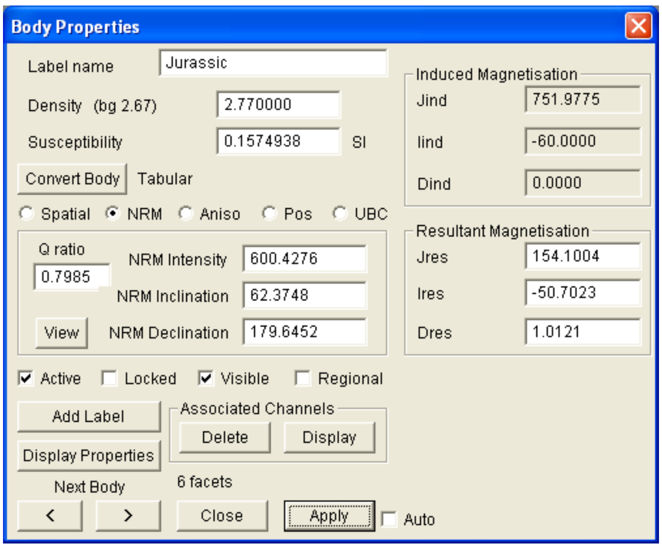
Inversion parameters prior to starting inversion.

After inversion you should end up with a result as shown below.



Inverted response (red) using resultant magnetization inversion.

Check the NRM property panel after the inversion and you will see that the resultant magnetization vector has an inclination of approximately -51 degrees and declination of approximately 1 degree. This represents a departure of approximately 10 degrees from the vector direction of the inducing magnetic field.



The resultant magnetization vector has an inclination that is approximately 11 degrees different to the induced field vector direction after inversion.

Tutorial 14 – Remanence from Magnetic Moments

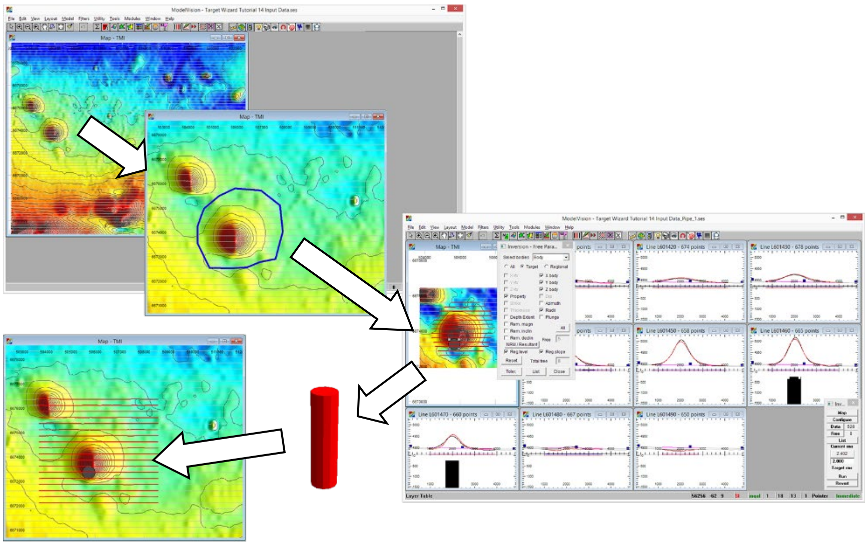
Refer to “Sample Synthetic Dataset” on p. 372 of the ModelVision User Guide.

Tutorial 15 – Target Wizard

Aim

This tutorial will teach you how to use the Target Wizard to quickly perform multi-line 3D modelling and inversion of individual anomalies or anomaly groups. The Target Wizard combines many separate processes into a single intelligent workflow that reduces the total time by up to 90 percent.

The figure below shows the basic steps of anomaly data selection, modelling and inversion of the data subset and restoration of the new body to the master session.



Intended User Level

This tool will be used by interpreters that have already learnt how to create models and run the inversion tools. The Target Wizard takes a commonly used workflow for 3D modelling and combines all the processes into a semi-automated procedure which allows the interpreter to focus on the geological problem.

Topics Covered

- Loading the session file
- Selecting the target anomaly
- Running the Target Wizard
- Creating a starting model
- Running inversion to produce a quality circular pipe solution
- Use inversion to determine the resultant magnetization vector
- Restoring the inverted body to the master session.

Background Information

The example session file has many magnetic anomalies that are suited to interpretation using steeply dipping pipe like bodies or tabular bodies. The geological objective is to estimate quickly the depth of cover, target shape and magnetic properties including resultant magnetization departure angles.

Tutorial Data

The data is a subset of a much larger survey near Bourke NSW. The flight line data includes sensor elevation, terrain data and total magnetic intensity. A session file called `TARGET WIZARD TUTORIAL 14 INPUT DATA.SES` is provided with the line data and magnetic grid.

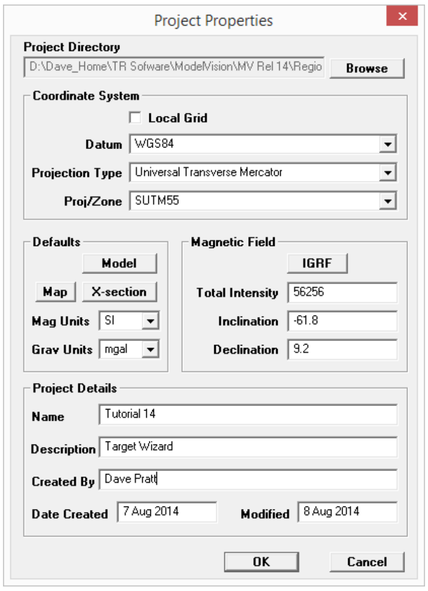
Tutorial Steps

The tutorial will take you through the process of loading the session file, selecting the anomaly of interest, running the Target Wizard, running forward modelling and inversion followed by restoration of the master session file with the new body appended.

Step 1 Loading the Session File

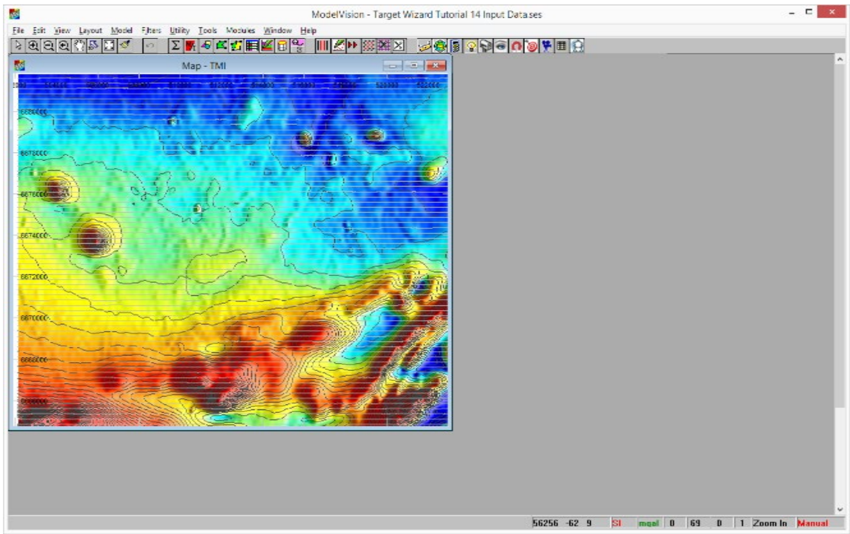
Load the project file from `..\TUTORIAL\TUTE 14`.

The settings should be similar to those shown below:

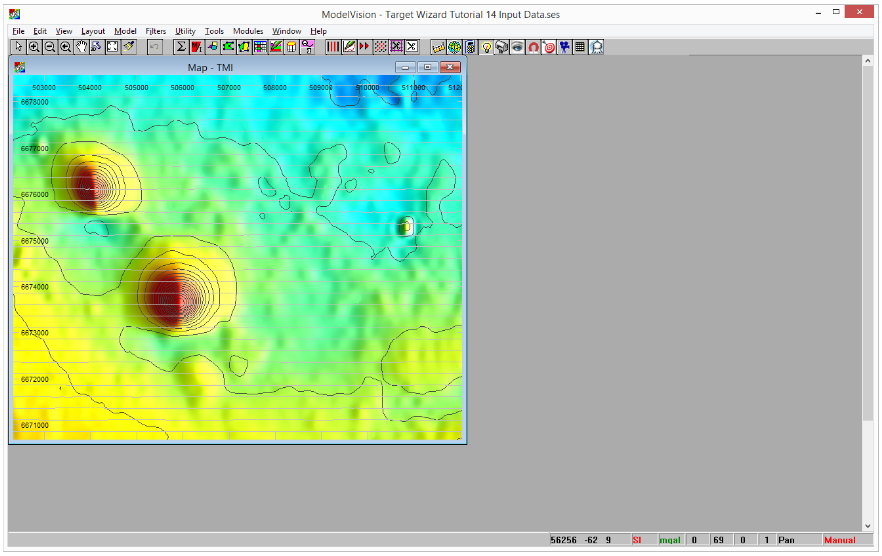


Load the session file called `TARGET WIZARD TUTORIAL 14 INPUT DATA.SES`.

You should see a map window containing an image of the magnetic data subset overlain by the flight lines.

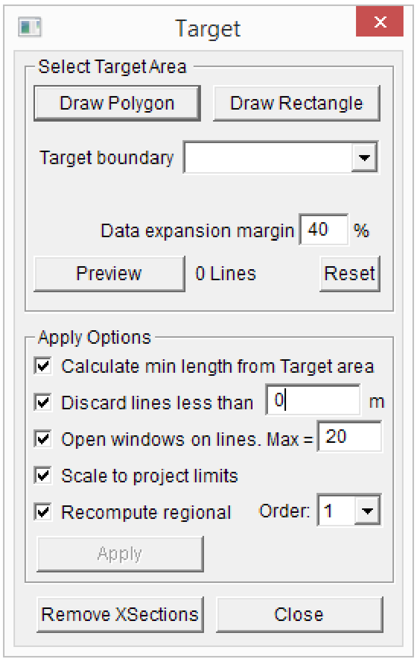


Now use the Zoom tool to select the area around the two circular magnetic anomalies.

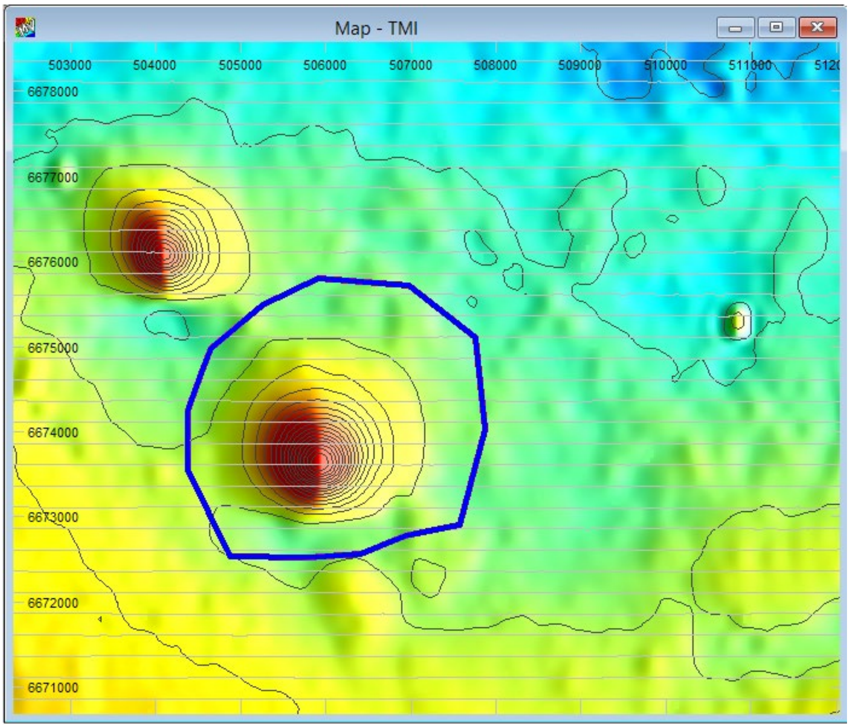


Step 2 – Running the Target Wizard

The **Target Wizard** can be accessed from the toolbar of ModelVision. When this button is selected the Target Wizard dialog appears.

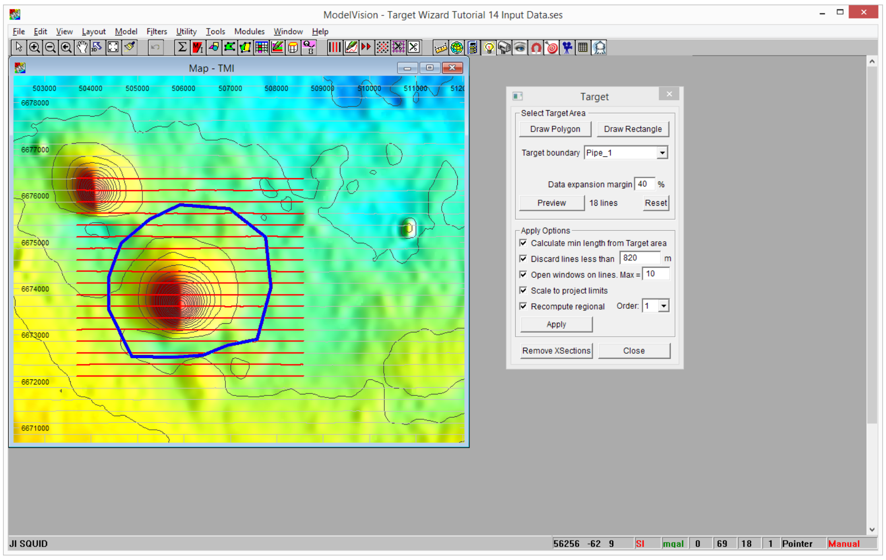


Now draw a polygon around the anomaly as shown below and give it the name Pipe_1. Note that there should be no spaces in the name.



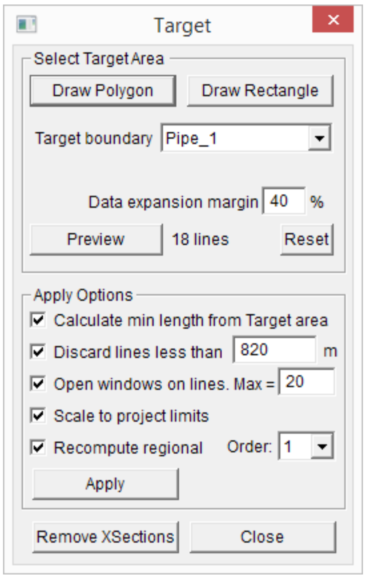
Your polygon is defining the area that will be used during inversion, so it is helpful to avoid the small anomalies on the southern side.

Select the **Preview** button which updates some dialog fields and the map area is highlighted with the data that will be used during the Target Wizard session.



This highlighted data will be isolated for calculation of an initial regional surface and the data points within the polygon will become active for modelling and inversion.

The dialog is updated with a minimum line length calculation and the name of the polygon that you entered. In this example, there are less than 20 lines available to open as cross sections, so leave the number at 20. You can delete unwanted sections from the display later.

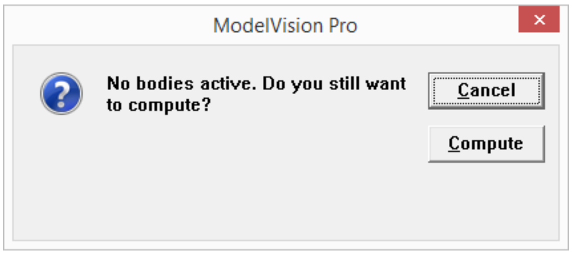


The Target Wizard uses an algorithm to optimise the line selection if you choose a number less than the available lines. For example, you may choose 10, but get 8. The algorithm tries to select every 2nd, 3rd or 4th line to maintain an even line spacing. Note that all the highlighted data is used for modelling and inversion even if it is not shown in a cross section.

A first order regional is a good choice for this anomaly and it is always better to use the lowest possible order you can to represent the local regional from interfering sources. If you have multiple overlapping anomalies, then you can use the Target Wizard to model them together.

Select the **Apply** button and the following will happen:

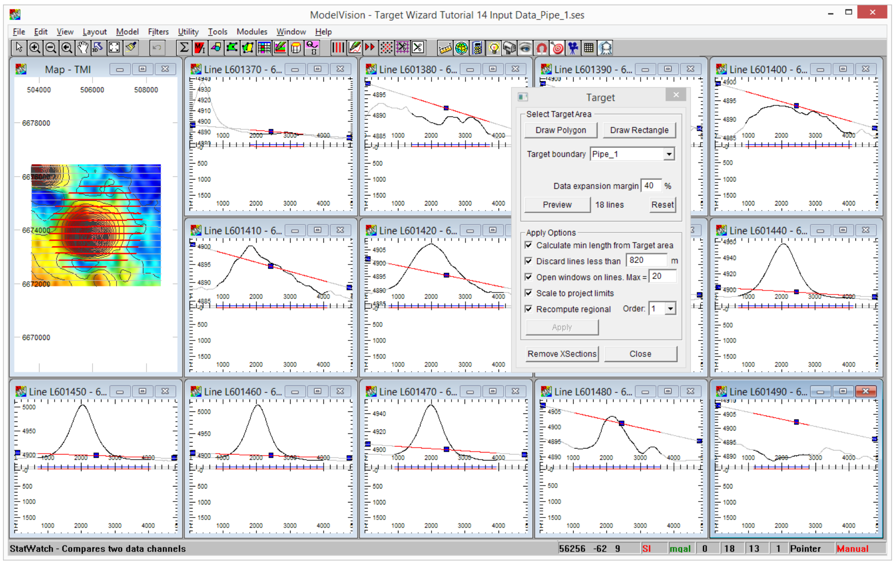
- The master session is saved.
- Crop the project to the bounding rectangle defined by the red lines.
- Deactivate and hide all existing bodies (none at this time).
- Open the new sub-session data and selected cross sections.
- Ask you to compute (the regional).



You need to select the **Compute** button to produce a regional for all the cross sections. The regional is calculated for the complete data subset rather than the data just within the polygon boundary.

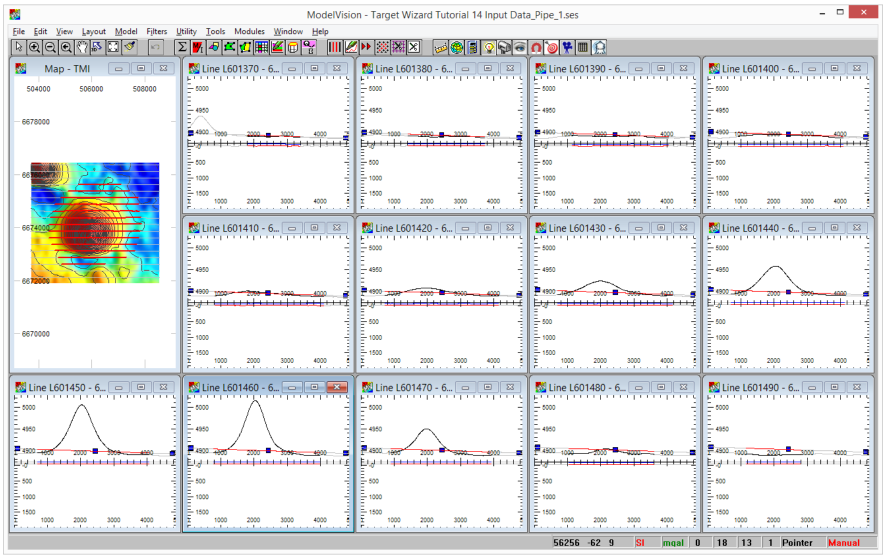
Step 3 – Running Modelling and Inversion

ModelVision opens the new sub-session with all the selected lines as cross sections and one extended map view. Note that the Target Wizard has applied the menu function **View>Tile windows (custom)** to the available windows. If your workspace looks different to this, then you may have different setting for the custom tiling. Check your settings in **View>Tiling options**.

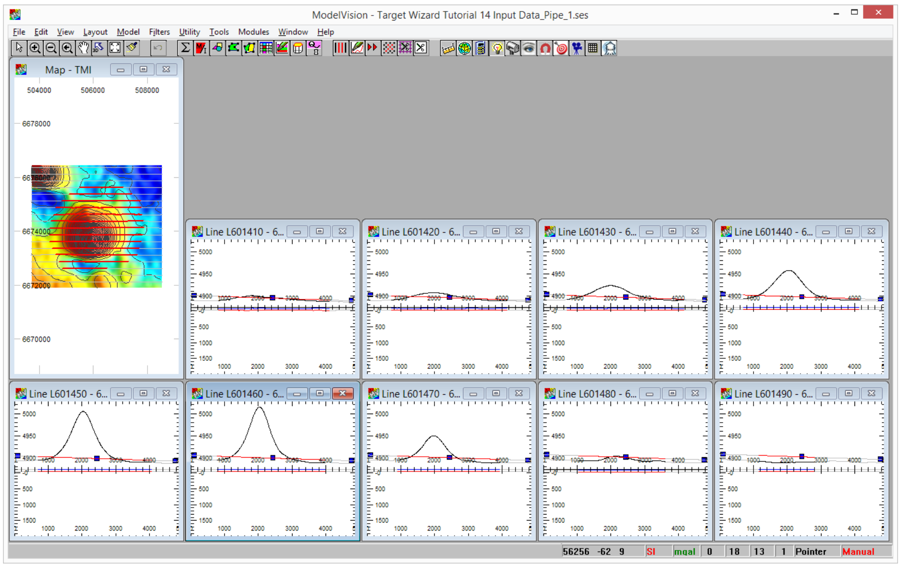


Now you can **Close** the Target Wizard and perform your modelling and inversion as you would normally within ModelVision.

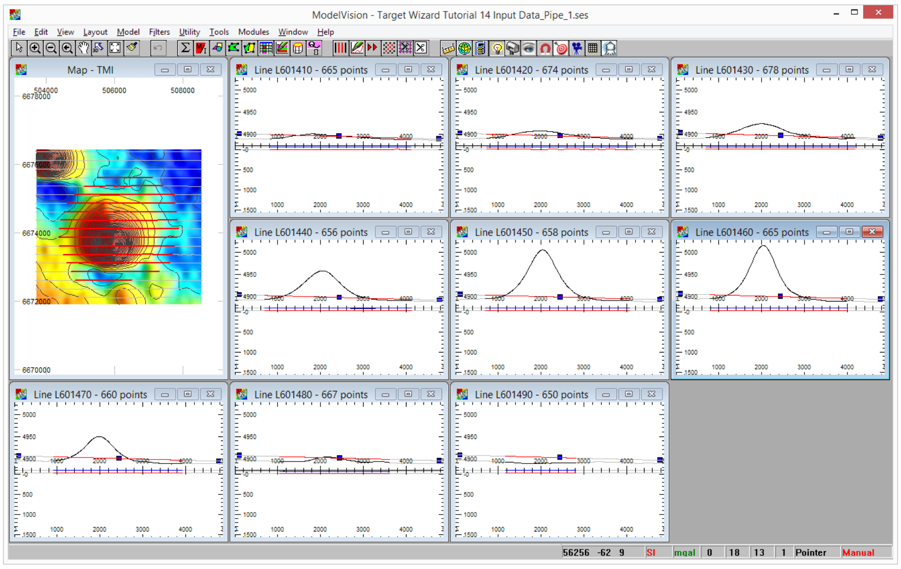
All cross sections have been scaled to suit the dynamic range of the individual section, but for Wizard modelling that is not ideal. It is better to have all sections scaled the same. Select the window Section L601460 and use the Right Mouse Click to bring up the properties menu. Select **Apply Scale to All** and all of the cross section windows will be redrawn to that scale.



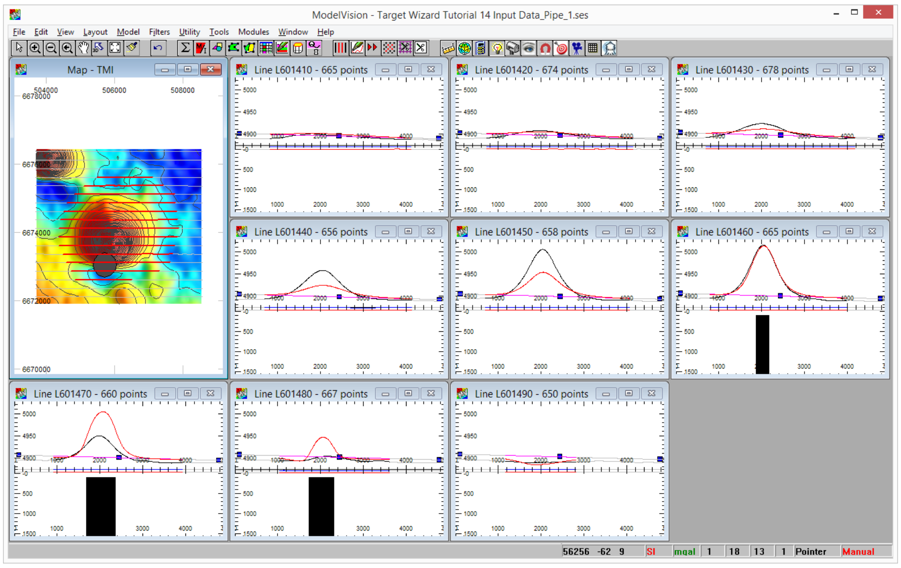
Delete the top four cross sections as there is little useful information in these lines.

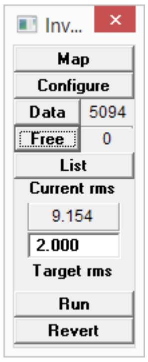


Then use the **View>Tile windows (custom) View** followed by rescaling of the sections using **Apply Scale to All** and all the sections will be redrawn.



You are now ready to start the modelling process. Select the map view and then insert a circular pipe between the high and low. Set the default depth to top at 100 metres, depth extent to 2000 metres and then compute the model. Your result will be similar to that shown below, but will depend on the radius you selected and x, y body position.

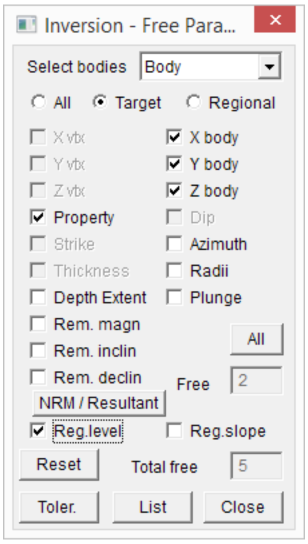




The next step is to run inversion on the model by selecting the menu option **Tools>Inversion – Magnetic**. Depending on the polygon you drew, you will have around 5,000 data points active for inversion. Too many for normal operations and you should only need 200 to 500 data points for a single anomaly target of this type.

Select the **Data** button and set the **Compression** to 10, which reduces the data to 1 in 10 data points.

Now select the **Free** button and open up the free parameter dialog. You can leave Free Parameter dialog open during the inversion process.

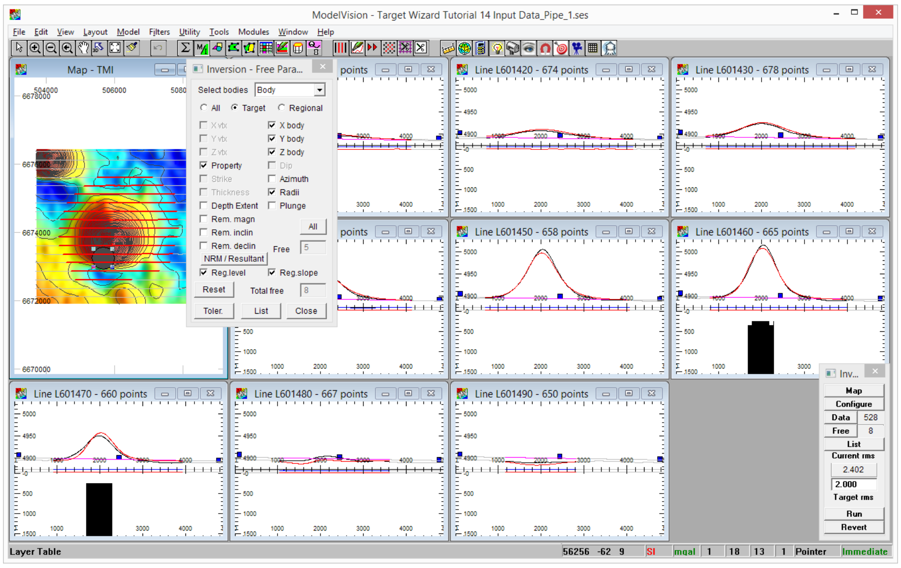


Select the body in the map view to make sure that the map window is active rather than an individual profile and the name of the body will appear in the Free Parameters dialog. Now set **X, Y, Z, Property** and **Reg. Level** free.

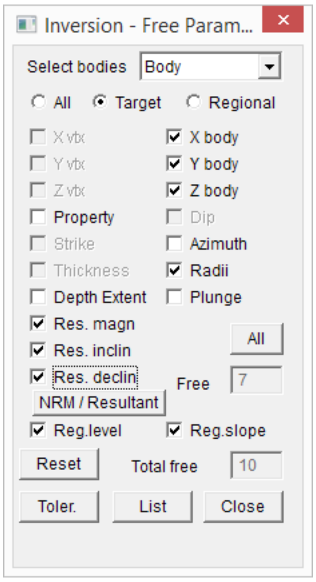
Don't set the regional slope free until later when you are approaching a good match. Run the inversion with these parameters and you should see the RMS reduce below 3%. You will see that the depth of the pipe has also increased.

Once you cannot get any further improvement, set the **Radii** to free then do another run with the **Reg. slope** set to free.

By this time you should have a reasonable match with an RMS between 2% and 3%.



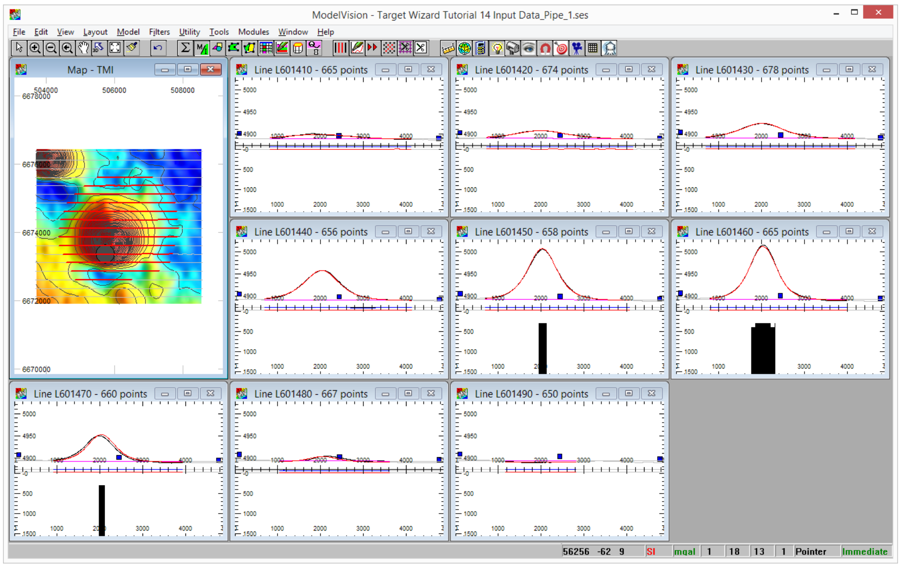
Step 4 – Inverting for Resultant Magnetization



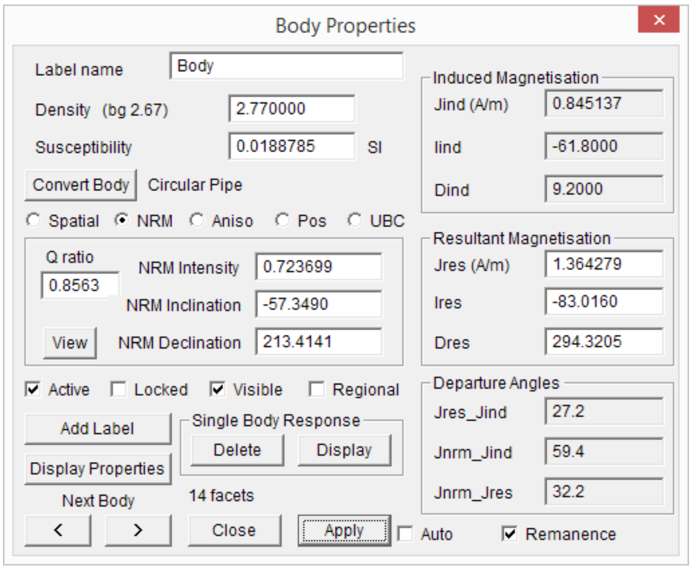
At this stage the match is excellent, but the background regional and Line 601470 show some residual differences that could be explained by magnetic remanence. To explore this possibility, toggle the **NRM/Resultant** button to **Res.** and check all three of the resultant magnetisation parameters. Make sure **Property** is turned off as this is incompatible with inversion for the resultant magnetisation. You can leave the **Reg.slope** free although there is always a possibility that it will interact with the magnetization vector inclination and declination. You can always control this using the regional handles if necessary.

You should also set your inversion RMS target to 1.0 as it is likely that the RMS will drop below the default setting of 2.0.

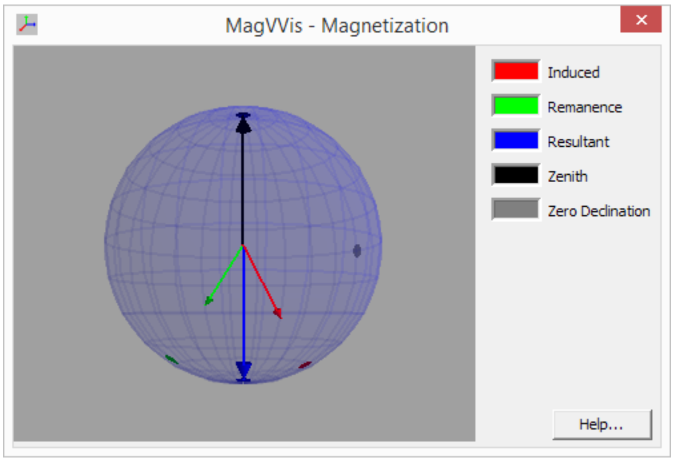
After you run the inversion for remanence, you should have all features of the anomaly and the regional well resolved as shown below.



Now open up the body properties dialog for the circular pipe and select the **NRM** tab. You should see the ARRA departure angle (**Jres-Jind**) around 27 degrees.



Now select the **View** button to obtain a 3D view of the Induced, Resultant and estimated Remanence vectors.

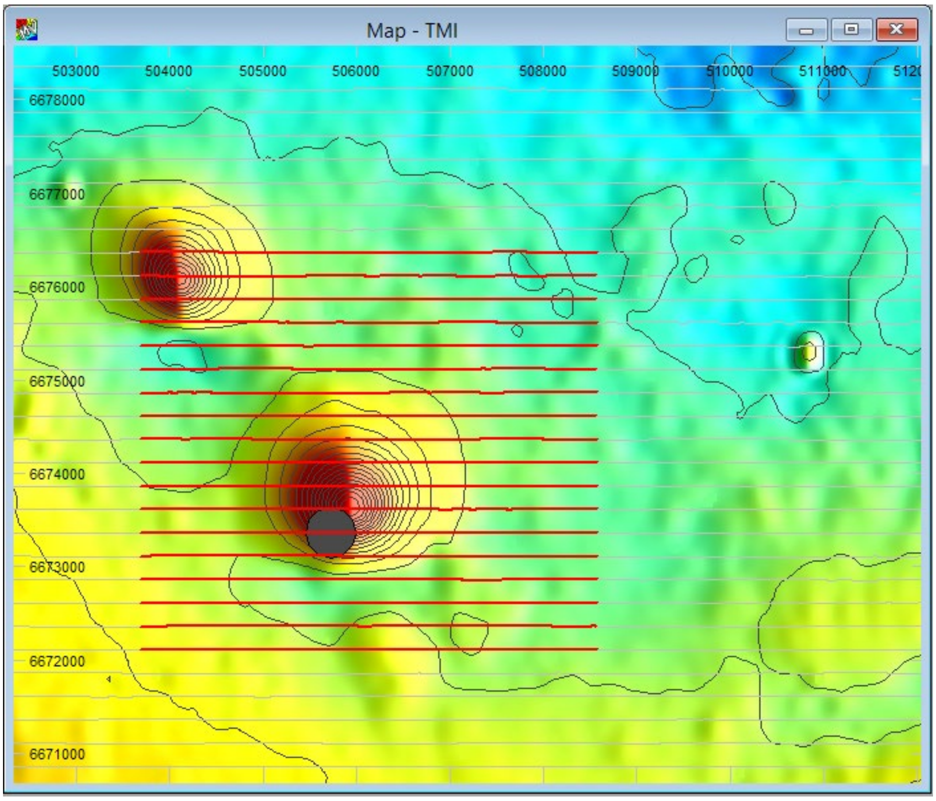


The additional step required to obtain the resultant magnetization vector is easy with the Target Wizard and is recommended as part of the routine procedure for interpreting limited strike length magnetic anomalies.

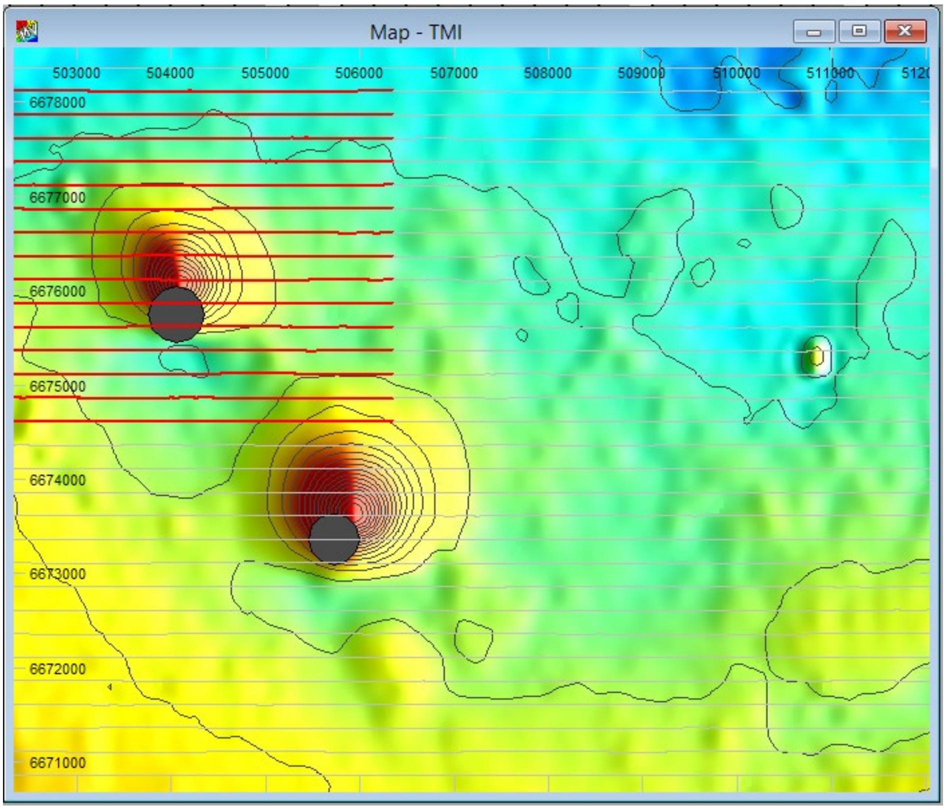
Step 5 - Returning to the Master Session

Now that the modelling of the isolated magnetic anomaly is complete it is time to pass the new body back to the master session. Any bodies that were passed into the sub-session were deactivated and hidden during the current session unless reactivated. They will be returned to the master session along with the new body. Note that when you first run this tutorial there are no bodies present in the master session, but after your first run the bodies will accumulate in the master.

Select the menu **File>Return subsession to master** and when asked to save the session file click OK and it will overwrite the initial sub-session with the file name `TARGET WIZARD TUTORIAL 14 INPUT DATA_PIPE_1.SES`. ModelVision appends the polygon name to the master file session name. This is useful if you want to open a sub-session later. You can choose not to save the sub-session, but it is recommended that you keep the sub-sessions while learning how to use the wizard.



After running Target Wizard on the adjacent anomaly, the master session changed as shown below with the data sub-set highlighted in red.



Tutorial 16 – UBC Module

This tutorial is designed to provide an overview of using the UBC module MAG3D with ModelVision. You will create an artificial but complex body magnetic response and use this as the input to the UBC MAG3D module. The voxel result of the inversion should reasonably match the initially used target body.

In a real-world case, you must have observed data plus a reasonable idea of the shape and property distribution of the target source. The creation of an appropriate 3D mesh and property distribution is shown in the tutorial.

Note UBC Model Mesh Designer is an ModelVision module and is not automatically available. A license message will be displayed if the module has not been licensed. For licensing details, please contact Tensor Research Pty Ltd.

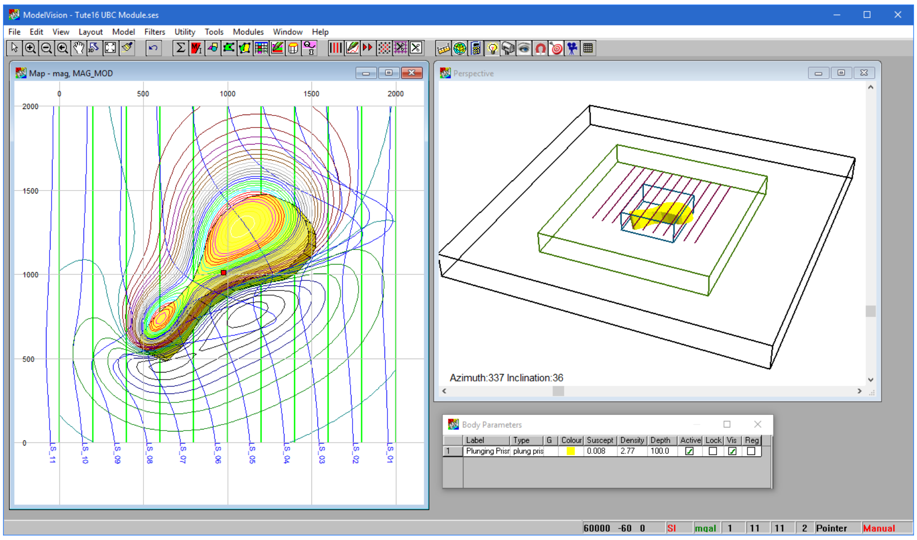
Tutorial Data

For this tutorial you are required to load the ModelVision session file located in the UBC software directory as installed from the UBC Installation CD. Refer to UBC-GIF Software Installation for instructions on this. The session file is located in the `C:\PROGRAMDATA\ENCOM\MVIS\TUTORIALS\TUTE16 UBC MODULE` folder and is called `TUTE16 UBC MODULE.SES`.

Running the Tutorial

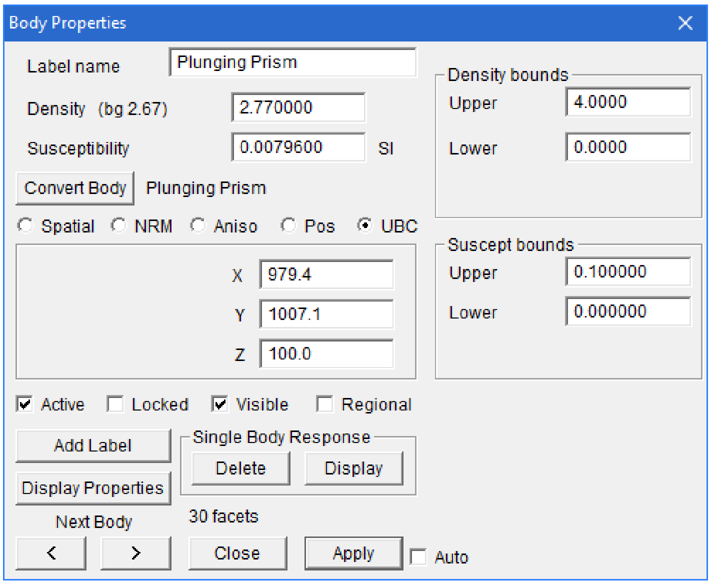
Step 1

Start ModelVision and use the File>Open>Session menu options to browse for and open `TUTE16 UBC MODULE.SES`. When it is loaded a preliminary plunging prism and computed response should appear as a map and 3D view similar to below:



Opening display from the TUTE16 UBC MODULE.SES session file

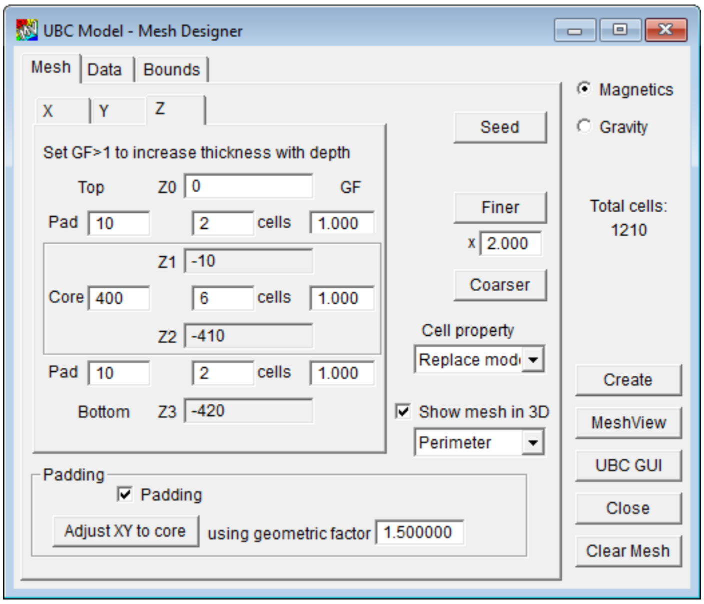
To display the properties of the body, click the Property button. This shows that only a single Plunging Prism body is shown in the window. From the Body Parameters table double click the ‘1’ at left side and the properties dialog of the body is displayed. Note in the display that upper and lower bounds for magnetic susceptibility and density can be specified. In this version, the bounds for magnetics are not operational but they can be set for density.



Dialog with UBC bounds settings available

Step 2

With the bounds, model and data displayed and specified, select the Modules>UBC Model Mesh Designer menu option. This option displays a dialog as below. Examine the dialogs and tabs presented by this option.



Export UBC model dialog with Mesh, Data and Bounds specifications

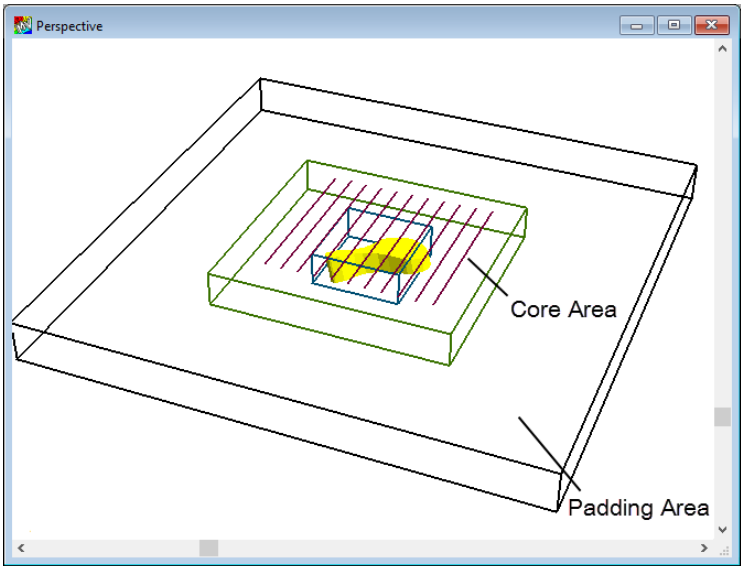
The Export UBC Model dialog has tabs for defining the mesh, data and bounds.

Mesh

For each of the three orthogonal directions of a mesh (X,Y and Z), you can specify the number of cells and their size for the core and padded areas surrounding the core.

Various controls allow the padding of cells to use a geometric factor for distribution.

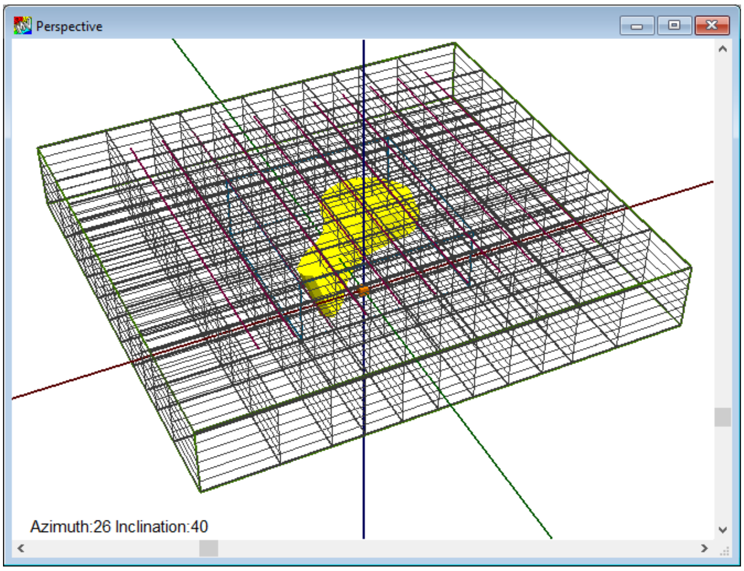
Use the Show mesh in 3D option to display the mesh design in Perspective View in two different designs. The default view is Perimeter which will show a bounding box for the core area of the UBC model and another bounding box for the padding area.



The Plunging Prism model with core and padded area specification

The other display option from the drop down list is to show All Cells, which when selected displays a preview of all cells within the UBC mesh being designed.

The Perspective View will update in real time when a change is made in the UBC Model–Mesh Designer dialog.



The Plunging Prism model with a 1.5 geometric weighting applied to the padding

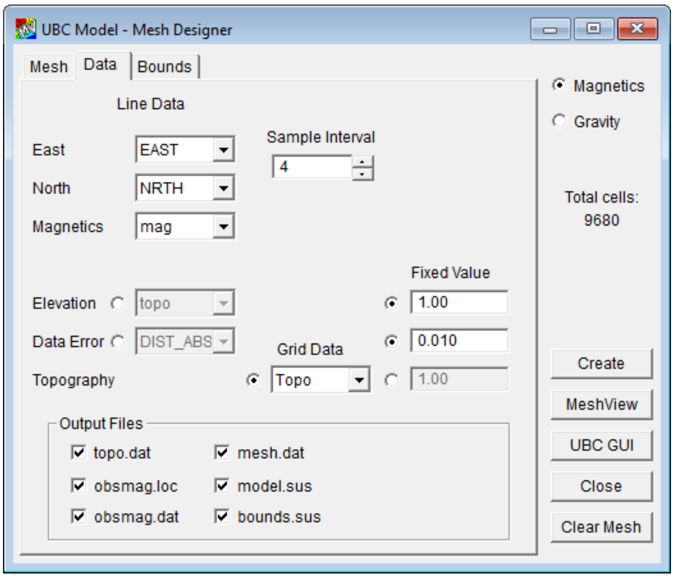
Data

The Data dialog allows you to specify the data channels, topography and elevation used for the UBC inversion. Ensure that the correct modelling method (magnetics or gravity) is selected before modifying the data definitions.

Each data reading can be used or every nth reading using the Sample interval setting. The computed response can be at a flat surface elevation above the model (at a Fixed value of elevation) or using a topographic gridded surface.

Note If you are using a topographic surface, the area of the surface must completely cover the core and padded area of the mesh.

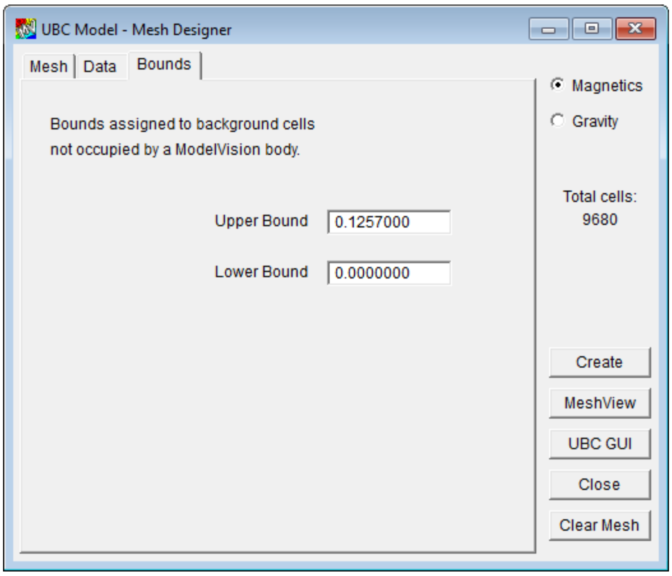
Output files required can be nominated for saving after the inversion.



Data control tab for the UBC modelling

BoundsI

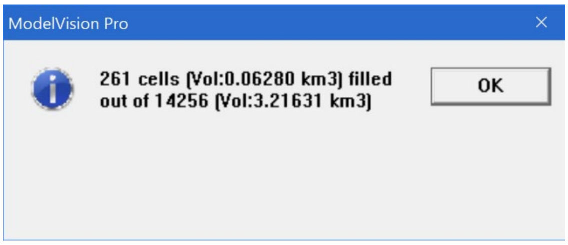
The Bounds control tab is used to specify the background cells surrounding the ModelVision seed body. The bounds are specified for the properties of the relevant method.



Bounds control tab setting the upper and lower bounds for the UBC modelling

Step 3

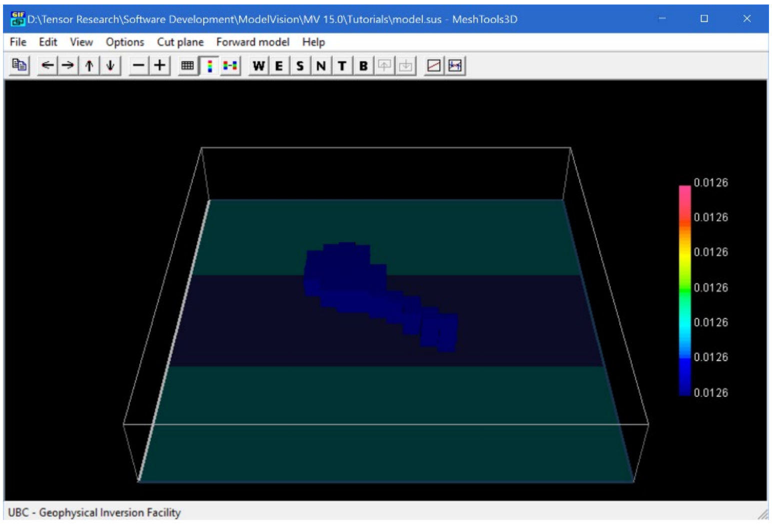
Using the defaults of the `TUTE16 UBC MODULE` session file, from the right side of the dialog, select the Create button. This operation writes the relevant files into the same directory as the initial session and model. As the files are created, ModelVision also reports the number of mesh cells, volume and padded area details.



UBC model and mesh report after creating the control and data files for the inversion

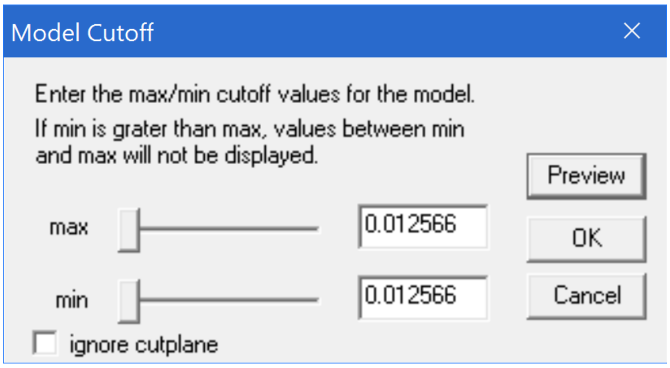
Step 4

Click the MeshView button. This executes the UBC 3D viewing software (MeshTools 3D).



MeshTools 3D view of the created mesh and seed model

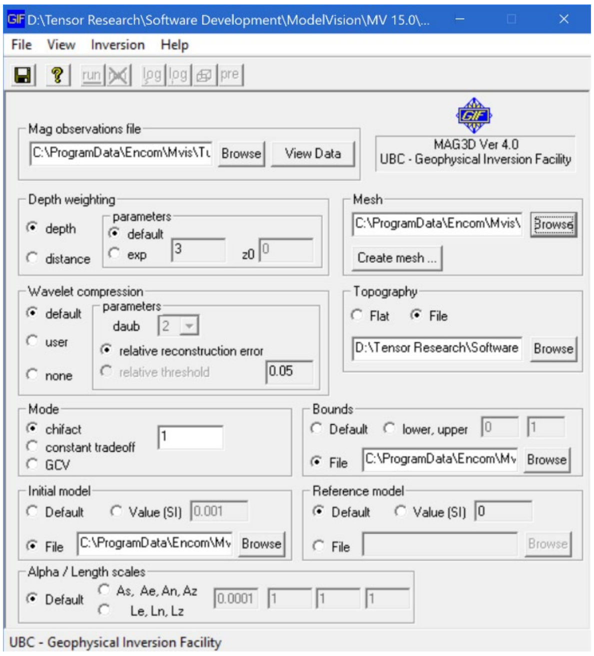
Refer to the MeshTools 3D help for information on operating the utility. You may need to rotate the mesh to optimally view the seed body, but you may also need to adjust the threshold properties of viewing. These are controlled from the Options>CutOff menu item. Adjust the slider bars to suit the model's magnetic susceptibility properties.



Slider controls of MeshTools 3D to control the view of the body

Step 5

Return to ModelVision (you do not have to close down the MeshTools 3D utility). Select the UBC GUI button. Initially a UBC-GIF copyright notice is displayed followed by the main MAG3D interface dialog.



MAG3D interface dialog used to specify the model, data and inversion methodology

For this tutorial, the main options are selected for you. The magnetics observation file is predetermined as are the settings for Data weighting and Wavelet compression. Refer to the UBC help for additional information on these settings. For fastest and optimal response, we recommend you select the computation Mode to be GCV and set the Initial model to be Default. The Mesh and Reference model can be left as defined.

Step 6

Once the above settings are specified, click the Save button (or the File>Save menu item). By default the control file selected is called `MAG_INV.INP` in the default directory. After the control input file is saved, the Run button becomes active. You can click the Run button and the inversion commences.

The inversion is executed in a DOS window that presents the progress of the computations. You will find that inversions depending on the speed of the computer, available memory, model/data complexity and size vary considerably in time required for a solution. For this case, about half an hour on a 7-800 mHz machine is required.

Note The UBC-GIF software is installed but has not been licensed, an error message is displayed. For more information, "Installing UBC Software" in the ModelVision User Guide PDF document.

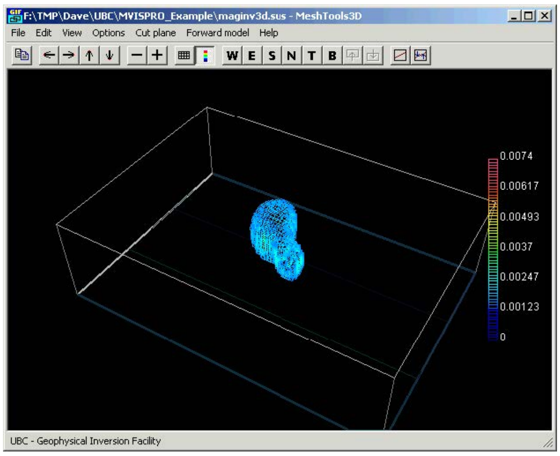
Step 7

Once the inversion has completed, the output, inverted solution files are written and these are available for examination. The inverted mesh solution can be viewed in Discover PA (with Voxel Model option) or in the MeshTools 3D utility.

In this case, the default file output is called:

Model file – `MAGINV3D.SUS`

Mesh file – `MESH.DAT`



MeshTools 3D representation of the output, inverted model