# Welcome to EMIGMA

EMIGMA is a powerful interpretation platform for many kinds of non-seismic geophysical data. It offers versatile solutions for Gravity, Magnetic, Electromagnetic, Magnetotelluric, Induced Polarization, Resistivity, and CSAMT applications and integrates data processing, simulation, inversion and imaging software as well as other associated tools.

Within one and the same framework you are offered a comprehensive set of functionalities:

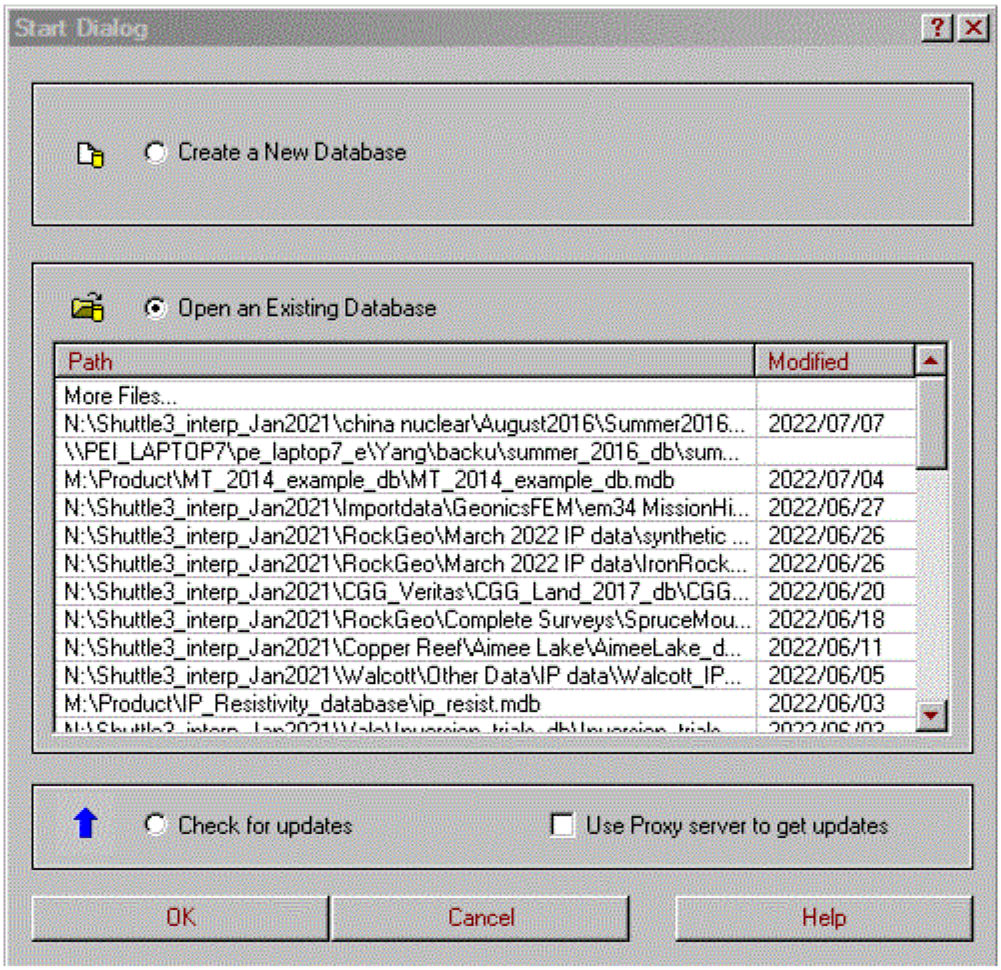
- Various kinds of imports both in the manufacturer's format and as ASCII columnar files
- Data quality control including editing, filtering, cleaning, trend removal, comparison analyses, and various corrections
- 3D Visualization intended for displaying any (measured, simulated, inverted) data in 3D space as profiles, vectors, true 3D surfaces, or 3D contoured surfaces, but also used for building and editing 3D models and inversions
- Gridding based on five interpolation algorithms and incorporating a ProfileViewer tool
- GridPresentation, MultiGrid, and Contour used for viewing the results of interpolations; with Contour allowing for both 2D and 3D representation of your data
- Plotting providing comparison between measured, simulated and processed data as well as a quick assessment of a single data set
- 3D Modeling allowing for unlimited prism, plate and polyhedra targets, complex topography, multiple body interactions and featuring fast and accurate 3D simulations, with a batch mode tool fully integrated into the platform
- 1D Inversions for frequency-domain EM, MT/CSAMT, Resistivity, and time-domain EM; 3D Inversions for Magnetics, Gravity, and Resistivity; 3D Euler Deconvolutions with Rodin Post Processing.

The objective of the present online help is not to explain the theoretical basis of EMIGMA or the complex algorithms lying behind each of the above-mentioned functionalities, but rather to offer the description of the user interface and all the options available therein. Being task-based, the help guides you step by step through every tool and application inside EMIGMA. The chapters, which, as a rule, have the same name as the tool or application they are describing, are divided into a number of topics, each covering a certain function or operation performable under this tool or application. An easy system of hyperlinks, related topics, and other references facilitates your navigating in the help and makes you aware of the relationships between different functionalities within the platform.

# Getting Started

Start **EMIGMA** from its group in the Start menu.

You will then be welcomed by the **Start** dialog:




<u>To open an existing database:</u>

The **Open an Existing Database** option will be selected by default. In the field below you will see the list of the demo databases and databases you created in previous times. You can also add other existing databases to your list:

- Double-click the **More Files** item that precedes the list to display the **Open Database File** dialog
- Select a database and click **Open**

To create a new database:

- Select the **Create a New Database** option in the upper part of the **Start** dialog. The **Save New Database** dialog appears

- Enter the name for the new database, select the folder to save the database in and click **Create**. The blank **Database** dialog will appear bearing the name you assigned to it

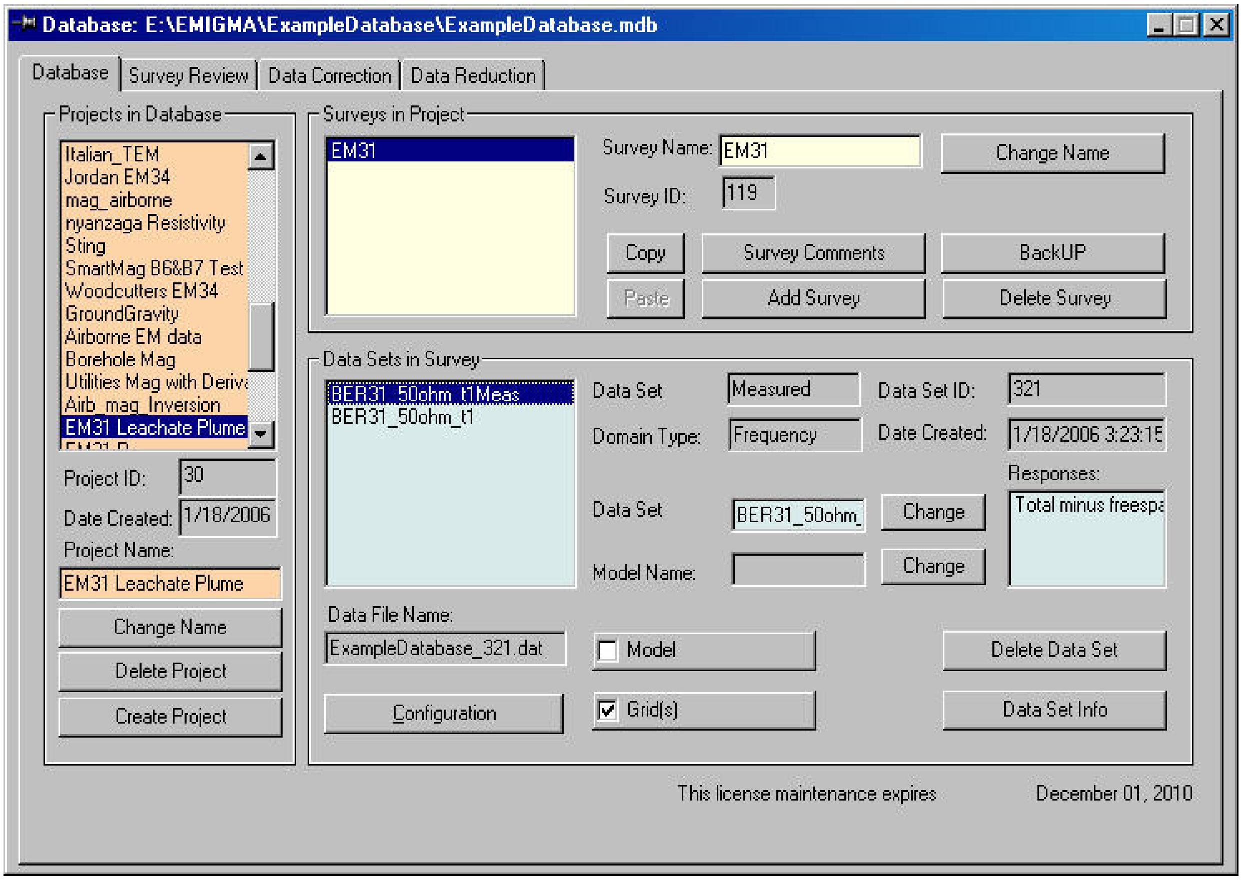
***Note***. *If you select an already existing .mdb file, it will be overwritten to produce a blank database*

To download EMIGMA updates:

- Select **Check for updates**
- If you have any problems connecting to the update server, activate the **Use Proxy** checkbox. Otherwise, leave it unchecked since it will noticeably increase the length of the download process.
- Click **OK**, EMIGMA will close and the Update window will be launched.

# Database Dialog: EMIGMA's Starting Point

The **Database** dialog that appears on starting EMIGMA is where you can manage all levels of your data organization, create or delete projects, view project-related information:

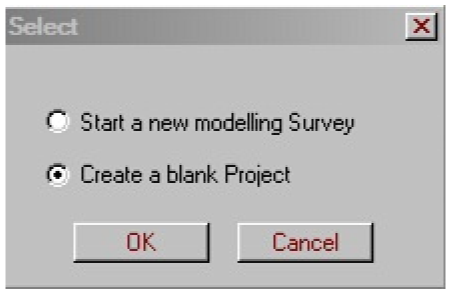



There are three levels of data organization. The first level, **Project,** is divided into one or more **Surveys**, which, in their turn, may contain one or more **Data Sets**. In one project, the surveys may represent different systems or data types, being associated with each other only, for instance, by a territorial criterion; as for the data sets in each survey, they may be numerous, measured or simulated, but must have the same strict structure defined by the system geometry, number of locations, etc.

**Related Topics:**

# Creating Projects From Scratch

- Click the **Create Project** button in the bottom left-hand section of the [Database] dialog. In the **Select** dialog to appear, select the **Start a New Modeling Survey** button:




- In the **Name for New Project** dialog to appear, type in the name of your project and click **OK:**


The **Configuration** window opens with a number of steps allowing you to specify your system geometry, waveform, profile information and output.

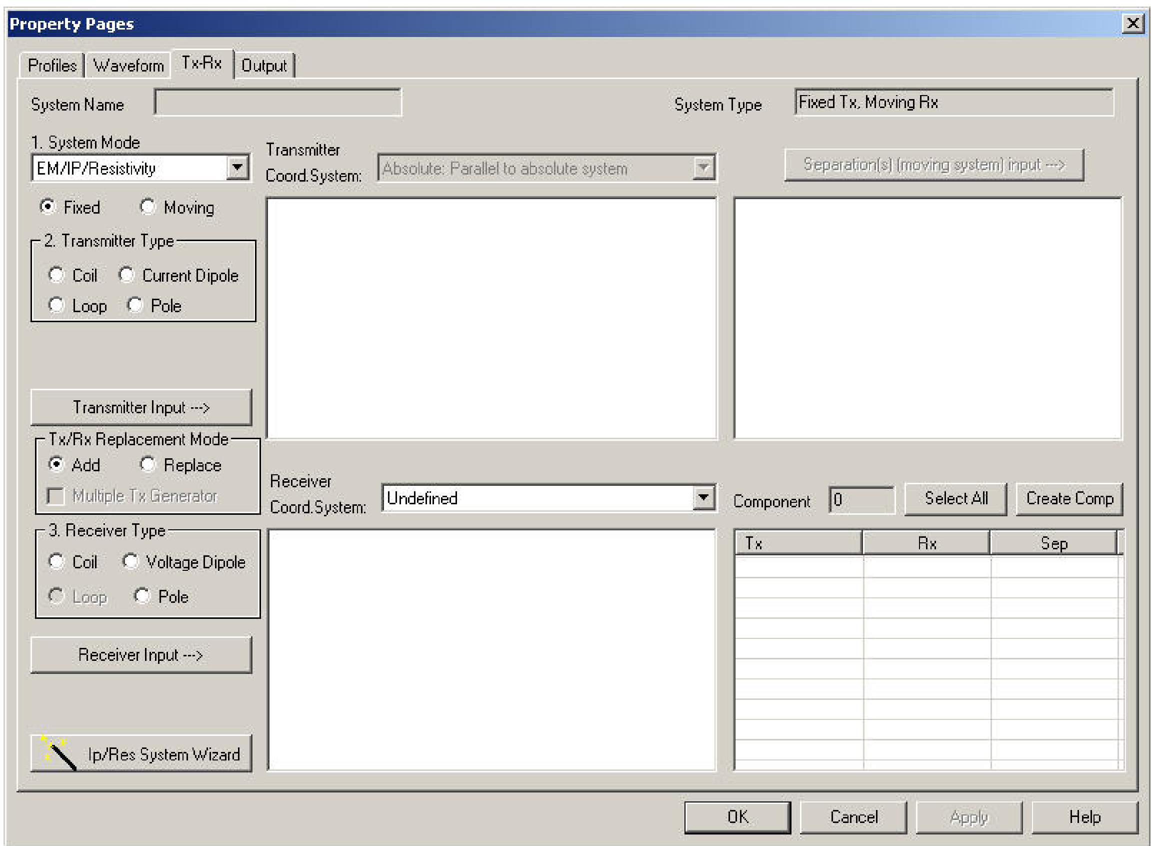
**See**

| **System Configuration:** | **Model Configuration**: |
|---|---|
| Specifying System Geometry | Specifying the Number and Properties of Layers |
| Specifying Waveform | Specifying the Number and Properties of Targets |
| Specifying Profile Information | Saving Model Configuration |
| Specifying Output | |
| | |

# System Configuration

## Specifying System Geometry

The **Tx-Rx** tab of the **Property Pages** dialog offers you to specify the system. If you are in the process of creating a new survey, it will open blank automatically after you have given a name to your project. Otherwise you can reach this page by selecting an existing project/survey and clicking the **Configuration** button in the bottom of the **Database** tab




**Tx-Rx** tab of the **Property Pages**

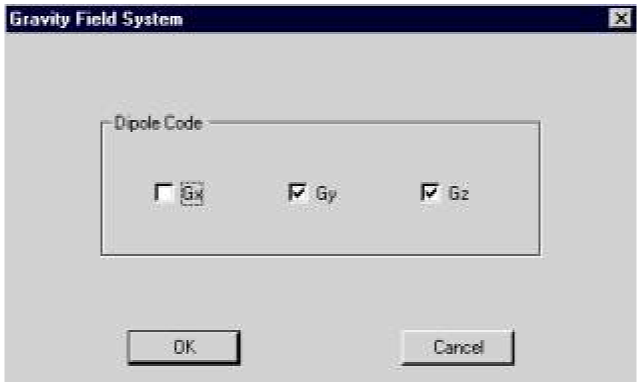
**Related topics:**

Specify a Gravity System
Specify a Magnetic System
Specify an MT System
Specify a CSEM System
Specify a ZTEM System
Define System Components for Modeling

| | |
|---|---|
| Specify a CSAMT System | Specify the Coordinate System |
| Specify an EM/IP/Resistivity System | Multiple Tx Generator |
| Specify a Land CSEM System | Specify a MMR System |
| Specify a VLF-R System | Specify a VLF System |

## Specify a gravity system

- Select **Gravity** from the **System Mode** list in the [Tx-Rx] tab of the **Property Pages** dialog. The **Gravity Field System** dialog appears

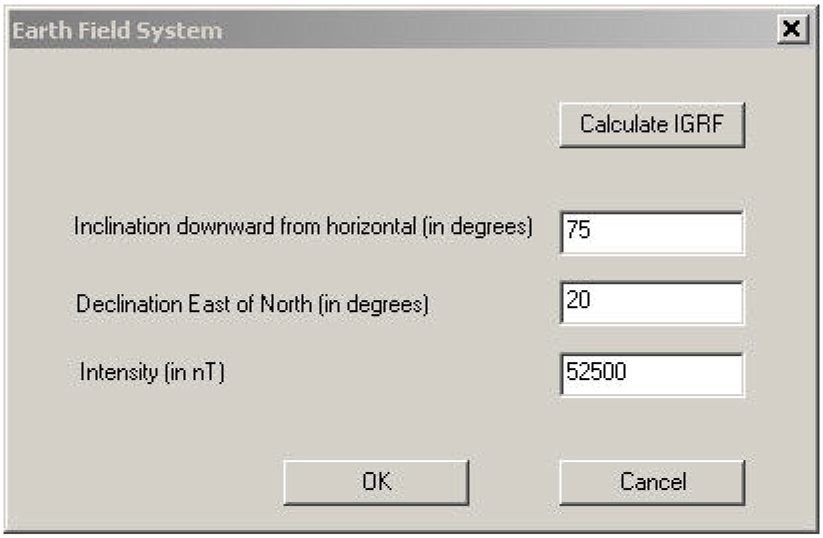



- Specify the number and orientation of the dipoles and click **OK**. The rest of the **Tx-Rx** tab will be filled out automatically
- To add a dipole, click the **Receiver Input** button to reopen the **Gravity Field System** dialog and check dipoles as needed

## Specify a magnetic system

- Select **Magnetic** from the **System Mode** list in the **Tx-Rx** tab of the **Property Pages** dialog.

  The **Earth Field System** dialog appears:

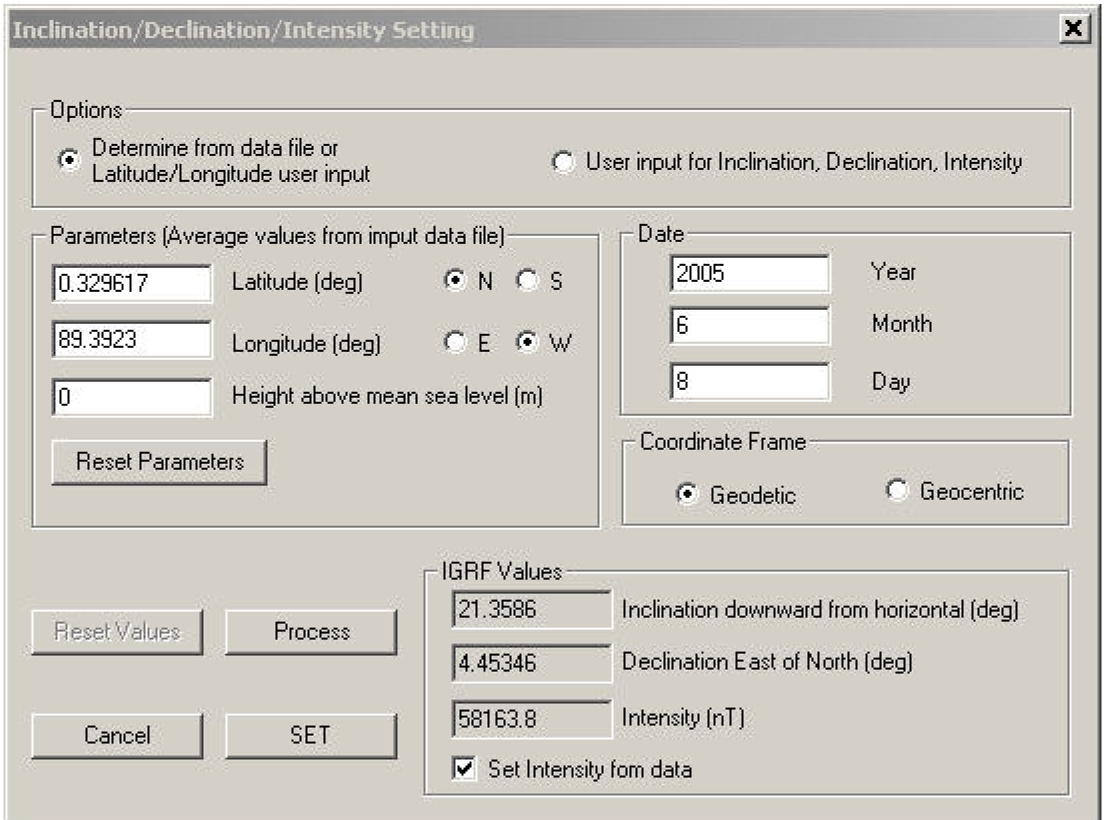



- Specify the inclination, declination and intensity in the respective fields

  OR

- Click the **Calculate IGRF** button to compute these parameters from your latitude and longitude.

  The **Inclination/Declination/Intensity Setting** dialog appears:




- Select the **Determine from Data File or Latitude/Longitude User Input** option to activate the **Parameters**, **Date** and **Coordinate Frame** sections below. The **Date** section contains the current date.

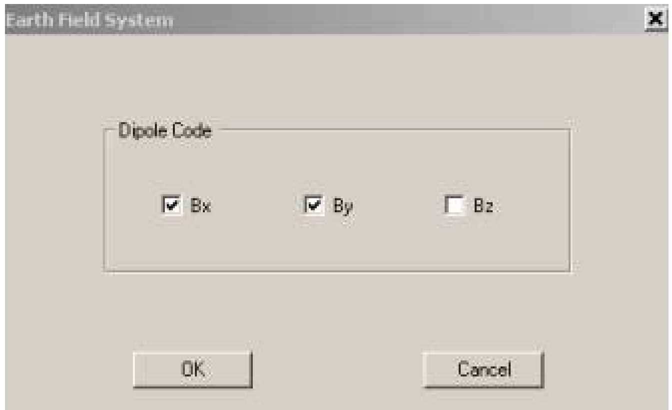
- Enter Latitude, Longitude, and Height above sea level in the respective fields of the **Parameters** section.

- Select between **Geodetic** and **Geocentric** in the **Coordinate Frame** section.

- Click **Process.** The **IGRF Values** section will update accordingly.

- Click **Set** to return to the **Earth Field System** dialog.

- Click **OK** in the **Earth Field System** dialog to confirm the values and proceed to the next dialog with the same name:

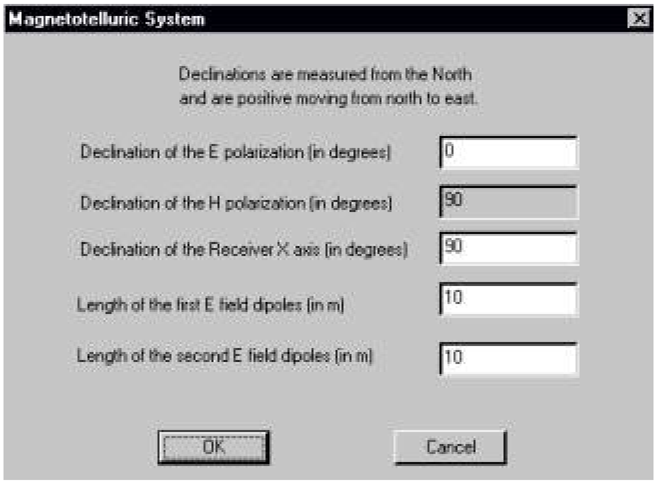



- Set dipole codes and click **OK** to close the dialogs and view the created system back in the **Tx-Rx** tab.
- To reopen the dialogs, use the **Transmitter Input** and **Receiver Input** buttons in the left-hand part of the tab.

## Specify an MT system

- Select **MT** from the **System Mode** list in the **Tx-Rx** tab.

  The **Magnetotelluric System** dialog will appear:




- Specify the declination of the E polarization in the respective field. This is the polarization of the electric field in the source plane wave. The declination of the H polarization will change accordingly by 90 degrees
- Set the value for the X-axis declination of the receiver. The measurement setup may be oriented at any angle
- Set the lengths of the first and second E field dipoles
- Click **OK**

To change your settings, reopen the **Magnetotelluric System** dialog by clicking **Transmitter Input** or **Receiver Input** buttons in the **Tx-Rx** tab of the **Property Pages** dialog.

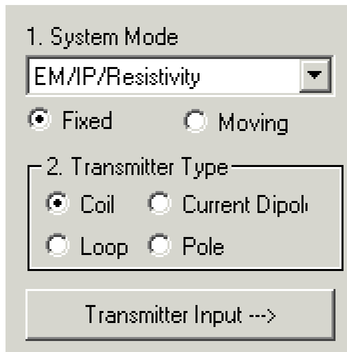
## Specify a CSAMT system

Select **CSAMT** from the **System Mode** list in the **Tx-Rx** tab of the **Property Pages** dialog. The **Bipole Length** dialog appears.

Specify the parameters of the transmitter as described in the Current Dipole section. The rest of the **Tx-Rx** tab will be filled out automatically.

## Specify an EM/IP/Resistivity system

- Select **EM/IP/Resistivity** from the **System Mode** list in the **Tx-Rx** tab of the **Property Pages** dialog:




- Select between the **Fixed** and **Moving** systems
- Specify a transmitter type in the respective section and click **Transmitter Input**. The dialog to appear depends on the transmitter type you selected

**Related Topics**

Specify Transmitter Type: Coil Transmitter

Specify Transmitter Type: Current Dipole

Specify Transmitter Type: Loop

Specify Transmitter Type: Pole

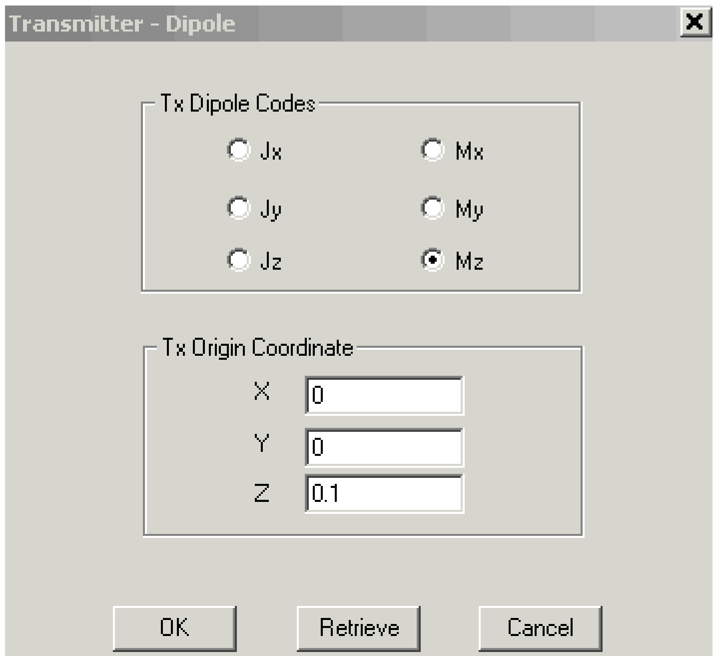
Specify Receiver Type

Specify Separation

IP/Resistivity Wizard

## Specify Transmitter type: Coil Transmitter

Select **Coil** in the **Transmitter Type** section of the **Tx-Rx** tab. The **Transmitter – Dipole** dialog will appear:



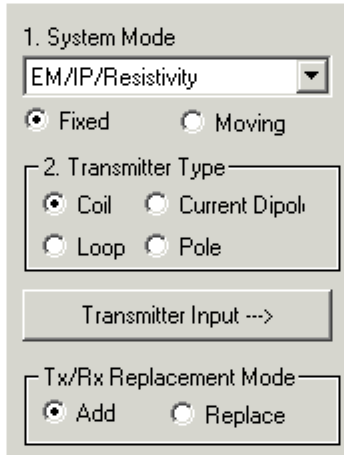

- Specify the Tx dipole codes

- For fixed systems, specify the origin coordinates in the respective section

  Should you have an electric dipole, its z coordinate will automatically set to its maximum, which is –0.1. In the case of a magnetic dipole, its z coordinate will automatically be set to its minimum, which is 0.1.

- Click **OK** to close the dialog and view your transmitter information on the **Tx-Rx** tab

- To add a new transmitter, make sure that the **Add** button is on in the **Tx/Rx Replacement Mode** section of the **Tx-Rx** tab, click the **Transmitter Input** button to reopen the **Transmitter – Dipole** dialog and specify the codes of a new dipole:

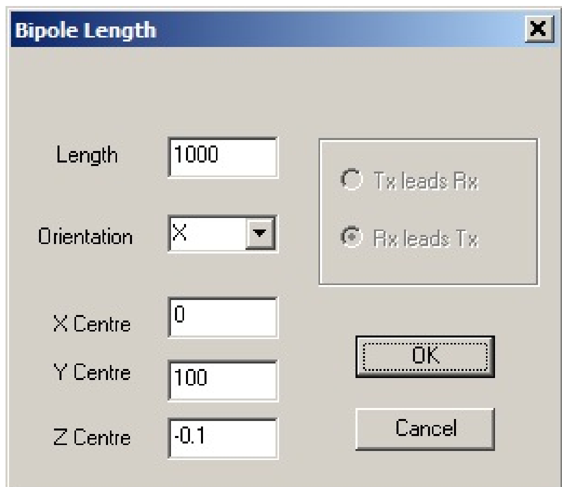



- To replace a transmitter, switch to the **Replace** option in the **Tx/Rx Replacement Mode** section, click the **Transmitter Input** button to bring up the **Transmitter – Dipole** dialog and change your settings. If you decide to return to initial values at this stage, click the **Retrieve** button.

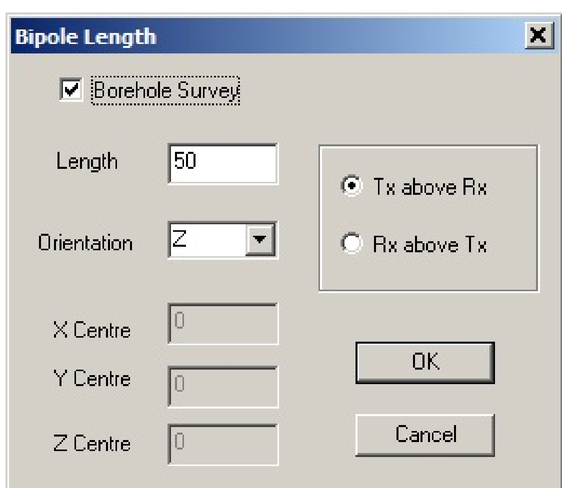
## Specify Transmitter Type: Current Dipole

If you select **Current Dipole**, a box to specify the current will appear below the **Transmitter Type** section.

The **Transmitter Input** button will bring up the **Bipole Length** dialog:




In case of **moving current dipole**, dialog has additional option - **Borehole** checkbox. When it is checked, user can select Tx and Rx position against each other:




- Specify the bipole length and select its orientation from the list.

The list contains 4 items: X, Y, Z following a blank item provided for those cases when bipole orientation is to be specified each separate time

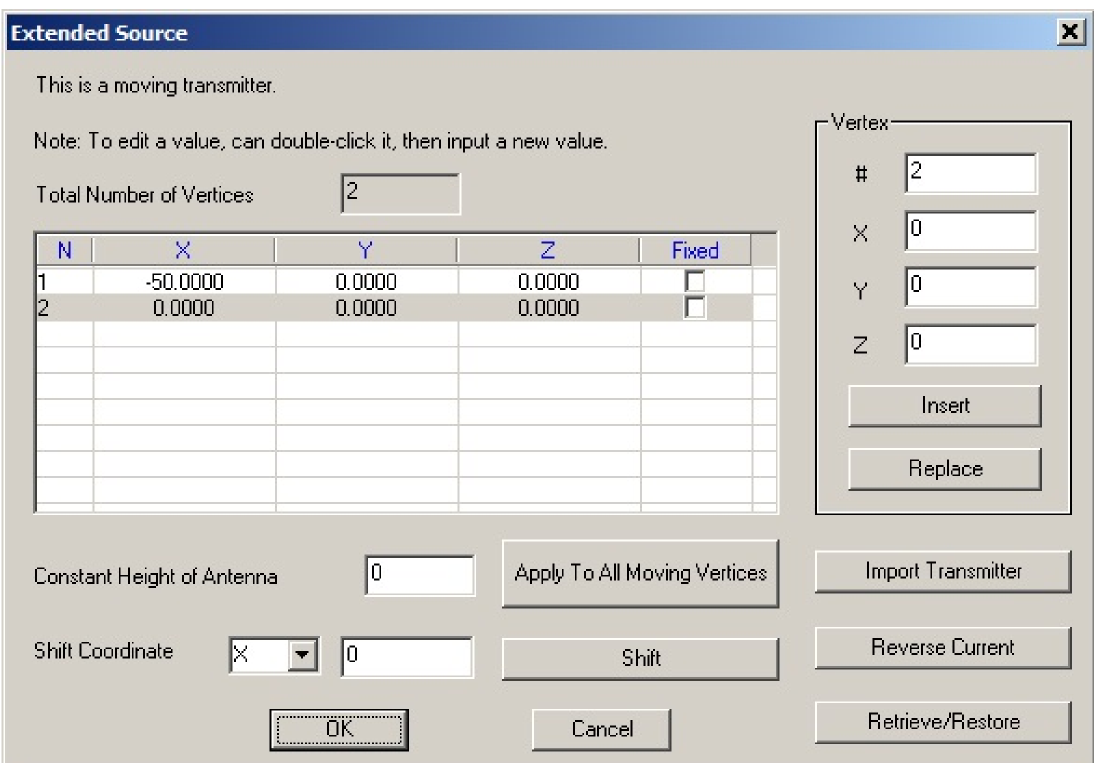
- In a fixed system, specify the position of the bipole center in the respective fields

  Should it be an X orientation, the **X Center** value will define the position of its center relative to the X-axis. The z coordinate will be set automatically to –1

- In a moving system, specify the position of the transmitter and receiver relatively to each other in the right-hand section of the dialog

- Click **OK**

The **Extended Source** dialog appears containing the settings defined in the **Bipole Length** dialog:



Last column "Fixed" with checkboxes in the grid control is present only for **Moving** transmitters. If box is checked in some row, it means that respective node is not moving.

To change the settings:

- Select the vertex row and type in new coordinates
- Switch to **Replace** in the **Mode** section and click **Add to the List**

  ***Note***. *If you change your mind, click the **Retrieve/Restore** button to return to your previous values*

- If necessary, alter the value of the antenna height in the respective box and click **Change** or, if it is to vary, check the **Variable** box

To replace the coordinate:

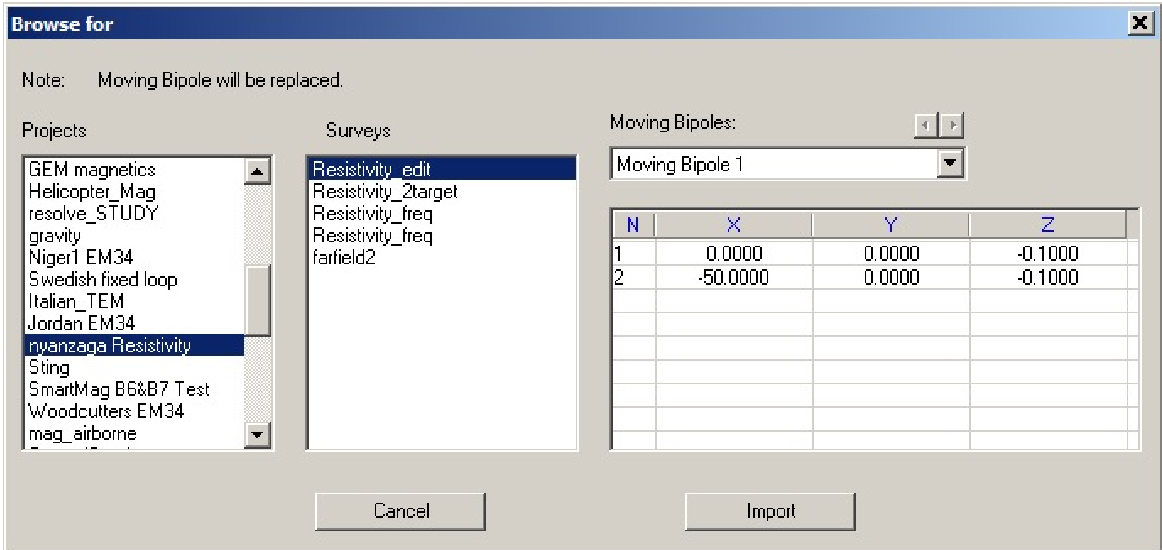
- Select it and turn the **Replace** button on in the **Mode** section of the dialog
- Change the coordinate as needed and click **Add to the List**

To change the coordinate for all vertices by the same value:

- Select the axis from the **Shift Coordinate** list
- Type in the value you want to change the coordinate by in the **Value** box and click **Apply**.

To import a bipole from another survey within the same project:

- Click **Import Transmitter** to open the **Browse For** dialog




- Select the required survey and bipole in this dialog and click **Import**

To change the current settings to reverse:

- Click the respective button in the bottom right-hand corner of the **Extended Source** dialog
- Click **OK** to close the dialog and view your transmitter information on the **Tx-Rx** page

To replace a bipole:

- Select the **Replace** option in the **Tx/Rx Replacement Mode** section of the **Tx-Rx** tab and click **Transmitter Input**

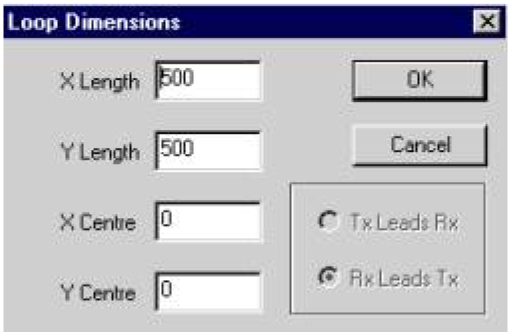
  This will bring up the **Extended Source** dialog

- Make your changes in the dialog as described above, click **Add to the List** and **OK**

## Specify Transmitter Type: Loop

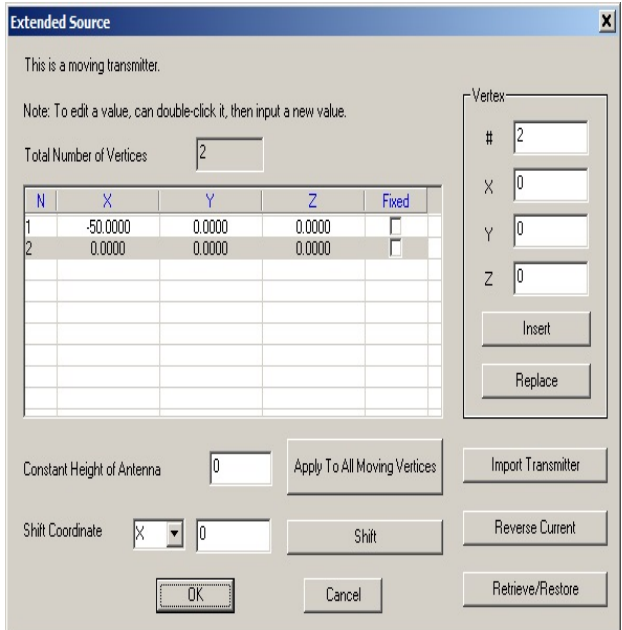
Select **Loop** in the **Transmitter Type** section of the Tx-Rx tab and click the **Transmitter Input** button.

In the **Loop Dimensions** dialog to open:




- Specify the length of the loop sides in the **X Length** and **Y Length** fields and the position of the loop center relative to the X- or Y-axis
- In a moving system, specify the position of the receiver and transmitter relative to each other in the activated bottom right-corner section
- Click **OK**.

The **Extended Source** dialog appears:




- To add more loop vertices, specify the X and Y coordinates in the respective section and click **Add to the List**
- To replace a loop with one from another survey within the same project
- Click **Replace Transmitter** to open the **Browse For Tx Loop** dialog
- Select the required survey and loop in this dialog
- Click **Replace** to bring the new loop into the **Extended Source** dialog

- To change the settings:
- Select the vertex row in the table and change one or more coordinates in the **Coordinates of Vertices** section
- Switch to **Replace** in the **Mode** section and click **Add to the List**

*Note*. *Click the **Retrieve/Restore** button to return to initial values*

- If necessary, alter the value of the antenna height in the respective box and click **Change** or, if it is variable, check the

**Variable** box

- To change the coordinate of all vertices by the same value
- Select the axis from the **Shift Coordinate** list
- Type in the value you want to change the coordinate by in the **Value** box and click **Apply**.
- To change the current to reverse
- Click the respective button in the bottom right-hand corner of the **Extended Source** dialog
- Click **OK** to close the dialog and view your transmitter information on the **Tx-Rx** tab

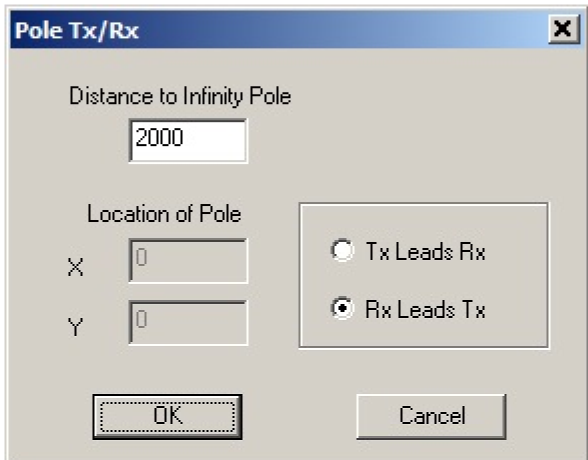
**To Add or Replace Loops**

To add a new loop, make sure that the **Add** button is on in the **Tx/Rx Replacement Mode** section of the **Tx-Rx** tab and click the **Transmitter Input** button to reopen the **Loop Dimensions** and through to the **Extended Source** dialog to specify the dimensions and vertices of the new loop

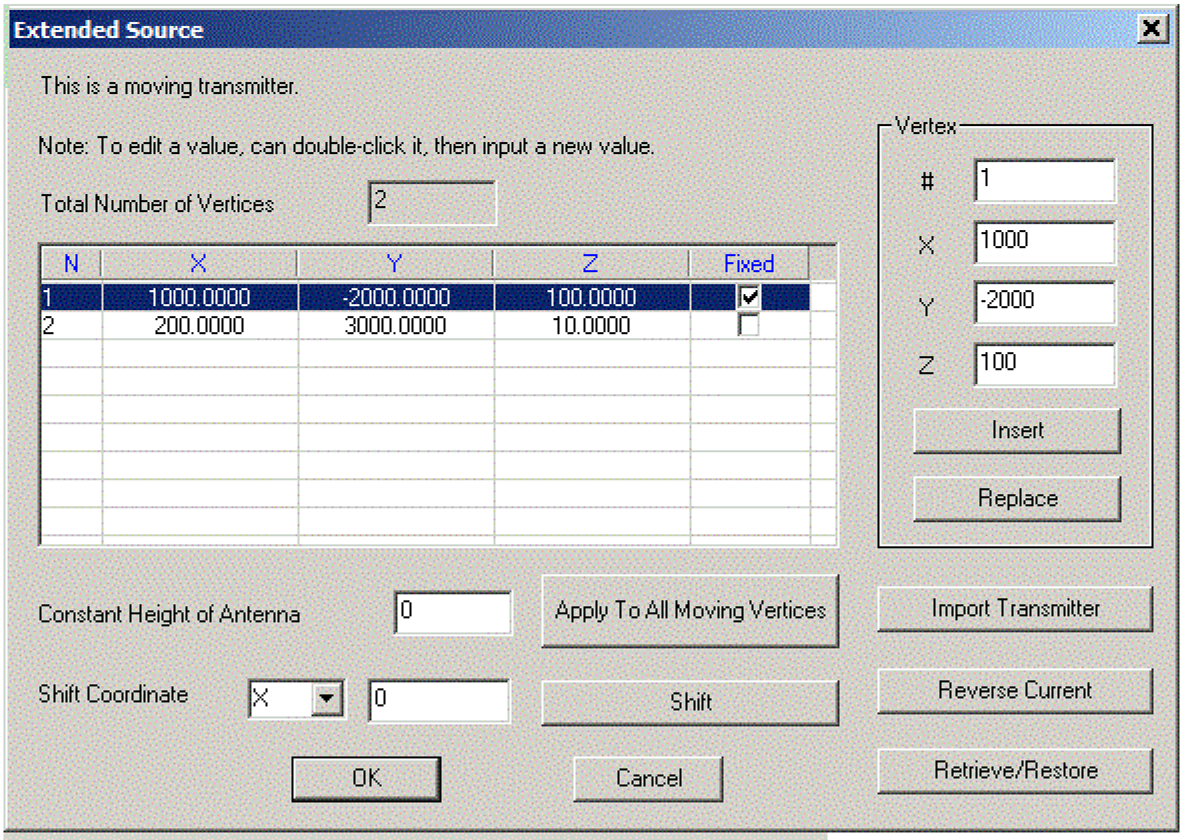
To replace a loop, select the **Replace** option in the **Tx/Rx Replacement Mode** section of the **Tx-Rx** tab and click **Transmitter Input**. This will bring up the **Extended Source** dialog right away. Make your changes, click **Add to the List** and **OK**.

## Specify Transmitter Type: Pole

Select **Pole** in the Transmitter Type section of the [Tx-Rx ]tab and click the **Transmitter Input** button. In the **Pole Tx** dialog to appear:




- Specify the distance to the infinity pole and its coordinates
- Click **OK**. The **Extended Source** dialog appears containing the settings you defined in the **Pole Tx** dialog:




Last column "Fixed" with checkboxes in the grid control is present only for **Moving** transmitters. If box is checked in some row, it means that respective node is not moving.

To change the settings:

- Select the vertex row in the table and change one or more coordinates in the **Coordinates of Vertices** section
- Switch to **Replace** in the **Mode** section and click **Add to the List**

*Note. Click the **Retrieve/Restore** button to return to initial values*

To change the coordinate of all vertices by the same value

- Select the axis from the **Shift Coordinate** list
- Type in the value you want to change the coordinate by in the **Value** box and click **Apply**.

To input a transmitter from another survey within the same project

- Click **Replace Transmitter** to open the **Browse For** dialog
- Select the required survey and transmitter in this dialog and click **Replace**

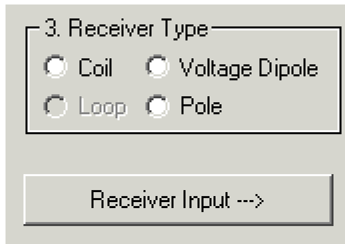
To change the current to reverse

- Click the respective button in the bottom right-hand corner of the **Extended Source** dialog

Click **OK** to close the dialog and view your transmitter information on the **Tx-Rx** tab

## Specify Receiver Type

Select the receiver type in the respective section of the **Tx-Rx** page and click **Receiver Input:**




The dialog to appear depends on the receiver type you selected.

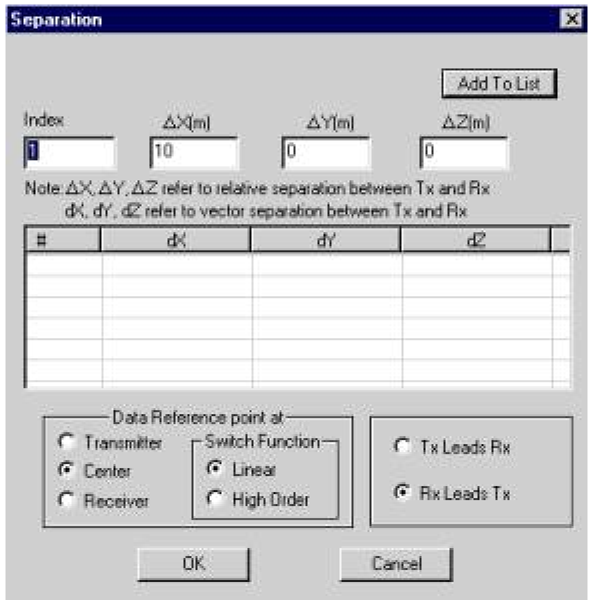
- If you selected **Coil**, the **Receiver – Dipole** dialog will appear. Specify dipole codes in the respective section. To change back to initial settings, click the **Retrieve/Restore Data** button

- If you select **Voltage Dipole**, the **Bipole Length** dialog will open to be followed by the **Extended Source** dialog, see Specify Transmitter Type: Current Dipole to go through all the necessary steps
- If you select **Pole**, the **Rx Pole** dialog will open to be followed by the **Extended Source** dialog, see Specify Transmitter Type: Pole to go through all the necessary steps

In case your system is fixed, you will return to the **Tx-Rx** page right from the above dialogs. If it is moving, you will have to specify the separation parameter, see Specify Separation.

## Specify Separation

In moving systems, after having specified the receiver type, you will be brought to the **Separation** dialog:




In this dialog:

- **Index** is the record number of separation and is 1 by default.

- Specify the relative separation in the **ΔX**, **ΔY** and **ΔZ** boxes and click **Add to List**. Repeat as many times as the number of separations you need

- **Select** between the **Transmitter**, **Center** or **Receiver** options, which are conventions being used as reference points

  The center point convention means that the transmitter is located +1/2 the separation parameter from the plot point. In the transmitter point convention, the transmitter and the plot point occupy the same position, whereas in the receiver point convention, it is the receiver that coincides with the plot point

- To switch between the three options, select a required switch function in the respective section. **Linear** is set by default
- Select between the **Tx leads Rx** and **Rx leads Tx** options to define the sign (negative or positive) of separation
- To delete a separation, select it in the table and press the Delete key
- Click **OK** to close the dialog and return to the **Tx-Rx** tab.

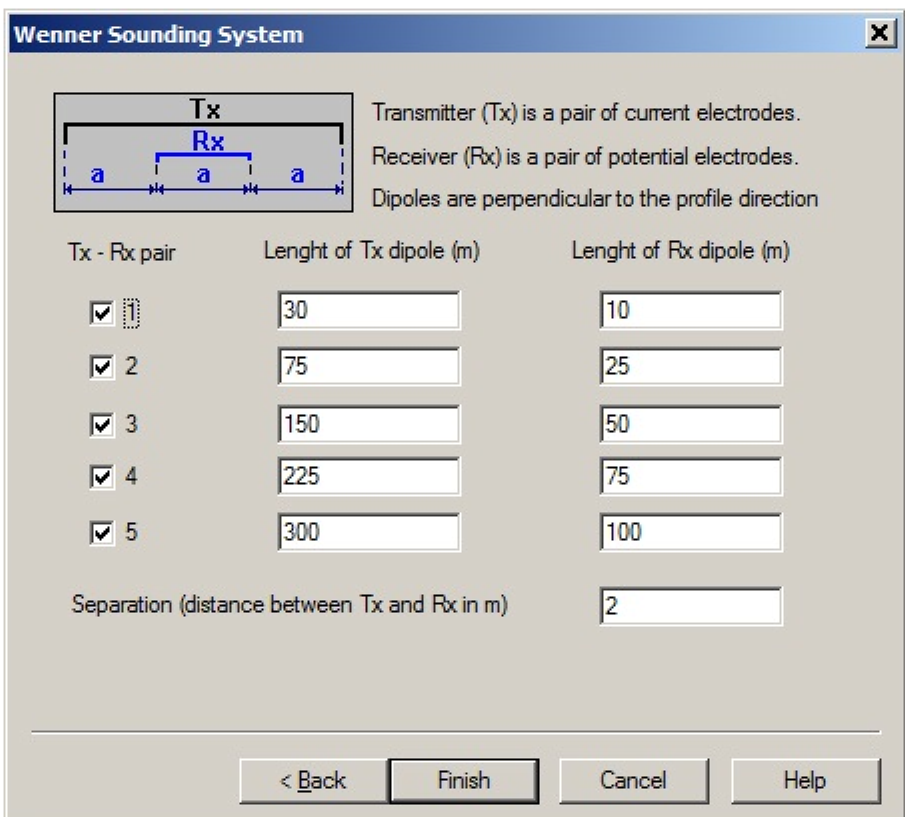
You can see the specified separation data in the upper right-hand field of the **Tx-Rx** tab. To change these data, click the **Separation(s) (Moving System) Input** button above this field. This will bring up the **Separation** dialog.

**IP/Res Wizard**

You can also use the **IP/Res Wizard** to automatically set your system geometry. As of now, only four standard configurations are available: Wenner, Schlumberger, Dipole-Dipole, Pole-Dipole:

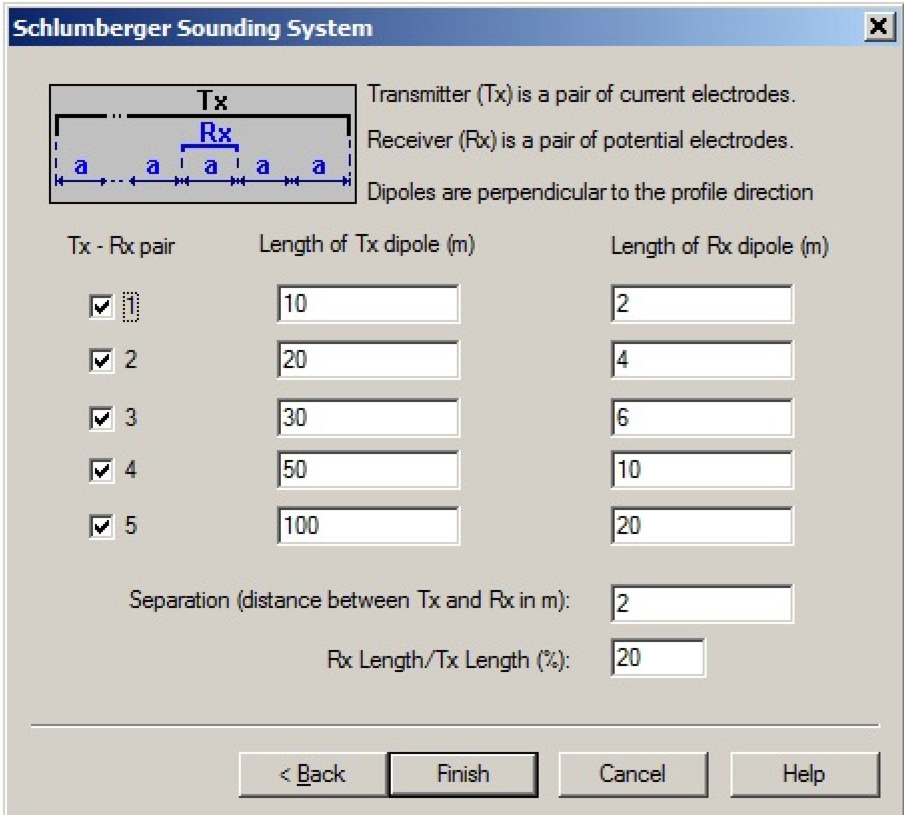
- Click the **IP/Res Wizard** button in the bottom left-hand corner of the **Tx-Rx** page. The **Select System** dialog appears containing the four options
- Select one of the options and click **Next**

If you select **Wenner**, the **Wenner Sounding System** dialog will appear containing the respective diagram.



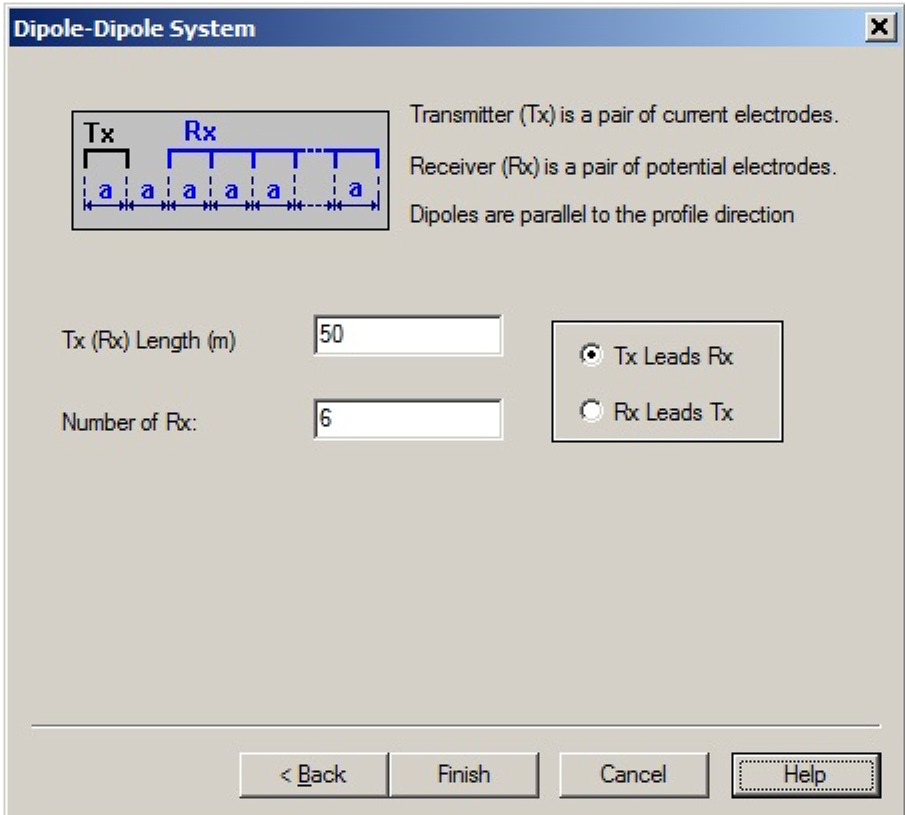

Change the suggested length of the transmitter dipole as needed; this will automatically change the length of the receiver dipole. Specify the separation and click **Finish**

If you select **Schlumberger**, the **Schlumberger Sounding System** dialog will appear containing the respective diagram.



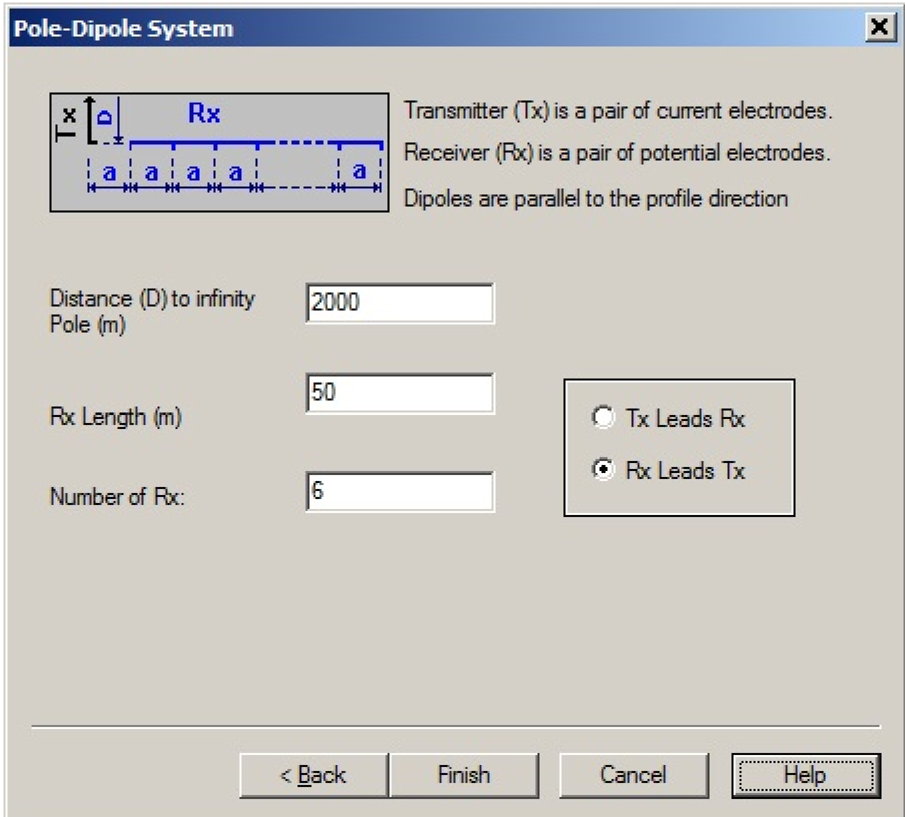

Change the suggested length of the transmitter dipole as needed; this will automatically change the length of the receiver dipole. Specify the separation and click **Finish**

If you select **Dipole-Dipole**, the **Dipole-Dipole System** dialog will appear containing the respective diagram.




Change the suggested length of the transmitter/receiver dipole and specify the number of receivers. Select which of the two – transmitter or receiver – is leading and click **Finish**

If you select **Pole-Dipole**, the **Pole-Dipole System** dialog will appear containing the respective diagram.




Change the suggested distance to the infinity pole, length of the transmitter/receiver dipole and specify the number of receivers. Select which of the two – transmitter or receiver – is leading and click **Finish**

## Specify a CSEM system

Select **CSEM** from the **System Mode** list in the **Tx-Rx** tab of the **Property Pages** interface.

Click **Transmitter Input** and the **Bipole Length** dialog appears.

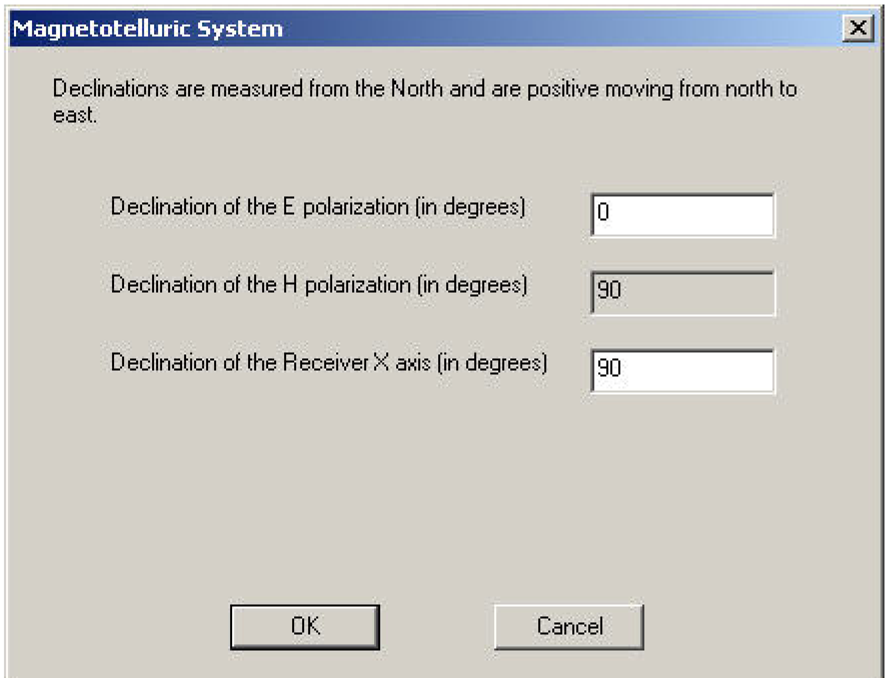
Specify the parameters of the transmitter as described in the Current Dipole section.

Click **Receiver Input** and the same interface that was used to specify the transmitter parameters is used to specify the receiver parameters.

## Specify a ZTEM system

- Select **ZTEM** from the **System Mode** list in the **Tx-Rx** tab of the **Property Pages** interface.

  The **Magnetotelluric System** dialog will appear:



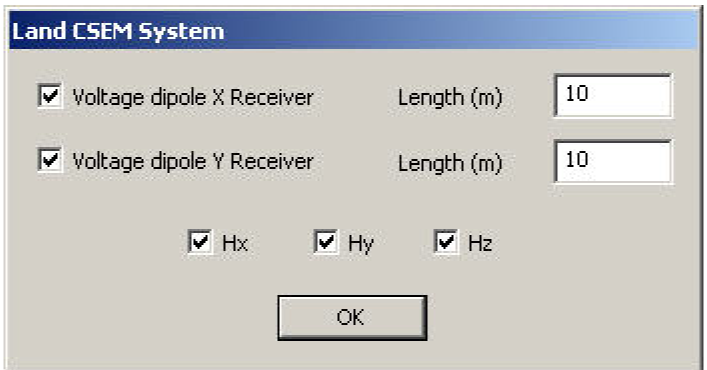

- Specify the declination of the E polarization in the respective field. This is the polarization of the electric field in the source plane wave. The declination of the H polarization will change accordingly by 90 degrees
- Set the value for the X-axis declination of the receiver. The measurement setup may be oriented at any angle
- Click **OK**

To change your settings, reopen the **Magnetotelluric System** dialog by clicking the **Transmitter Input** or **Receiver Input** buttons on the **Tx-Rx** tab.

### Specify a land CSEM system

- Select **Land CSEM** from the **System Mode** list in the **Tx-Rx** tab.
- Specify the parameters of the transmitter as described in the Current Dipole section.

  The following window then appears:

- Select at least one coil receiver and at least one dipole receiver. Confirm the length of the dipole receiver and click OK.
- To change your settings, click the **Transmitter Input** or **Receiver Input** button on the **Tx-Rx** tab.

**Specify an MMR system**

- Select **MMR** from the **System Mode** list in the **Tx-Rx** tab.

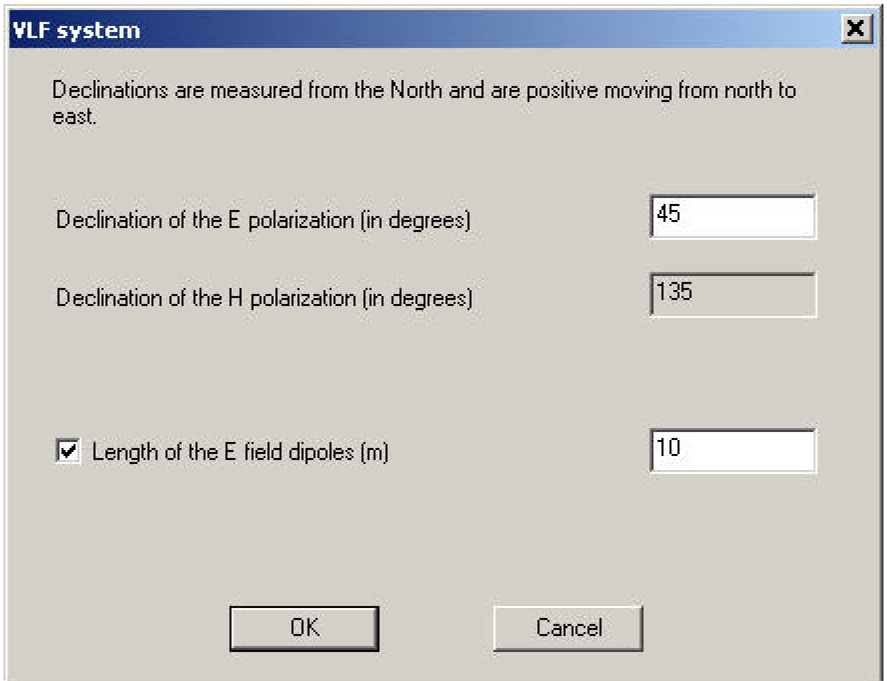
Specify the parameters of the transmitter as described in the Current Dipole section. The rest of the **Tx-Rx** tab will be filled out automatically.

To change your settings, click the **Transmitter Input** button on the **Tx-Rx** tab.

**Specify a VLF-R system**

- Select **VLF-R** from the **System Mode** list in the **Tx-Rx** tab.

The following window will appear:



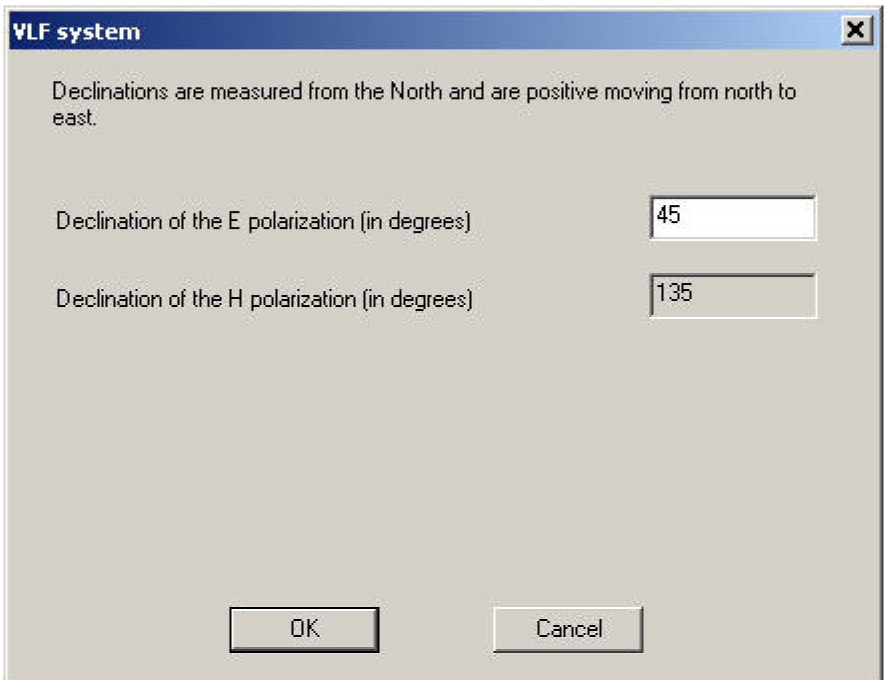

- Specify the declination of the E polarization in the respective field. This is the polarization of the electric field in the source plane wave. The declination of the H polarization will change accordingly by 90 degrees
- Set the length of the E field dipole
- Click **OK**

To change your settings, click the **Transmitter Input** or **Receiver Input** button on the **Tx-Rx** tab.

**Specify a VLF system**

- Select **VLF** from the **System Mode** list in the **Tx-Rx** tab.

The following window will appear:




- Specify the declination of the E polarization in the respective field. This is the polarization of the electric field in the source plane wave. The declination of the H polarization will change accordingly by 90 degrees
- Click **OK**

To change your settings, click the **Transmitter Input** button on the **Tx-Rx** tab.

## Define system components for modeling

If you have a number of transmitters, receivers or separations, you may choose to use either all of them or only their concrete combinations in further modeling:

- Click the **Select All** button to add all transmitters, receivers and separations into the **Component** table in the bottom right-hand corner of the **Tx-Rx** tab. This will result in all possible combinations
- Select a required transmitter, receiver or separation in the respective field and click the **Create Component** button. This will add only the combinations you selected

***Note***. *In the case of gravity, magnetic and magnetotelluric systems, the list of components will be generated automatically.*

## Specify the coordinate system

The **Tx-Rx** tab has the **Transmitter Coordinate System** and **Receiver Coordinate System** dropdown lists. The first is not active, so to change it, you have to specify the coordinate system in the second. However it is true only for moving transmitter geometries. In the case of fixed geometries, the transmitter will always be located in the absolute coordinate system and the selection from the respective list will read **Absolute: Parallel to Absolute system** no matter which coordinate system is chosen for the receiver.

In gravity and magnetic systems, the **Transmitter** and **Receiver Coordinate System** lists will both be **Absolute: Parallel to Absolute system**, whereas in MT, the magnetotelluric option will be selected.

For more information on the coordinate systems in EMIGMA, see Coordinate System in the **Specifying Output** section.

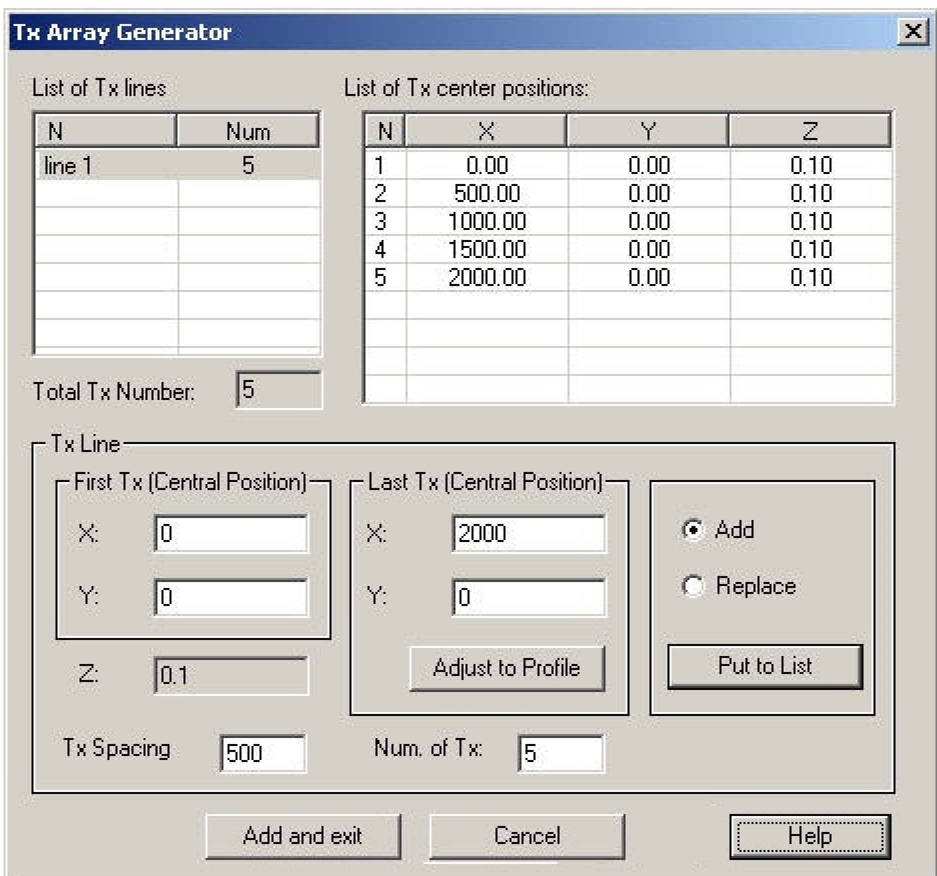
## Multiple Tx Generator

Add a group of evenly spaced transmitters or receivers using the **Multiple Tx Generator**. This option changes to the **Multiple Rx Generator** for CSEM systems.

In the **Tx/Rx Replacement Mode** section on the waveform tab, enable the **Multiple Tx Generator** by selecting the **Replace** option and then checking the box labelled **Multiple Tx Generator**.

*Note: Multiple transmitters can only be generated for a fixed system.*

Click the **Transmitter Input** button to launch the **Tx Array Generator** interface:



The **Tx Line** section lets you enter the positions of the first and last transmitters

**Tx Spacing** specifies the distance between two adjacent transmitters

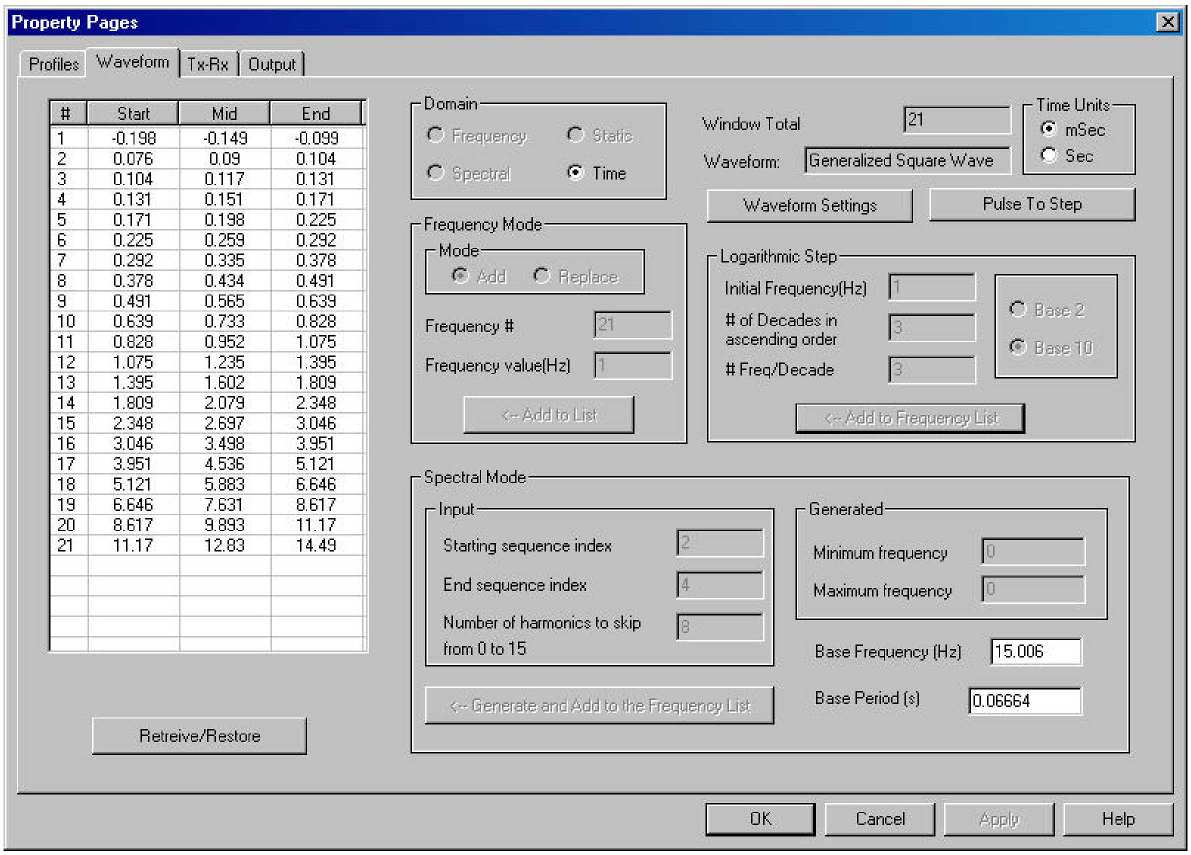
**Num. of Tx** is the number of transmitters to be generated.

Click **Put to List** to generate a group of transmitters. The locations of the transmitters will appear in the **List of Tx center positions**. The group is given a name and added to the **List of Tx lines**. The **List of Tx center positions** will be updated when clicking on entries in the **List of Tx lines**.

Click **Add and exit** to add the new transmitters to the survey and return to the main Tx-Rx page.

## Specifying Waveform

In the **Waveform** tab of the **Property Pages** dialog, you can specify waveform information. If you are in the process of designing a survey from scratch, move over to this page after having specified the system geometry on the [Tx-Rx] tab. Otherwise you can reach this page by clicking the **Configuration** button in the [Database] dialog.




**Waveform** tab of the **Property Pages**

When creating a survey, you can specify only the frequency-domain and spectral mode settings. If your data are magnetic or gravity, you will have the **Static** option selected for you automatically, with all the rest sections of the dialog being inaccessible. If you want to model a time-domain system, you have to specify the spectral mode, run the forward simulation and then transform your simulated spectral data to time-domain. The time-domain option will also be available for imported time-domain surveys with measured data.

**Related Topics**

Specify Frequency-Domain Mode

Specify Spectral Mode

Specify Time-Domain Mode

**Specify the frequency-domain mode**

This mode is useful for direct frequency-domain calculations as it allows a list of desired frequencies to be inputted directly. Select **Frequency** in the **Domain** section of the **Waveform** tab. This will activate the **Frequency Mode** and the **Logarithmic Step** sections.

In the **Frequency Mode** section:

- Leave the **Add** option on to create a new frequency in the list of frequencies to the left

If you are creating your first frequency, the #="**Frequency #**" number box will be set automatically to 1.

- Specify the value of your first frequency in the respective box below and click **Add to List**.

The consequent number and value of frequency will be added to the table, whereas the **Frequency #** and **Frequency Value** boxes will switch to the next number and value. Create as many frequencies as needed

- To replace a frequency, select the frequency to change from the list of frequencies, turn the **Replace** button on, specify the value of the frequency in the respective box and click **Add to List**
- To delete a frequency, select it from the list of frequencies and press the **Delete** key.

In the **Logarithmic Step** section:

- Choose the initial frequency in the respective box
- Specify the number of frequency decades in the ascending order in the respective box and the number of steps within one decade in the **#Freq/decade** box.
- Click **Add to the Frequency List**. In the example below, the initial frequency is 10.00, the number of frequency decades is 3 (10-100, 100-1000, 1000-10000) and the number of steps is 3, as well:

| # | Freq |
|---|---|
| 1 | 10.000 |
| 2 | 21.5443 |
| 3 | 46.4159 |
| 4 | 100.000 |
| 5 | 215.4430 |
| 6 | 464.1590 |
| 7 | 1000.000 |
| 8 | 2154.4399 |
| 9 | 4641.5898 |
| 10 | 10000.000 |

- To change the step, type in your new values in the above boxes and click **Add to the Frequency List**. The message will warn you that the previous data will be lost. Click **OK** to display a new list of frequencies.

**Specify the spectral mode**

The spectral mode was designed to provide faster time-domain simulations. Spectra are generated at a fixed number of frequencies, which are sampled linearly in every decade. After having passed forward simulation, they are subject to the standalone fast frequency to time domain transform (see **FSEMTRS**).

- Select **Spectral** in the **Domain** section. The **Generate Spectral Suite** dialog opens

- Specify the base frequency in this dialog and click **OK**.

  The frequencies will be added automatically to the list on the left. The **Input** section of the **Waveform** tab will display the starting and ending spectral sequence indices dependently of the base frequency you indicated. For example, if your base frequency is less than 20, you will automatically have 1 and 3 set for the starting and ending sequences, respectively; if the base frequency is over 20, these indices will be 2 and 4. The number of harmonics will be set to 8 by default. It means that each sequence (decade) will contain 12 frequencies.

- If necessary, change the spectral settings and click **Generate and Add to the Frequency List**

  In the **Generated** section, you will see the minimum and maximum frequencies dependent on the index of the first and last sequences as well as the base frequency and base period.

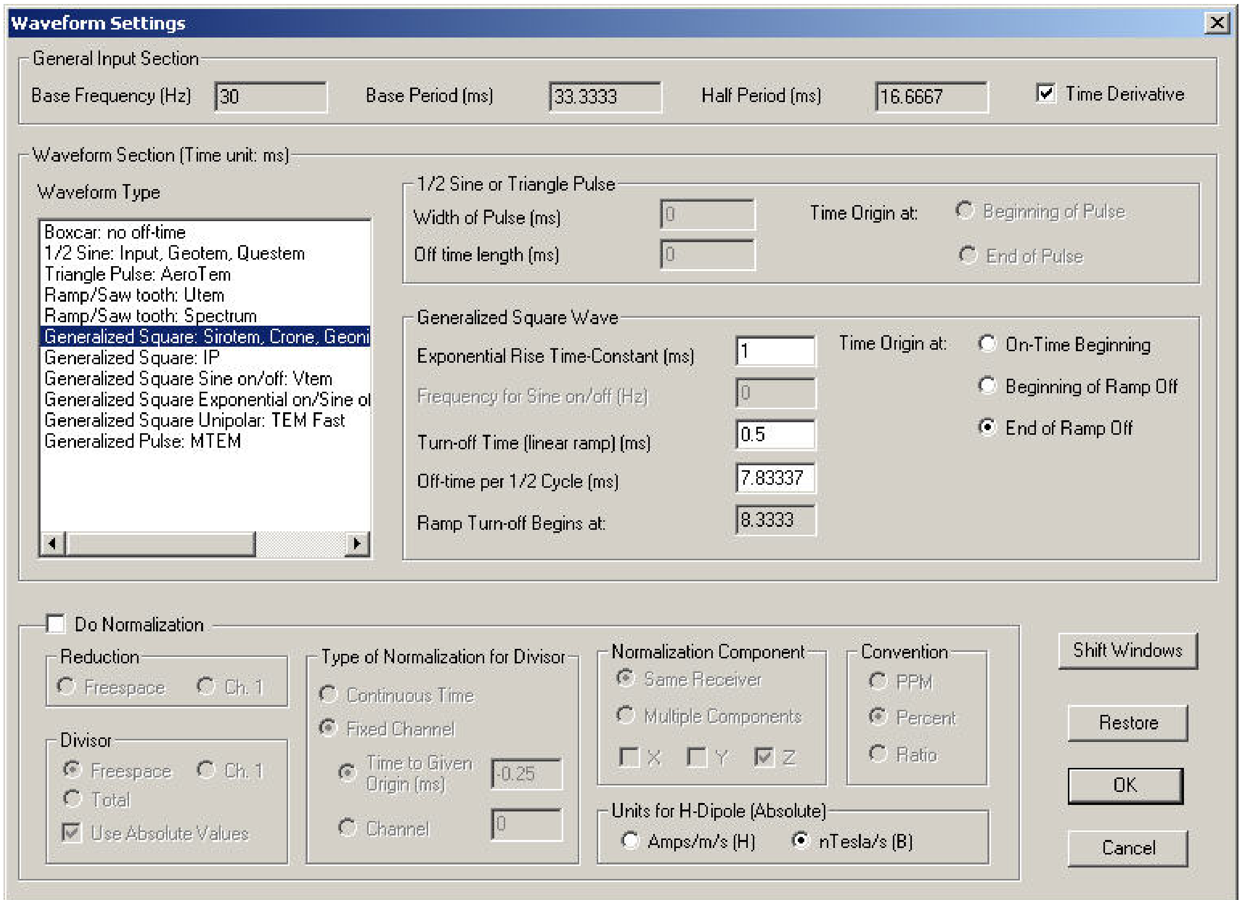
**Specify the time-domain mode**

If you are using the property pages to create a survey, it will not be possible to select the Time Domain option. This option will only be available for spectral surveys that have been converted to time domain using the Transform tool and time domain surveys with measured data that have been loaded into the EMIGMA database.

On this page it is possible to change the units for the time windows between milliseconds and seconds.

The **Pulse To Step** function will convert the data from derivative to non derivative data. There are three algorithms to choose from. After processing, click the **Apply** button at the bottom of the window and a new survey with the converted data will be added to the database.

The **Waveform Settings** button will give the ability to edit the properties of the waveform. It launches the following window:




The **General Input Section** displays some basic information about the waveform.

The **Time Derivative** checkbox indicates whether the current survey contains derivative data or not. This information about the data can also be determined by looking at the **Units for H-Dipole** section at the bottom of the window. The available units will be Amps/m and nTesla for non-derivative data.

The **Waveform Section** contains more information about the waveform currently being used. It is possible to change the waveform type as well as the parameters that describe its shape. The **Time Origin** used for the time windows can also be selected in this section.

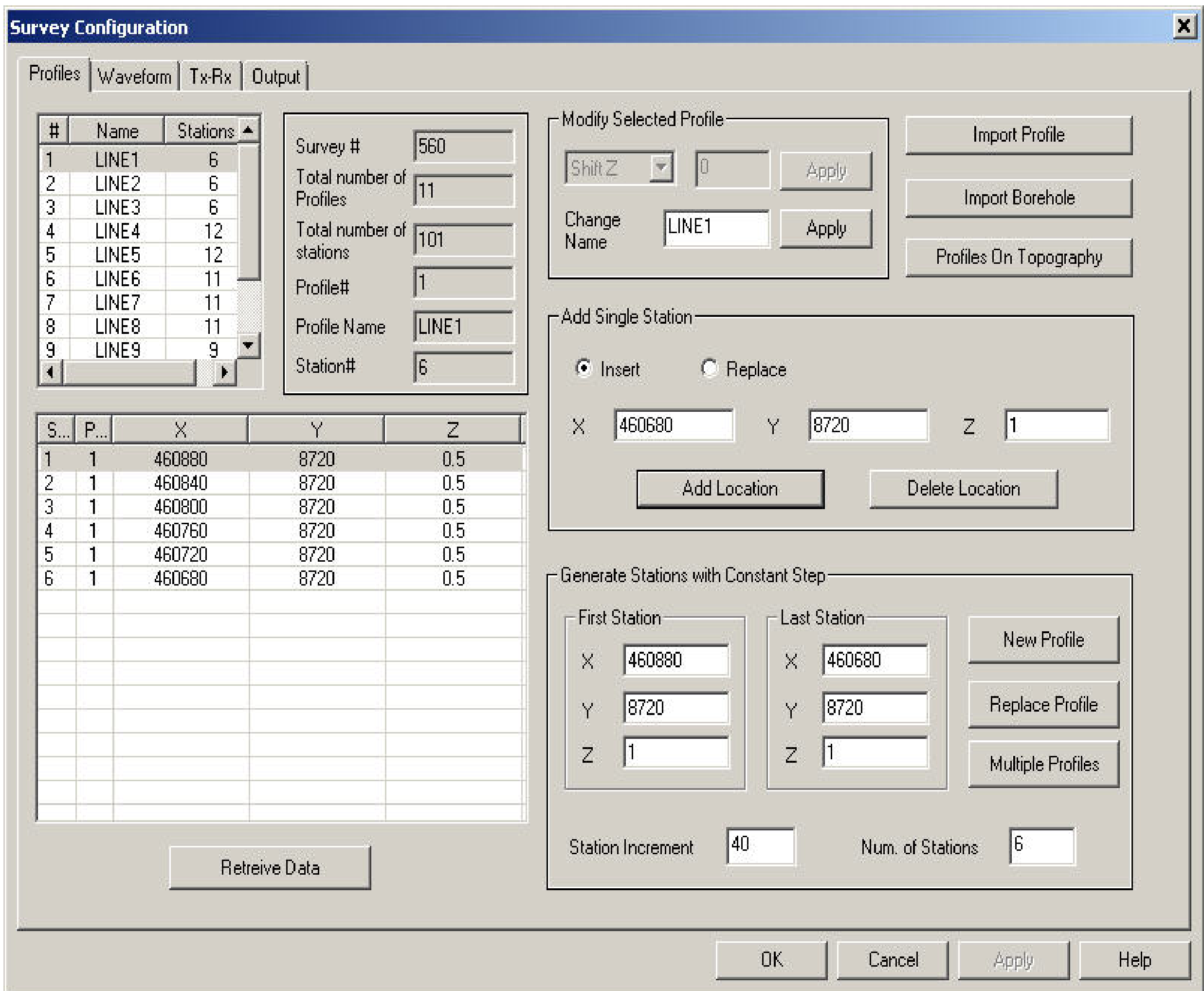
The **Normalization** section is enabled by selecting the **Do Normalization** checkbox. The controls that are enabled are used to specify how the normalization is to be performed.

To change the position of all the time windows, click the **Shift Windows** button and enter the amount of time in milliseconds to apply a shift to the current time windows.

To return to the original settings, press the **Restore** button.

# Specifying Profile Information

You can specify profile information in the **Profiles** tab of the **Property Pages** dialog. If you are in the process of creating a new project/survey, fill out this tab after having specified system geometry and waveform. In future, to reach this page, select your project/survey and click the **Configuration** button in the bottom of the [**Database**]() dialog.




**Profiles** tab of the **Property Pages**

## Related Topics

[Add/Delete Profiles]()

[Rename a Profile]

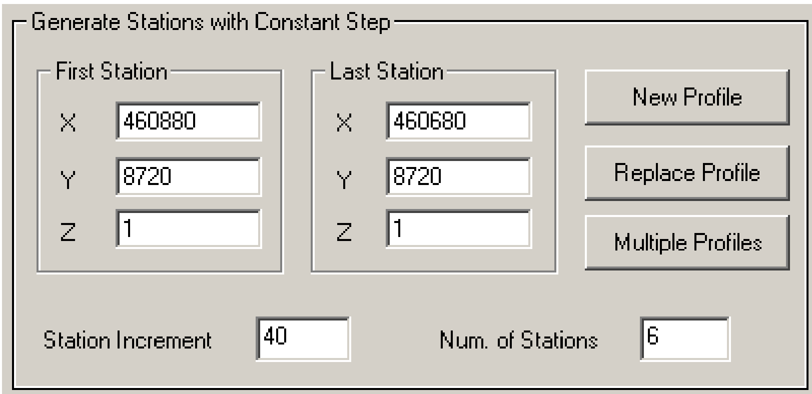
[Replace a Profile]

[Import a Profile]

[Import Topography]

## Add a new profile

In the **Generate Stations with Constant Step** section of the **Profiles** tab:




- Specify the **X**, **Y** and **Z** coordinates of the first station and click in the respective boxes for the last station to generate its coordinates automatically. However you can always change them manually as required
- Specify the step in the **Station Increment** box and click in the **Num. of Stations** box to change the number of stations accordingly
- Click **New Profile**.

The new profile will appear in the table in the top left-hand corner of the Profiles tab (*List of Profiles)*. The spreadsheet-like table below will show all the stations and their coordinates (*List of Stations)*.

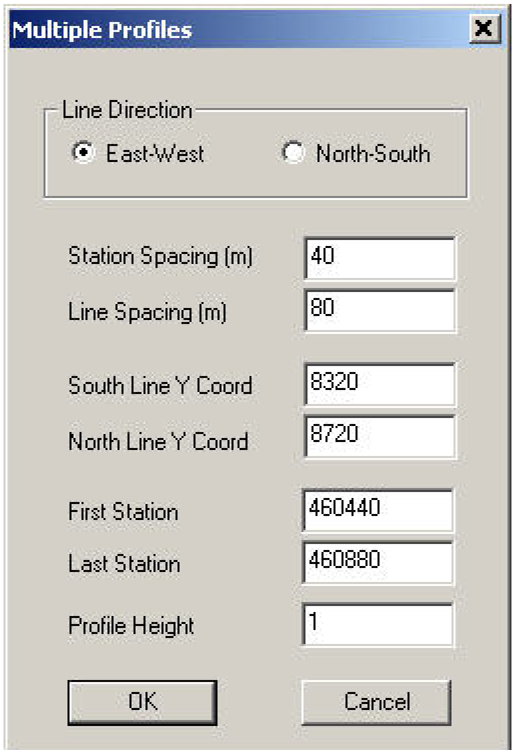
Information on each profile can be viewed in the section to the right of the *List of Profiles*. Select a required profile in the *List of Profiles* and this information will be displayed automatically.

## Delete a profile

Select a profile to delete in the *List of Profiles* and press the Delete key

## Add multiple profiles

The current profiles will be replaced with regularly spaced profiles that you specify. Click the **Multiple Profiles** button. The following interface appears:



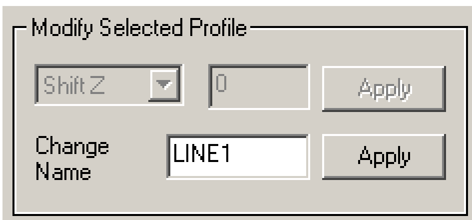

You can select the **Line Direction** by clicking on one of the two options at the top of the window.
**Station Spacing** and **Line Spacing** refers to the distance between the coordinates.
The next four values specify the positioning of the profiles. **Profile Height** is the value given to z.

### Rename a profile

The profile number will be generated automatically, starting with Line 0 for the first profile

- Select the profile whose name you want to change in the list of profiles in the top left-hand corner of the **Profiles** tab

Its name will be displayed in the **Change Name** box of the **Modify Selected Profile** section of the tab:




- Replace the former name with a new one and click the **Apply** button to the right. The name of the profile you selected in the table will change respectively

## Replace a profile

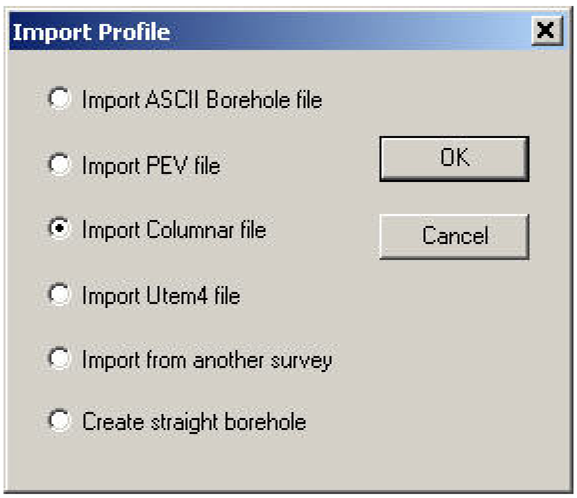
- Select the profile you want to replace from the list of profiles in the top left-hand corner of the **Profiles** tab
- Edit the coordinates, station increment and number of stations as required in the **Generate Stations with Constant Step** section of the tab and click **Replace**

The profile you selected will be replaced with the one you edited. If necessary, change its name: see Rename a Profile.

## Import Profiles

Click **Import Profile** in the top right-hand corner of the **Profiles** tab

The **Import Profile** dialog will open offering you to select one of the six options available:




**Related Topics:**

Import ASCII Borehole Files

Import PEV Files

Import Columnar Files

Import Utem4 Files

Import from another survey

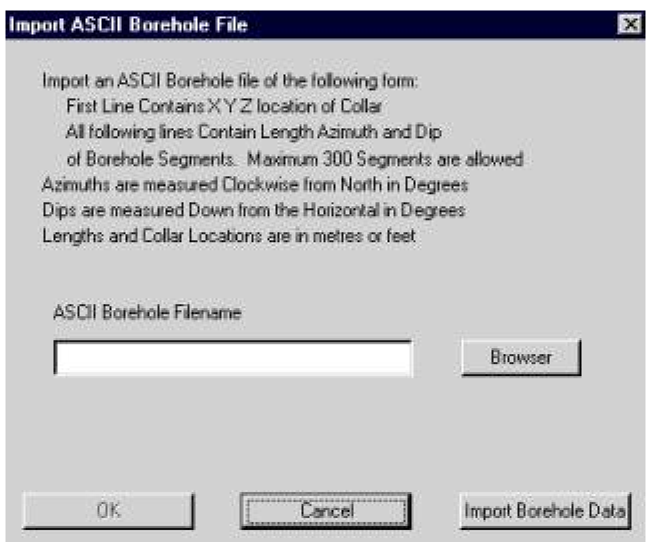
Create straight borehole

# Import ASCII Borehole File

To be efficiently imported, your file should be a text file of the following appearance:

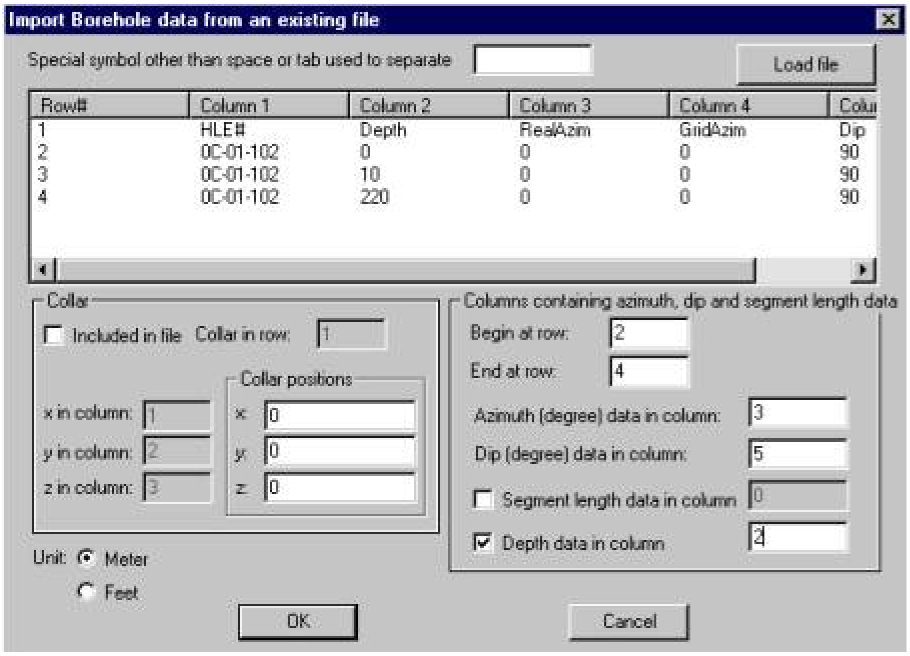

- Select **Import ASCII Borehole File** in the [Import Profile] dialog and click **OK**

  The **Import ASCII Borehole File** dialog appears describing the format of the file to be imported:




- Click **Browser** to open the **Borehole Data File** dialog, a standard Windows-style dialog, find and select a required file and and click **Open**. Your filename will appear in the **ASCII Borehole Filename** field in the **Import ASCII Borehole File** dialog
- Click **Import Borehole Data**

The **Import Borehole Data from an Existing File** dialog appears:




- If in the file you are importing, a symbol other than a space or tab is used to separate data, specify this symbol in the upper box and click **Load File.** The table below will show the data you are about to import

In the **Collar** section of the dialog:

- If your data contain collar information, leave checked the **Included in File** box (it is checked by default) and specify the row and columns containing this information
- If your data do not contain collar information, de-select the **Included in File** box and specify the collar position in the respective section

In the **Columns containing azimuth, dip and segment length data** section:

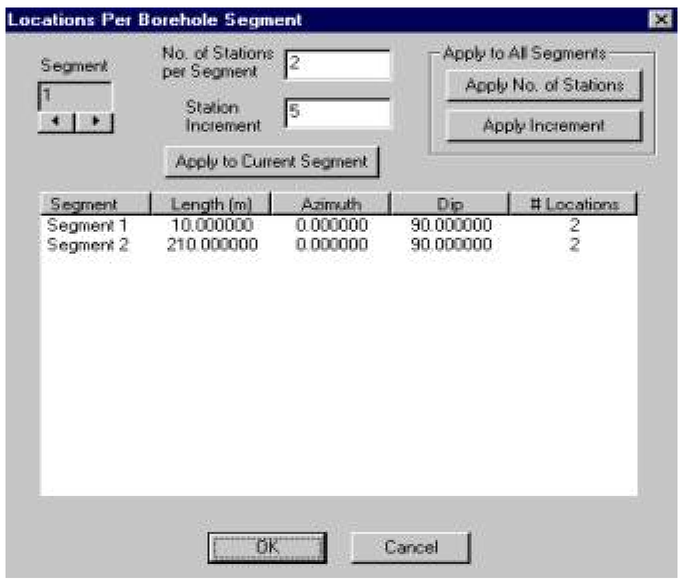
- Specify the columns containing azimuth and dip information

- Select between depth and segment - two ways of determining location coordinates in a borehole

- Specify the units (meters or feet) in the bottom left-hand corner of the dialog, and

- Click **OK**.

  The **Import ASCII Borehole File** dialog reappears. Click **OK** in this dialog to confirm import and proceed to the **Locations per Borehole Segment** dialog:




In this dialog, to change data of a segment:

- Select a segment in the table or from the **Segment** list using the scroll arrows
- Specify the number of stations per segment in the respective box and click **Apply to Current Segment**. The segment data in the table will change accordingly as well as the step in the **Station Increment** box

  You can also choose to edit the station increment, which will lead to the respective change in the number of stations per segment and the table data

To change data of all segments:

- Specify the number of stations per segment in the respective box and click **Apply No. of Stations** in the **Apply to All Segments** section

OR

- Specify the station increment in the respective box and click **Apply Increment** in the **Apply to All Segments** section

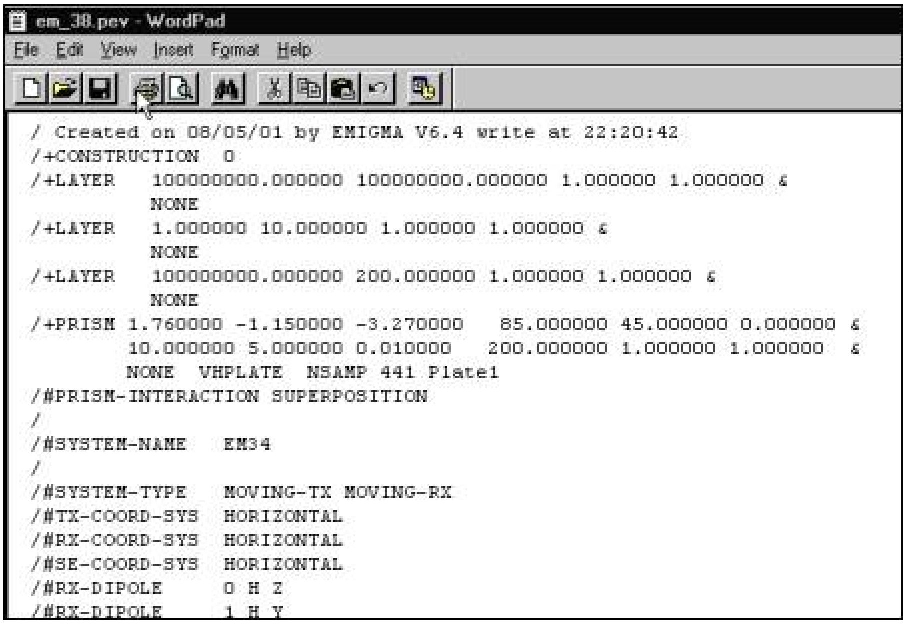
  As a result all segments will have the same number of locations and step

- Click **OK** to complete the import procedure and return to the **Profiles** tab of the **Property Pages** dialog

Your ASCII profile will be added to the list of profiles in the upper left-hand corner of the tab. If necessary, change its name as described in Rename a Profile.

# Import PEV files

PEV is originally the PetRos EiKon format. PEV files can be imported, for example, from GeoTutor. In WordPad, they have the following appearance:




```
/ Created on 08/05/01 by EMIGMA V6.4 write at 22:20:42
/+CONSTRUCTION  0
/+LAYER   100000000.000000 100000000.000000 1.000000 1.000000 &
          NONE
/+LAYER   1.000000 10.000000 1.000000 1.000000 &
          NONE
/+LAYER   100000000.000000 200.000000 1.000000 1.000000 &
          NONE
/+PRISM 1.760000 -1.150000 -3.270000   85.000000 45.000000 0.000000 &
        10.000000 5.000000 0.010000   200.000000 1.000000 1.000000  &
        NONE  VHPLATE  NSAMP 441 Plate1
/#PRISM-INTERACTION SUPERPOSITION
/
/#SYSTEM-NAME   EM34
/
/#SYSTEM-TYPE   MOVING-TX MOVING-RX
/#TX-COORD-SYS  HORIZONTAL
/#RX-COORD-SYS  HORIZONTAL
/#SE-COORD-SYS  HORIZONTAL
/#RX-DIPOLE     0 H Z
/#RX-DIPOLE     1 H Y
```

- Select **Import PEV file** in the **Import Profile** dialog and click **OK**

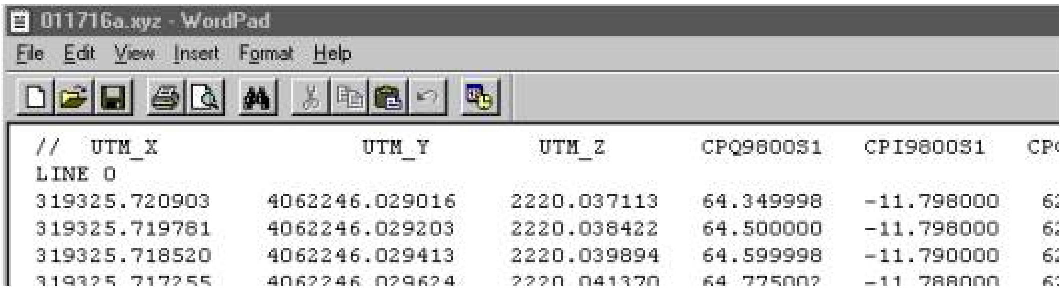
  This will bring up the **Import ASCII File with Profile Data** dialog, a standard Windows-style dialog for searching and opening files

- Select a required .pev file and click **Open**

  The profile data from the file will be imported into the list of lines in the upper left-hand corner of the **Profiles** tab automatically.

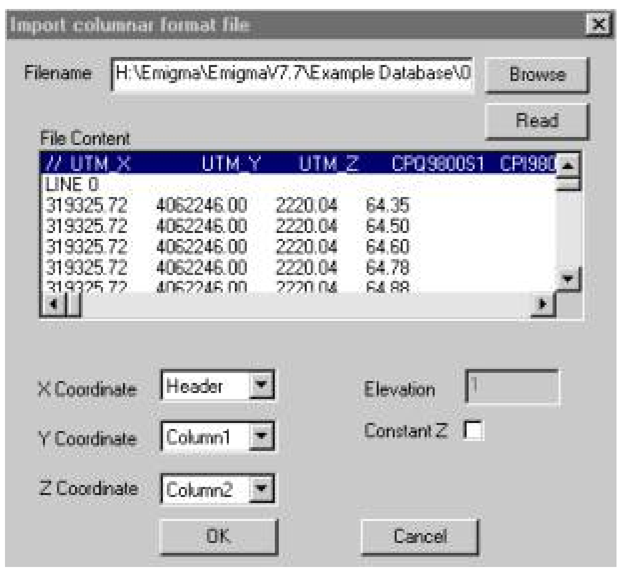
## Columnar files

Columnar files have an xyz format and in WordPad they look as follows:




- Select **Import Columnar File** in the [**Import Profile**]() dialog and click **OK**

  This will display the **Import columnar format file** dialog:




- Click **Browse** to search for a file to import. In a standard Windows-style dialog to open, select a required file and click **Open**

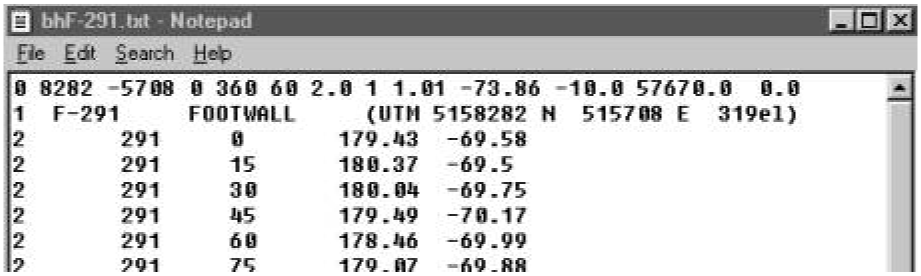
  The selected filename will appear in the respective field of the **Import columnar format file** dialog

- Click **Read** to display the profile information in the **File Content** field

- Select the columns for X and Y coordinates from the respective dropdown lists; in the present version, **Header** stands for the first column
- If you want the Z coordinate to be constant, leave the **Constant Z** box checked (it will be checked by default). If you want it to be taken as is, de-select the **Constant Z** box. The **Z Coordinate** dropdown list will become active
- From this list, select the column for the z coordinate to appear
- Click **OK** to complete import and see the results in the **Profiles** tab.

# Import UTEM4 format

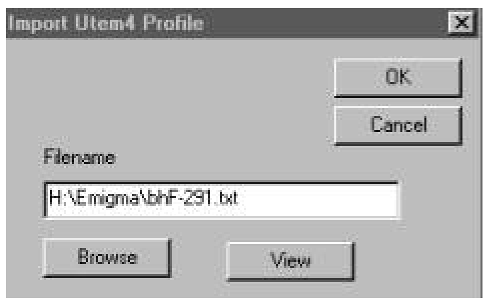
UTEM4 files contain borehole data in the following format:




```
0 8282 -5708 0 360 60 2.0 1 1.01 -73.86 -10.0 57670.0  0.0
1  F-291      FOOTWALL      (UTM 5158282 N  515708 E  319el)
2        291      0        179.43  -69.58
2        291      15       180.37  -69.5
2        291      30       180.04  -69.75
2        291      45       179.49  -70.17
2        291      60       178.46  -69.99
2        291      75       179.07  -69.88
```

- Select **Import UTEM4 file** in the **Import Profile** dialog and click **OK**

  This will bring up the **Import UTEM4 Profile** dialog:

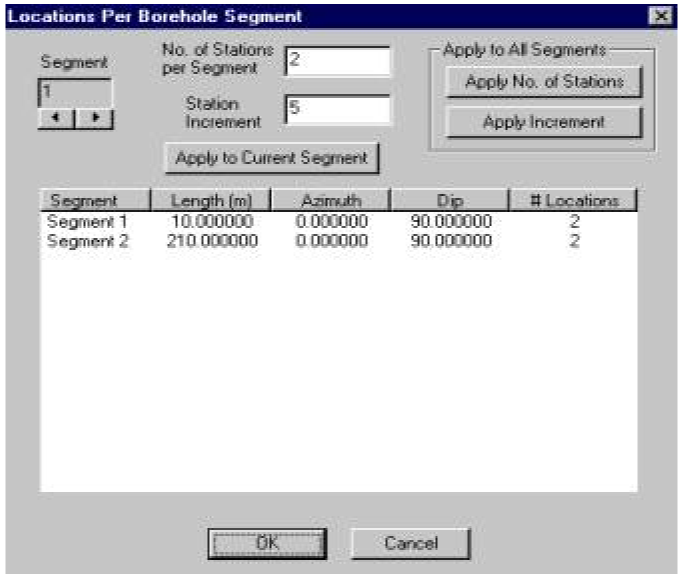



- Click **Browse** to display the **UTEM Data File dialog**, a standard Windows-style dialog for searching and opening files. Select a required file and click **Open**

  The selected filename will appear in the respective field of the **Import UTEM4 Profile** dialog

- Click **View** to open the file in its original (text) format

- Click **OK** to proceed to the **Locations per Borehole Segment** dialog:




To change data of a segment:

- Select a segment in the table or from the **Segment** list using the scroll arrows

- Specify the number of stations per segment in the respective box and click **Apply to Current Segment**. The segment data in the table will change accordingly as well as the step in the **Station Increment** box

  You can also choose to edit the station increment, which will lead to the respective change in the number of stations per segment and the table data

To change data of all segments:

- Specify the number of stations per segment in the respective box and click **Apply No. of Stations** in the **Apply to All Segments** section

  OR

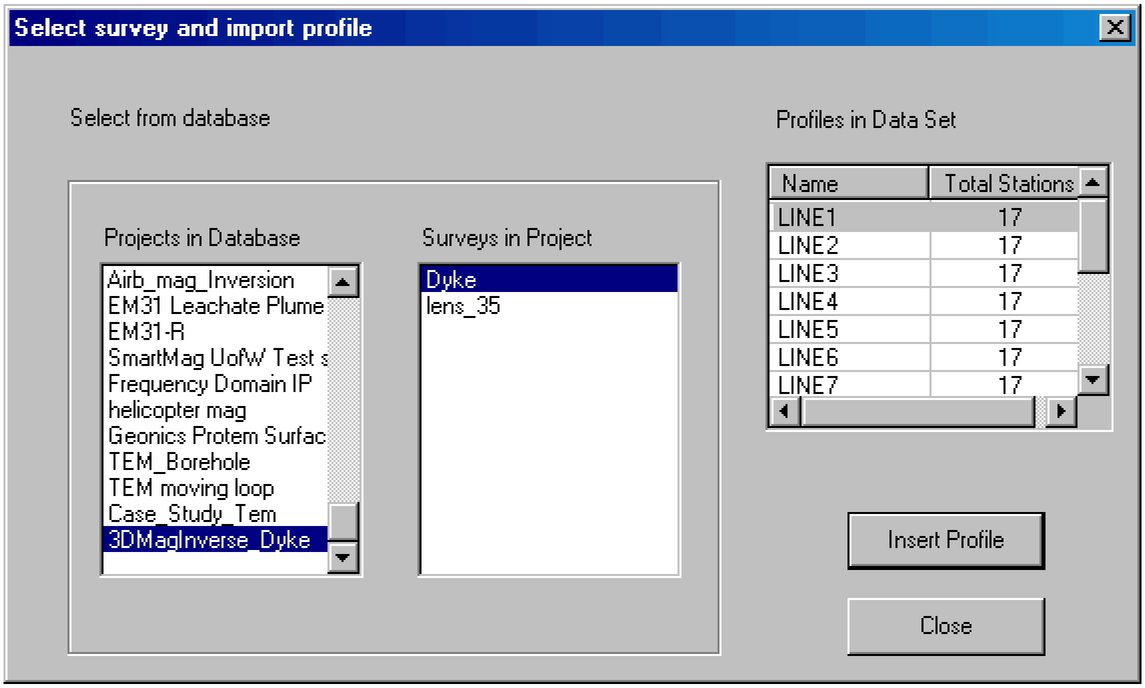
- Specify the station increment in the respective box and click **Apply Increment** in the **Apply to All Segments** section

  As a result all segments will have the same number of locations and step, each changed accordingly relative to the other

- Click **OK** to complete the import procedure

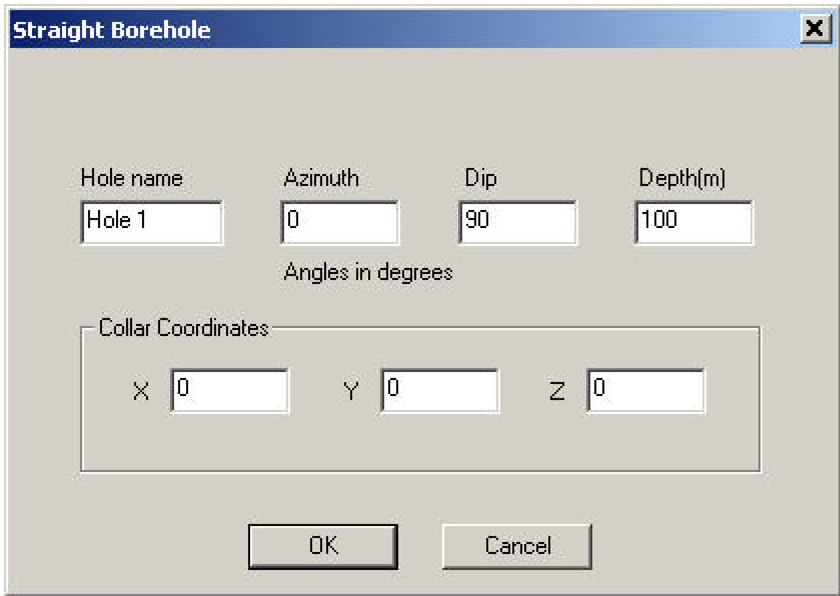
  The profile data from the file will be imported into your **Profiles** tab of the **Property Pages** dialog. You can see them displayed in the tables (list of profiles and list of stations), with the coordinates of each location computed based on azimuth and dip.

## Import from another survey




- Select the project and survey from the respective lists and the entries in the **Profiles in Data Set** list will be updated accordingly.
- Select the profiles you would like to import and click **Insert Profile**

## Create straight borehole

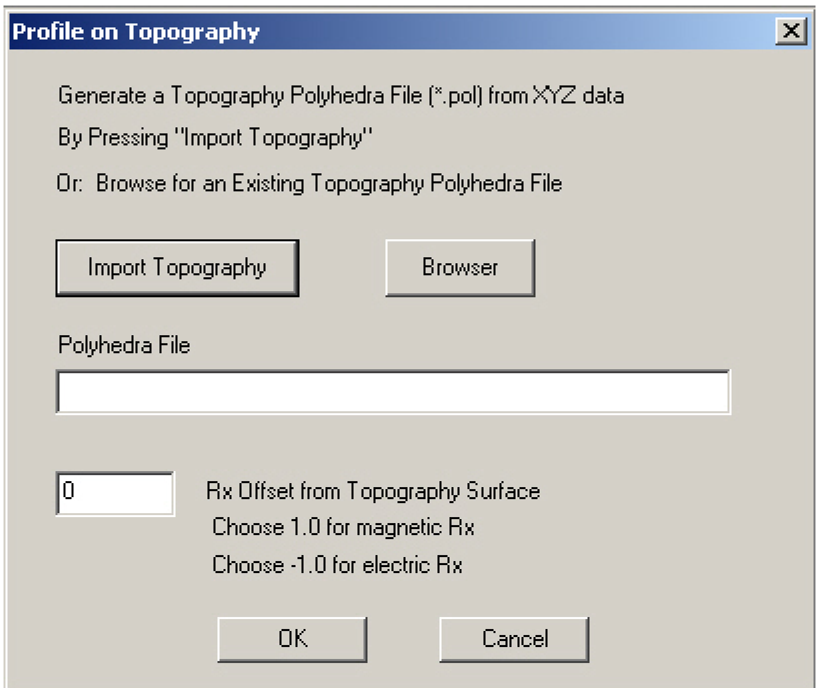



- Specify the **Hole name**, **Azimuth** in degrees measured from clockwise from north, **Dip** in degrees measured down from the horizontal, **Depth** in metres
- You may also specify the **Collar Coordinates**
- Click **OK** and the hole will be added to the profile list.

## Import topography

- Click the **Profiles on Topography** button in the upper right-hand corner of the **Profiles** tab

  The **Profile on Topography** dialog will open:

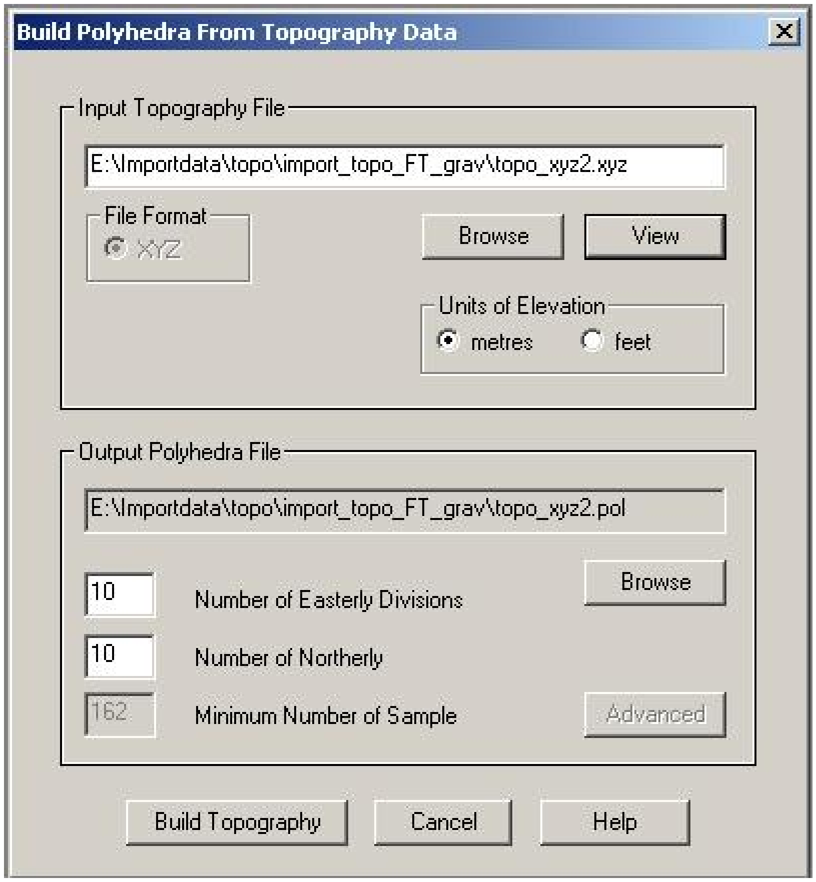



- Click **Browse** to find an existing polyhedra file (.pol)

  OR

- Click **Import Topography** to create a polyhedra file from an available topography (.xyz) file

  In the latter case, the **Build Polyhedra from Topography Data** dialog appears:




- Click **Browse** in the upper **Input Topography File** section to display the **Topography Data File** dialog, a standard Windows-style dialog for searching and opening files. Select a required .xyz file and click **Open**

  **See Also** Topography file format

  The filename will appear in the **Input Topography File** field. Simultaneously, a polyhedra file (.pol) will be generated and saved in the same directory, as seen in the example. Its path will be written in the **Output Polyhedra File** field of the dialog

- In the **Input Topography File** section:
- To see the input topography file in the WordPad format, click **View**
- Select the units of elevation

- In the **Output Polyhedra File** section:
- Specify the number of Easterly or Northerly divisions (conventional units into which a polyhedron is divided for the simulation purposes) in the respective boxes. The minimum number of samples below will change accordingly. To properly carry out simulation, this number cannot exceed 250 (for ILN) and 1000 (for LN)
- Click **Build Topography**

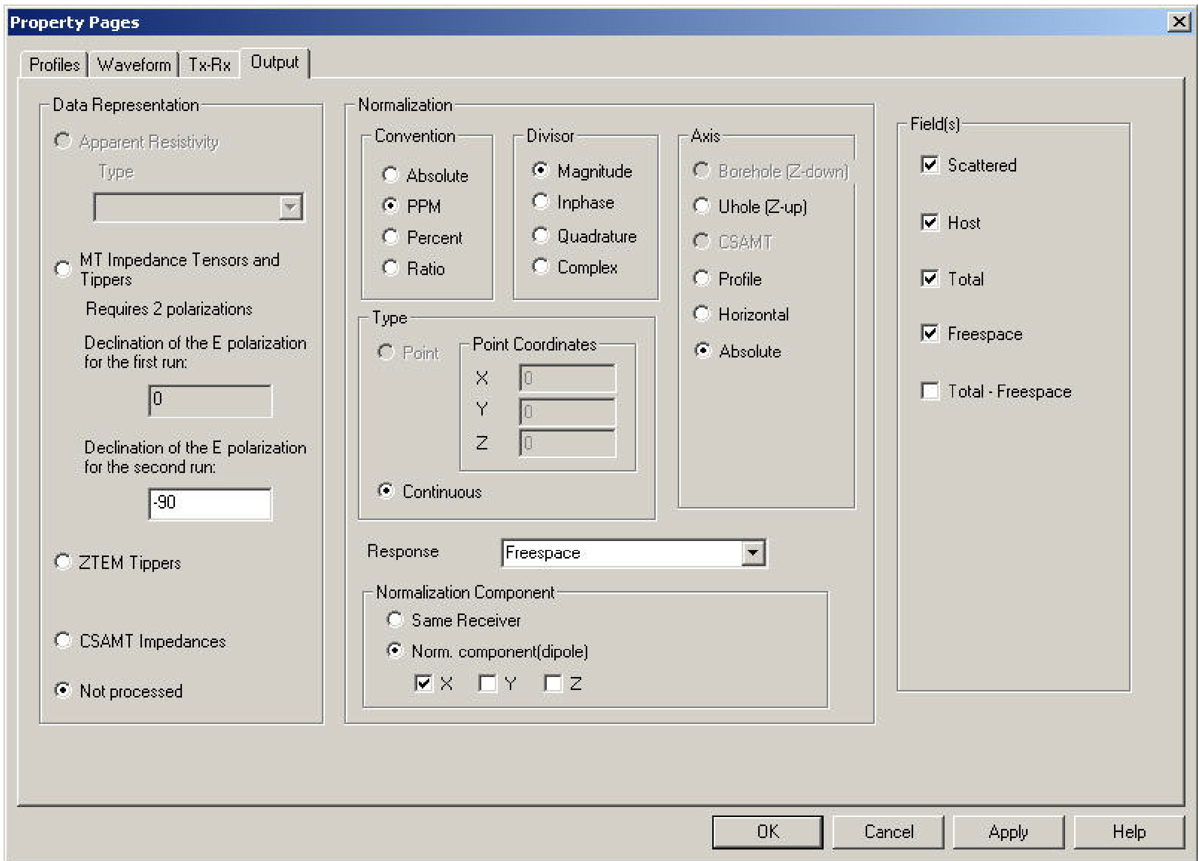
This will bring back the **Profile on Topography** dialog, with the newly created .pol file displayed in the **Polyhedra File** field

- Specify the receiver offset from the topography surface and click **OK**

Back in the **Profiles** tab, view the changes in the *Z* column of the list of stations (the table in the bottom left-hand corner)

# Specifying Output

The parameters related with data representation, normalization and fields can be set in the **Output** tab of the **Property Pages** dialog:




**Output** tab of the **Property Pages**

If you are in the process of designing a survey and have already specified your system geometry, waveform mode and profile information, this tab will be filled out automatically. However you will be able to edit settings, if necessary. In frequency-domain systems, you can change settings in all sections. In magnetic or gravity systems, where no normalization is used, the **Normalization** section will not be accessible, but you will be able to specify different fields. In the spectral mode, all the sections of the dialog will be disabled. In this case, normalization is performed in FSEMTRS, and since the choice of the fields is very specific, it is done for you and cannot be changed.

To reach this tab from the Database dialog, click the **Configuration** button.

**Related Topics**

Output Data Representation

Normalization

Coordinate System

Output Fields

## Output Data Representation

Currently, three options are available in the **Data Representation** section of the **Output** dialog:

1. If you have MT data, the **MT Impedance Tensors** option will be selected for you automatically. The declination of the E polarization boxes will contain the settings you specified in the **Tx-Rx** tab, see Specify an MT system

2. If you have CSAMT data, the **CSAMT Impedances** option will be selected automatically

3. If you have ZTEM data, the **ZTEM Tippers** option will be selected automatically.
4. If you have VLF-R data, the **VLF-R Impedance** option will be selected automatically.
5. If you have VLF data, the **VLF Ratio** option will be selected automatically.

6. In all other cases, the **Not processed** button will be turned on

## Normalization

In the **Normalization** section of the **Output** tab, you can specify normalization convention, normalization type and normalization component.

In the **Convention** subsection of the tab:

- To apply no normalization, click **Absolute**. This will disable all other buttons and boxes in the **Normalization** section

- To apply normalization:

- Select **PPM**, **Percent** or **Ratio** to specify the units of your normalized data representation. All other subsections will become active
- Select the component to normalize your data to in the **Divisor** subsection. You can normalize to the complex field, its magnitude or in-phase and quadrature components by selecting the respective option in this subsection

*__Note__. When you normalize to the magnitude of the field, the sign may be lost and outputs may be reversed from what is expected*

In the **Type** subsection, two types of normalization are available:

- Select **Point** to perform the point normalization

  This type of normalization is, as a rule, calculated using a free-space model with the same transmitter-receiver geometry (moving transmitter-receiver modes) or the same transmitter loop (fixed transmitter modes) as was used in the simulation. In the latter case, specify the coordinates of a point from which to normalize the data

- Select **Continuous** to perform the continuous normalization

This type of normalization is only meaningful in fixed transmitter modes. The normalizing fields are recalculated for each orientation and position occupied by a receiver. Thus the fields are normalized to the fields that would have been present if there had been no anomalous structures. It is especially useful for amplifying anomalies that may be a significant fraction of the normalizing field, but have low amplitude because of a very small host (incident) field.

In the **Normalization Component** subsection:

- Select **Same Receiver** if you want to normalize the field of a receiver to the equivalent free-space field modeled for the same receiver

- Select **Norm.Component (dipole)** to specify one-, two- or three-component normalization. In three-component normalization, the sum of the total free-space field from all components is used as the normalizing field

Since you normalize fields to equivalent free-space fields, **Freespace** will be selected automatically in the **Response** list of the **Normalization** section.

## Coordinate System

You can set a required coordinate system in the **Axis** subsection of the **Output** tab. If you have already done it in the **Tx-Rx** tab while specifying the system geometry (see Specify the Coordinate System), the axes will be set for you automatically. Otherwise, four choices are available: absolute, profile, borehole and horizontal:

- Select **Absolute** to have the orientation of your coordinate system parallel to absolute model coordinates. This coordinate system is the simplest to visualize and is useful for model studies. However it is not very practical for modeling actual field data

- Select **Profile** to have the orientation of your coordinate system parallel to the direction of the profile; at that, the y-axis will be horizontal and on the left-hand side when viewing down the profile and the z-axis will be pointing up and perpendicular to the x- and y-axes. This coordinate system is useful when the orientation of system components is relative to the profile, even if the profile has a variable elevation

- Select **Uhole (Z-Up)** to model a borehole system. You will have the same orientation as in the profile system, but with the z-axis being in the line of the profile. The x-axis will be horizontal, whereas the y-axis will be orthogonal to the two and its projection on the horizontal will be opposite the azimuth direction of the borehole (i.e. it will point in the direction of the collar).

- Select **Horizontal** to have a coordinate system in which the x-axis will be the horizontal projection of the profile, the y-axis is also horizontal and on the left-hand side when viewing down the profile and the z-axis is parallel to the absolute z-axis. Thus, it is just a rotation of the absolute system about the absolute z-axis by an angle equal to the strike angle of the profile. It is useful for modeling systems oriented along the profile, but leveled with respect to true vertical and horizontal.

## Output Fields

In the **Field(s)** section of the **Output** tab, you can specify the field you want to model:

- **Scattered** fields are generated from currents flowing inside prisms, plates or other anomalous structures. In the absence of these structures, no scattered field will be present. Scattered fields can be thought of as "anomalies" sought after in exploration geophysics

- **Host** (incident) fields are reflected from the electrical property contrasts in the layered earth.

EMIGMA offers the choice of outputting the sum of scattered and host fields. It is especially useful when you want to simulate the signal from surface or airborne prospecting systems that remove the primary field.

- **Total** fields include scattered and host fields plus the field transmitted directly from the transmitter to the receiver through the layer interfaces (if the transmitter and the receiver are in different layers) or the field that could be measured were the transmitter and the receiver in a uniform whole space (if the transmitter and the receiver are in the same layer). In other words, the total field represents the entire signal sensed by the receiver and is often useful for simulating surface-to-borehole and cross-hole surveys.

- **Freespace** fields are generated as the response of the system to an absolutely resistive environment or vacuum. In EMIGMA, both half-spaces are given the same resistivity ($10^8$ Ohm.m), so in effect the free-space field is the response to the system in a whole-space air model.

- **Total minus Freespace** field is useful for simulating some borehole surveys

In surface and airborne surveys, one would expect that the total minus freespace output should be equivalent to the sum of the scattered and host fields. In principle, it is true, but numerical round off errors can make difference, and the calculation of the sum of the scattered and host fields from the difference of the total and free-space fields can yield poor results.

## Saving System Configuration

After having specified your system geometry, profile information, waveform mode and output, click **Apply** in the bottom of the **Property Pages** dialog to save your configuration. If you are in the process of creating a survey from scratch, you are most likely to do it from the Output tab of the dialog. However if you decide to edit your configuration later on, you can save it from any of the four tabs.

Your newly created configuration saved, the **Model Configuration** dialog opens offering you to build a model.

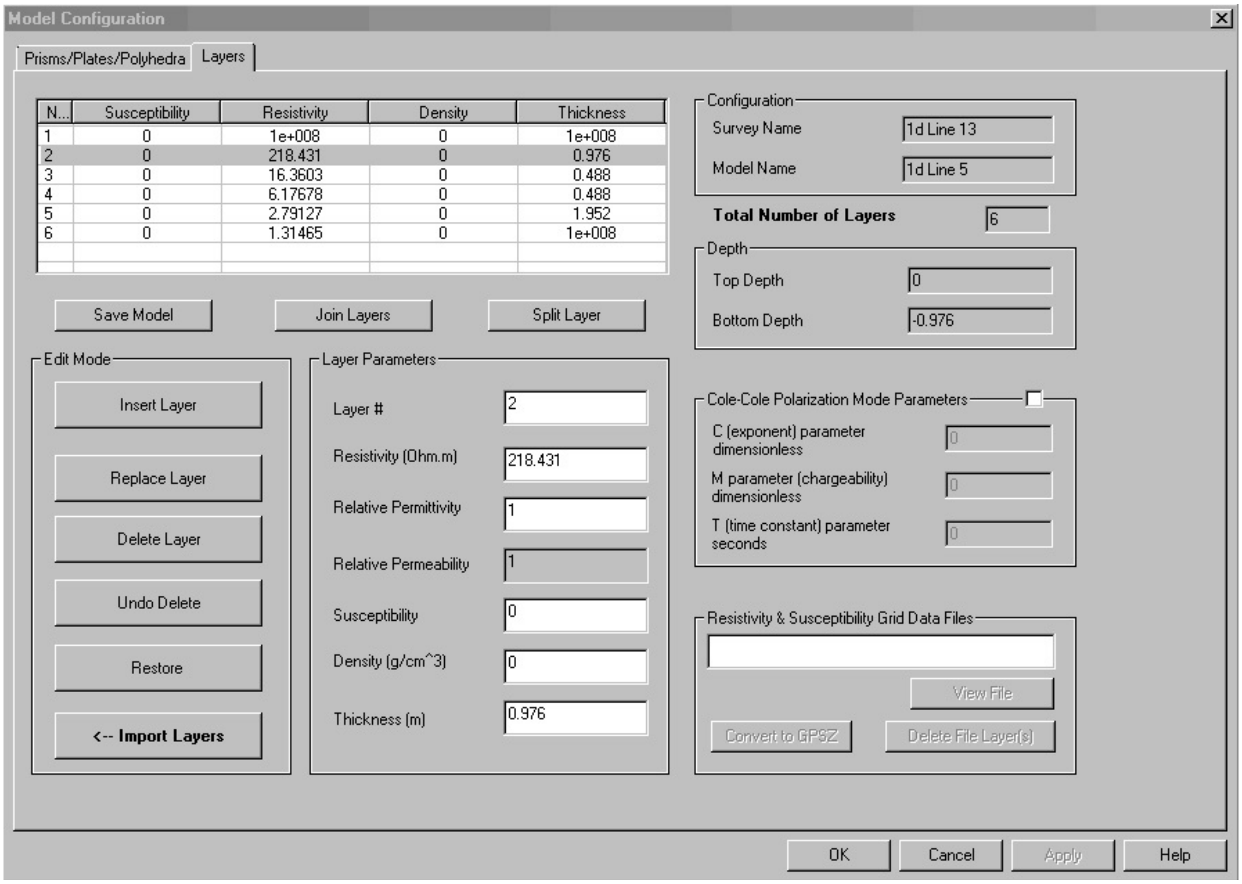
# Model Configuration

## Specifying the Number and Properties of Layers

In the **Layers** tab of the **Model Configuration** dialog, you can define the settings of a layered-earth model to simulate the general background structure of the geology. This model is conceptually simple and accounts for such geological effects as weathering, lateralization, overburden and conductive ground water.

To build such a model, one has to specify the number of layers and assign certain physical properties (conductivity, magnetic permeability and electrical permittivity) to each layer. There is an option to include Cole-Cole polarization parameters in each layer, as well. The model always consists of two half spaces with a number of layers sandwiched in between. For gravity data, you can insert a top layer to apply a dc shift to the simulated data.

If you are creating your survey from scratch, the **Layers** tab will open automatically after you have specified and saved system configuration. In its top left-hand corner, you will see a list of layers containing one layer by default. It is the upper half space (usually air except for special surveys such as marine or underground). The index (**N**) of the upper half-space will always be 1, whereas the index of the lower half-space (ground) to be added will depend on the total number of layers requested in the model. Since the half-spaces are compulsory, the list of layers is to contain no less than 2 items.




**The Layers tab**

To reach the **Layers** tab, click the **Model** button near the bottom of the main [Database](#) interface.

**Save Model**: Exports the model to a ascii file.

**Join Layers**: Select 2 or more layers and when join is selected the resulting layer is the combined thickness with the resistivity weighted by the thickness of the initial layers

**Split Layer: Splits the selected layer into 2 layers of the same resistivity and with combined thickness of the original layer.**

**For a more complete explanation, please see the *Forward_Modelling* tutorial in the /tutorials directory.**

**Related Topics:**

[Insert a Layer]

[Replace a Layer]

[Delete/Restore a Layer]

[Import a Layer]

[Build the Cole-Cole model]

[View Resistivity & Susceptibility Grid Data Files]

[Save a Layer Model]

[Join Layers]

[Split a Layer]

## Insert a layer

In the **Layers** tab, the list of layers will contain the first half-space by default. To add a new layer:

- Specify the index (N) and properties (resistivity, permittivity and susceptibility) of the new layer in the **Layer Parameters** section and click **Insert Layer** in the **Edit Mode** section

  The new layer will appear in the row as specified by the index

**Replace a layer**

In the **Layers** tab:

- From the list of layers in the upper left-hand corner of the dialog, select the layer you want to replace and change the properties in the **Layers Parameters** section as required
- In the **Edit Mode** section, click **Replace Layer**
- Check the properties of the layer you replaced in the table.

## Delete/restore a layer

In the **Layers** tab:

- From the list of layers, select the layer you want to delete and click **Delete Layer** in the **Edit Mode** section

  The layer will be removed from the list

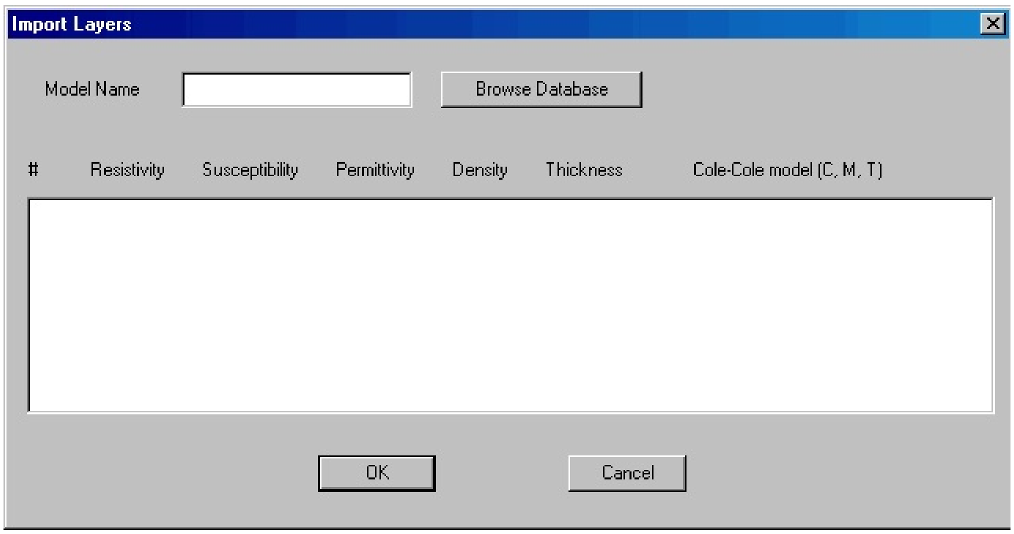
- To restore the layer, click **Undo Delete**.

  The layer will reappear in the list

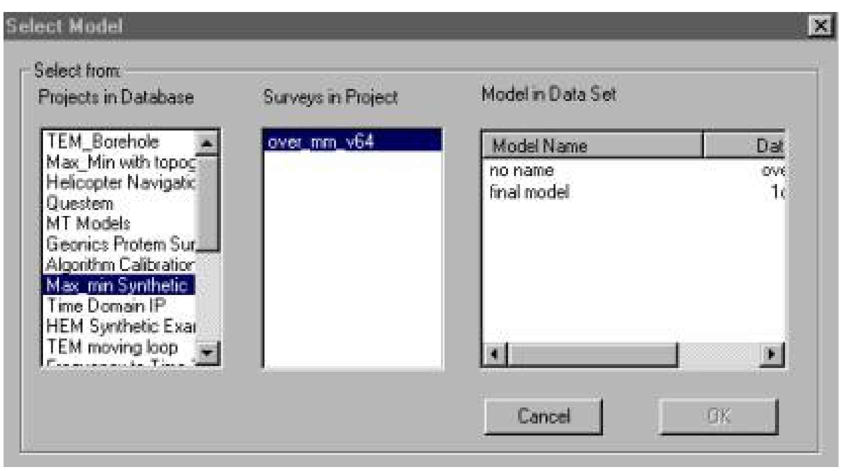
## Import a layer

In the **Layers** tab:

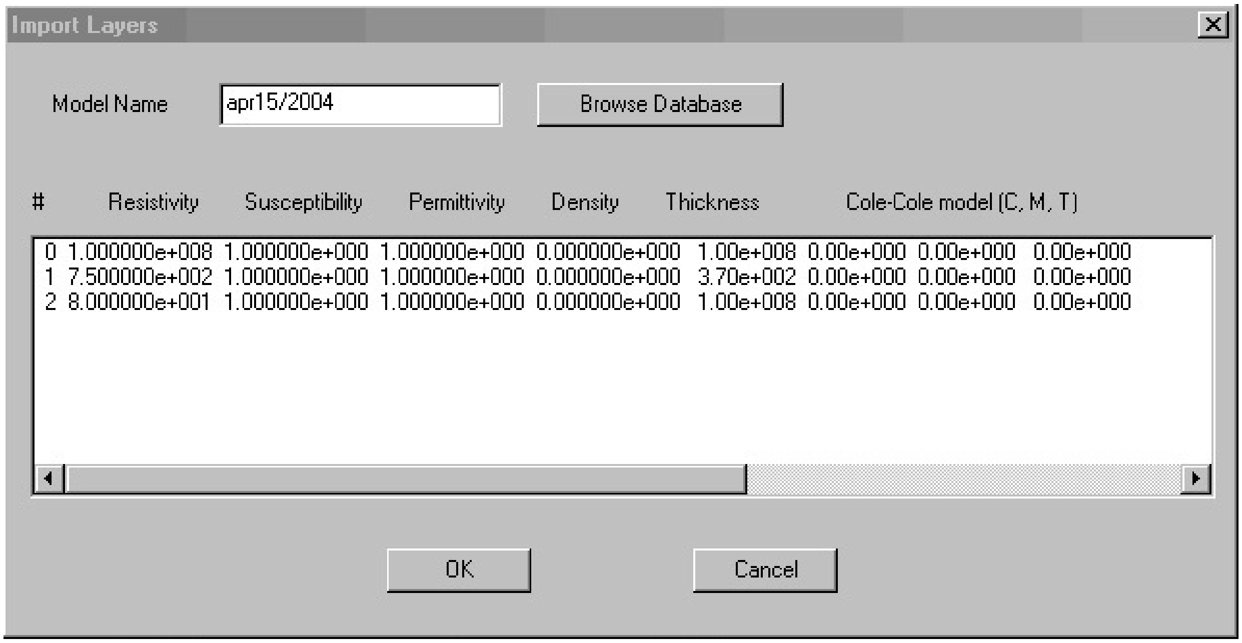
- Click **Import Layers** in the **Edit Mode** section. The following window will appear:




- Click **Browse Database** to bring up the **Select Model** window and search for the model within the current database:




And an interface such as this will appear




- Click **OK** to return to the**Layers**window or **Cancel** if not correct

  The list of layers and their properties will be displayed in the respective field of the window

- Click **Apply** to complete import and a request to create a new dataset or overwrite will appear and you will return to the main interface
- Simulation of the new model can now be performed

## Build the Cole-Cole model

The Cole-Cole model is useful for representing the electrical conductivity or resistivity of a polarizable material and allows one to account for frequency dependent conductivity effects associated with low frequency IP phenomena.

In the **Layers** tab

- Check the **Cole-Cole Polarization Mode Parameters** box to activate the respective section
- Specify chargeability, $m$, a time constant, $\tau$, and a frequency dependence, $c$, - the three parameters that describe conductivity/resistivity, magnetic permeability and electric permittivity in EMIGMA. *Please see the EMIGMA manual for a more complete description of the Cole-Cole parametrization of electrical polarization.*

## View Resistivity & Susceptibility Grid Data Files

In the **Layers** tab:

The **Resistivity & Susceptibility Grid Data Files** section allows viewing *.pex files created as a result of 1D inversion within EMIGMA. The names of such files, if available, will be displayed in the left-hand field of the section

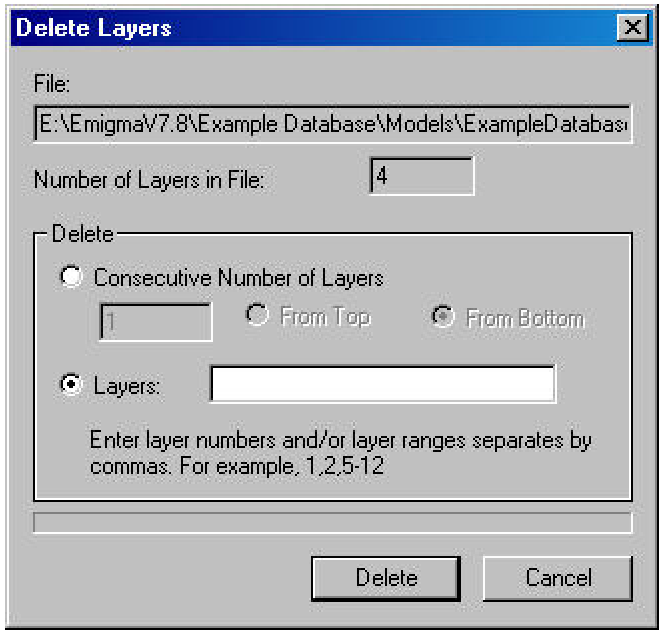
- Select a required file and click **View** to look at the data in the text format. A text file will contain resistivity, coordinate and depth information.

  ***Note***. *To view these data as a grid, save your model and close the* ***Model*** *dialog. In the* ***Database*** *dialog, select the PEX file from the* ***Data Sets in Survey*** *list and click the* ***CDI Viewer*** *button on the main toolbar (see CDI Viewer)*

- Click **Convert to GPSZ** to include topography in the current resistivity pattern. This command will be active if your PEX file contains GPS data. When launched, it recalculates Z coordinates by subtracting depth values from the GPS data

  ***Note***. *A new GPSZ data set will be added to the* ***Data Sets in Survey*** *list of the* ***Database*** *dialog. Click the* ***CDI Viewer*** *button on the main toolbar to see the results of the conversion.*

- Click **Delete File Layer(s)** to remove a number of layers from the PEX file. This command will launch the **Delete Layers** dialog:

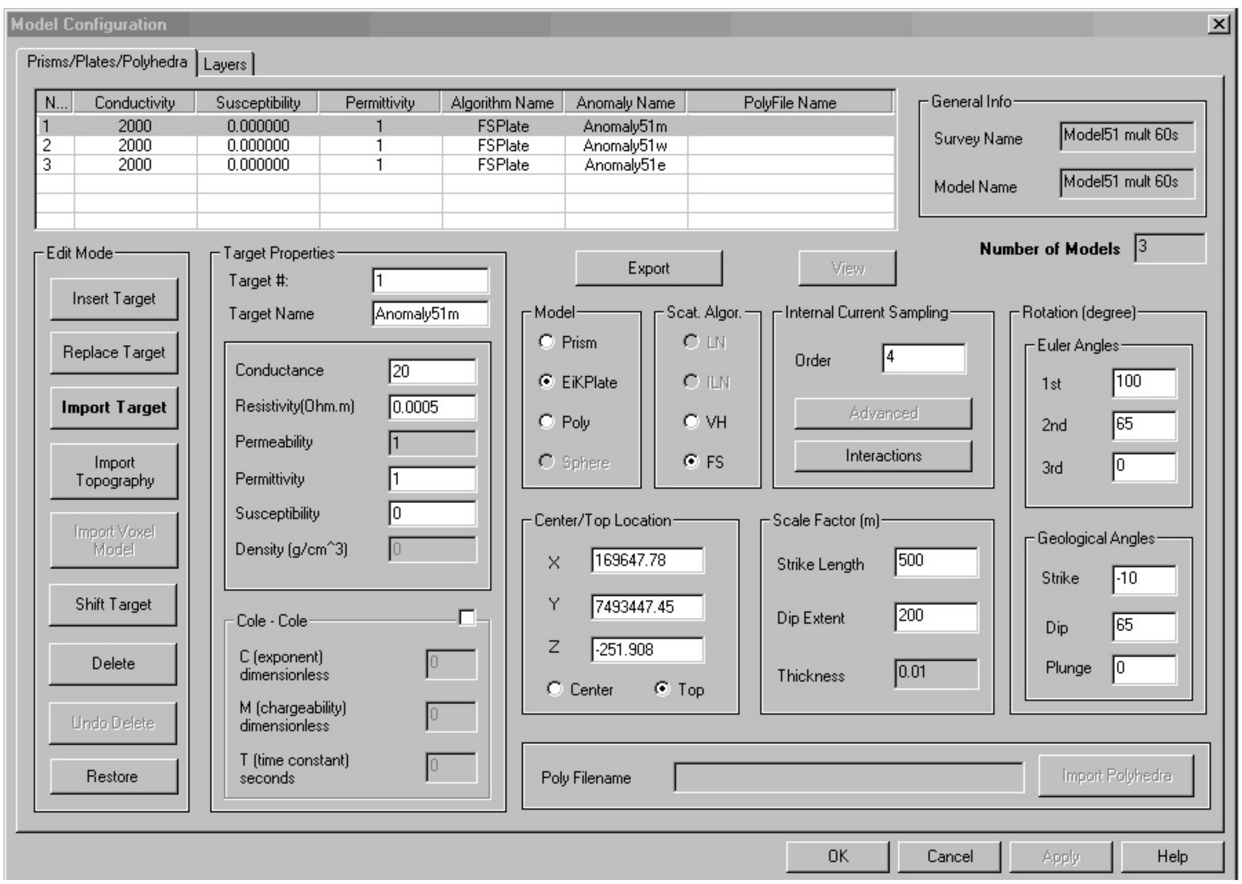



- The number of layers in the PEX file are displayed under the filename. If **Layers** is selected, the layer numbers can be entered as ranges or individually separated by commas. Layers are numbered from ground level to the basement.
- A shortcut option is available if the layers to be deleted are consecutive and include an extreme layer. Simply select **Consecutive Number of Layers**, enter the number of layers to be deleted and whether they are located at the top or bottom of the layer stack.
- Press **Delete** to delete the layers. The resulting model is placed in a newly created data set.

# Specifying the Number and Properties of Targets

EMIGMA allows simulation of different complex conductivity structures that can be represented by right rectangular prisms (overburden thickness variations, fractures, lithology changes, etc.) and plates (dipping tabular bodies) or complex structures via the use of polyhedral volumes. The prisms and plates can further be converted into polyhedra in the Visualizer to build more complicated geometrical shapes. Polyhedra can also be created from scratch.

If you are creating a survey from scratch, go to the **Prisms/Plates/Polyhedra** tab of the **Model Configuration** interface after having specified the layered-earth model:




**The Prisms/Plates/Polyhedra tab**

To reach this tab from the **Database** window, click the **Model** button in the bottom part of the interface.

A relatively complete description is given in the *Forward_Modeling* tutorial and more description in the full EMIGMA manual

4 basic shapes: Rectangular Prisms, thin-sheet plates, Polyhedra and Spheres. When selecting the shape, the offered algorithms are enabled

Position: Center of mass or Top Point. If the top point isn't unique then it is the middle of the top line or the center of the top surface.

Strike and Dip length and Thickness can be used as the user desires. There is no thickness to the thin-sheet and no unique lengths for the polyhedra. A sphere has a radius

Multiple objects can be in the model but only certain algorithms can be mixed.

The plate and prism orientations in space are defined by the standard Euler angles. The geological angles are a geological interpretation of these angles. Strike being the first Euler angle, Dip the second and Plunge the third.

**Related Topics**

Insert a Target

Delete/Restore a Target

Import a Target

Import Topography

Specify the Number of Sampling Points

Specify the Type of Interaction between Targets

[Specify the Model Size and Position]

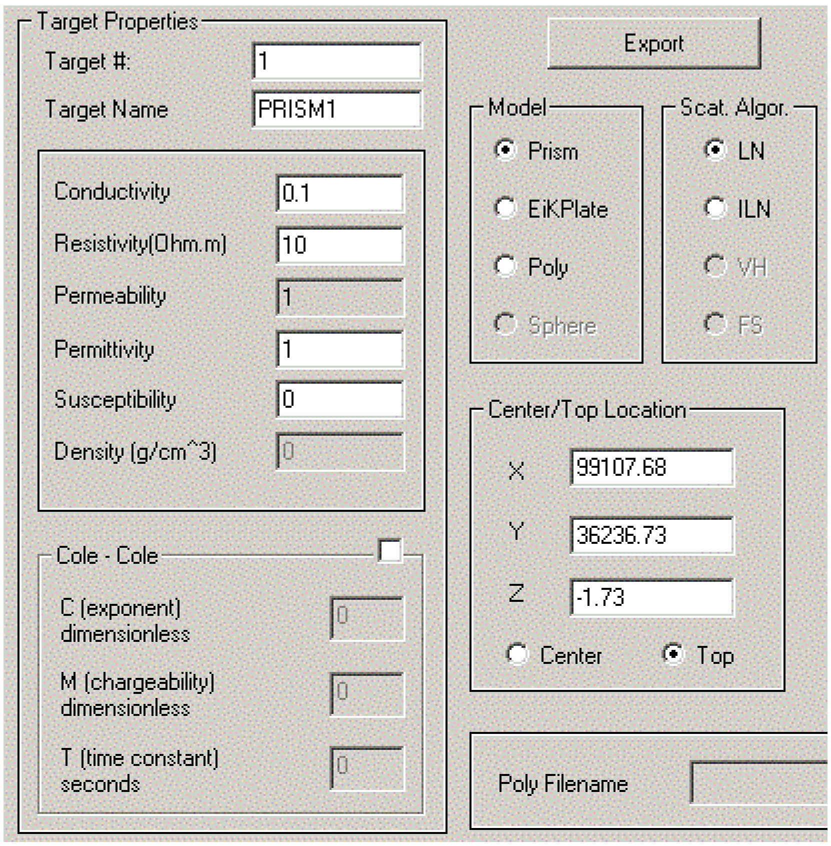
[Replace a Target]

[Import Voxel Model]

**Insert a target**

In the **Prisms/Plates/Polyhedra** tab:

- Select between **Prism**, **Plate**, **Poly** and **Sphere** in the **Model** section




The **Target Properties** section will automatically display the number and name of the model. To change the name, delete it in the **Target Name** box and type in a new one.

- Specify the scattering algorithm to be used in the **Scat. Algor.** section. If your model is a prism or polyhedron, two options are available, **LN** and **ILN**. In the case of a plate, the **VH** and **FS** options are available.

*Note. For more information on these algorithms, see the section titled "Inside the Scattering Algorithms" in the EMIGMA manual. It is located in EMIGMA's Manual directory. Also, find more details* *regarding the algorithms and the interfaces inside the **Forward Modeling** tutorial in the /tutorials directory.*

- In the case of a plate, select between the **Center** and **Top** options in the **Center/Top Location** section. Both options represent the reference points of a plate in the absolute frame, the first coinciding with its center and the second – with its top center
- In the case of a polyhedron, click the **Import Polyhedra** button in the bottom right-hand corner of the **Prisms/Plates/Polyhedra** tab. This will open the standard Windows-style **Open** dialog offering you to select an existing polyhedra file (.pol). Select a required file and click **Open**. The file path will appear in the **Poly Filename** box. (See Creating Polyhedra Files)

- In the **Model Properties** section:
- Specify the physical properties of the target (conductivity, magnetic permeability, electrical permittivity)
- Set density in the **Density** box, which becomes active in gravity systems
- To account for frequency dependent conductivity effects which are often associated with low frequency IP phenomena, check the **Cole-Cole** box to activate the respective section and specify chargeability, *m*, a time constant, *t*, and a frequency dependence, *c*.

  **Chargeability** is the proportional drop in voltage immediately after the current is turned off. **1** is a maximum, **.6** is a large chargeability, **.004** is common for many background materials

  **Frequency dependence (exponent)** generally ranges from 0.5 to 1.0, with 1 indicating one IP decay for the anomaly

  **Time constant (decay in seconds** for a scattering object) generally ranges from 0.5 to 3.0 seconds

- Click **Insert Target** in the **Edit Mode** section. Your target will be added to the list of targets in the upper left-hand corner of the

**Prisms/Plates/Polyhedra** tab.

## Delete/restore a target

To delete a target:

- Select it from the list of targets in the upper left-hand part of the **Prisms/Plates/Polyhedra** tab and click **Delete Target**

To restore a target:

- Click **Undo Delete** if you have changed your mind and want to bring it back

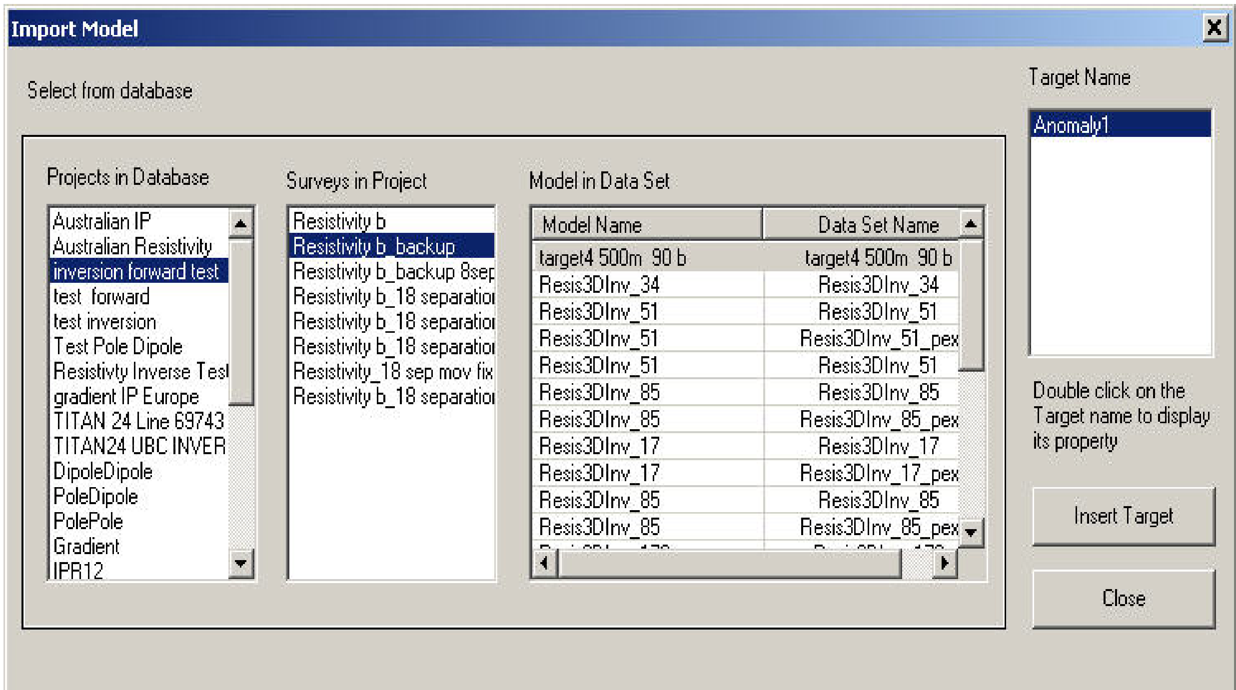
  OR

- Click **Restore**, if you deleted, replaced or changed your target and now want to cancel all the changes and return to the initial version

### Import a target

In the **Edit Mode** section of the **Prisms/Plates/Polyhedra** tab:

- Click the **Import Target** button. The **Import Model** window appears




To import a model from the current database:

- Select the project, survey and model from the respective lists

  The names of the targets will be displayed in the **Target Name** field on the right. Double click on a target to see its properties. Select the desired targets.

- Click **Insert Target** to complete the import

  When you click **Insert Target**, you will see the imported model added to the list of targets on the main Prisms/Plates/Polyhedra tab. The target properties can then be edited if required.

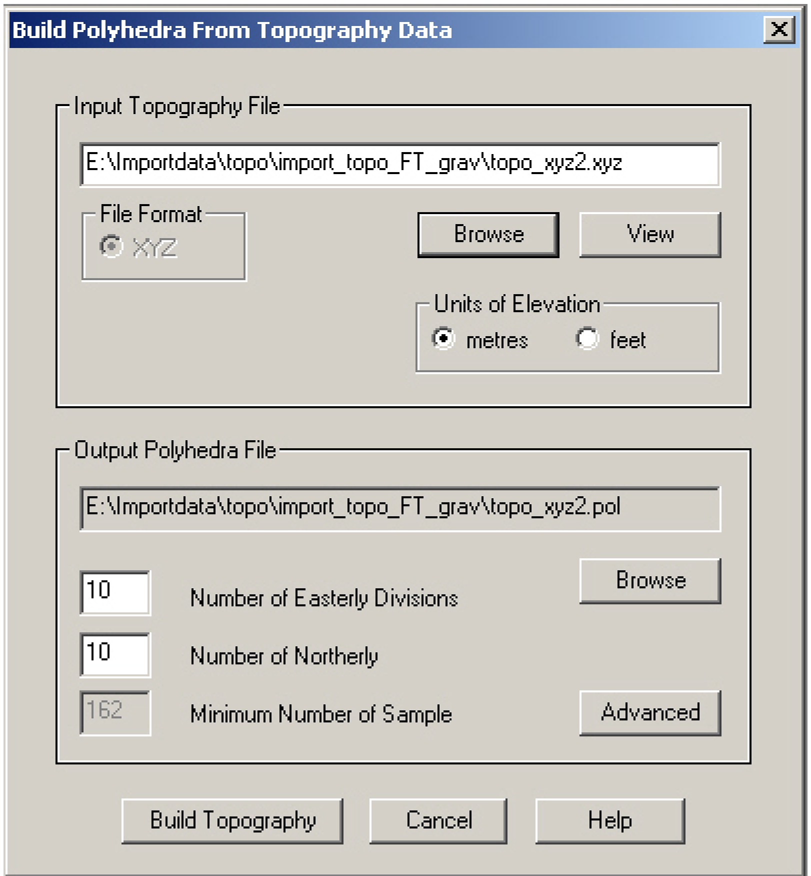
**Import topography - XYZ file**

It is possible to build a model of the topography in the vicinity of your survey. This can be done using either an .egr grid file, a .qdem file or an .xyz file. For the .xyz format, the first three columns must be the x co-ordinate, y co-ordinate and elevation in that order. Line labels are also required.

Once you have built a model of the topography, you can simulate its effect by either placing the profiles on the same topography model under configuration, or by running the simulation according to GPS_Z (potential fields).

On the **Prisms/Plates/Polyhedra** tab, click **Import Topography** in the **Edit Mode** section.

- To use an .xyz file to build topography, click **ASCII column xyz file**




- The above window appears:

In the upper **Input Topography File** section

- Click **Browse** to display the **Topography Data File** dialog, a standard Windows-style interface for searching and opening files. Select a required topography XYZ file and click **Open**

  **See Also** Topography file format

  The filename will appear in the **Input Topography File** field. Simultaneously, a polyhedra file (.pol) will be generated and saved in the same directory, as seen from the example. Its filename will be written in the **Output Polyhedra File** field of the window

- Click **View** to see the input topography file in the WordPad format

- Select the line direction and units of elevation in the respective subsections

In the **Output Polyhedra File** section:

- Specify the number of **Easterly** or **Northerly** divisions (conventional units into which a polyhedron is divided for the simulation purposes) in the respective boxes.
- The minimum number of samples is for standard numerical simulation of the model ( for gravity and magnetics and resistivity the maximum is 32,000 , for AC EM, the maximum is 2000)
- However, for all DC techniques the default simulation is via analytic techniques and thus the sampling is not an issue

In the **Advanced** section: the spatial limits of the polyhedra are given from the input file but can be reduced if desired

Click **Build Topography** in the bottom part of the **Build Polyhedra from Topography Data**. This will return you to the **Prisms/Plates/Polyhedra** tab. In the **Poly Filename** field, you will see the newly created .pol file

- In the **Edit Mode** section of the tab, select material properties and **Replace Target** and then **Apply**

## Import topography - EGR file

It is possible to build a model of the topography in the vicinity of your survey. This can be done using either an .egr grid file or an .xyz file.

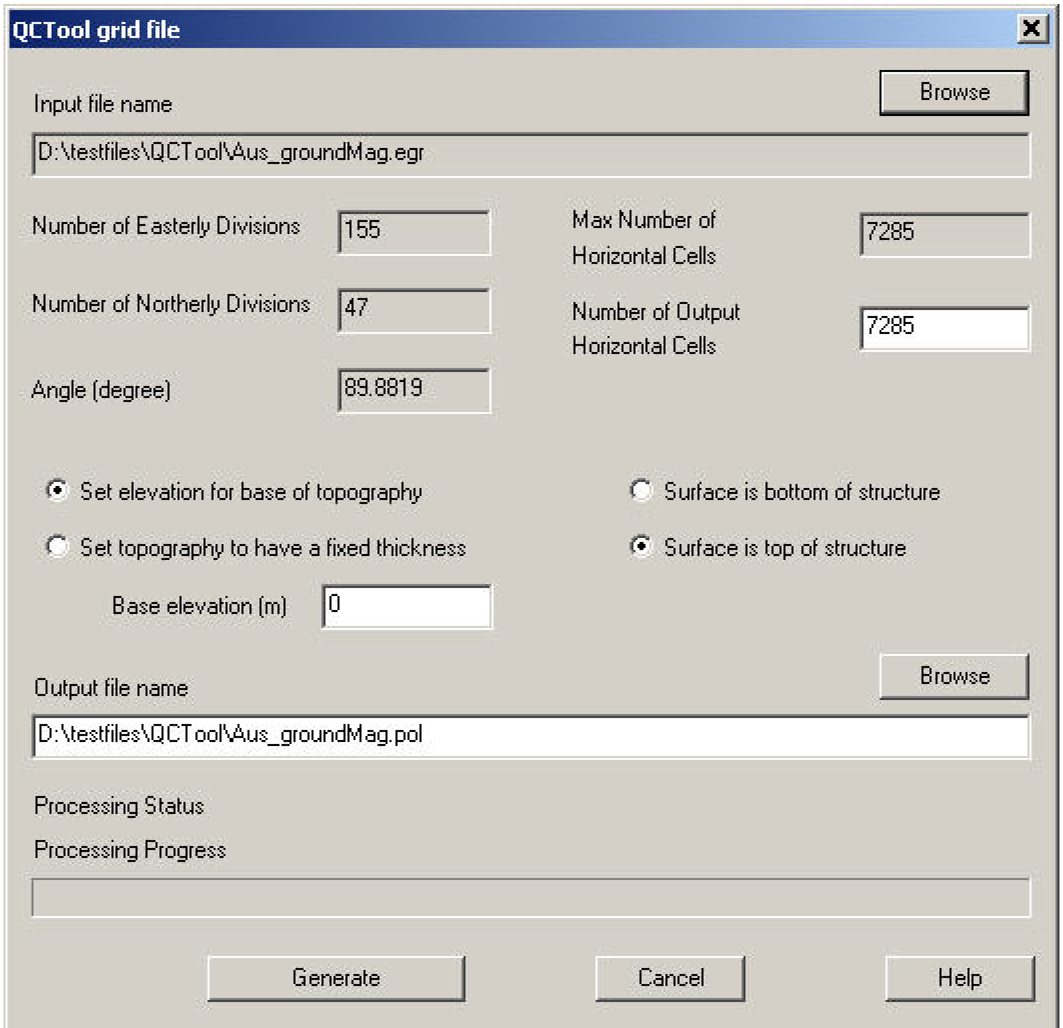
Once you have built a model of the topography, you can simulate its effect by either placing the profiles on the same topography model under configuration, or by running the simulation according to GPS_Z (potential fields).

An appropriate .egr file is created by interpolating elevation data (in meters) in QCTool and saving the resulting grid (under Grid Mesh). Keep in mind that if you use a large number of grid cells, the model may run slowly in EMIGMA.

In DC cases (gravity, magnetic, resistivity, MMR), the simulation can be done analytically and then it is very quick

On the **Prisms/Plates/Polyhedra** tab, click **Import Topography** in the **Edit Mode** section.

- To use an .egr file to build topography, click **QCTool Grid (.egr) file**
- The following window appears:




**Input file name** Click Browse to locate the egr file. The dimensions of the grid and its angle will be displayed directly below the egr file name.

**Set elevation for base of topography (Base elevation)**
This is the datum above which the topography model will be built. For example, if the DC elevation is set as 0 m, then the model will be built upwards from 0 m to the given elevation at a point. If the DC elevation is set as 50 m, then the model will be build from 50 m up to the elevation at the point. If **Surface is bottom of structure** is selected the model will be built downward from the given elevation.

**Set Topography layer thickness**
Select **Set topography to have a fixed thickness** and you will have topography on both the top and bottom of your structure with the specified

thickness.

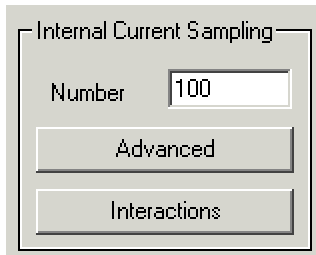
**Number of Output Horizontal Cells**
You may limit the number of cells in your topography grid by entering a smaller number here.

**Output file name**
This is the polyhedral file in which the topography model created from the .egr file is saved. It is only a temporary file. When you click **Apply** on the Prisms/Plates/Polyhedra tab, a new .pol file will be created in the Polyhedron folder of your database

Once you click OK, you will return to the Prisms/Plates/Polyhedra tab. In the **Poly Filename** field, you will see the newly created .pol file. In the **Edit Mode** section of the tab, change the properties as desired of the model and then **Replace** and then **Apply**

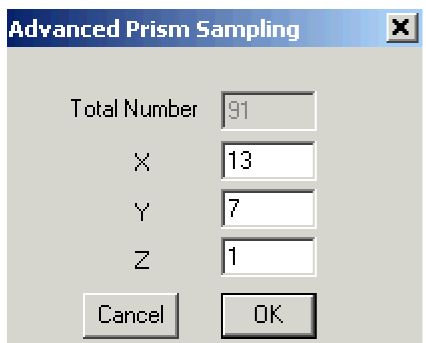
## Specify the number of sampling points

If your model is a prism or polyhedron, the **Internal Current Sampling** section of the **Prisms/Plates/Polyhedra** dialog will be enabled suggesting 100 points as a default value for LNPRISM. You can change it as needed, with a maximum of 600, which is often more than enough.




The **Advanced** button allows for a more accurate setting:

- Click **Advanced** to open the **Advanced Prism Sampling** dialog:

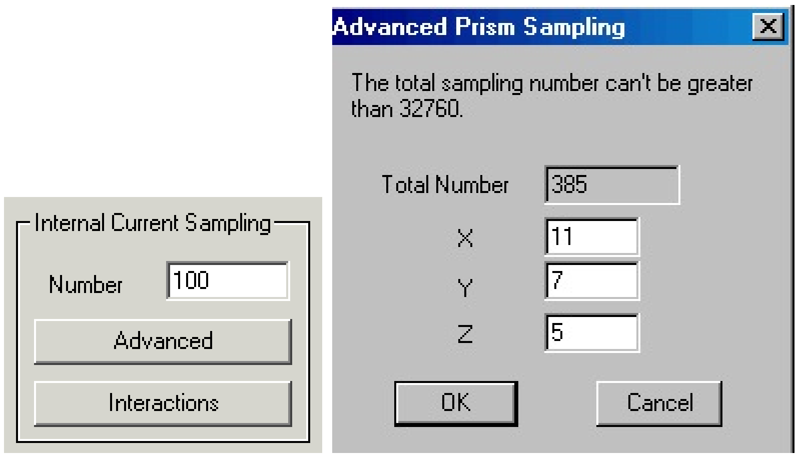



- Type your values in the **X**, **Y** and **Z** boxes. The multiplication product of these values can be viewed in the **Total Number** field on top of the dialog
- Click **OK** to close the dialog and view the result in the **Number** box of the section

**Note**. If your model is a plate, the **Internal Current Sampling** section will be disabled and the respective number will be set to 441. This value represents a mesh of 21 by 21 used to calculate the scattered current distribution in the VH scattering algorithm and cannot be changed.

## Specify the type of interaction between targets

In the **Internal Current Sampling** section of the **Prisms/Plates/Polyhedra** tab:




**Prism**: The internal secondary source sampling along each aspect can be set in the advanced mode in order to check convergence.

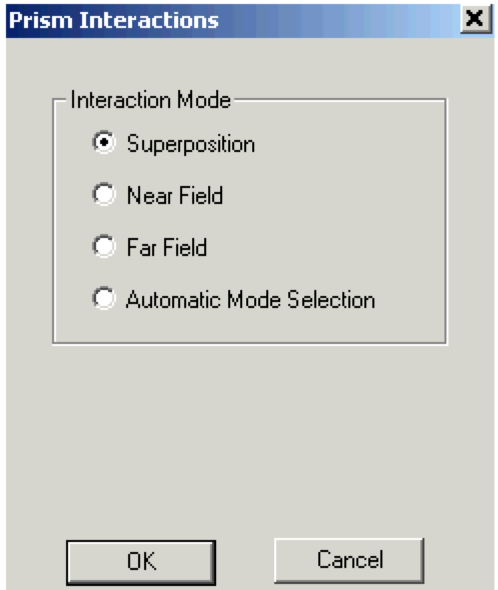
**Poly**: only the total sampling can be defined. The algorithm automatically utilizes our tetrahedral volume gridding which attempts to make all grid cells of equal volume. Secondary sources are computed at the centre of each tetrahedron.

**Plates**: VHplate utilizes 2 sets of polynomials as bases functions and there are exactly 441 sample points. FSPlate (freespace plate) uses eigenfunctions which are often called eigencurrents. You can use between 2 and 11 of these functions.

**Sphere**: Sphere uses spherical harmonics. The maximum number is 200.

- Click the **Interaction** button. This brings up the **Prism Interactions** window
  This is available when using combinations of LN or ILN targets.
  This is available for EM (DC, FEM or TEM), Magnetic and Gravity surveys.

- Select between the four options available in this window:




The **Superposition** option simply adds the target responses that do not interact

The **Near-Field** option accounts for the current flow between your targets and regional conductors in contact with them

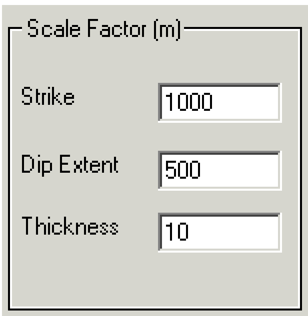
The **Far-Field** option includes the primary effects of multiple conductors not in contact with your target

The **Automatic Mode** option sets interaction for you automatically

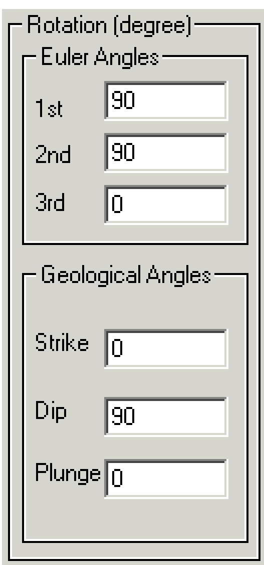
- Click **OK**

## Specify the model size and position

In the **Scale Factor (m)** section of the **Prisms/Plates/Polyhedra** tab, specify the geological dimensions of your target, including its strike, dip and thickness:




In the **Rotation (degree)** section of the tab, specify the Euler and geological angles in the respective fields to provide the required orientation of the model:



**Replace a target**

- Select the target you want to replace in the list of targets of the **Prisms/Plates/Polyhedra** tab
- Change settings in the rest of the sections as needed, see

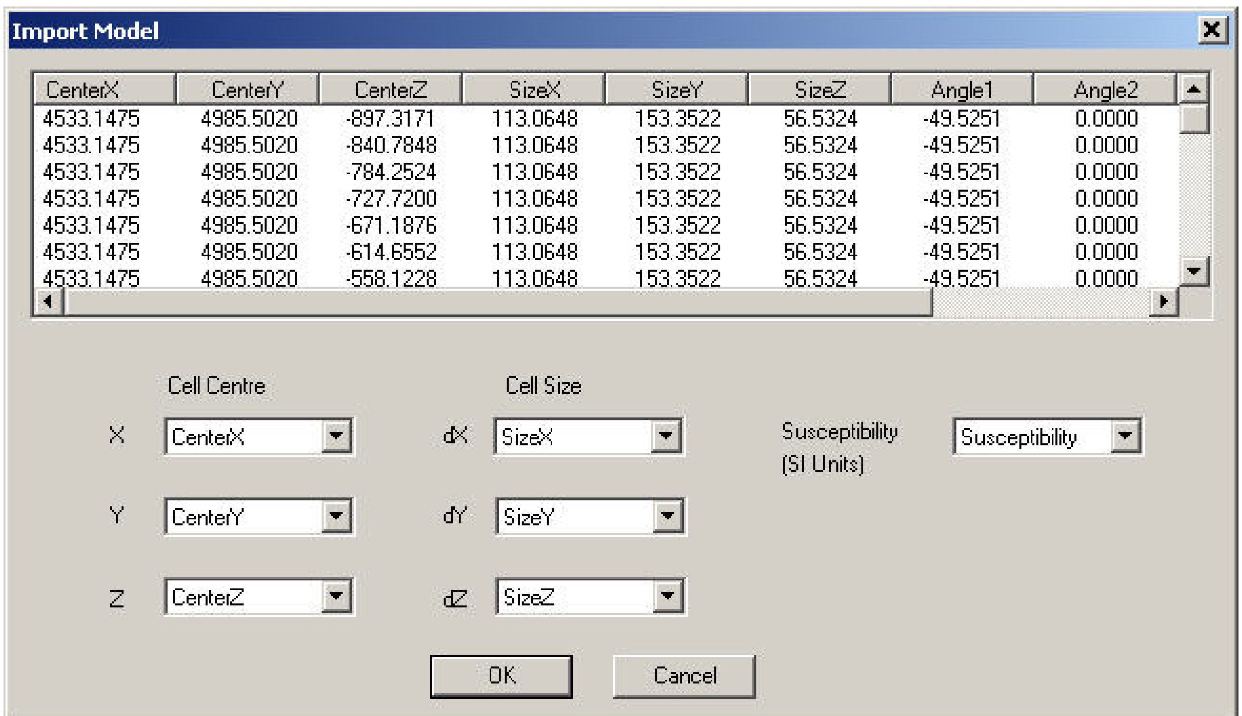
  Import a Target, Import Topography, Specify the Number of Sampling Points, Specify the Type of Interaction between Targets, Specify the Model Size and Position

- Click **Replace Target**

## Import voxel model

A model of prism cells can be imported from a .qct file for gravity and magnetic surveys. Channels required in the .qct file are the x,y,z coordinates of the cell centre, the width, length and height of the cell and susceptibility for magnetics or density for gravity.

Click **Import Voxel Model** in the **Edit Mode** section to open the standard Windows-style **Open** interface. Select your file and the following window appears:




Select the appropriate channels, click OK and the model will be added to the anomaly list box. Only one voxel model is allowed per data set. If you would like to replace the current model, you need to delete the current model first.

## Saving model configuration

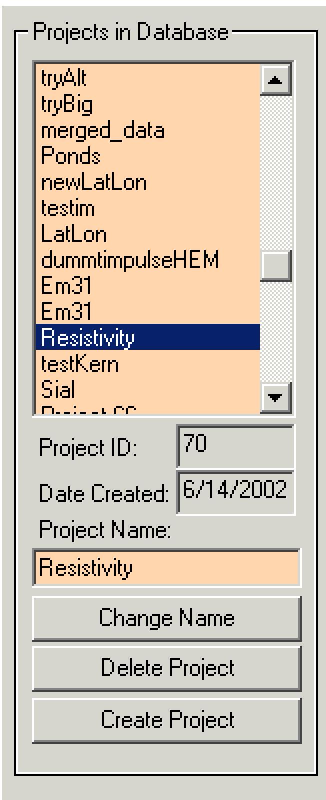
- Click **Apply** in the bottom right-hand corner of the **Model Configuration** dialog (either in the **Prisms/Plates/Polyhedra** or the **Layers** tabs)
- Click **OK** to close the **Model Configuration** dialog and return to the initial **Database** dialog. The box on the **Model** button in the bottom of this dialog will be checked to signal that your data set now contains a model

***Note***. *To reopen the **Model Configuration** dialog, click the **Model** button*

# Managing Project Information

## Projects

In the **Projects in Database** section of the **Database** dialog, you can create and delete projects, change their names. You can also see the project ID generated by the system and the date of its creation.




## Related Topics

Create a New Project

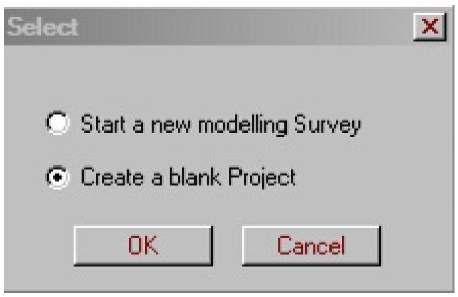
Change the Name of the Project

Delete a Project

## Create a New Project

- To create a new project, click the **Create Project** button in the bottom of the **Projects in Database** section.

  The **Select** dialog appears:

- Select **Import data from file** if you want to create your project from imported data
- Select **Start a new modeling Survey** if you want to create your project from scratch

  See Creating Projects from Scratch

- Select **Create a blank Project** if you want to create a project but do not want to go to the configuration page right away.

## Change the name of a project

- Select a project from the list in the upper left-hand corner of the **Projects in Database** section and type a new name in the **Project Name** box below
- Click the **Change Name** button

## Delete a project

- Select a project from the list in the **Projects in Database** section and click the **Delete Project** button.

  The message will appear asking you to confirm deletion

- Click **OK**

# Surveys

In the **Surveys in Project** section of the **Database** dialog, you can see a list of surveys available in each project. Here, you can add or delete surveys, provide them with comments, change their names or create back up files. The **Survey ID** number is generated automatically.

**Related Topics**

Add/Delete a Survey

Rename a Survey

Write, View and Edit Survey Comments

Create a BackUP Survey

Copy/Paste a Survey

## Add a new survey

- Click **Add Survey** in the **Surveys in Project** section of the **Database** dialog. The **Tx-Rx** tab of the **Property Pages** dialog will open offering you to specify your survey parameters
- If you create your survey from scratch, see Creating Projects from Scratch

All done, you can see your new survey added to the **Surveys in Project** list

## Delete a survey

- Select the survey to delete from the **Surveys in Project** list and click the **Delete Survey** button.

  The message will ask you to confirm deletion

- Click **OK**. The survey will be removed from the **Surveys in Project** list

## Rename a survey

- Select a survey from the **Surveys in Project** list of the **Database** dialog. Its name will appear in the **Survey Name** box to the right
- Replace the old name with a new one and click **Change**

## Write survey comments

- Select the survey in the **Surveys in Project** list of the **Database** dialog and click the **Survey Comments** button.

  The respective dialog will open. For your convenience, you can pin it to prevent it from closing during other operations, if any, in the **Database** dialog

- Write necessary comments in the empty field of this dialog, click **Save** and **Exit**

## View or change survey comments

- Double-click the survey you want to display your comments for or select it in the **Surveys in Projects** list and click the **Survey Comments** button

  The **Survey Comments** dialog will open containing the earlier written comments

- View or change them as needed, click **Save** and **Exit**

## Create a backup survey

You can create a duplicate of your survey to experiment with your data while keeping the original file intact.

- Select a survey in the **Surveys in Project** list of the **Database** dialog and click the **BackUP** button. This will automatically create a backup file and add it to the **Surveys in Project** list

## Copy a survey

- Select the survey you would like to copy from the list in the **Surveys in Project** section of the main database window.
- Click the **Copy** button in this section and the selected survey will be copied to the EMIGMA clipboard.

## Paste a survey

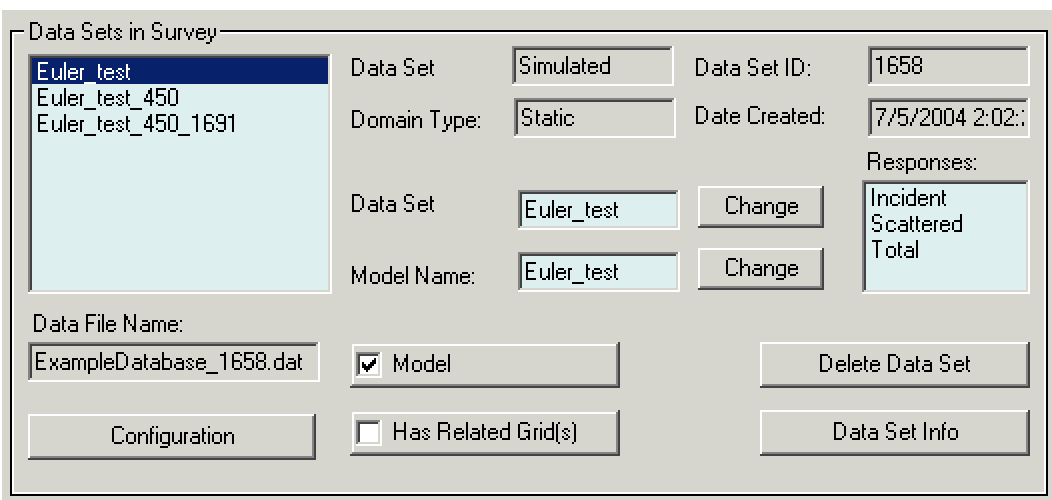
- You may add the survey on the EMIGMA clipboard to a project by first selecting the project from the **Projects in Database** list.
- Click the **Paste** button in the **Surveys in Project** section.
- A message will ask you to confirm that you want to copy the survey. Answer Yes and the survey will be added to the selected project.

# Data Sets

In the **Data Sets in Survey** section of the **Database** dialog, there is a list of data sets available in each survey. They all are characterized by the same configuration (system geometry, waveform, profile and output settings). Whenever you do any operations with your data set – simulation, transform or inversion - the program will ask your permission to overwrite it; otherwise it will create a new data set, with a name differing from the initial one by an automatically generated number attached to its end.

The **Data File Name** box below the list contains the name, which EMIGMA assigns to each data set to store it with in the respective database directory

The boxes to the right of the list display information on the type of data (measured or simulated; frequency-domain, time-domain, spectral, static, gravity), data set ID (EMIGMA internal number), the date of creation, and output fields.




If needed, you can change the name of your data set or model, edit survey and model configuration, view related grids and data set info - all from the Data Sets in Survey section of the Database dialog.

**Related Topics**

Rename a Data Set

Rename a Model

Delete a Data Set

Edit or Add a Model

View and Edit Survey Configuration

View Data Set Information

View, Edit and Export Grid Information

## Rename a data set

- Select a required data set in the **Data Sets in Survey** list of the **Database** dialog. Its name will be displayed in the **Data Set** box
- Type a new name in the box and click **Change**

## Rename a model

- Select the data set containing a model in the list. The model name will be displayed in the **Model Name** box to the right
- Type a new name in the box and click **Change**

This option may be very useful, since it allows you to provide additional details about your data set.

## Delete a data set

- Select from the **Data Sets in Survey** list of the **Database** dialog and click the **Delete Data Set** button

  A message will appear asking to confirm deletion. Click **OK**

## Edit or add a model

- Select a data set from the **Data Sets in Survey** list of the **Database** dialog and click **Model**. The **Model Configuration** dialog will open
- Edit (or just check) your layered earth model on the **Layers** tab of the dialog and your target on the **Prisms/Plates/Polyhedra** tab
- To add a new model, follow the steps as described in Specifying the Number and Properties of Layers and Specifying the Number and Properties of Targets

***Note**. Once you added a model to your data set, the box on the **Model** button in the **Database** dialog will be checked.*

## View or edit survey configuration

Select a survey or data set in the respective lists of the **Database** dialog and click **Configuration**. This will display the **Tx-Rx** tab of the **Property Pages** dialog. Check and edit the contained system geometry, profile, waveform and output information

See

Specifying System Geometry

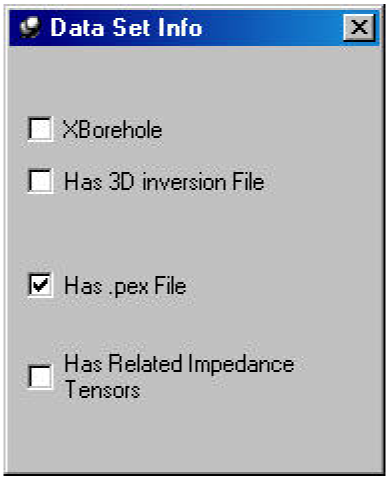
Specifying Waveform

Specifying Profile Information

Specifying Output

## View data set information

To check what kind of information a data set may contain, click the **Data Set Info** button in the bottom of the **Database** dialog. The window below will appear:

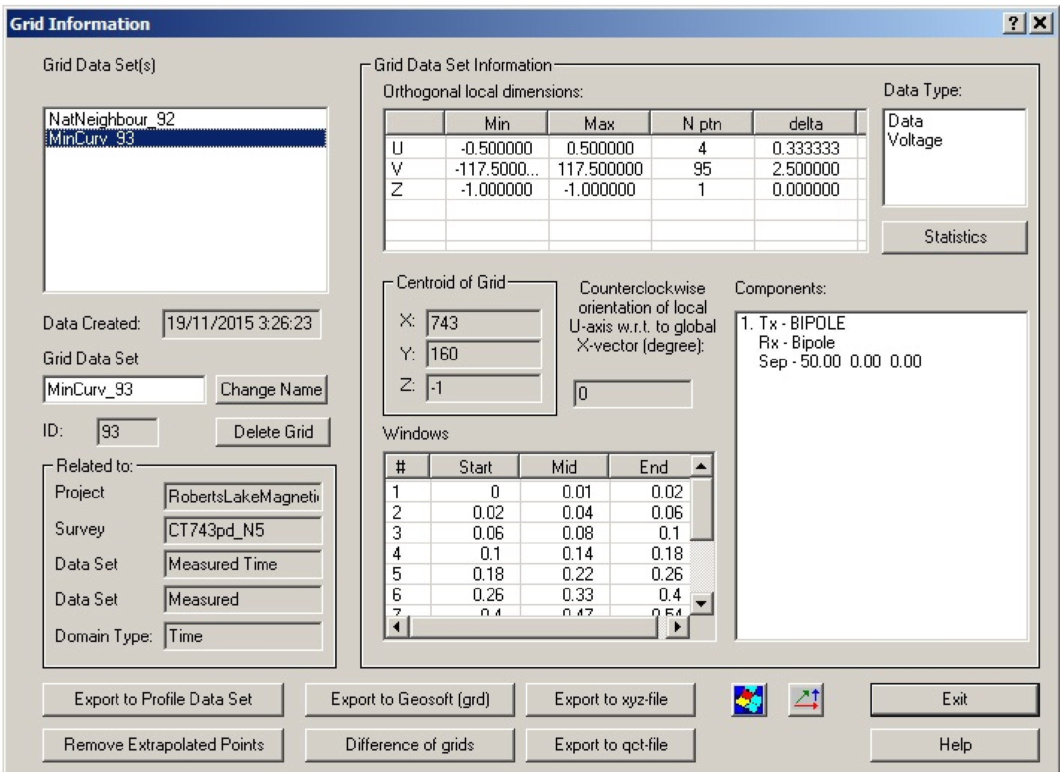



To search through a number of data sets without the window closing, it can be pinned by clicking on the pin icon in the upper left hand corner.

- A checked **Xborehole** box means that the data set represents a crosshole survey.
- A checked **Has 3D inversion File** box means that the data set has an attached file resulting from one of the 3D inversion tools within EMIGMA
- A checked **Has .pex File** box means that the data set has an attached PEX file resulting from 1D inversion within EMIGMA
- A checked **Has Related Impedance Tensors** box means that the data set contains files for different configurations of an MT/CSAMT survey.

# View, edit and export grid information

If your data have earlier been interpolated, the resulting grid is attached to the initial data set. Whether or not a data set has a grid will be seen from the **Grid(s)** button in the [Database](Database) dialog: the box on the button will be checked to signal the availability of a grid. To view grid information, select the data set from the **Data Sets in Project** list and click the **Grid(s)** button. This will bring up the **Grid Information** dialog:



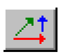

In this dialog, the **Grid Data Set(s)** list contains all the grids available with your data set

- To view information on a given grid, select it from the list. The rest of the fields and sections of the dialog will display thorough information on this grid and the data set it belongs to. This information cannot be edited
- To see more details, i.e. the grid statistics per channel or configuration component (transmitter, receiver, and separation) in a desired format (fixed or exponential), click the **Statistics** button

The **Grid Statistics** dialog will open. Toggle through all the available channels and configuration components to view respective grid parameters

- To change the name of the grid, type the new name in the **Grid Data Set** box in the left-hand part of the **Grid Information** dialog and click **Change Name**
- To delete a grid, select it from the **Grid Data Set(s)** list and click the **Delete Grid** button

The  button will launch the **Grid Presentation** tool and allow you to view the grids in the **Grid Data Set** list.

The  button will launch the **FFT** tool.

See

Remove "No Data" Points from your Grid

Export a Grid into a Profile Data Set

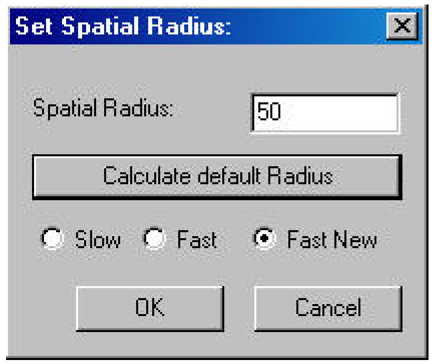
Export a Grid into the Geosoft Format

Export a Grid into the XYZ or QCT format

Calculate the Difference of Two Grids

**Remove non-data points from your grid**

- Click the **Remove Extrapolated Points** button in the bottom of the Grid Information dialog. This will bring up the **Set Spatial Radius** dialog:




- Set a required spatial radius to restrict the area of interpolation. You can also click the **Calculate Default Radius** button to use the radius set by default

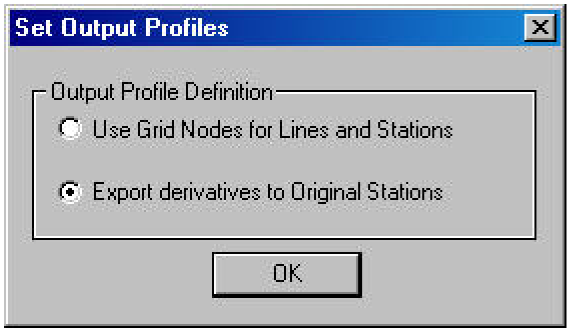
  In the present example: a spatial radius of 50 means that if there are no data in the radius of 50 m around a given point – a grid cell center, – this cell will be removed from interpolation

- Select between the slow, fast and fast new interpolation algorithms (slow is more accurate, but fast is almost always sufficient) and click **OK**

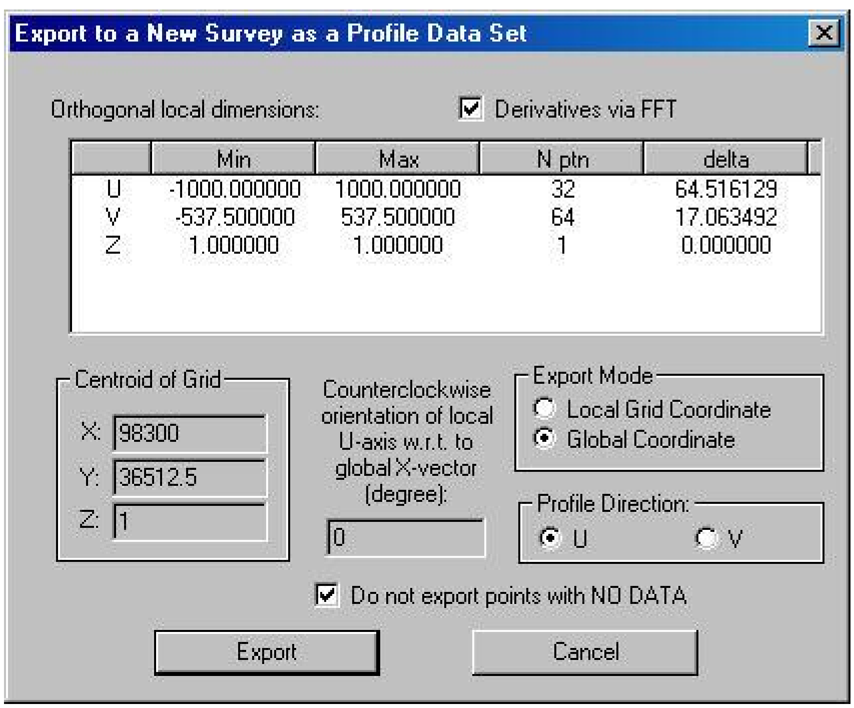
Your new grid will appear in the **Grid Data Set(s)** list of the **Grid Information** dialog.

## Export a grid into a Profile Data Set

To export your grid into a separate survey/data set, click the **Export to Profile Data Set** button in the bottom of the **Grid Information** dialog. The **Set Output Profiles** dialog might appear depending on the type of data.




To create a new survey with derivatives associated with the original stations as opposed to the grid nodes, select **Export derivatives to Original Stations** Otherwise, select **Use Grid Nodes for Lines and Stations** to proceed to the **Export to a New Survey as a Profile Data Set** dialog with the **Orthogonal Local Dimensions** field displaying the coordinates of your grid.



To export as a profile data set using the local grid coordinates:

- Select the **Local Grid Coordinate** button in the **Export Mode** section
- Select between the two options for profile direction in the respective section. If you turn the **U** button on, the profile direction will coincide with the X-axis; with the **V** button on, the profile direction will coincide with the Y-axis

  In the example above, if you select the **Local Grid Coordinate** option, with the **U** option turned on in the **Profile Direction** section, your profile in the resulting data set will start at –1000 and end at 1000, with 41 stations spaced at 50 m.

To export as a profile data set using the coordinates of the original data set:

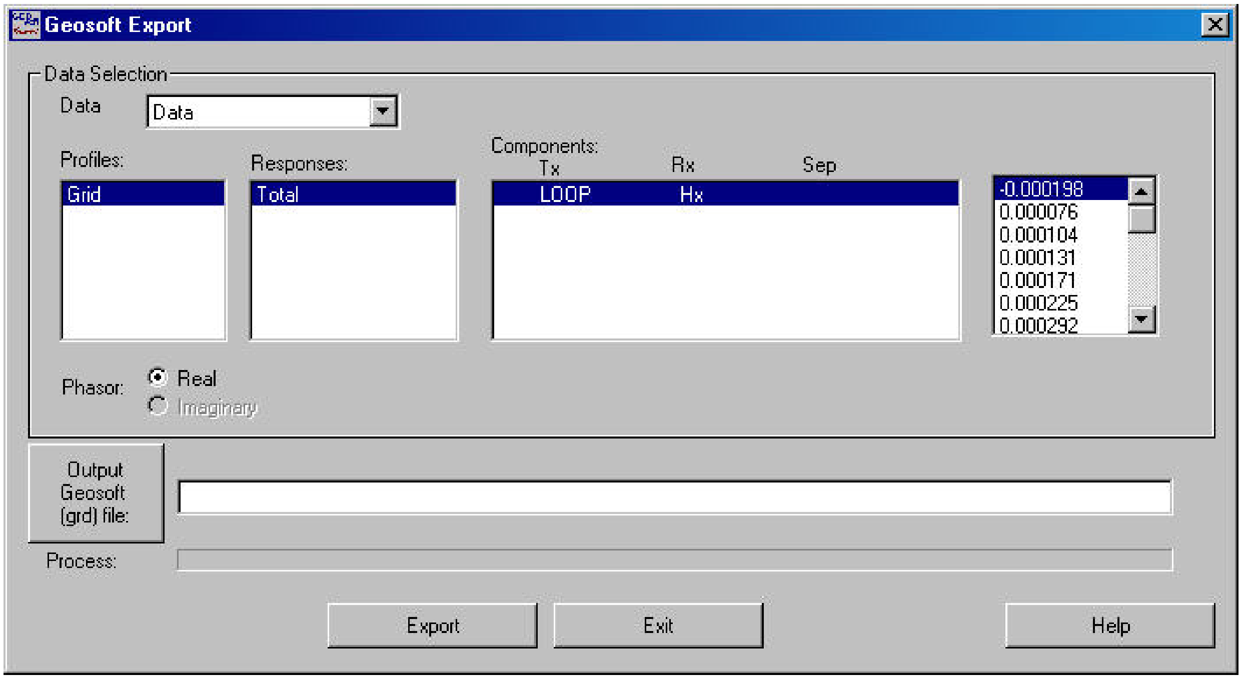
- Select the **Global Coordinate** button
- Select between the two options for profile direction in the respective section.

  In the example above: the profile in the original data set, as seen from the number of stations (32), the step (64.5 m) and the X coordinate of the profile center (98300) in the **Centroid of Grid** section, starts at 97300 and ends at 99300. The exported profile will have the same coordinates and, since the **U** option in the **Profile Direction** section is turned on, will coincide with the X-axis

- To prevent export of points containing no data, check the respective box in the bottom of the dialog
- If the grid has attached derivatives, the **Derivatives via FFT** checkbox can be used to indicate whether or not this data should exported.
- Click **Export**. The new survey appears in the **Surveys in Project** list of the **Database** dialog containing a new data set. Change the name of the survey and data set, if necessary

## Export a grid into the Geosoft format

To use your data in Geosoft's Oasis Montaj, export them into the Geosoft format. Click the **Export to Geosoft (.grd)** button in the bottom of the **Grid Information** dialog. This will bring up the **Geosoft Export** dialog:




- Select the data type you want to export from the **Data** dropdown list

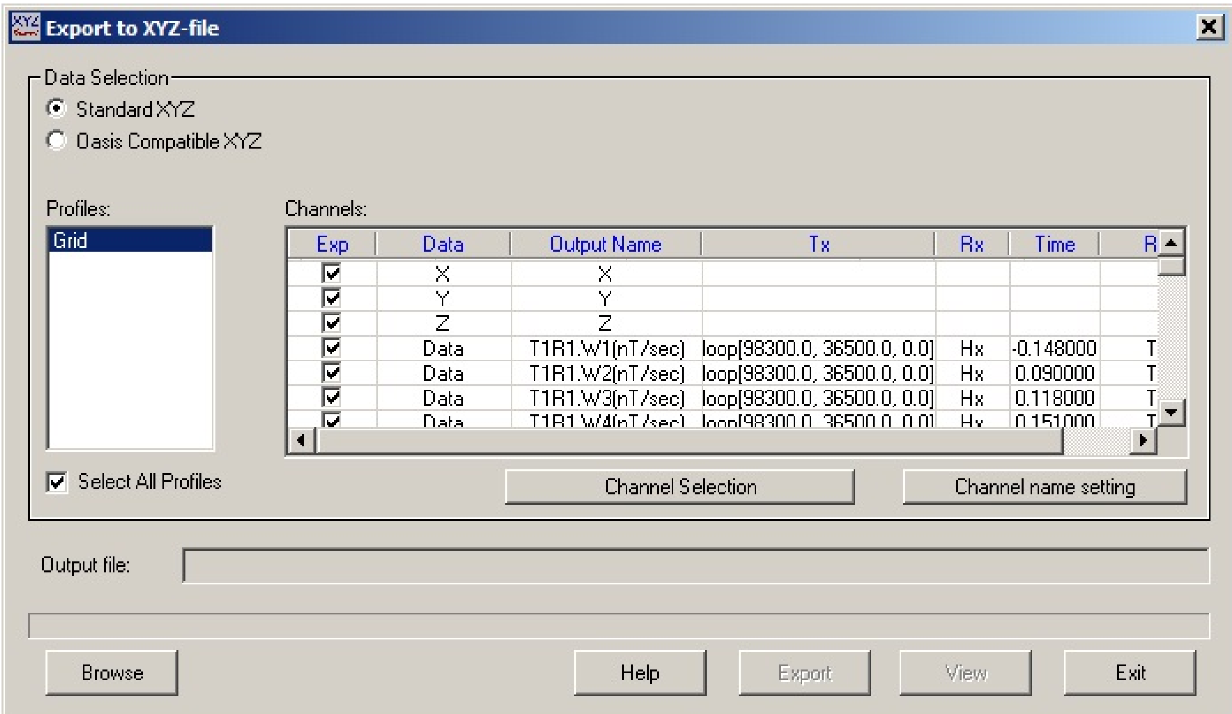
  All data obtained by means of import or simulation in EMIGMA, subjected to normalization, etc., are considered as core data and are referred to as ***Data***; all the rest - calculated through various algorithms - are considered as optional and are referred to in accordance with their type, e.g. ***Apparent Resistivity***, ***Apparent Depth***, ***IP***, etc.

- Make necessary selections in the **Profiles**, **Responses** and **Components** boxes. The Geosoft format can take only one component or channel at a time
- Click the **Output Geosoft (.grd) File** button to bring up a standard Windows-style **Open** dialog, browse for the directory and folder to save the output file into and click **Open**. The output filename will appear in the respective box

- Click **Export** to complete export to the Geosoft format

## Export to XYZ format

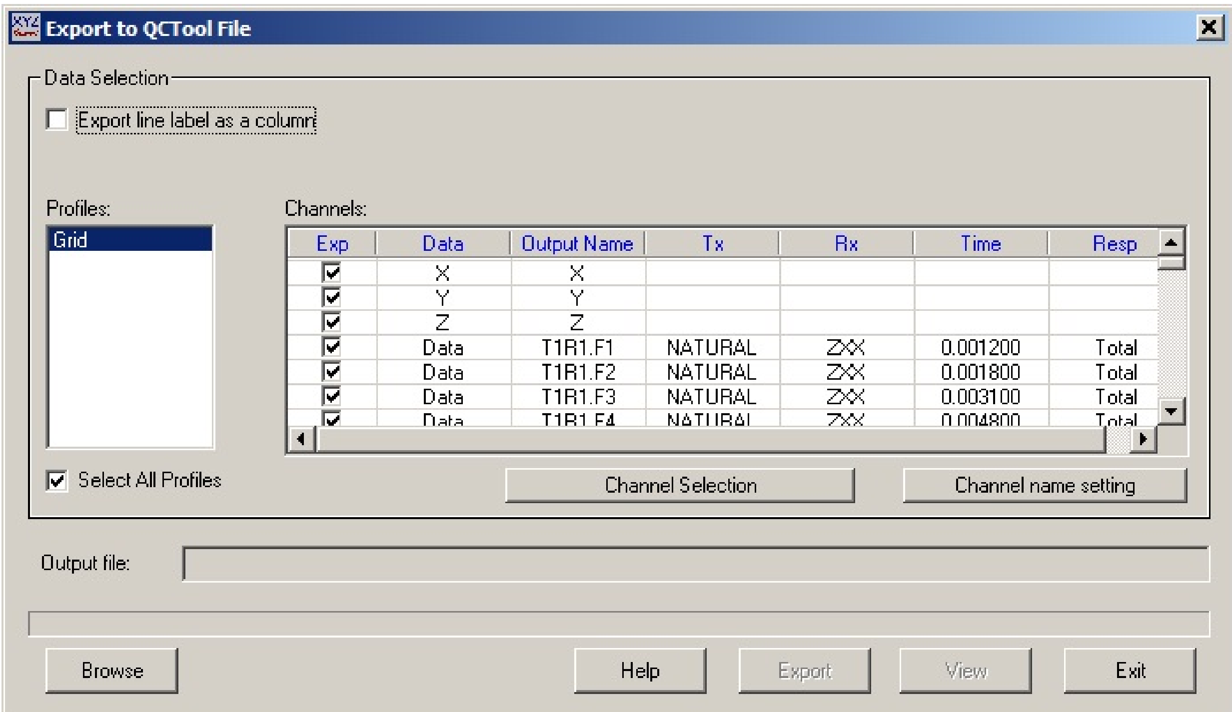
The XYZ Export can be accessed by clicking the **Export to xyz-file** button in the bottom of the [Grid Information]() window. The **XYZ Export** window will open offering two different options of export: one to transfer data to the standard XYZ format and the other to the Geosoft's Oasis Compatible XYZ format.




- Select channels to export from the respective lists. It can be done in this dialog by checking boxes in "Channels" list, or in the dialog called on pressing button **Channel selection**. You can choose to select all available channels by checking the respective boxes below the lists.
- Click **Browse** to bring up a standard Windows-style **Open** dialog, specify the directory and folder to save the output file into and click **Open**. The output filename will appear in the **Output File** box
- Click **Export** to create the output file.
- To view the output file in the text format, click **View** in the bottom of the dialog.

## Export to QCT format

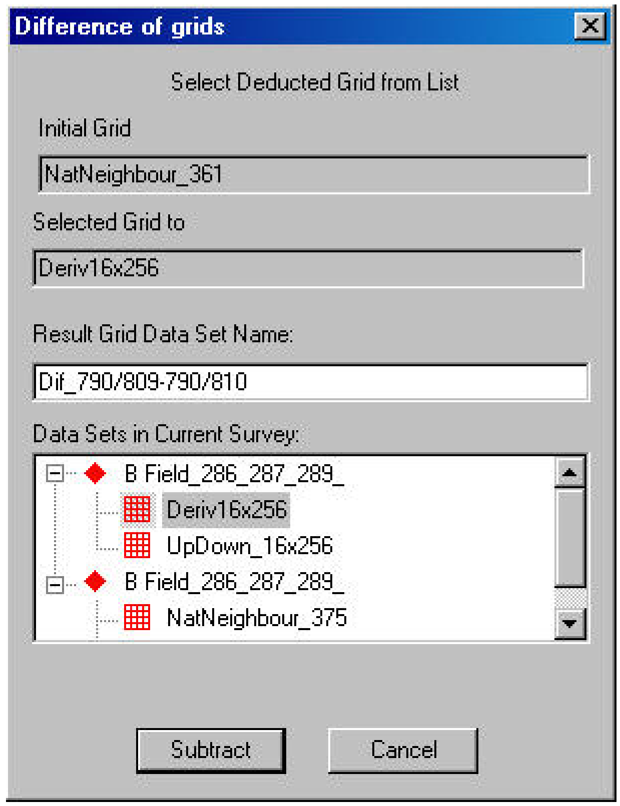
From **Grid Information** window, export to QCT format can be accessed by clicking the **Export to qct-file** button at its bottom. The **QCT Export** window differs from its XYZ version described above only by checkbox in the top left corner:




Further procedure is identical to XYZ case.

## Calculate the Difference of Two Grids

To create a new data set which contains the difference of two grids, first select the grid data set that you would like to subtract from, then click the**Difference of grids** button at the bottom of the [Grid Information]() dialog. The **Difference of grids** dialog appears:




In this dialog, the grid data set that you selected is displayed under **Initial Grid**. The name of a default grid which is to be subtracted from the initial grid is displayed under **Selected Grid To**. This grid can be changed by selecting a different one from the **Data Sets in Current Survey** list. The data sets which are not selectable are displayed with a red diamond next to the name. The selectable grids are listed under the data set name. The name of the resulting data set containing the difference of data is displayed under **Result Grid Data Set Name**. This default name may modified. Click **Subtract** to create the new grid data set.

# Processing Tools

## 1D Frequency Domain Inversion

### Starting FEM Inversion

To perform FEM Inversion, select a frequency domain data set from the main Database page. Click on the  button on the main EMIGMA toolbar and the window for Step 1 of the inversion wizard will appear.

## Step 1. Style and Data Selection.

**Inversion Algorithms** Trust Region: A type of steepest descent intended for when model parameters are less than number of data.
Enhanced Conjugate Gradient: A smooth Occam Conjugate Gradient algorithm
Standard Least Squares: a simple Marquardt approach

L1 Linear Regression: Model functional is a L1 functional and a linear regression technique is utilized

**Frequencies**
List of available frequencies. Select the frequencies to be used when calculating the inversion.

**Components and Separations**
Includes the selection of different data vectors (Mx,My,Mz) as well as separations.
List of available components. Selected frequencies will be updated when selected components are changed so a frequency will not be selected if a corresponding component has not been selected.
The example above is for an EM34 data set, which has 3 frequencies and 3 TX-RX separations.

## Step 2. Starting Model Specification.

**Generate uniform layering** A layer model is created by defining the total number of layers and their combined thickness as well as an initial resistivity or susceptibility. The thickness of each layer will be calculated to be all equivalent. The halfspace layer is not used in calculating the individual layer thicknesses.

**Inversion Parameters**
Inversion will calculate resistivity, susceptibility or both. Only the inversion technique, *Enhanced Conjugate Gradient Occam with Susceptibility Extension* allows for joint or susceptibility inversion.

If susceptibility is to be a model parameter then the Inphase of the data must be selected.

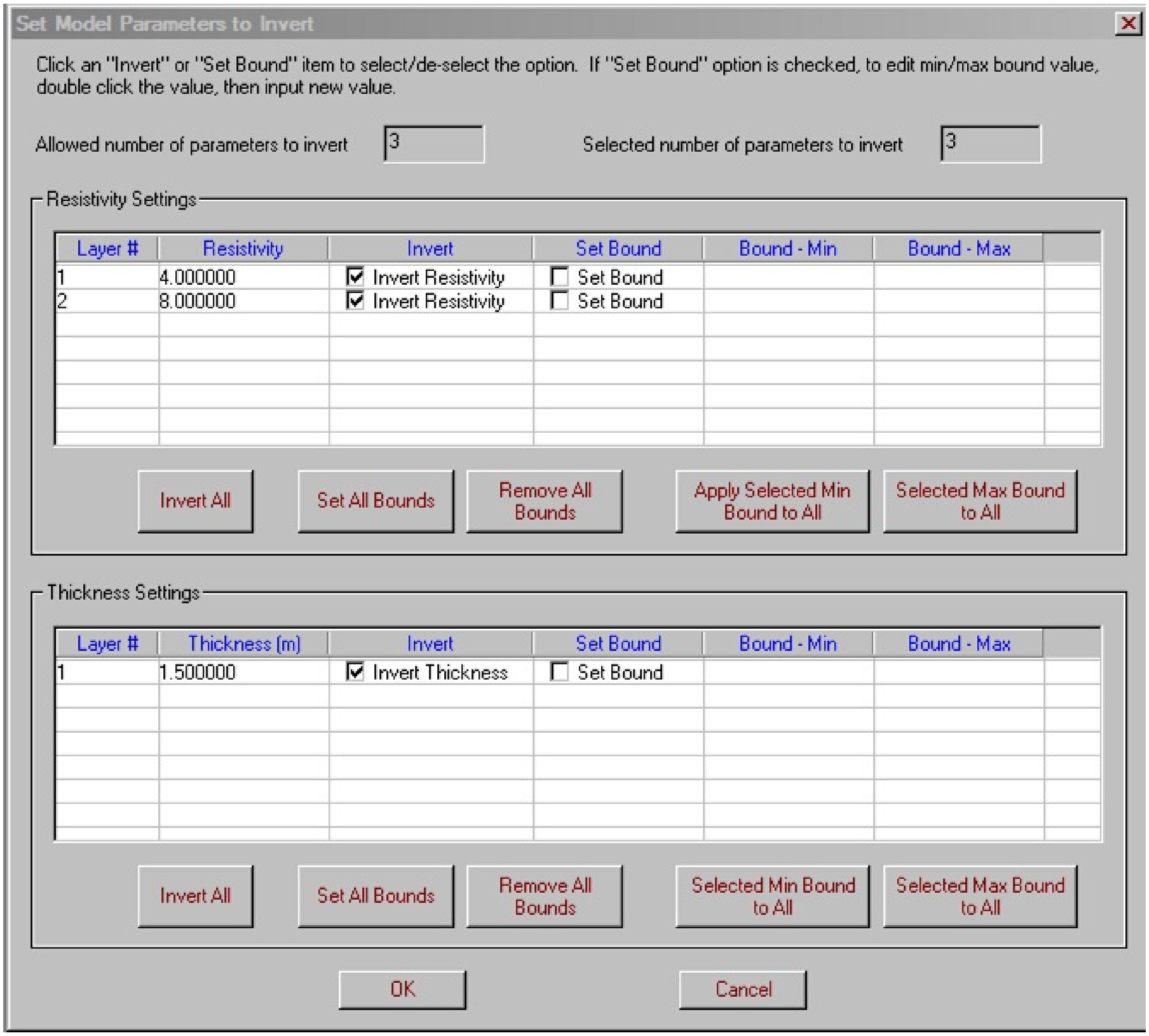
**Edit or Add to List**
Add a layer to the list of layers. The layer indicated by Layer # will be replaced or inserted. To edit a layer, first select the layer and alter its resistivity and thickness and click Replace.

**Import a layer model**
Attempts to model the data various forward modeling may be imported as a starting model. It is recommended that the user first try to forward model the data and utilize this information in their starting modelings.

**Remove**
To remove a layer select it and press Delete key.

**Model Constraints**
If utilizing the Trust Region technique then the model parameters may be constrained. Also, one can select which parameters are to fixed during the

inversion.


Set Model Parameters to Invert
Click an "Invert" or "Set Bound" item to select/de-select the option. If "Set Bound" option is checked, to edit min/max bound value, double click the value, then input new value.
Allowed number of parameters to invert
3
Selected number of parameters to invert
3
Resistivity Settings
Layer #
Resistivity
Invert
Set Bound
Bound - Min
Bound - Max
1
4.000000
Invert Resistivity
Set Bound
2
8.000000
Invert Resistivity
Set Bound
Invert All
Set All Bounds
Remove All Bounds
Apply Selected Min Bound to All
Selected Max Bound to All
Thickness Settings
Layer #
Thickness (m)
Invert
Set Bound
Bound - Min
Bound - Max
1
1.500000
Invert Thickness
Set Bound
Invert All
Set All Bounds
Remove All Bounds
Selected Min Bound to All
Selected Max Bound to All
OK
Cancel

## Step 3. Inversion Settings.

For more detailed information about the settings on this page, see Review of 1d Inversion Methods. This page varies depending upon the inversion technique.

### Data Type

Choose the kind of data that needs to be fit. Quadrature only will be automatically selected if there is no inphase data. Otherwise Inphase/Quadrature is selected.

**Inversion Parameters**
***Max Iterations***
Number of times the algorithm will try to match the data with a resistivity. The maximum number of jacobian calculations while converging to the solution.

***Target Fit***
Algorithm will continue running until the model norm becomes smaller than the target fit.

Data_norm = RMS(observed_data – calculated_data)
Check if (data_norm<target_fit) then
Check model epsilon

***Model Epsilon***
Once the target fit has been met, if subsequent inversion produce an rms change in model parameters less than epsilon then inversion is deemed to be accomplished.

Change = RMS((model_resistivity – last_model_resistivity)/max_resistivity)
If (change<model_epsilon) then successful fit

***Fit Tolerance***
Determines how close to determine the final fit

***Min Tolerance***
Determines how accurately the search algorithms determine minima in the fit.

**Reset Default**
Restore all the inversion parameters to their default values.

**Resistivity/Susceptibility Limits**
Calculated resistivity will lie between the upper and lower bounds. One or both of the bounds can be ignored but this is not recommended.

**Join adjacent layers within a specified contrast**
Reduces the number of layers by looking at the resistivity of adjoining layers (starting from top to bottom). If the relative difference(in %) between them is less than the user defined "percent", then it replaces both layers by one having the average resistivity.

**Ratio of susceptibility to conductivity parameters**
Available only for joint inversion. With ratio s,
Weight related to conductivity parameters----> 1/(1 + s)
Weight related to permeability parameters----> s/(1 + s)
The resulting parameters are multiplied by these weights.

**Log(Conductivity)**
Decides whether to take the logarithm of conductivity instead of conductivity as the model parameter. NOTE: this option is available only for conductivity not for susceptibility.

## Step 4. Run and Output.

**Use inversion result from previous location as initial model for current location** As this is a 1D inversion and for this approach to be appropriate, the resistivity structure of the earth should change very slowly. Thus, we can initiate the inversion of the next station with the model from the previous station.

Click on **Run** button to perform the inversion.

## Review of 1d Inversion Methods.

(Based on information provided by 1d-inversion source code, Numerical Recipes in C , and paper by Constable et al, Geophysics 52 (1987) 289)

The inversion methods implemented in the 1-d inversion code can be classified within two categories: Regularization (L1 and L2) and Damped least squares (M).
The first one obtains a solution by minimizing a combination of data residual and model properties,
the latter acts as a regularization operator that deems the process to converge.
In the last one there is no requirement on the model properties, which means that convergency should be achieved by dumping the process in some way.

The following presents a review of the methods and their connection with the inversion code parameters. Testing and comparison is provided at the end of the document.

REGULARIZATION

This method seeks to minimize an objective function (φ) of the form

where model properties (φm) are minimized provided the data residual (φd) achieved a given misfit value (φ*). Depending on the norm used to describe each functional the L1 and L2 methods are introduced:

**L1-inversion:**

The objective function is defined in terms of L1-norm:

0'th derivative :

(1)

1st derivative

(2)

and where

with: di : datum-i, i=1,N

σi : variance of datum-i

pj : model parameter-j, j=1,M

Fi : forward model that connect parameter space (p) into the model datum-i

**L2-inversion:**

The objective function is an L2 norm:

1st derivative

(3)

2nd derivative

(4)

In both cases, the goal is realized subject to the constraint that the model must fit the data to a desired accuracy ( RMS misfit between the observed data and model data).

The solution is modelled after the Occam's inversion algorithm. Because the Jacobian is non-linear, a new "linearized" data set d' is defined as follows:

where **ci** is the model data. On each iteration of the inversion, a new Jacobian is calculated, based on the latest model. At each iteration a sweep through values of **μ** determines the value which minimizes the data norm.

Once solutions within the data norm threshold are found, the inversion continues until the model converges to a fixed solution. Convergence is controlled by an epsilon parameter(input). If the RMS change between model parameters on successive iterations falls below epsilon, then convergence is assumed to have occurred, and the routine returns successfully.

The relevant parameters that controls this inversion from the front end are:

***Minimize(L) (param_deriv) (only for L1)***

- Absolute values (0)==> minimizes eq. (1)
- Absolutes values of differences (1)==> minimizes eq. (2)

Note that for L2 there is no input to choose between minimizing eq.(3) or (4). The current implementation in the code minimizes eq.(4) only

***Inversion parameters***

| | |
|---|---|
| *Max Iterations (L,M)==> max_iter* | the maximum number of iterations before returning. One jacobian calculation per iteration |
| *Model Epsilon (L,M) ==> epsilon* | once the *target_fit* has been met, if subsequent inversion produce an rms change in model parameters less than |

| | |
|---|---|
| | *epsilon* then conversion is deemed to be accomplished. |
| *Min Tolerance(L) ==> min_tolerance* | determines how accurately the search algorithms determine minima in the fit. |
| *Fit Tolerance(L) ==> fit_tolerance* | determines how close to determine the final L1 fit -a fractional value. |
| *Target Fit (L,M) ==> target_fit* | tries to find the solution with the smallest L1 model norm whose data norm is no larger than *target_fit* |

In terms of data and model parameters:

(5)

(6)

where k-represent the iteration number , and **pmax** = max { pi }

The tolerance parameters are needed for the 1-d linear search. Fisrt to find the **μ**-value that minimizes the data norm (*min_tolerance* an absolute value) at each iteration, and then to refine the search parameter to achieve the target fit ( *fit_tolerance* a fraction value)

The iteration process is driven by the *target_fit* and *epsilon* parameters. The goal is to achieve the *target_fit* , and then check parameter convergency via *epsilon*. The iteration stops when both criteria are met

**DAMPED LEAST SQUARE (Marquardt)**

This method minimizes an objective function which includes only data residual:

Because of the nonlinearity of the problem, a straightforward minimization is likely to diverge, and this tendency must be counteracted by damping the process. This consists of systematically reducing the size of the change in the model from one iteration to the next. The Marquardt method modifies the Jacobian by adding a constant value to the diagonal, thus decreasing the size of the computed perturbation; the constant is chosen to decrease the misfit at each iteration (similar to the **μ**-parameter in the regularized methods) , for a fixed Jacobian. Iteration continues until the misfit achieved the *target_fit* value. In contrast to the L-methods, iteration may stop before the misfit has been reached if the change in model parameters is below the threshold defined by *epsilon*.

**TESTING AND COMPARISON**

Setting the inversion process:

- In order to have a meaningful inversion is necessary to start with a model whose total thickness (aside of the upper and bottom layers having large resistivity, that is air and half space) covers the expected inhomogeneous area. The number of layers should define a minimum layer thickness below the expected resolution. Constable et at suggested a layer number of 20 and decreasing layer thickness with depth. The current implementation of the code limit the layer number to 10.

The starting model during the testing process is (beginning after air and ending in half space):

4 layers of thickness 10
5 layers of thickness 5

Initial resistivity (alike for all the layers) of value 5 (other values were tested with similar results)

- The *target_misfit* is the leading parameter for the L-inversions (M-type stop iteration if change parameters are less than *epsilon*). Its value will depend on the data uncertainties. If the uncertainties are due to a zero-mean, Gaussian process that is independent in each of the observations, and ói are the associated standard deviations, then the data misfit is well known to be distributed as xi square. The expected value for the misfit is then just N (data points) :

  This means to have a **target_misfit** = 1 (default value in code)

  However simulated data does not seem to have associated a Gaussian noise, which means data is known precisely and minimization should be done with zero (optimally) misfit. However due to numerical errors (machine precision, forward model, etc) that is not possible and the minimum value needs to be found explicitly.
  The code does not seek the minimum value for the misfit, but a target value within a more general minimization (which includes model properties). Since this target value can not be set equal to zero (successfully), the following strategy is followed:
  - Keep fix the following parameters:
    Max Iterations=20
    Model Epsilon=0.1
    Min Tolerance=0.1
    Fit Tolerance = 0.01
  - start with *target_fit*=0
  - since convergency will not be achieved pick up the *target_fit* value reached previously (rounded off) and check for convergency . This is important since the solution should be

stable in change parameters (control by Model Epsilon), which was not checked in the previous run

- *epsilon* is a secondary convergency parameter for the L-inversions and a primary (along with *target_misfit*) for the M-type. A reasonable value seems to be in the range 0.1-0.01. Smaller number are not achievable since given its definition eq.(6), the convergency will be driven by the largest model parameter (in general will be the layer related to half space)
- *min_tolerance* and *fit_tolerance* (needed only for the L-inversions) are related to 1d-search process (like bisection method or golden search). They define a tolerance threshold for the linear search. *min_tolerance* is related to the best ì that minimizes the data fit at each iteration, and the *fit_tolerance* to the fractional tolerance as the misfit is achieved. They do not produce a significant effect if they are change from their default values. However they should not be chosen too small (less than 1.e-4 in the case of *min_tolerance* ) since they will delay the linear search unnecessarily

**TESTING:**

The following two models will be described (existing simulated data from EMIGMA):

1. Testmax_min.pev

| Thickness (T) | Resistivity(R) |
|---|---|
| 50 | 20 |
| half space | 10000 |

Frequency (F)

222
444
888
1777
3560

2. Testem34.pev

| Thickness (T) | Resistivity(R) |
|---|---|
| 10 | 20 |
| 20 | 5 |
| half space | 10000 |

Frequency (F)
400
1600
6400

After running according the previous setting, the best behavior was obtained by the L1 algorithm with the option "absolute differences":

Test_maxmin.pev

| (20) | (R) | |
|---|---|---|
| 4x10 | 20 | |
| 2x5 | 20 | |
| 3x5 | 8300 | iter=5 |
| half space | 8300 | target_fit=1.19-3 |

Test_em34.pev

| (21) | (R) | |
|---|---|---|
| 10 | 20 | |
| 2x10 | 5 | |
| 10 | 518 | iter=3 |
| 5x5 | 830 | target_fit=1.79-3 |
| half space | 830 | |

All the frequencies were considered in the solution. 1 frequency data did not resolve the problem. The model test_em 34 presented several choices for "separation&components", only the first one gave meaningful result (if all or any of the other were chosen the result will go nowhere)

The initial value for the resistivity did not matter to find the solution (only affect the iteration number).

Next method was the L2 algorithm which in general showed more structure than needed. It gave good resolution for the upper layers but failed in describe the gradient resistivity. This method may need more layers to be more efficient. It has the numerical advantage over the L1, that its gradient (over the model parameters) is a continuous function

Marquardt failed to give more information than the input as the initial solution from the L1 or L2 result. According to *Constable et al* , in this method the solution lies close to the initial guess, because the modified Jacobian keeps the changes small at each step of the process; thus the resultant model is strongly influenced by the initial choice.

The option "absolute norm" eq.(1), for the L1 inversion gave good results provided the layer number be small.

## Step 1. Style and Data Selection.

**Frequencies** List of available frequencies. Select the frequencies to be used when calculating the inversion.

**Components**
List of available components. Selected frequencies will be updated when

selected components are changed so a frequency will not be selected if a corresponding component has not been selected.

**Inversion Techniques**

Overparametrized Occam smooth inversion or Trust Region (under parametrized)

## Step 2. Starting Model Specification.

**Generate uniform layering** A layer model is created by defining the total number of layers and their combined thickness as well as an initial resistivity or susceptibility. The thickness of each layer will be calculated to be all equivalent. The halfspace layer is not used in calculating the individual layer thicknesses. Once generated, starting resistivities and thicknesses may be edited.

**Inversion Parameters**
Inversion will calculate resistivity, susceptibility or both.

**Add to List**
Add a layer to the list of layers. The layer indicated by Layer # will be replaced or inserted.

**Join Layers**
Join the selected layer with the following one.

**Import a layer model**
The results from all the successful inversion are saved so any one these can be used as a starting model.

**Remove**
To remove a layer select it and press Delete key.

## Step 3. Parameter Settings

For more detailed information about the settings on this page, see Review of 1d Inversion Methods.

**Data Type**
Choose the kind of data that needs to be fit. Inphase/Quadrature is selected

by default.

**Resistivity Limits**
Set the minimum and maximum bounds for resistivity.

**Inversion Parameters**
***Max Iterations***
Number of times the algorithm will try to match the data with a resistivity. The maximum number of jacobian calculations while converging to the solution.

***Target Fit***
Algorithm will continue running until the model norm becomes smaller than the target fit.

Data_norm = RMS(observed_data – calculated_data)/observed_data

**Reset Default**
Restore all the inversion parameters to their default values.

## Step 4. Run and Output. MT.

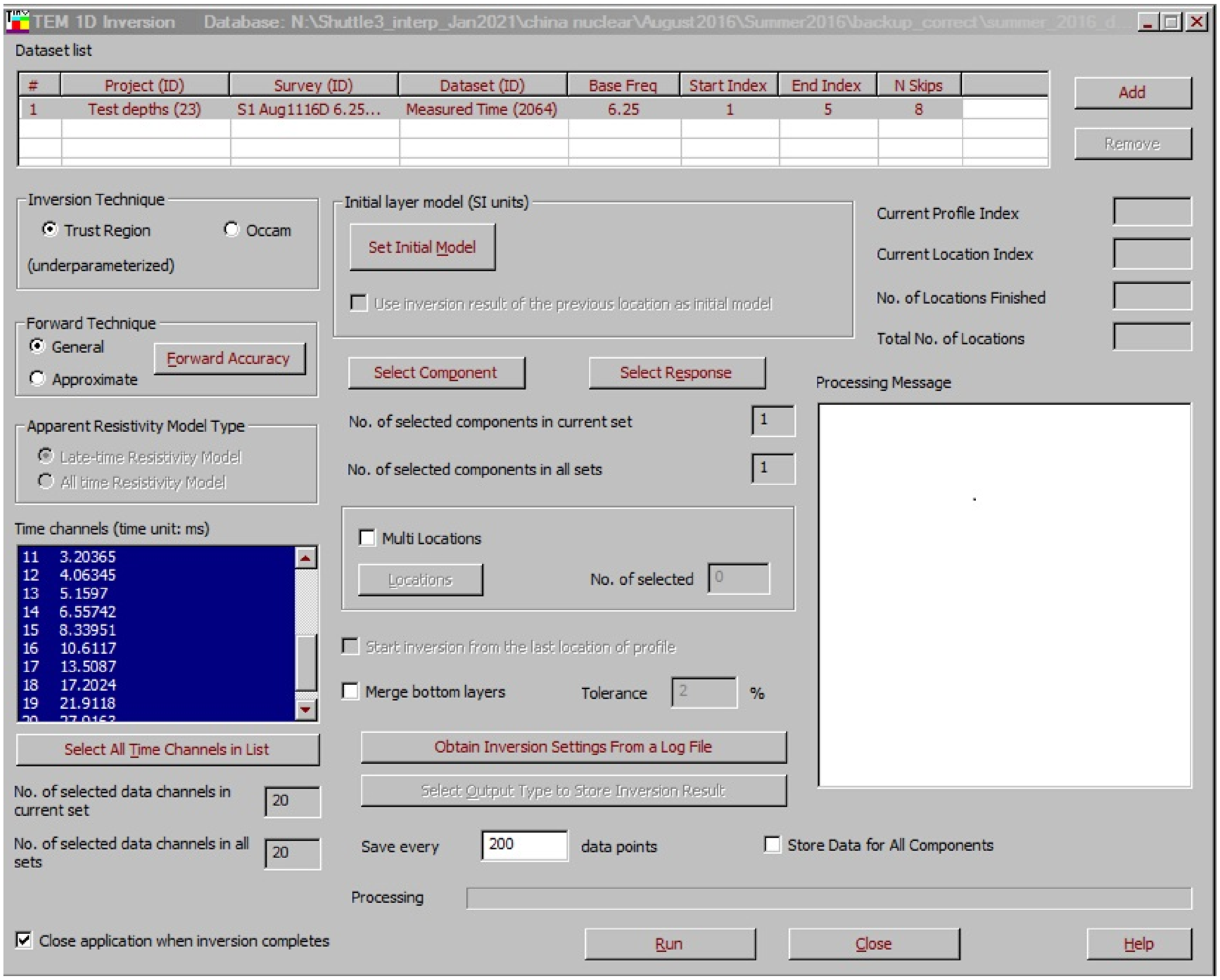
**Use inversion result of previous location as initial model** With this option selected, at the start of inverting a new location, the model from the previous inversion is utilized as a starting model. This ensures some spatial model smoothing and helps limit non-uniqueness.

**Run**
Start inversion process. Inversion is performed one location at a time and results of each inversion are displayed in the Output Info edit box.

**Clear List**
Erases contents of the Output Info edit box.

# Using 1D TEM Inversion

To start 1D TEM Inversion, select the data set with time domain data you wish to work with from the Database dialog. Click the  button on the main toolbar to launch the main window:



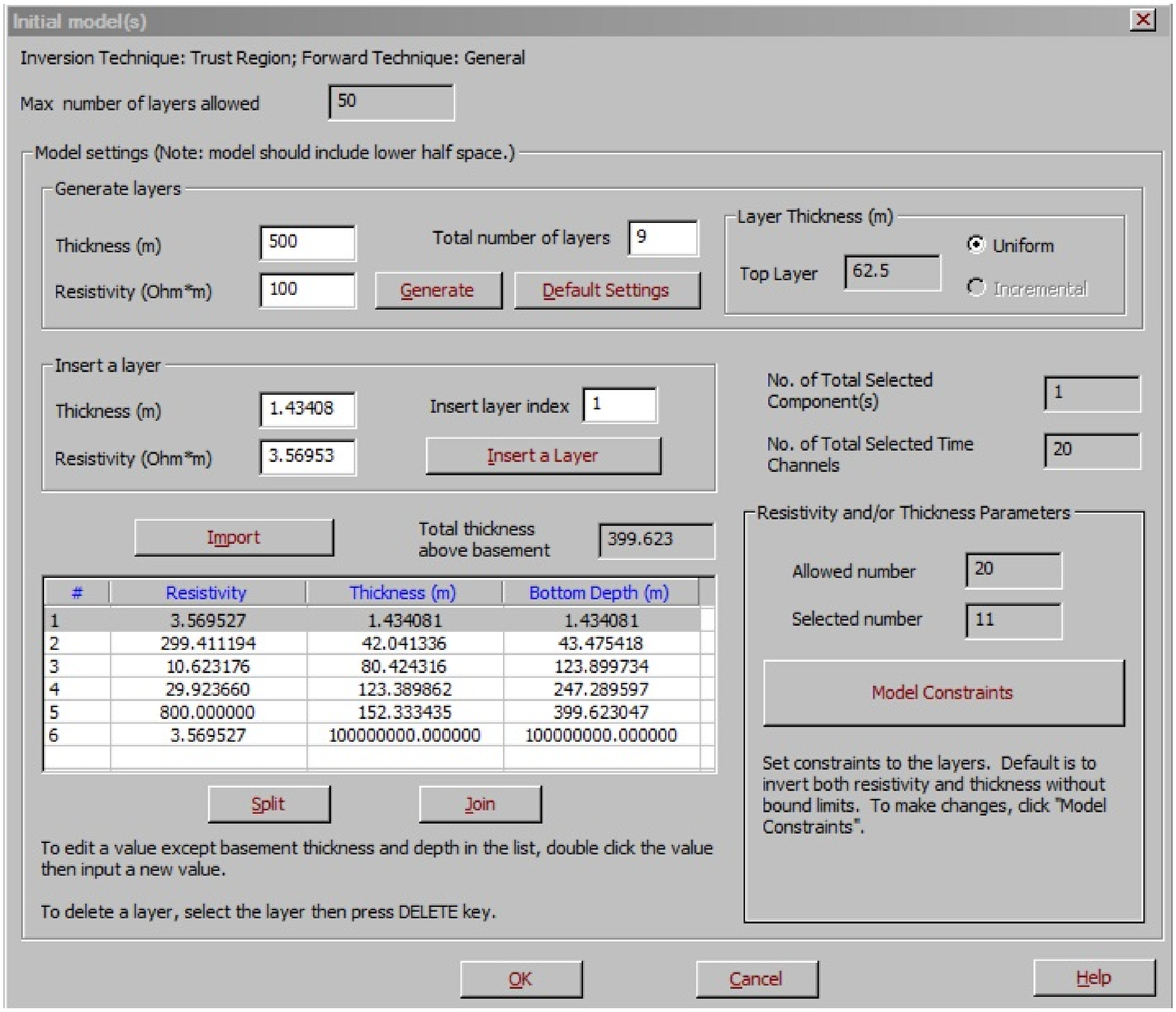

1. A log file is prepared each time an inversion is executed. If you would like to reuse the settings from the last inversion run, click the **Get Inversion Settings From a Log File** button, select the log file, make any modifications required or skip to step 9. Each data set has an ID number in the dataset which can be seen on the main interface. The log file for that inversion dataset is stored with the data set ID number. The last inversion also uses the default log file name.

2. Choose between the two inversion techniques:
   **Trust** - underparametrized technique that inverts for layer thickness and resistivity
   **Occam** - overparametrized technique that only inverts for resistivity with layers fixed
3. Choose a forward technique:
   **General** requires that the bandwidth and accuracy of the transform to time domain be specified by clicking **Forward Accuracy**.
   **Approximate** gives a choice of two **Apparent Resistivity Model Types** and is only for In-Loop data.
4. Select time channels that will be used for inversion.
5. Click the Select Component button to select which components you wish to fit in the inversion process. In the window that opens, a set of time channels may be chosen for each component. The time channel selection here will be reflected in the time channel list on the main window.
6. Click the **Select Response** button to choose the data response that will be used. Only applicable for synthetic data. Field data is stored with only one response type.
7. Multiple surveys may be chosen for joint inversion using the **Add** button at the top right.
8. Select the **Multi Locations** option to specify the locations to be used during the inversion. Selecting the **Advanced** option will allow you to select components and time channels for each location. Click the **Select Locations** button to make changes to your selections.
9. Specify a single initial model or multiple initial models.
10. Check **Use inversion result of the previous location as initial model** for the specified initial model to be used only for the first location. Subsequent locations will use the final model of the location before it. Most useful for large datasets.
11. Click **Run**
12. If the current survey contains inversion results, the **Select output type** window will appear. Choose **Create a new dataset to store the inversion result** to save to a new dataset otherwise see Completing an Unfinished Inversion
13. Inversion parameters are requested before processing begins. The default values should be quite appropriate.

14. Inversions are saved every N points as chosen by the user and also at the end of each profile.
15. Click **Close** to complete the active inversion and save the entire progress
16. Adjusting to GPS elevations
17. The inversion result can be viewed graphically in the PEX Show tool.

## Specifying a Single Initial Model

Click the **Set Initial Model** button and its window appears:




To load a layer model from another data set (e.g. forward model):

- Click **Import** to load an entire or part of a layer model from another data set within the database.

To create a set of identical layers:

- Specify the layer resisitivity and thickness as well as the number of desired layers then click **Generate uniform layers**. The deepest layer generated will be the basement and will have a thickness of 1e+8. An incremental layer thickness can also be chosen.

To insert one layer:
- Specify the layer resistivity and thickness.
- Click **Insert a layer** and a new layer will be inserted at the location specified by the **Insert layer index**.

To delete a layer:
- Select the desired entry from the list and delete with the DELETE key.

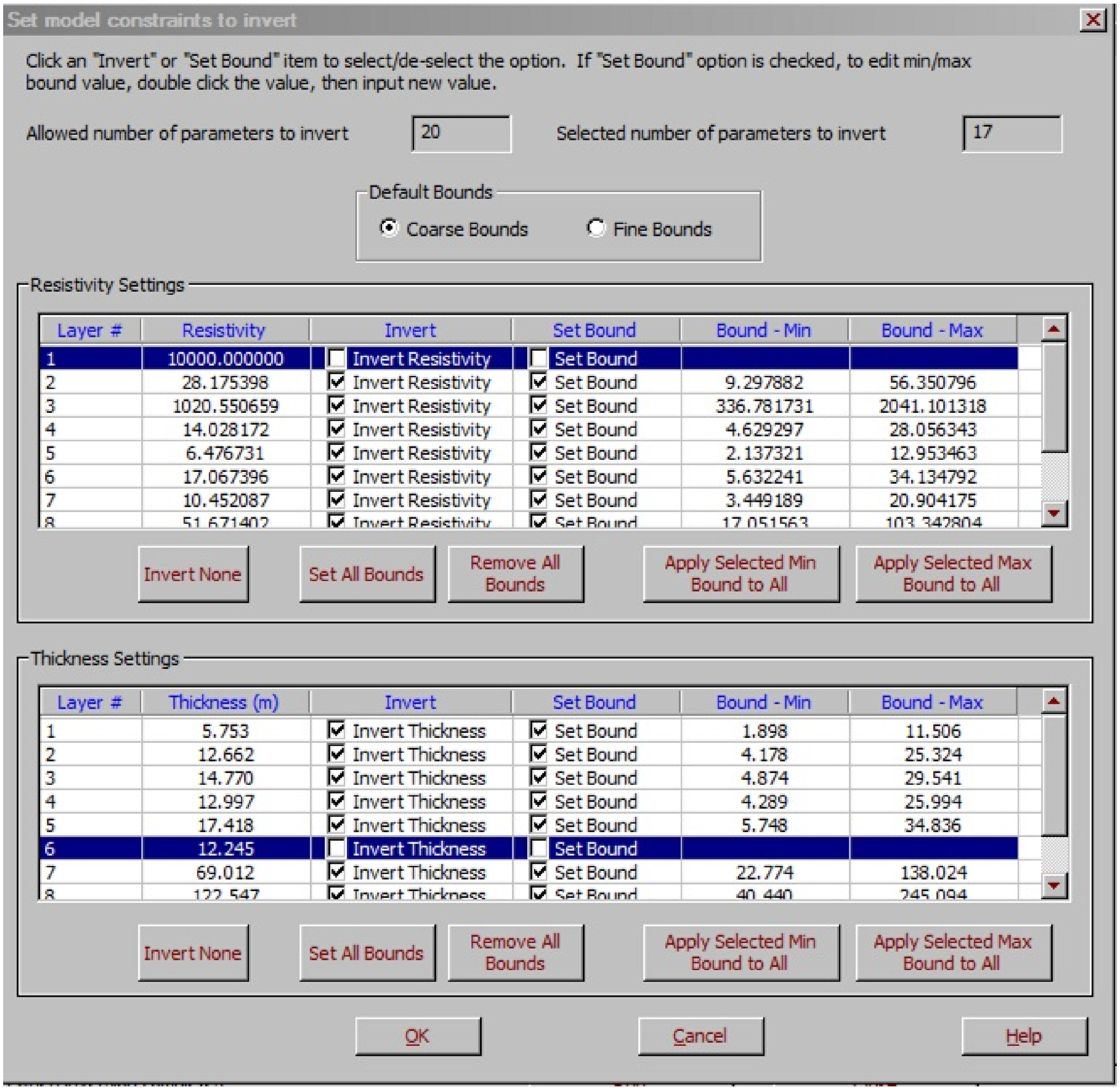
To edit a value in the list:
- Double click the value then input a new value.

To split a layer
- Select a layer, click **Split** and it is divided into two equal parts.

To join layers
- Select a layer, click **Join** and then select the 2 layers to join in the interface.

To constrain model parameters:
- Click the button labelled **Model Constraints**.

# Constraining Model Parameters



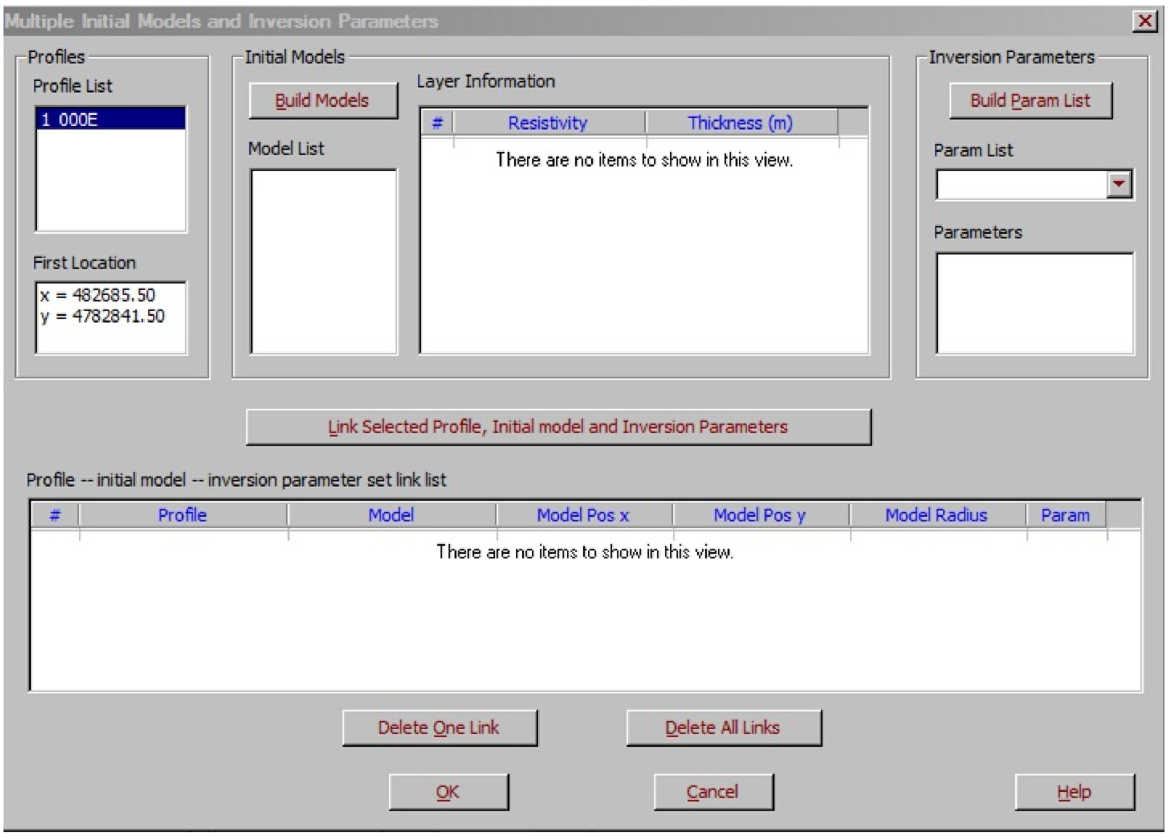

- If available, the **Invert** columns have checks to indicate which parameters will be inverted. **Selected number of parameters to invert** will be updated everytime an **Invert Resistivity** or **Invert Thickness** checkbox is checked or unchecked. That value must not exceed the **Allowed number of parameters to invert**.
- Click a **Set Bound** checkbox to set bound values for the parameter. The default bound values can be edited by double clicking on the value.

- Click the **Set All Bounds** button to give bound values to all the selected parameters.
- Click the **Remove All Bounds** button to let there be no bounds on the selected parameters.
- Click the **Apply Selected Min Bound to All** or **Apply Selected Max Bound to All** buttons as a shortcut to setting all the minimum or maximum bounds to the selected value.
- All bounds may be individually edited if required

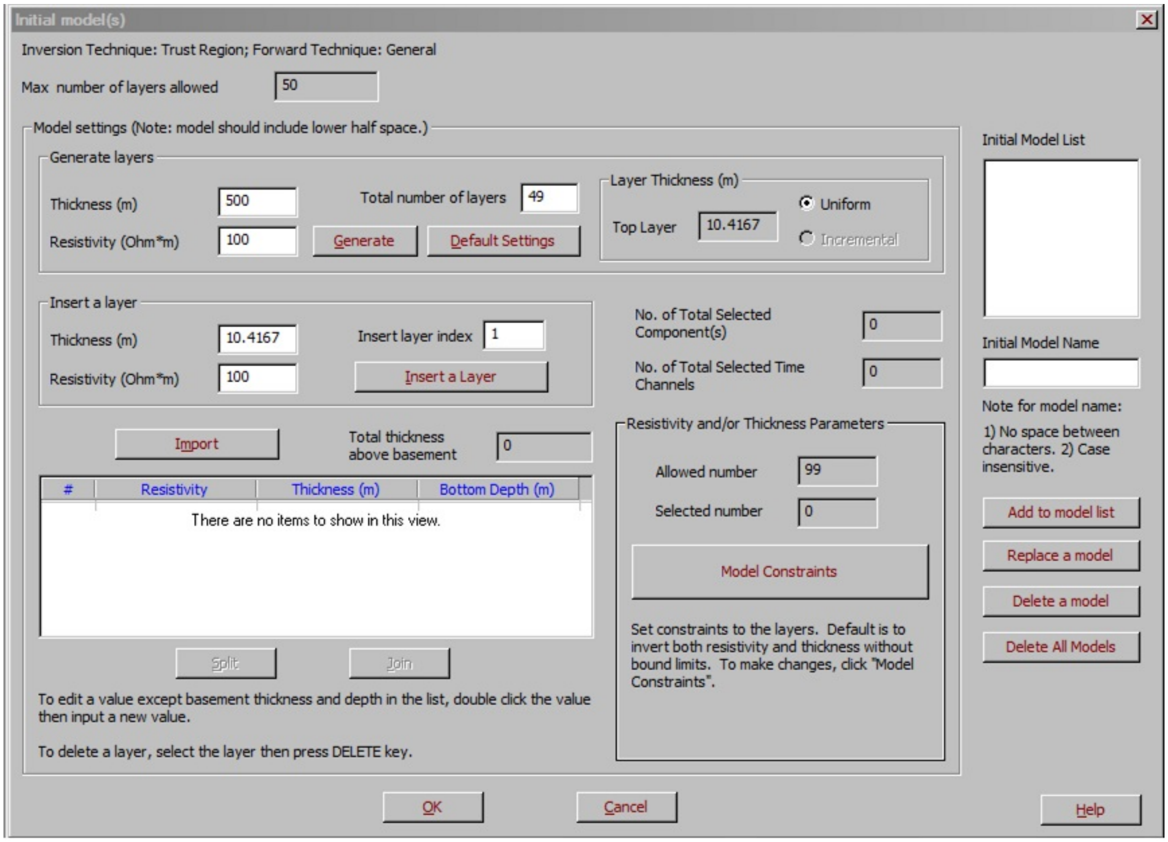
## Specifying multiple initial models

For a longer survey, the starting models may differ in different areas of the survey and this possibility can be utilized.

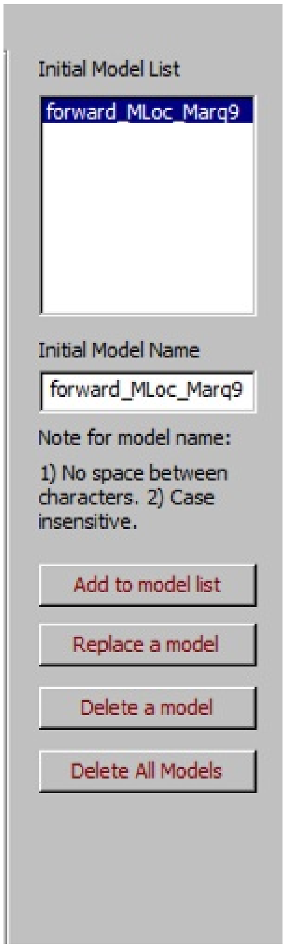
1. Check **Use advanced initial model settings** on the main window and click the **Set Initial Model** button next to it. The following window appears:




2. The layer models you would like set as initial models need to be selected. Click the **Build Models** button and the **Initial model** window will appear:

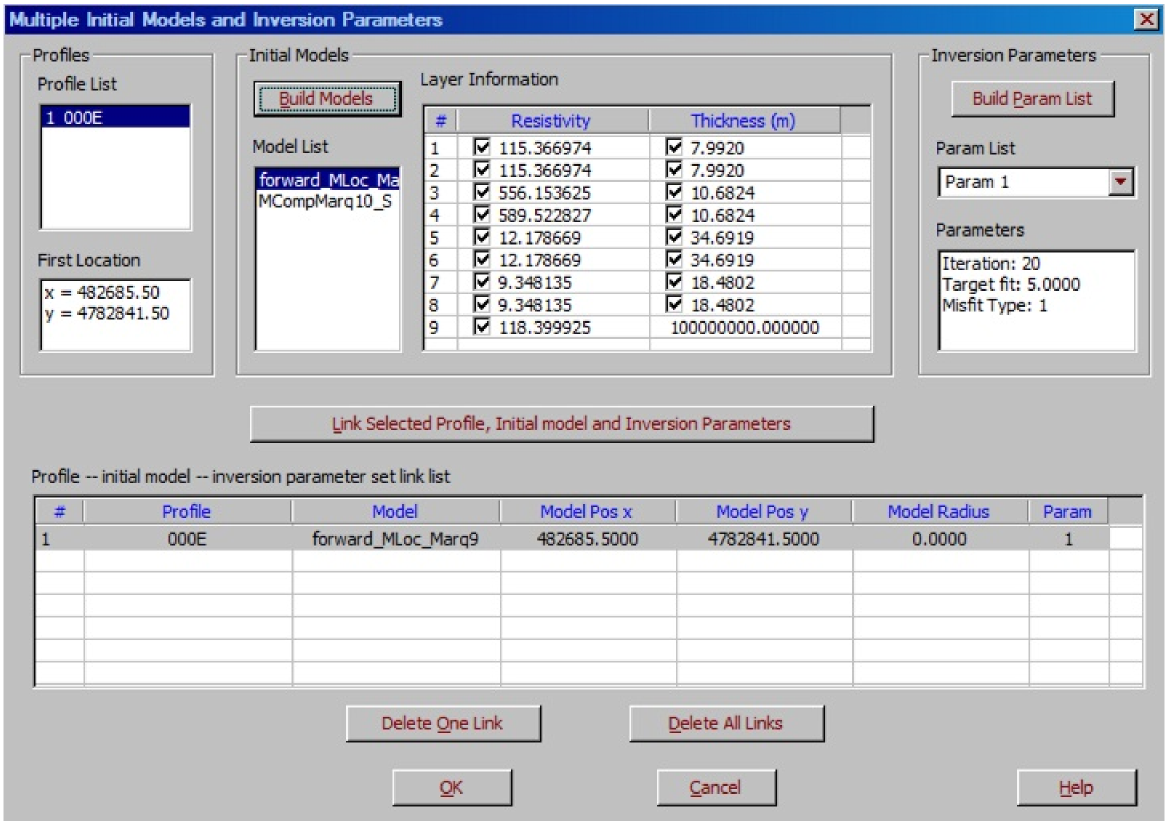



The following interface will be added:




Define a layer model either by importing or creating from scratch. Click on the **Add to model list** button and it will be displayed. Add additional models to the list in the same fashion.

3. Any model in the list can be modified by selecting it, making the necessary changes and clicking **Replace a model**. Models can be deleted from the list by clicking **Delete a model** after it has been selected or clicking **Delete All Models**. Once complete, click OK and you will return to this window:

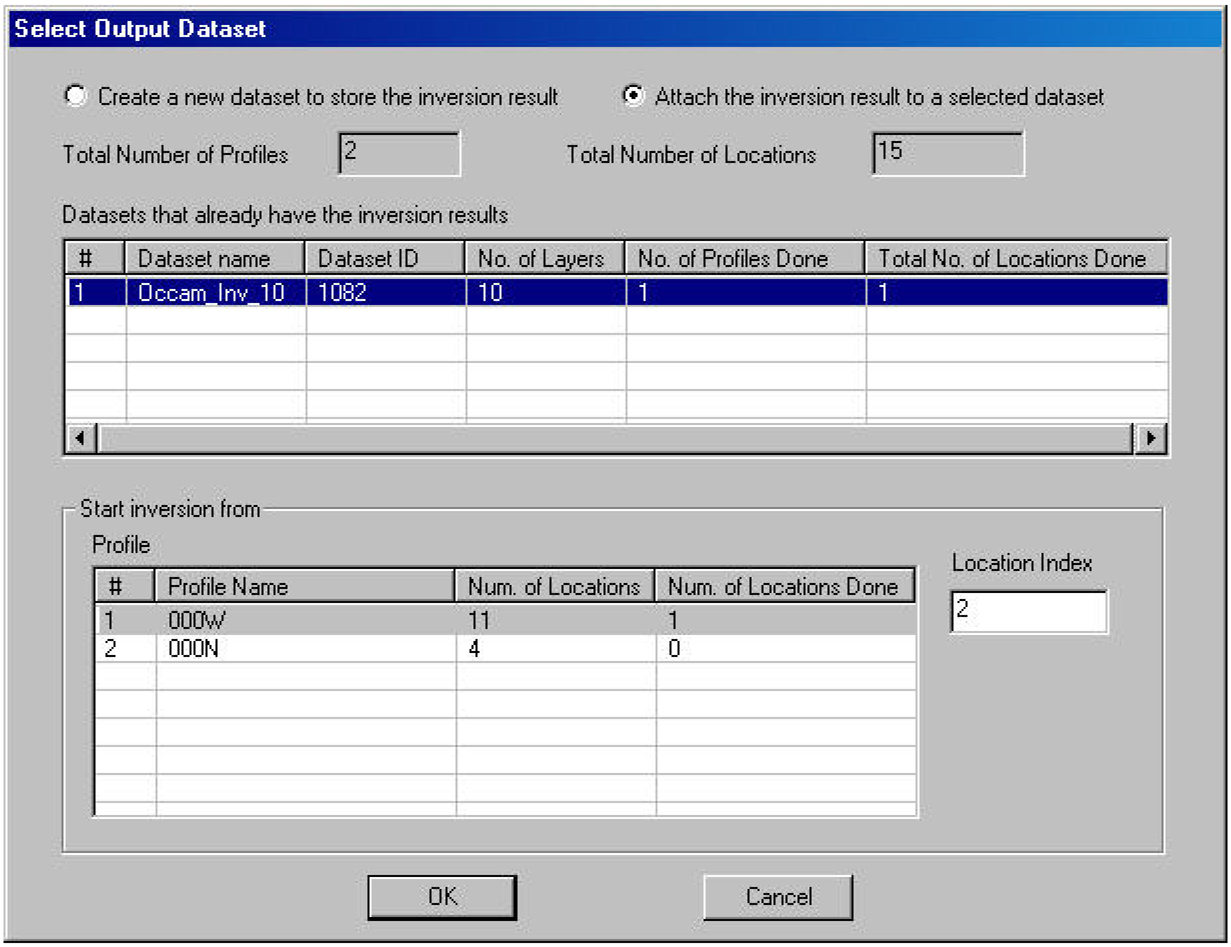



4. The desired parameters to be used must be specified by clicking the **Build Param List** button. In the resulting **Parameters for inversion** window, click **Add** to save the displayed parameters to the parameter list. Make changes to the parameters and click **Add** to save another set of parameters. **Replace** and **Delete** functions are also available.

5. Make a selection from the Profile List, the Model List and the Parameter List and click on **Link Selected Profile, Initial model and Inversion Parameters**. Different combinations of profile, model and parameter can be selected to create additional links.
6. A selected link can be deleted with the **Delete one link** button. All the links can be deleted with the **Delete All** button.
7. Click **OK**.

## Completing an Unfinished Inversion

It is possible to resume an inversion process after it has been interrupted or modify inversion results that have been completed. Select the survey which contains the inversion result you would like to resume. Start 1D Inversion. Click **Select Output Type to Store Inversion Result** on the main window and the **Select Output Dataset** window appears:

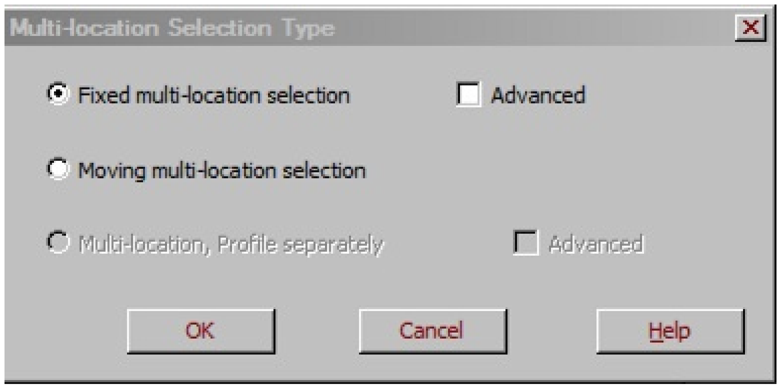



- Choose a dataset from the list labelled **Datasets that already have the inversion results**. The inversion result will be attached to this dataset.
- Edit the value labelled **Location Index** to specify which location to start the inversion process from. The number of locations processed so far is displayed in the lower list.
- Click **OK**.

- Make sure the settings are the same as those of the inversion result you are adding to and click **Run**.

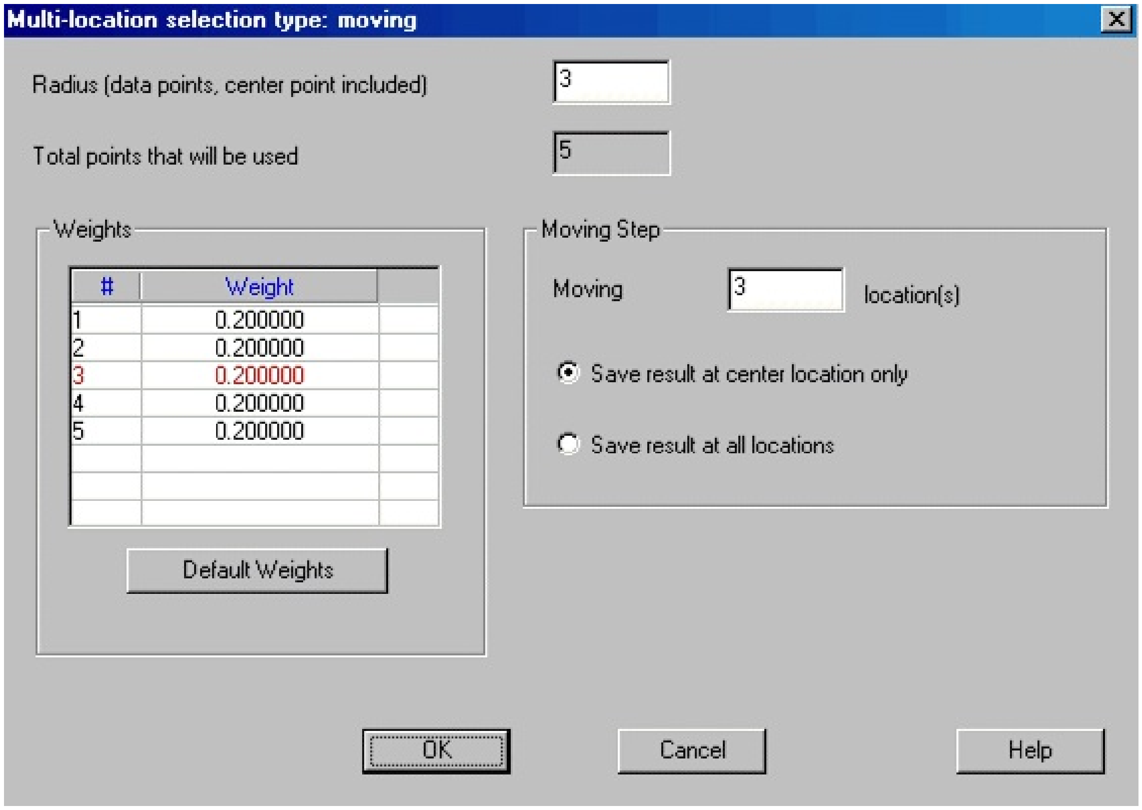
# Selecting locations for inversion

Select the **Multi Locations** option on the main inversion interface to specify the locations to be used during the inversion. The following window appears:



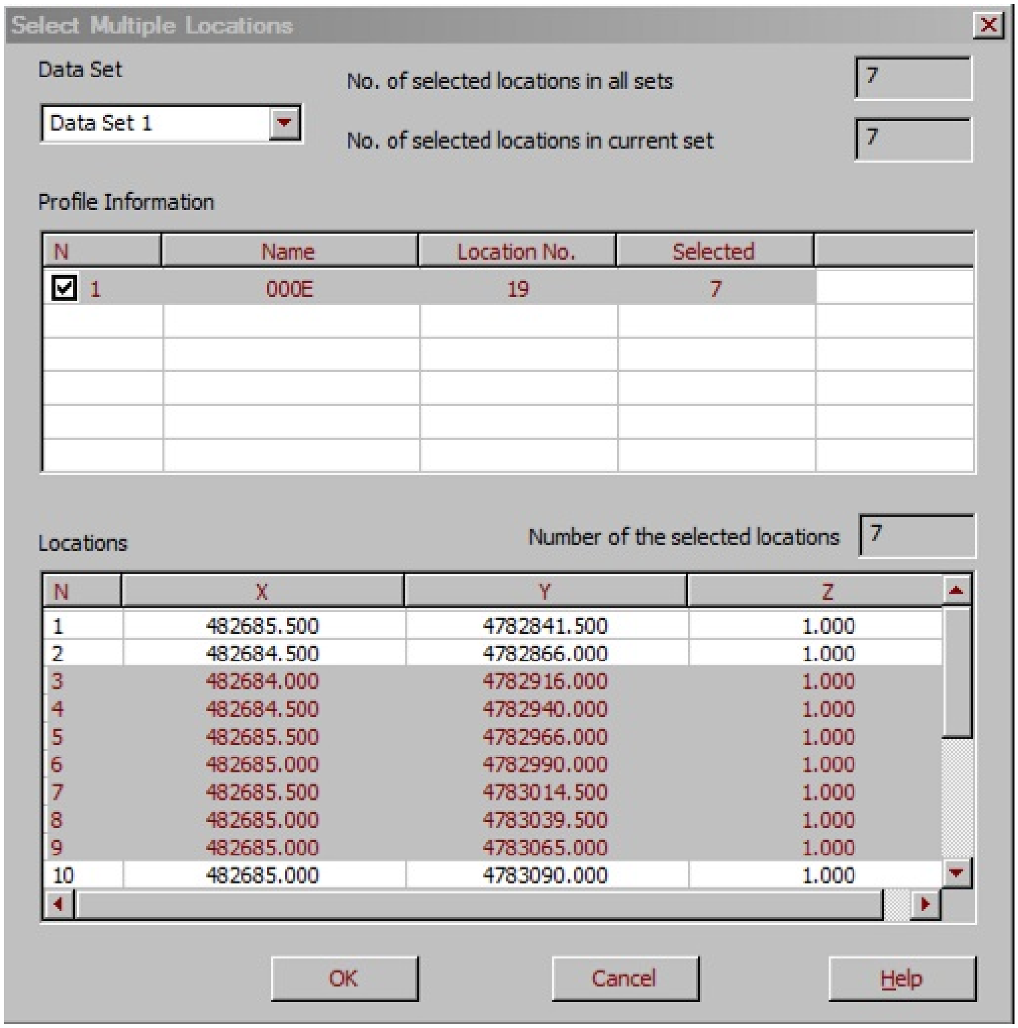

- If the dataset is configured as Moving and there are multiple data points, you can select either **Fixed multi-location selection** or **Moving multi-location selection**. These differ in that in the first case, there is one set of locations whereas in the second case, there is a moving window containing locations.
- If the dataset is configured as Fixed then **Fixed multi-location selection** should be chosen with **Advanced** selected if required

## Moving multi-location



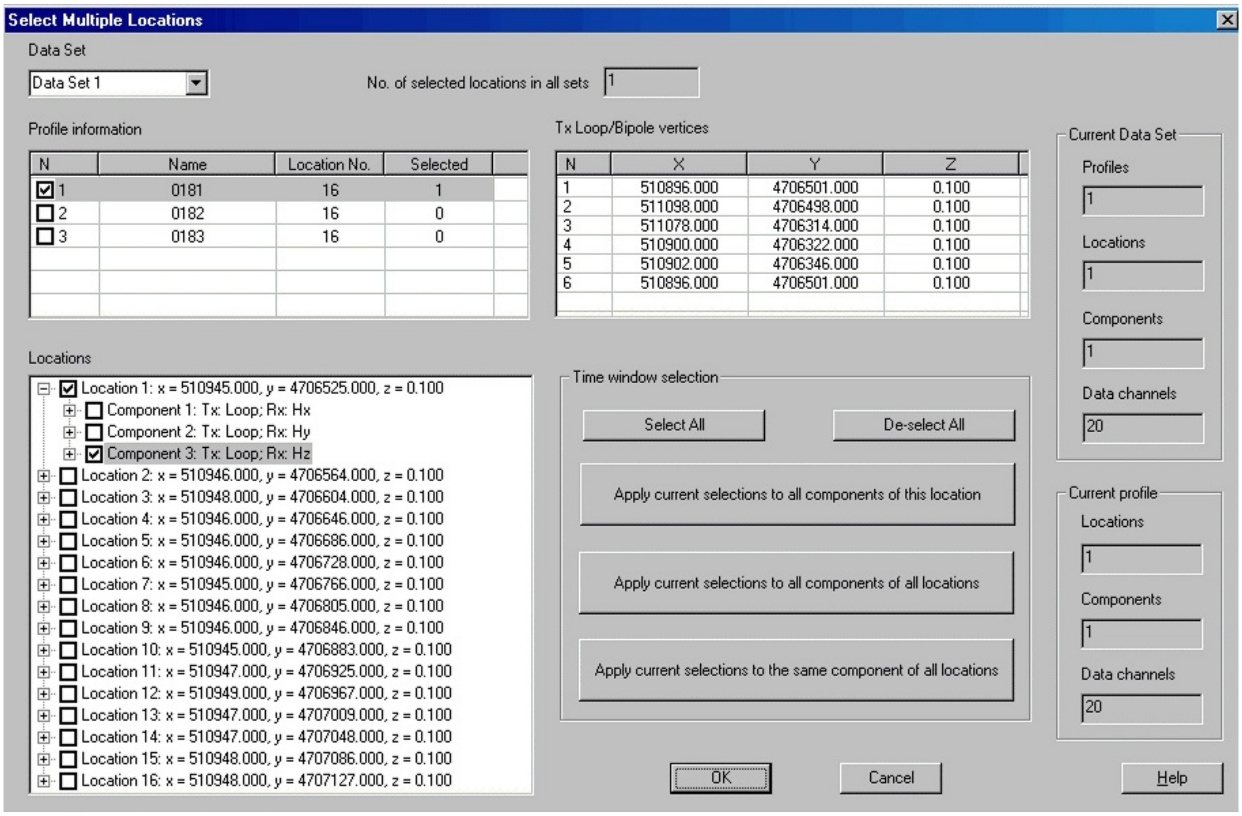

- With a moving system, the user selects a window containing a certain number of stations. The radius is the number of points outwards from the present data point being inverted. The window moves with each data point.
- User may define the Step of the Moving Window in number of datapoints along the profile
- Two Save options are provided
- Weights for different points in the moving window can be adjusted

## Fixed multi-location




- Each line in the **Profile Information** list has a checkbox. Click a profile's checkbox to include it in the inversion processing.
- The **Locations** list displays the coordinates of the profile that has been selected in the **Profile Information** list. Select the locations in the **Locations** list that you would like to include in the inversion processing.

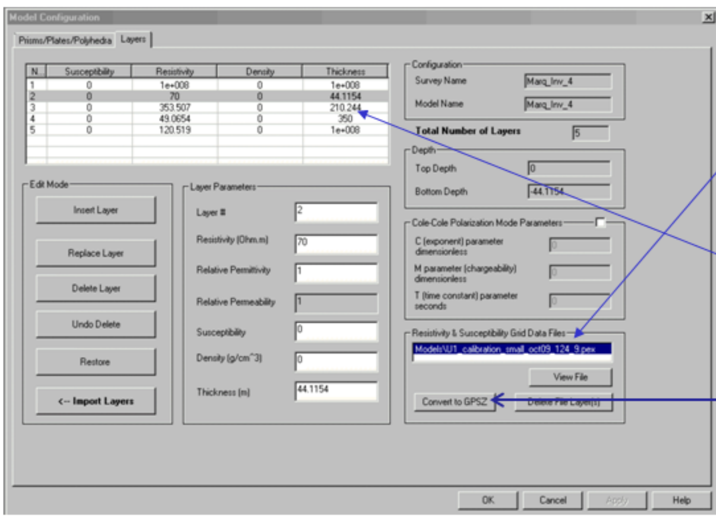
Selecting the **Advanced** option on the main window will allow you to select components and time channels for each location. Clicking the **Advanced** checkbox will display the following window:




- Each line in the **Profile Information** list has a checkbox. Click a profile's checkbox to include it in the inversion processing.
- The **Locations** list displays the settings for the profile that has been selected in the **Profile Information** list. Click the appropriate checkboxes in the **Locations** list to specify which locations, components and time windows will be included in the inversion processing.
- Clicking a **Component** line in the **Locations** list will display the transmitter vertex coordinates in upper right hand section of the window. This action will also enable the **Time window selection** interface.
- The **Time window selection** interface will allow you to select or deselect all time windows. Using this interface, you can also copy the time window selections of the currently selected component to other components or other locations.

## Adjusting to GPS elevations

Launch the **Model Configuration** interface by clicking the **Model** button on the main database page:



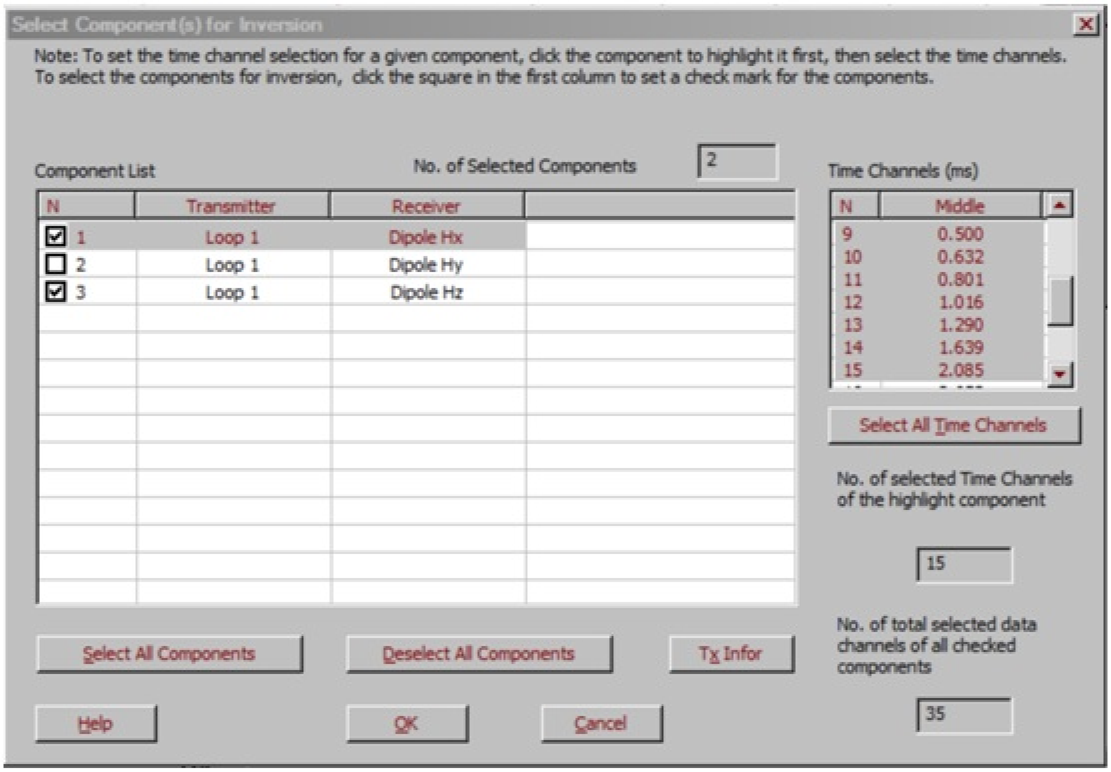

Attached to the database, in a subdirectory called "Models", are the inversion results in a simple ASCII XYZ file (*.pex) which may be viewed here. This file may easily be imported to another application although graphical viewing tools are provided within EMIGMA.

The 1D model for the final data point is also included.

The inversion may be adjusted w.r.t. GPS elevation in order to view in conjunction with topography. This is available ONLY when GPSZ is imported with the data.

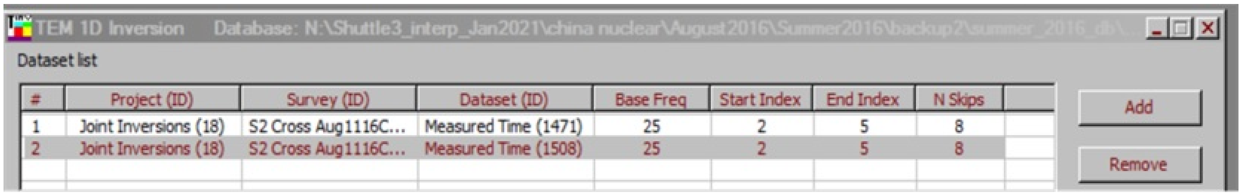
To view the results in EMIGMA close the window.

# Specifying Component Details




- Multiple Data Components: In this example, Hz is chosen using 20 channels and Hx using 15 channels. If moving loop data, then multiple vector components and multiple TX-RX separations may be selected. Also, it is possible to use data with different transmitters and select data associated with different transmitters.

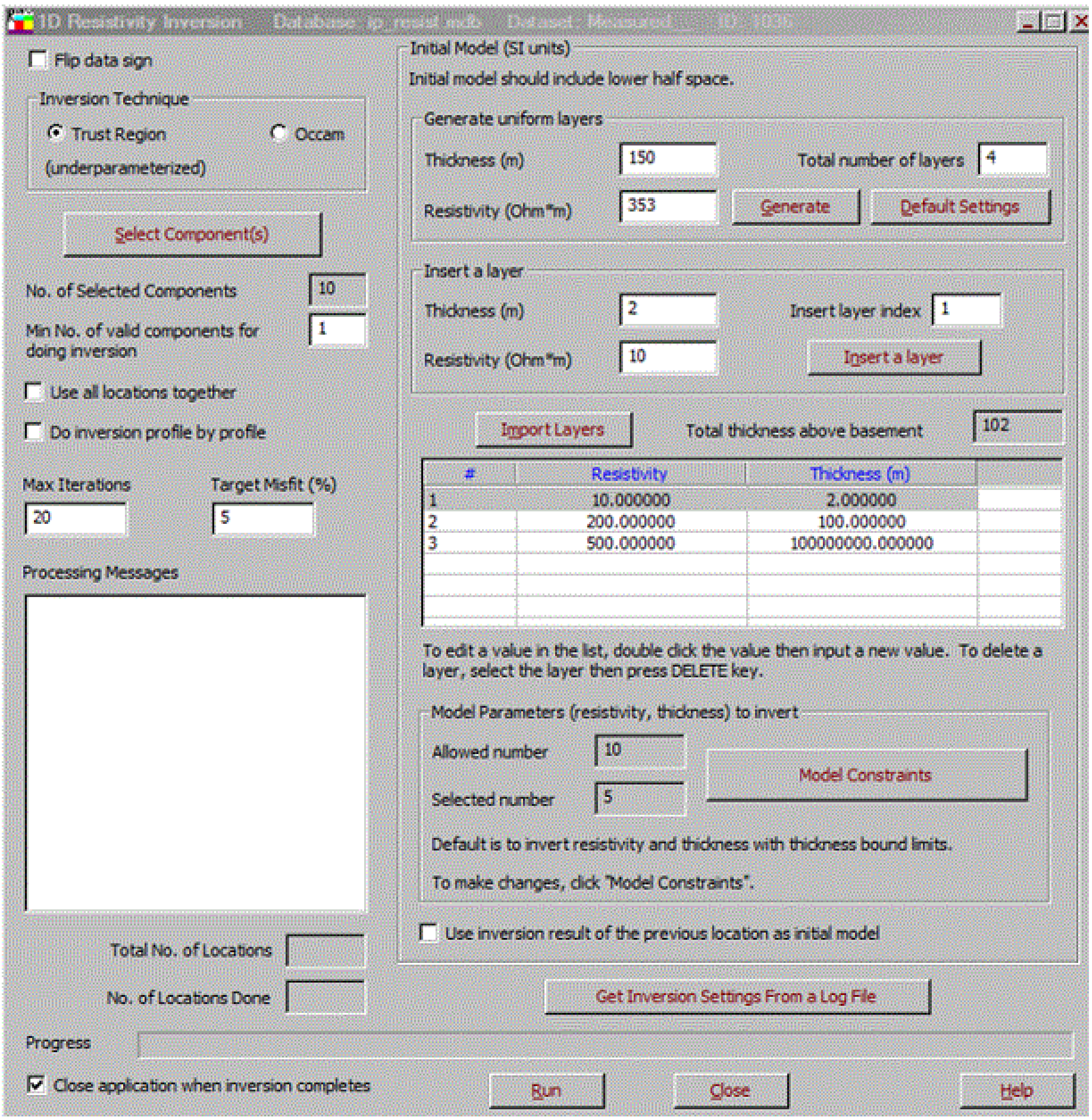
# Specifying Multiple Surveys




- Data may be chosen from multiple surveys to perform an inversion
- For example, a moving loop survey with 100m loop, 2 separations (inside and outside loop), 3 component 25Hz base frequency and the same but with 6.25Hz base frequency. Another example; 3 loops all with the same centre and a variety of measurement locations and 3 base frequencies and then all this data inverted together.
- This function has two main objectives. One is to be able to invert multiple base frequencies with the same survey setup and the second is to be able to invert fixed loop data with loops of different sizes and stations both close and far from the centre of the loops.

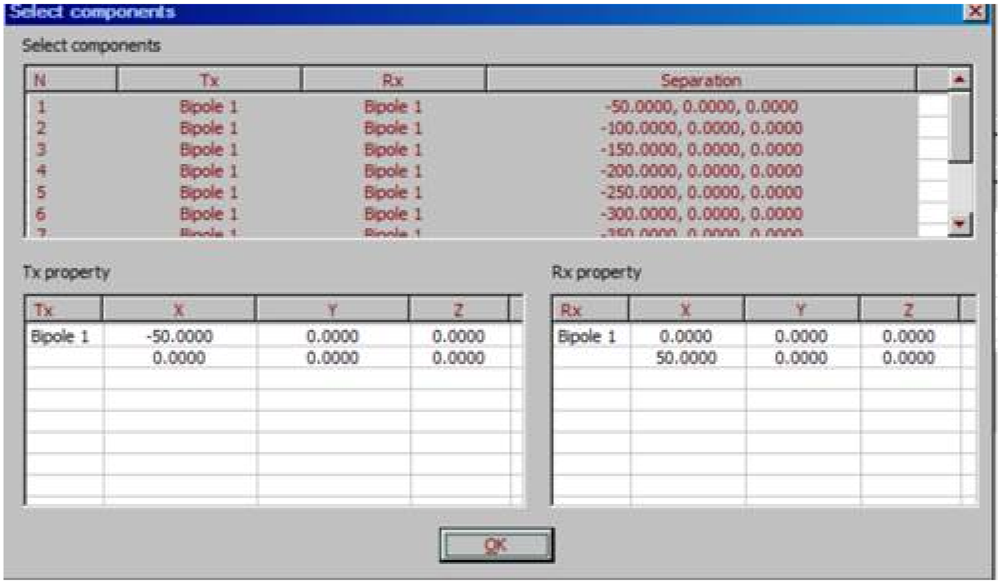
# Using 1D Resistivity Inversion

To start 1D Resistivity Inversion, select the data set with resistivity data you wish to work with from the main Database window. Click the  button on the main toolbar to launch the main window:



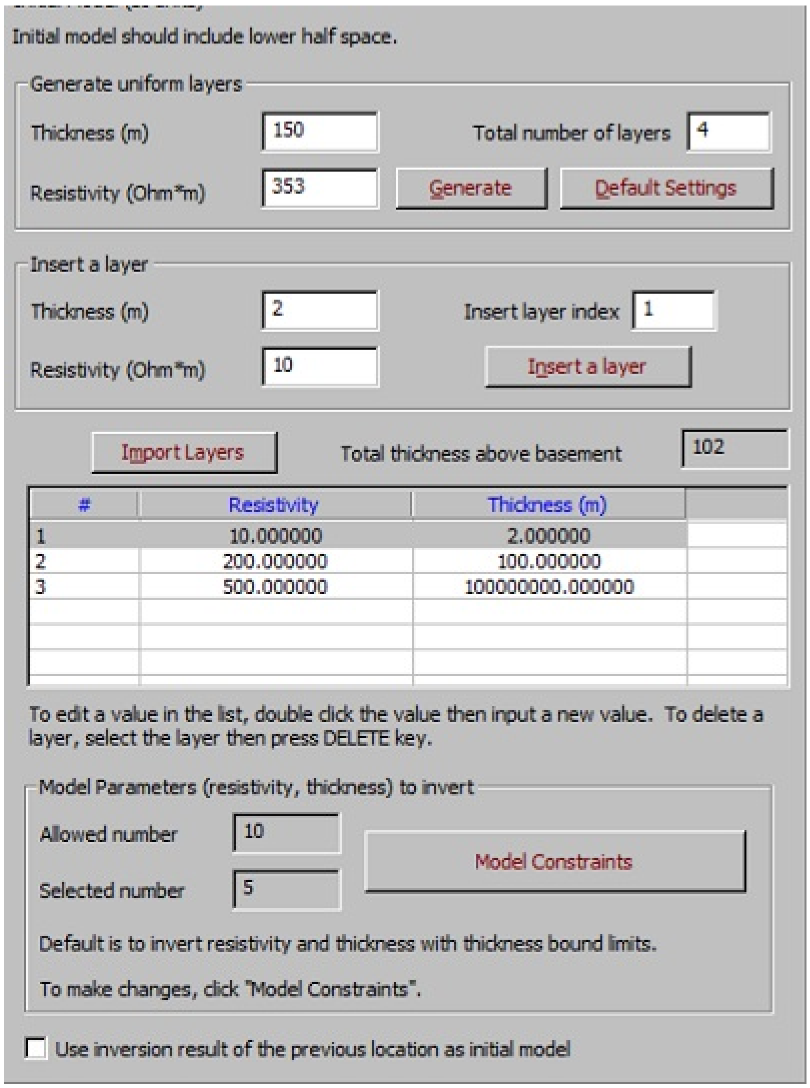

1. Choose between the two inversion techniques:
   **Trust Region** - underparametrized technique that inverts for layer

thickness and resistivity
**Occam** - overparametrized technique that only inverts for resistivity




2. Click the **Select Component(s)** button to select which components you wish to fit in the inversion process. In the window that opens, the selected components are displayed above the local coordinates of the transmitter and receiver.
3. **Use previous Inversion Settings**
   Select *Get Inversion Settings from a Log File*. A window will open to allow you to select a previous Log file. The log files are contained in the /log sub-directory of the database. The log files are identified with the data set number in the database. Click on the dataset and the ID is show to the right.

# Specifying an Initial Model




To load a layer model from another data set:

- Click **Import layers** to load an entire or part of a layer model from another data set.

To create a set of identical layers:

- Specify the layer resisitivity and thickness as well as the number of desired layers then click **Generate uniform layers**. The deepest layer generated will be the basement and will have a thickness of 1e+8.

To insert one layer:

- Specify the layer resistivity and thickness. Insure the Total number of layers displayed is the same as in the list.
- Click **Insert a layer** and a new layer will be inserted at the location specified by the **Insert layer index**.

To delete a layer:

- Select the desired from the list and delete with the DELETE key.

To edit a value in the list:

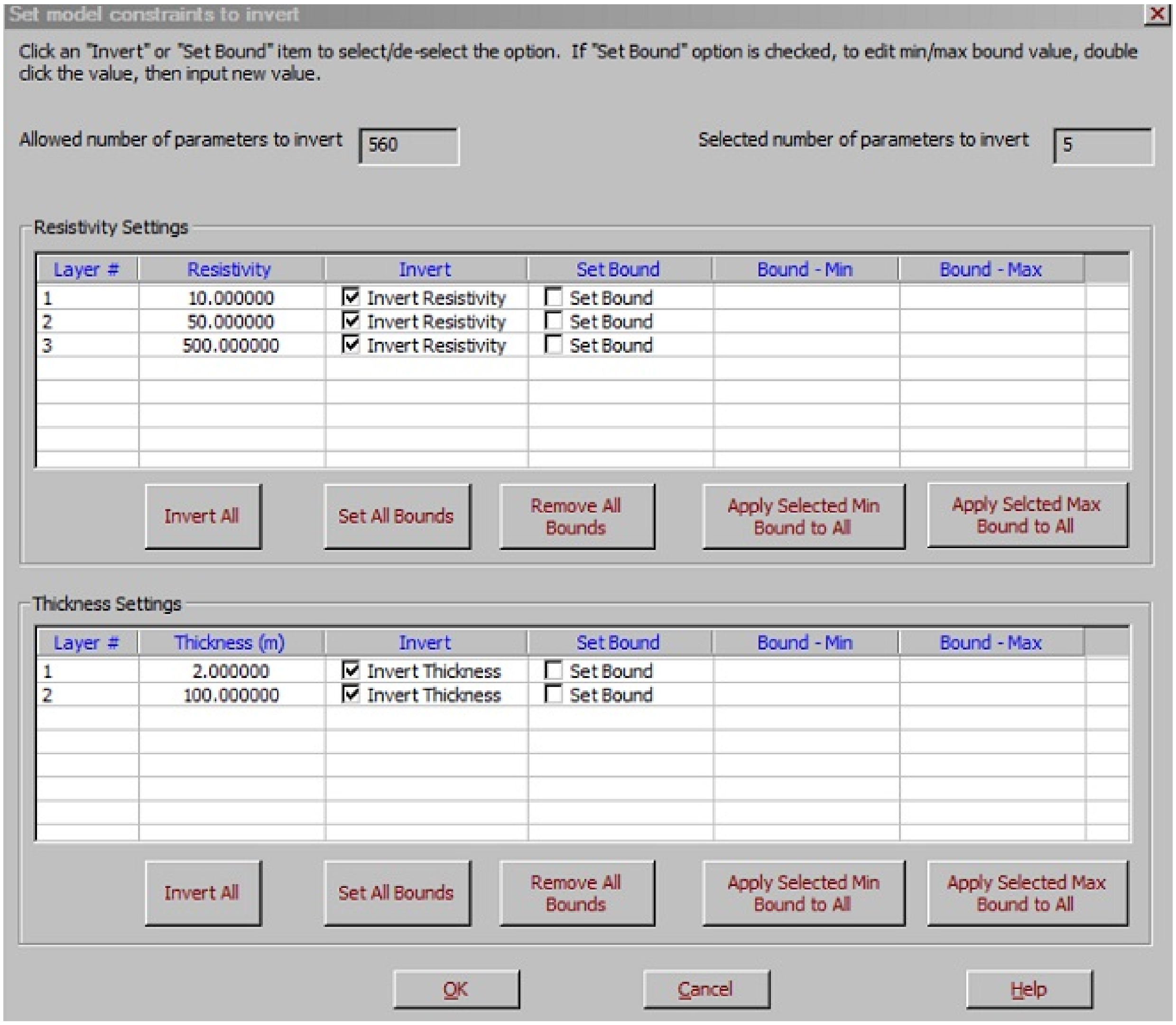
- Double click the value then input a new value.

To constrain model parameters:

- Click the button labelled **Model Constraints**.

Selecting **Use inversion of the previous location as initial model** will initiate the initial model of a new station to use the inverted model from the previous station.

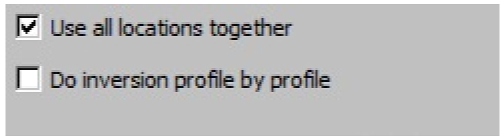
## Constraining Model Parameters




- If available, the **Invert** columns have checks to indicate which parameters will be inverted. **Selected number of parameters to invert** will be updated everytime an **Invert Resistivity** or **Invert Thickness** checkbox is checked or unchecked. That value must not exceed the **Allowed number of parameters to invert**.
- Click a **Set Bound** checkbox to set bound values for the parameter. The default bound values can be edited by double clicking on the value.
- Click the **Set All Bounds** button to give bound values to all the selected parameters.

- Click the **Remove All Bounds** button to let there be no bounds on the selected parameters.
- Click the **Apply Selected Min Bound to All** or **Apply Selected Max Bound to All** buttons as a shortcut to setting all the minimum or maximum bounds to the selected value.

## Joint Inversion Possibilities

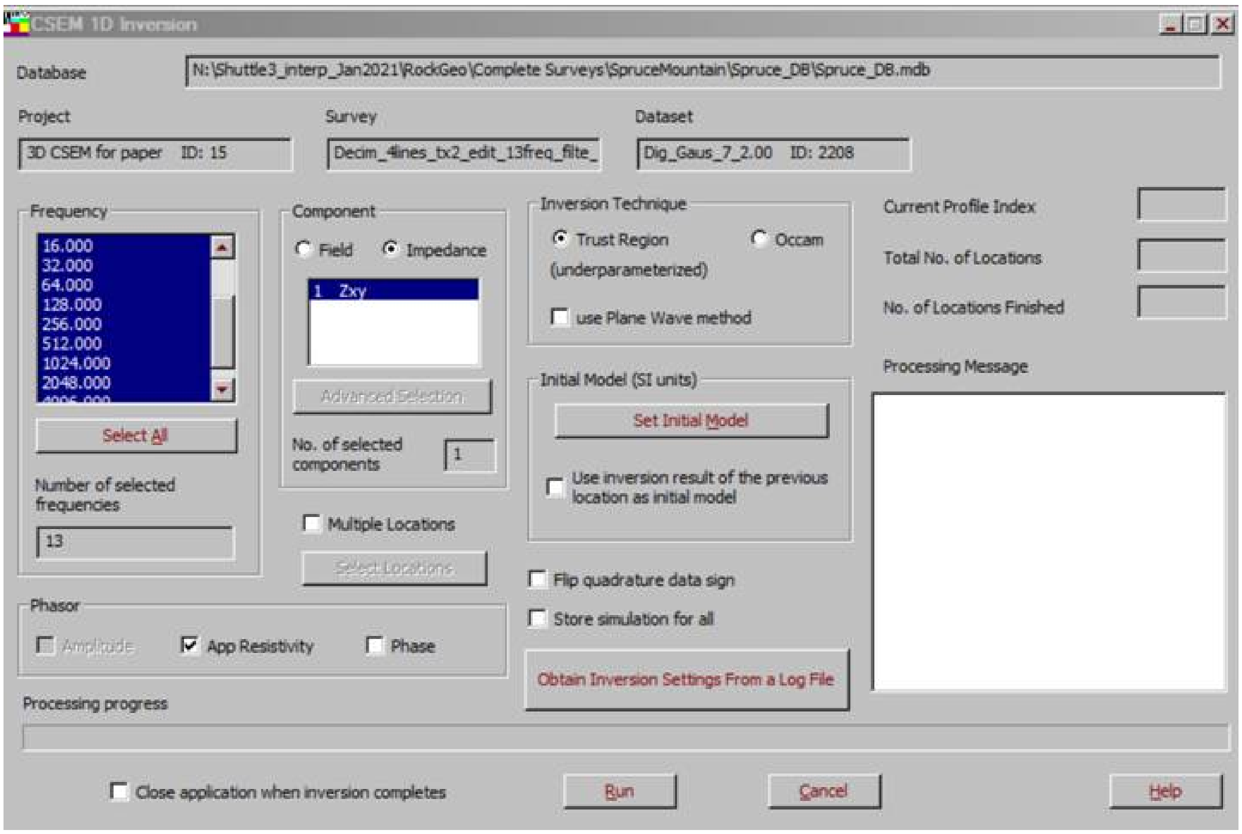



**Use all locations together** The inversion will attempt to find one (1) layered model to fill all of the data both selected components and all the stations

**Do inversion profile by profile**
All the stations on each profile are jointly inverted

# Using 1D CSEM/CSAMT Inversion

To start 1D CSEM/CSAMT Inversion, select the data set you wish to work with from the main Database window. Click the  button on the main toolbar to launch the main window:

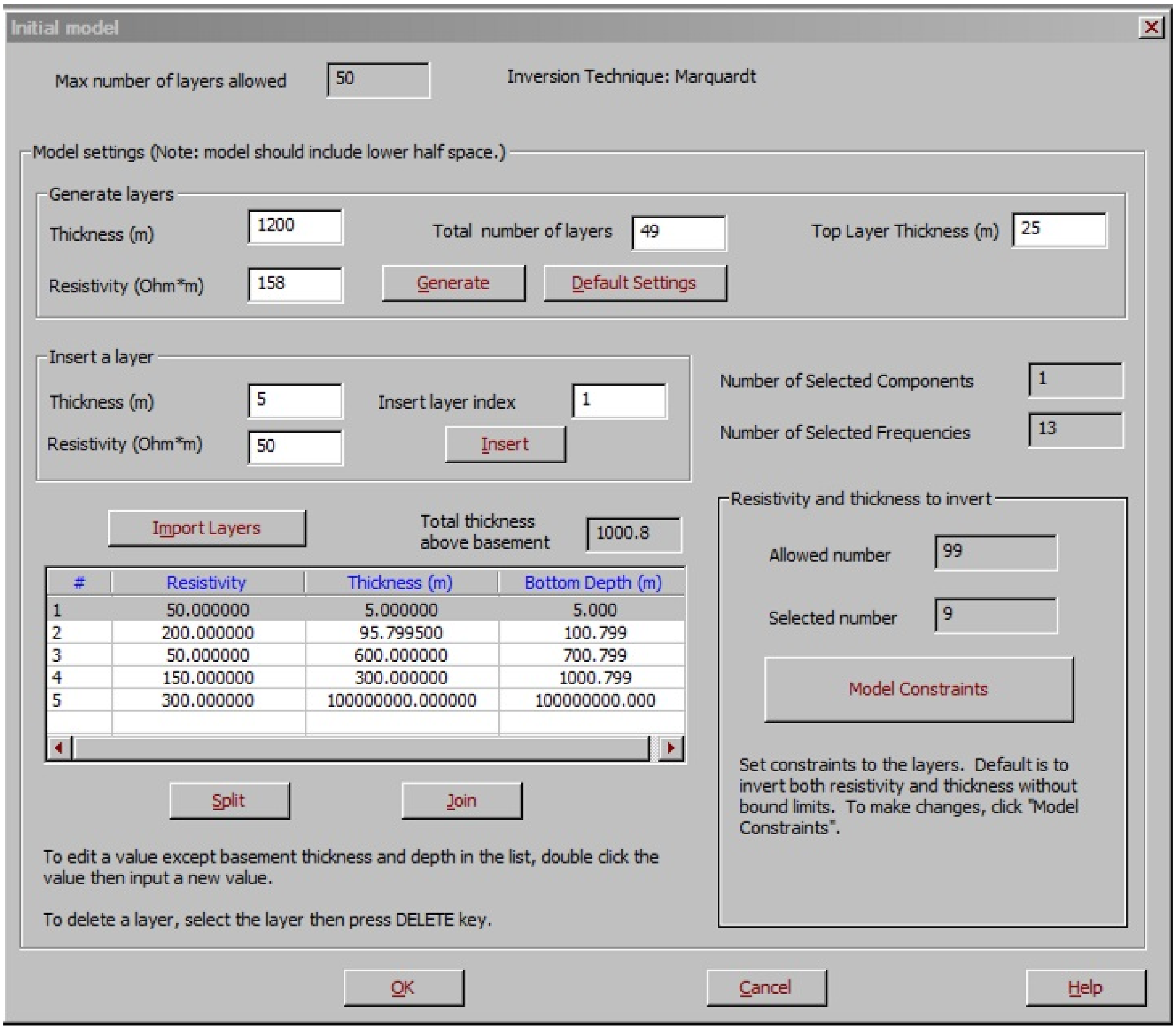



1. A log file is prepared each time an inversion is run. If you would like to reuse the settings from the last inversion run, click the **Obtain Inversion Settings From a Log File** button, select the log file and skip to step 7. Two log files are saved with every run, a default file called CSEM1dInversion.log and another called CSEM1dInversion_p*N*_s*Q*_d*MMMM*.log where *N* is the project number, *Q* is the survey number and *MMMM* is the dataset number
2. Select the frequencies to be used during the inversion from the list labelled **Frequency**. The number of selected frequencies is displayed below the list.

3. Choose between the two inversion techniques:
   **Trust Region** - underparametrized technique that inverts for layer thickness and resistivity
   **Occam** - overparametrized technique that only inverts for resistivity
   The default technique is to use the 3D source in the inversion but if you would like to try a conventional approach, then you may select Plane Wave Method. In this case, a uniform plane wave propagating exactly vertically is utilized.
4. Select the **Flip quadrature data sign** checkbox to multiply all the input quadrature data by -1 when it is used in the inversion processing. This is required when the data does not use the same convention for $e^{i\omega t}$ as the software.
5. In the **Component** section, specify whether you would like to use Field or Impedance components in the inversion process. Use the list box to select the specific components you wish to use. If the data set has multiple transmitters, the **Advanced Selection** button will be enabled.
6. Real and Imaginary or Amplitude can be selected in the **Phasor** section for Field components. Apparent resistivity and Phase can be selected for an Impedance component.
7. Specify an initial model which is normally determined by trial forward models. This process helps in the quality control of data as well as determining your data conventions such as sign and phase convention.
8. Select the **Multi Locations** option to specify the locations to be used during the inversion. Click the **Select Locations** button to make changes to your selections.
9. Check **Use inversion result of the previous location as initial model** for the specified initial model to be used only for the first location. Subsequent locations will use the final model of the location before it.
10. Click **Run**
11. The inversion result can be viewed graphically in the CDI Viewer tool.

## Specifying an Initial Model

Click the **Set Initial Model** button and the following window appears:




To load a layer model from another data set which is usually from your trial forward simulations:

- Click **Import layers** to load an entire or part of a layer model from another data set.

To create a set of identical layers:

- Specify the layer resisitivity and thickness as well as the number of desired layers in the **Generate uniform layers** section then click **Generate**. The deepest layer generated will be the basement and will have a nominal thickness of 1e+8.

To insert one layer:

- Specify the layer resistivity and thickness in the **Insert a layer** section.
- Click **Insert a layer** and a new layer will be inserted at the location specified by the **Insert layer index**.

To delete a layer:

- Select the desired layer from the list and delete with the DELETE key.

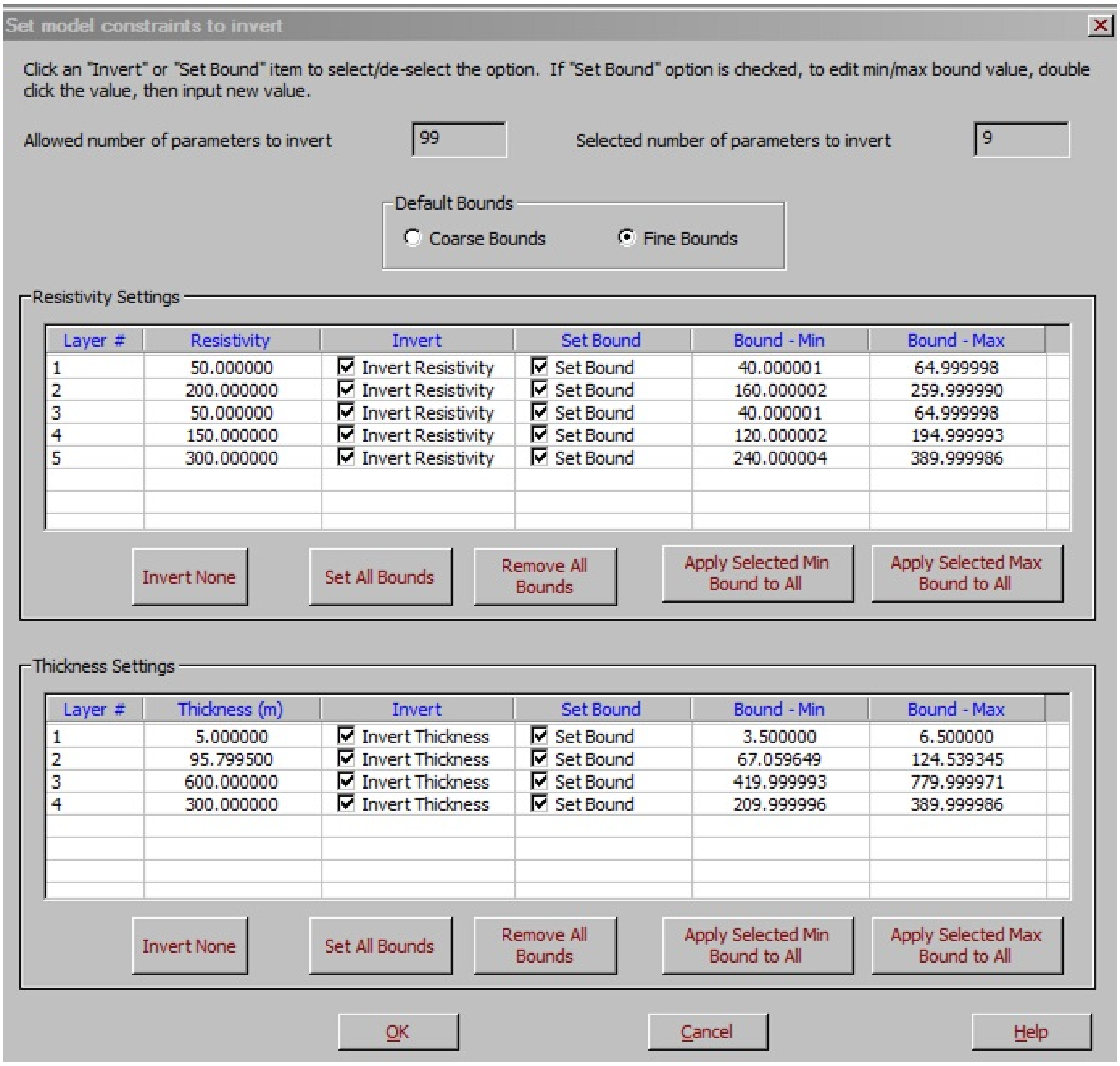
To edit a value in the list:

- Double click the value then input a new value.

To constrain model parameters:

- Click the button labelled **Model Constraints**.

# Constraining Model Parameters

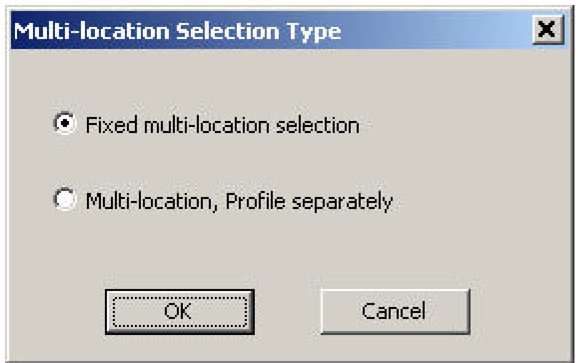



- If available, the **Invert** columns have checks to indicate which parameters will be inverted. **Selected number of parameters to invert** will be updated everytime an **Invert Resistivity** or **Invert Thickness** checkbox is checked or unchecked. That value must not exceed the **Allowed number of parameters to invert**.
- Click a **Set Bound** checkbox to set bound values for the parameter. The default bound values can be edited by double clicking on the value.

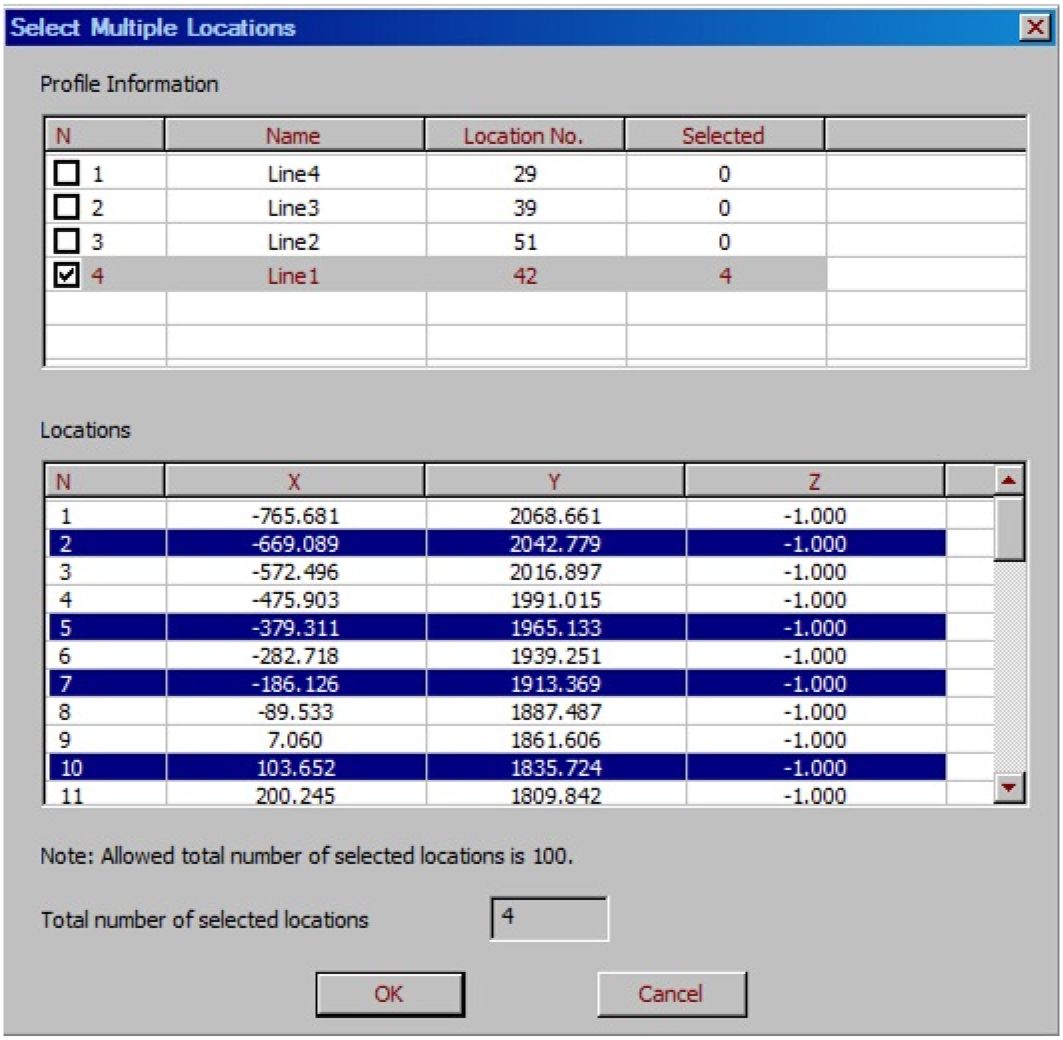
- Click the **Set All Bounds** button to give bound values to all the selected parameters.
- Click the **Remove All Bounds** button to let there be no bounds on the selected parameters.
- Individual bounds may be edited. Double click on the box to edit.
- Click the **Apply Selected Min Bound to All** or **Apply Selected Max Bound to All** buttons as a shortcut to setting all the minimum or maximum bounds to the selected value.

# Selecting locations for inversion

Select the **Multi Locations** option on the main inversion interface to specify the locations to be used during the inversion. The following window appears:




All selected locations are inverted simultaneously (Fixed multi-location) or all selected locations for each profile are inverted profile by profile.
Click **OK** to display the following window:




- Each line in the **Profile Information** list has a checkbox. Click a profile's checkbox to include it in the inversion processing.
- The **Locations** list displays the coordinates of the profile that has been selected in the **Profile Information** list. Select the locations in the **Locations** list that you would like to include in the inversion processing.
- The maximum number of allowed locations as well as the current number of selected locations are displayed at the bottom of window.

## Advanced Component Selection

Clicking the **Advanced Selection** button will launch the following window:

The available transmitters in the data set are listed in the box labelled **Transmitters**
The current number of selected components is displayed to the right of the **Receivers** list.

**To select a transmitter for inversion:**

- Click the square in the first column of the transmitter list and a check mark will appear

**To view the selected receivers related to a transmitter:**

- Click the transmitter in the **Transmitters** list.
- The related receivers in the **Recievers** list will be highlighted.

**To select the same receivers for each transmitter**

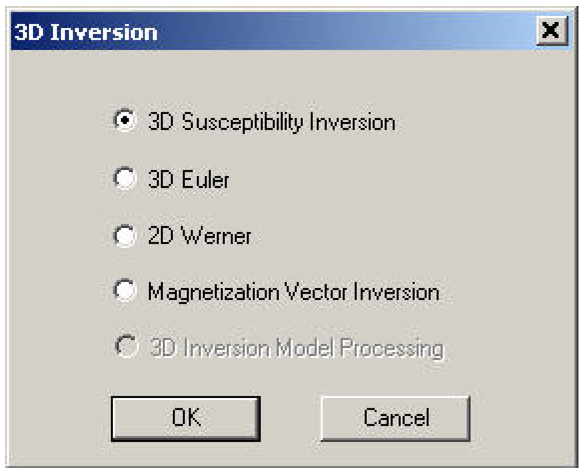
- Select the checkbox labelled **Set the selected Rx to all Tx**

# 3D Magnetic Data Inversion

## Selecting Inversion Type

Select the magnetic data set to subject to inversion in the **Database** window and click the **3D Inversion** button on the EMIGMA toolbar.

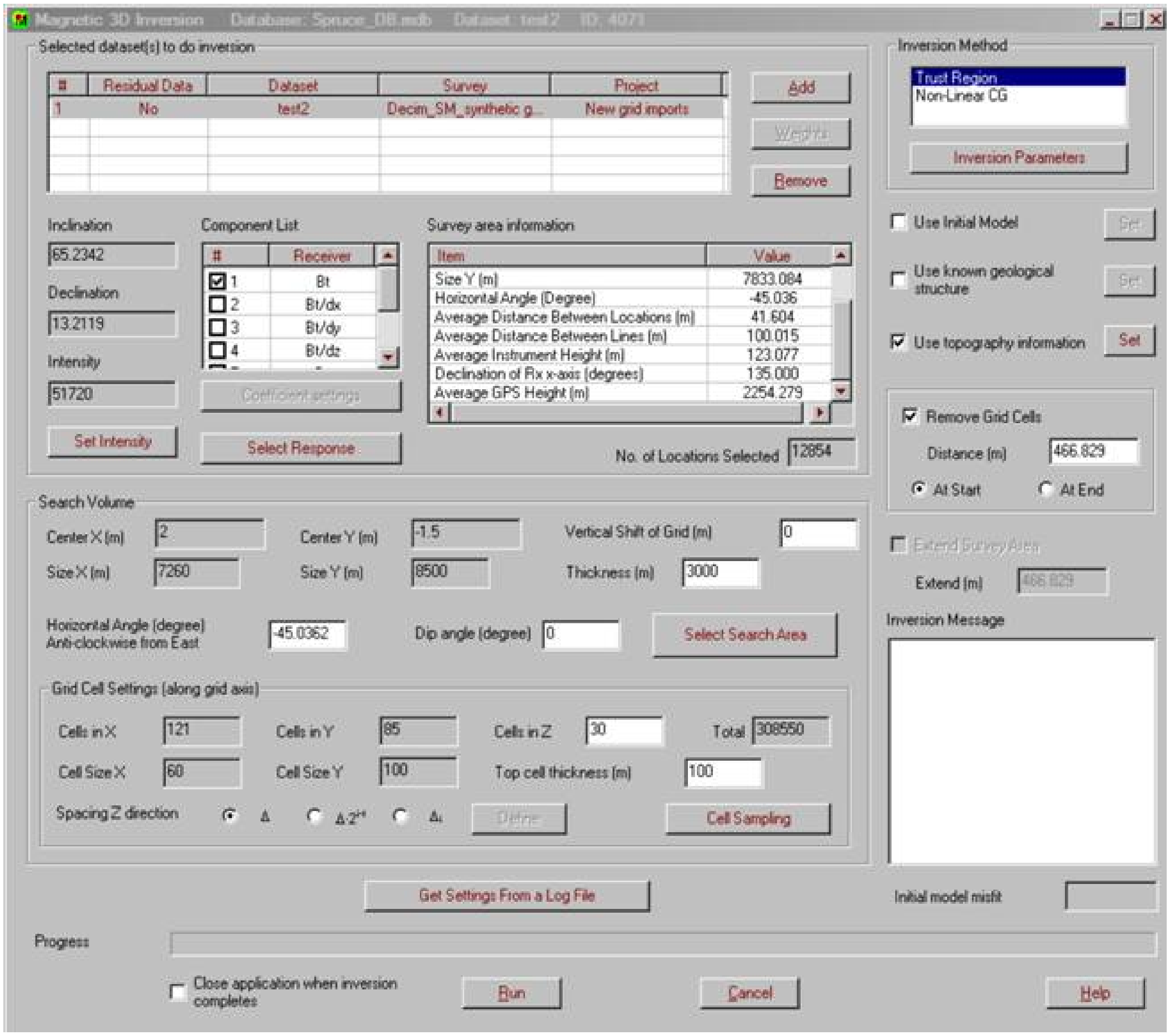
The **3D Mag Inversion** dialog will appear:




Select **3D Susceptibility Inversion** and click **OK**

The other inversions are different types of inversion producing inverted target points rather than a grid type inversion.

## Performing Inversion




When you have completed adjusting the settings, click the **Run** button to begin processing. When complete, a new data set will be created. Its name will be based on the inversion method you chose. The inversion results can be viewed graphically using EMIGMA's visualizer or 3D contour tool or simply opened into an ASCII editor. Most 3D visualization applications will read the inverted grid.

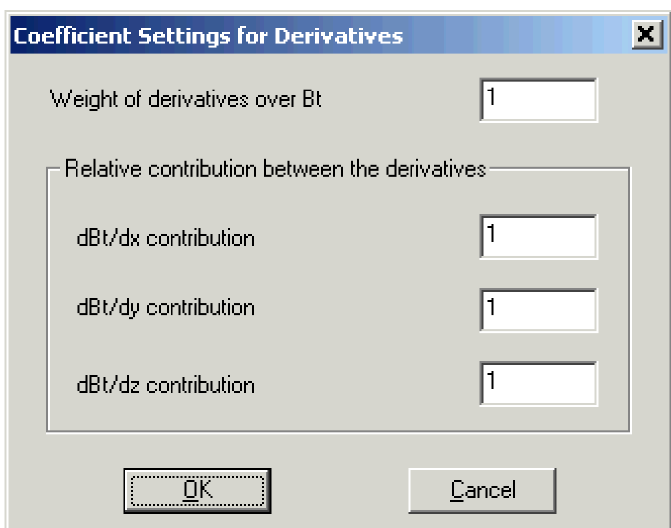
**Additional datasets** A dataset may be added for use in the inversion by clicking **Add**. For example, one may utilize an airborne and a ground dataset. Each dataset is given equal weight by default. This can be changed by clicking **Weights**.

**Components**

You can choose to invert on the combinations of [Bx, By, Bz], or [Btotal(TMI) and gradients of the TMI e.g. dBt/dx]. Developments are underway to allow mixing of data from total field sensors as vector sensors. Note: When dealing with vector data whether components or derivatives, the data are normally rotated to consistent grid azimuth or geographic coordinate. When using such data, be sure to import the grid azimuth at time of importing data. The database stores this azimuth and is utilized in both simulation and inversion.

**Coefficient settings**

This button will be enabled when gradient data is available and more than one derivative has been selected. It launches the following window where a weight can be assigned to each available derivative. A weight can also be assigned to all the derivatives so the Btotal field will not have an equal effect on the inversion results:




**Earth's background field**

You can choose between various methods to obtain a value for the background field by clicking **Set Intensity**.

- **Intensity in the dataset** - uses the value defined in the selected survey.

- **Average of data** - the value will be calculated from the data. The amount of data values used for the calculation depends on the option chosen.
- **User define** - simply enter a new value in the field intensity box.

**Survey Area**
Click the **Select search area** button to launch the graphical tool which enables you to specify the data points that will be used in the inversion calculations.

**Search Volume**
The default parameters in the **Search Volume** section will create a grid that covers the entire survey. You can modify the search area parameters by entering new values or by using the graphical tool available by clicking the button labelled **Select search area**.

**Cell Sampling**
Grid cells defined in **Search Volume** can be divided into smaller units when calculating the simulated data by clicking **Cell Sampling**. Type your values in the X, Y and Z boxes to specify the number of samples in the X, Y and Z directions.

**Grid Settings**
Confirm the number and layout of grid points to be used in the inversion in the **Grid Settings** area. The points will be evenly spaced in the x and y directions. Choose $\mathbf{\Delta}$ for evenly spaced points in the z direction or $\mathbf{\Delta \cdot 2^{i-1}}$ for exponentially spaced points. You may specify a custom spacing by selecting $\mathbf{\Delta_i}$. Your custom settings can be later modified by clicking **Define**.

**Log File**
A log file is created each time an inversion is run. Click **Get Settings From a Log File** to use the settings from a previous inversion. For inversions performed prior to the last inversion, select the .log file which is named with the data set id of the output inversion data set.

**Inversion Methods**
There are two inversion methods to choose from. Set parameters for your chosen technique by clicking the **Inversion Parameters** button.

- **Trust Region** - Faster than Non-Linear CG and has better handling of model constraints.
- **Non-Linear CG** - General concept is to start with an initial guess and go looking for the best fitting model by minimizing a given function using an iteration process.

**Initial Model**
Click the checkbox labelled **Use Initial Model** to specify an initial model. Return to the initial model window by clicking the **Set Initial Model** button. An initial model could be either a previous coarse inversion grid, a forward model defined either by the user or CAD modeling.

**Use topography information**
This option will be enabled if you imported your data with a gps z channel. In EMIGMA, the Z coordinate represents the altimeter data for the measurement. Select this option and the gps z values will be used when performing the inversion. When loading inversion results to the visualizer, a window will appear asking to display the survey according to z or gps z. Select gps z to see the inversion results with topography. If the user wishes to use an imported topographic model (DTM model), you may select a dataset which has this DTM model imported and a default susceptibility for the topography can also be given.

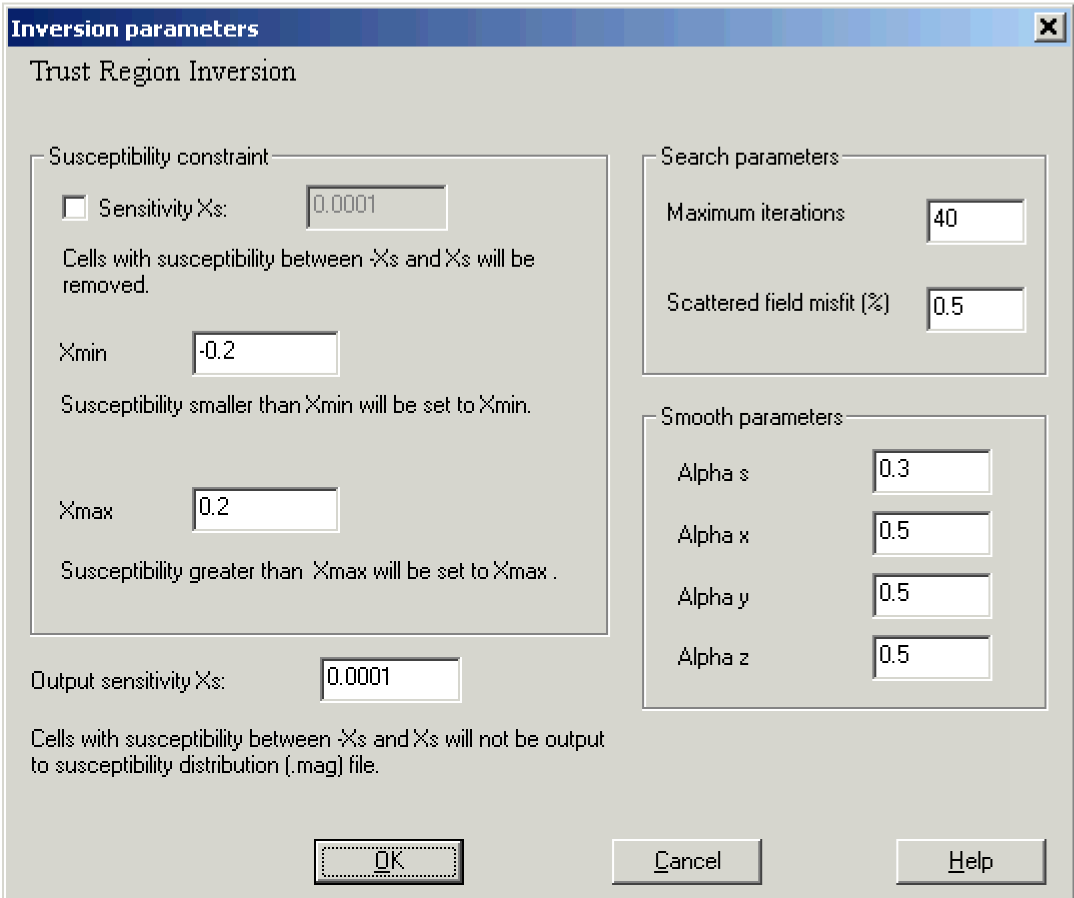
**Remove Grid Cells**
Any cells that are beyond the specified **Distance** from the closest data point will be removed from the inversion result.
Select **At Start** to remove grid cells when inversion begins. Select **At End** to remove grid cells when inversion has completed.

**Extend Survey Area**
Additional data points can be extrapolated around the selected survey to the distance specified in the edit box labelled **Extend**. This option is not available if removing grid cells when the inversion begins.

**Geological Structure**
Click Use known geological structure to define a structure that will apply constraints to the inversion result.

**Initial model misfit**
Defines how close the initial model fits the data. The closer the value is to 0, the better the fit.

## Inversion Parameters.




**Maximum iterations** The number of iterations the program will run to generate the final solution. In general, the default value is sufficient for the inversion.

**Scattered field misfit**
The inversion solution will be considered found when the difference between average measured and simulated scattered field divided by the scattered field falls below this value. Scattered field is the difference between the measured data and the IGRF values. Note: as the scattered or secondary field is often small compared to the IGRF, misfits can seem large.

**Sensitivity Xs**

Cells with susceptibility between –Xs and Xs will be removed after each iteration.

**Xmin**
Upon completion of each iteration, a susceptibility smaller than Xmin will be set to Xmin.

**Xmax**
Upon completion of each iteration, a susceptibility greater than Xmax will be set to Xmax.

**Smooth parameters**
Larger values will increase the smoothness of the inversion result.
**Alpha s** decreases the range in the gradient of all the susceptibility values. Values range from 0 - 40
**Alpha x, y and z** define relative smoothing in the three directions. Values are relative to the other directions.

**Output sensitivity Xs**
Cells with susceptibility between –Xs and Xs will be removed after the inversion has completed and will not be output to the susceptibility distribution (.mag) file.

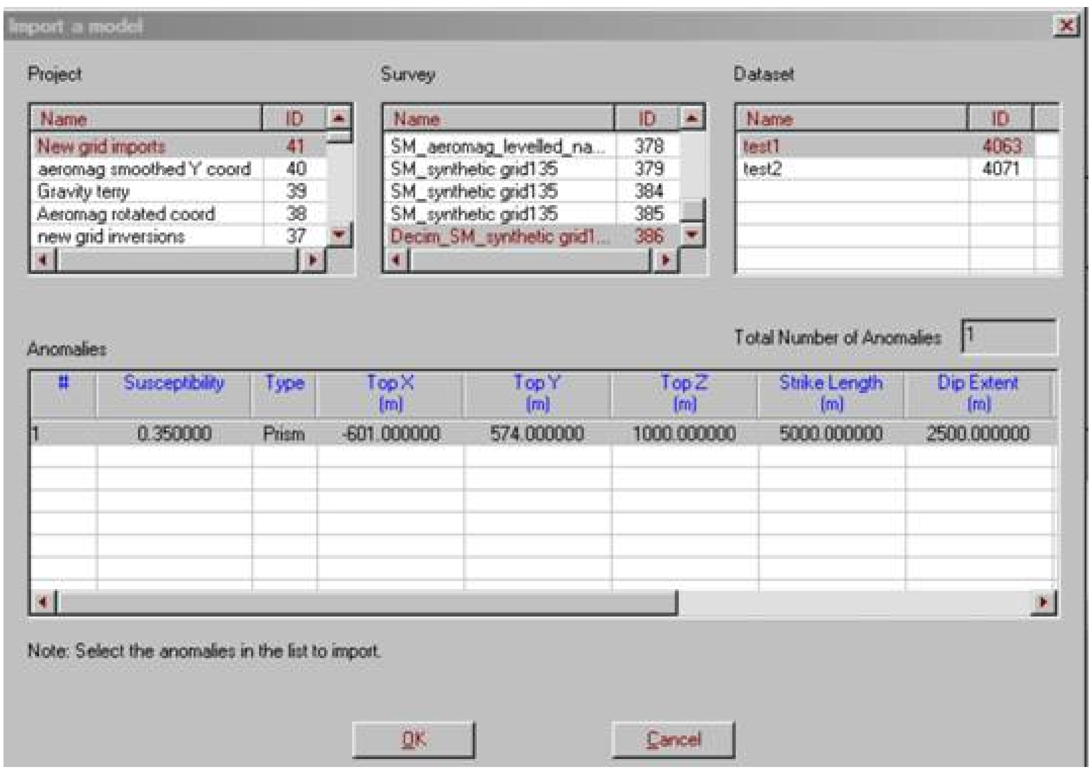
# Editing the Initial Model.

Clicking the **Use Initial Model** button will launch the following window:

The starting model is described by a list of prisms/polyhedra or grids with various properties.

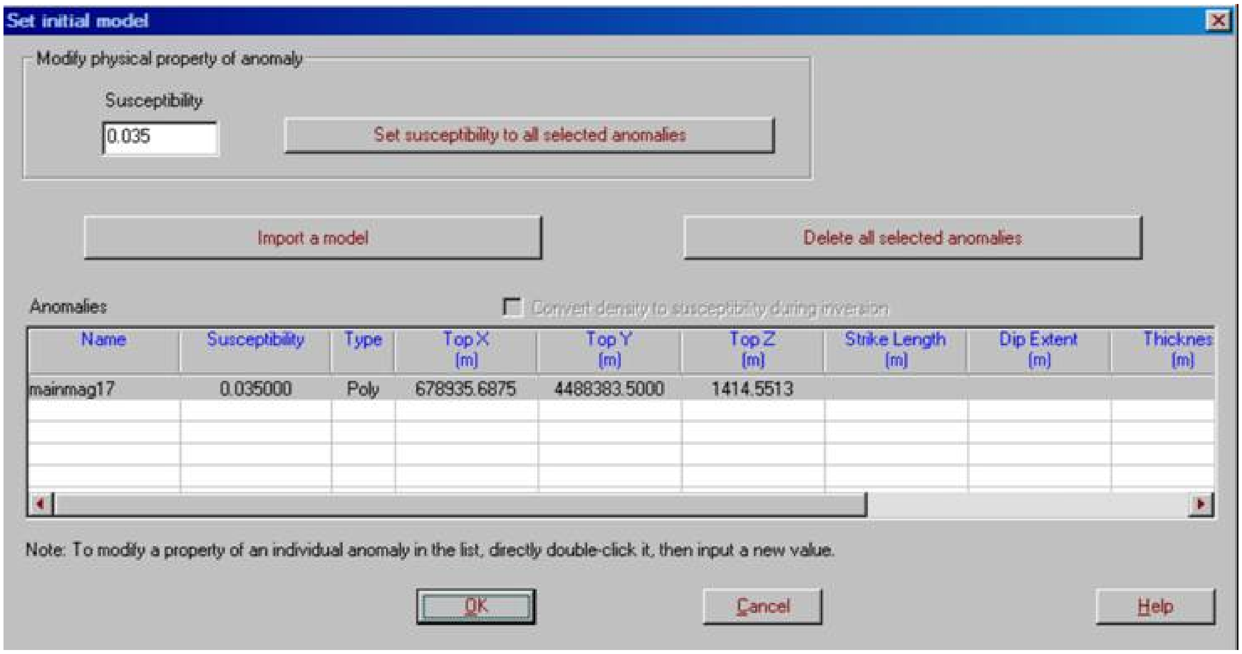
**To add a prism/polyhedra to the model list:**

- Click **Import a model**. Browse in your database to find the model
- Each parameter may be edited in each parameter box if desired




**To delete prisms from the model list:**

- Select the prisms to be deleted in the anomaly list.
- Click **Delete all selected prisms**




**To apply the same values for a group of selected prisms:**

- Click the **Set susceptibility to all selected prisms** button to modify the susceptibility.
- Click the **Set angles to all selected prisms** button to modify the angles.
- Click the **Set size to all selected prisms** button to modify the size.

## Editing the Grid Cell Thickness.

Selecting $\boldsymbol{\Delta_i}$ or clicking the **Define** button in the **Grid Settings** section will launch the following window:

The interface displays the total thicknesses before and after editing as well as the topmost z value. The cell sizes are listed in the **Search grid cell thickness** section.

**To modify an existing entry:**

- Specify the **Thickness** and select the entries to which you would like to assign this thickness.
- Click **Modify the selected**.

**To insert a new entry:**

- Specify the **Thickness**.
- Specify the line where the new entry should be inserted in the **Insert Index** box.
- Click the **Insert a thickness** button.

**To delete an entry:**

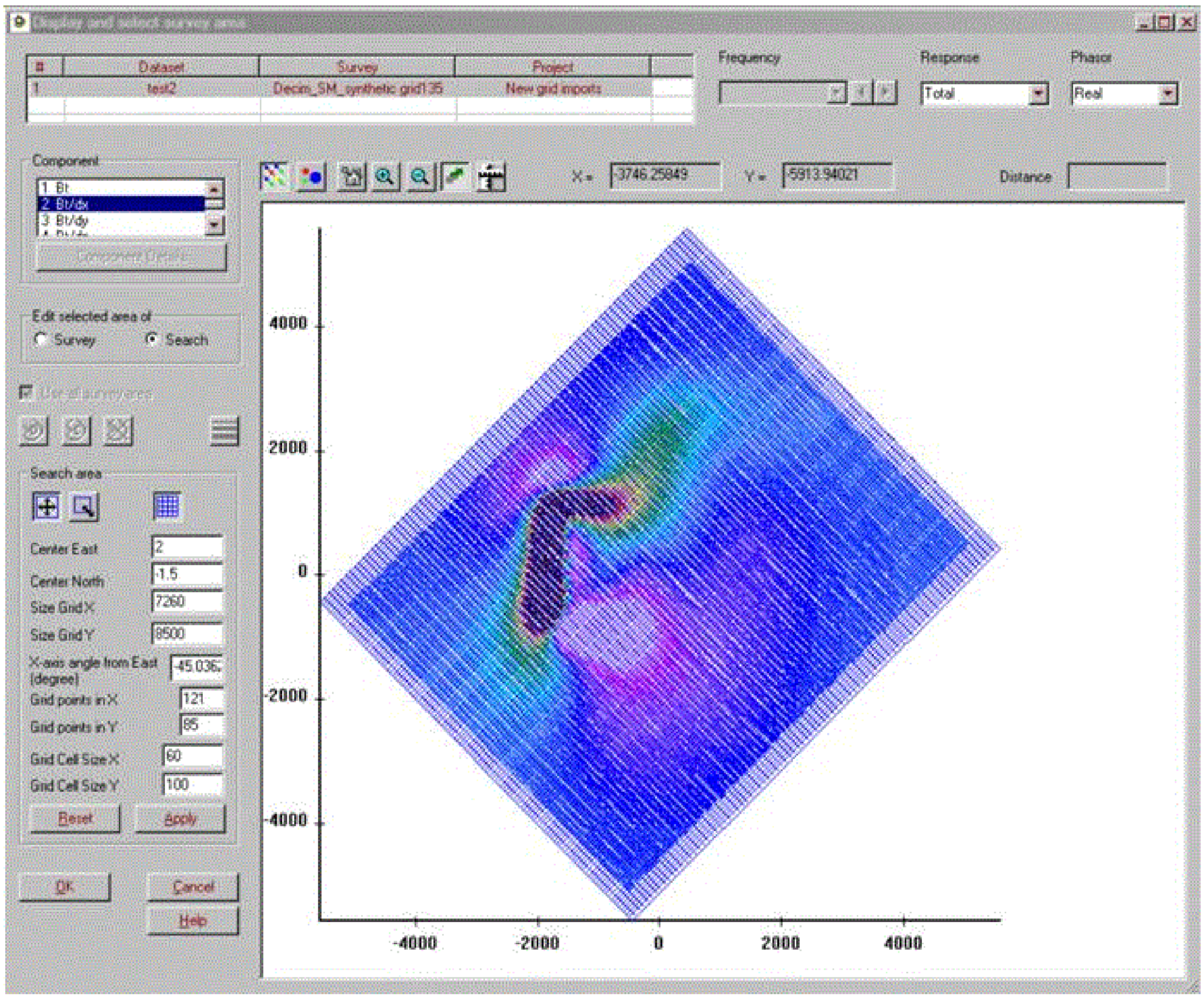
- Select the entries to be deleted.
- Click **Delete the selected**.

## Selecting Survey and Search Area

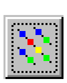
## Search Area: defines inversion grid

## Survey Area: defines the portion of the entire survey to be utilized

Clicking the **Select Search Area** button launches the following window:



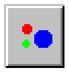
To toggle display of data values:

- Click the  button.

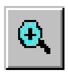
To change display of component or response:

- Make a selection from the **Component** and **Response** list.

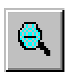
To change the size of the symbols for each data value:

- Click the  button. Use the slider on the window to appear to change the symbol size. Select **Show change immediately** to see the symbols change size as you move the slider.

To zoom in on a section of the survey:

- Click the  button. Drag the mouse on the survey display to draw a rectangle describing area you would like to zoom in on.

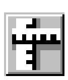
To zoom out to the previous zoom level:

- Click the  button.

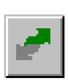
To zoom out to the original zoom level:

- Click the  button.

To find the distance between two points on the survey display:

- Click the  button. Drag the mouse between the two points on the survey display and the distance will be displayed at the top of the window in the box labelled **Distance**.

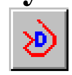
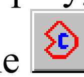
To display the survey in a proportional view:

- Click the  button.

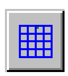
To find the coordinates of a location on the survey display:

- Move the mouse to the location and the coordinates will be displayed at the top of the window in the boxes labelled **X=** and **Y=**

To select only a section of the survey:

- Deselect the checkbox labelled **Use all survey area**. This will enable the three buttons underneath. Make sure the  button is pushed in. Click on the display of the survey at the vertices of the polygon which describes the area you would like to select. Click on the  button to close the polygon and complete your selection. Click on the  button to delete the polygon if you are unsatisfied with it.

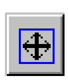
To toggle the display of the search area grid:

- Click the  button.

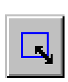
To select a search area:

- Make sure **Search** is selected in the **Edit selected area of** section. The parameters in the **Search Area** pane can be modified and the effect of any changes can be seen by clicking **Apply**.

To move the search area:

- Click the  button and drag the search area box with the mouse.

To resize the search area:

- Click the  button and resize the search area box with the mouse.

## Defining a Geological Structure

Clicking the **Use known geological structure** checkbox or the **Set Structure** button will launch the window below. The idea here is to use known susceptibility information most likely from drill cores but also from surface sampling.

Various properties of the structures are listed in the box labelled **Geology Structure List**

**To add a structure to the list:**

- Specify the susceptibility, position and radius of the spherical structure in the **Build or Edit Geology Structure** section.
- Click the **Add to List** button.

**To modify an existing entry in the list:**

- Click the line of the structure you would like to edit in the **Geology Structure List**.
- Enter the new susceptibility, position or radius values in the **Build or Edit Geology Structure** section.
- Click the **Modify** button.

**To delete from the list:**

- Select the entry to be deleted in the list.
- Click **Delete the selected structure**
  OR
- Click **Delete all structures** to clear the entire list.

**To save the list to an ascii file:**

- Click the **Save structures to a file** button.
- Specify a location and filename for the file.

**To import a previously saved list of structures:**

- Click **Load structures from a file**.
- Select the file and the **Geology Structure List** will be populated with the structures in the file.
- Format:

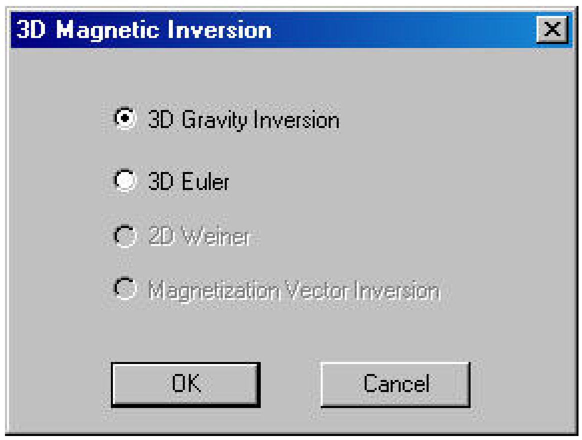
```
// Number of geological structures: 2
//  CenterX       CenterY       CenterZ
CenterGPSZ        Radius            k
    678521.       4487578.0     -1520.0000         735.0000
350.0000          0.035
    677501.       4487005.0     -1520.0000         985.0000
350.0000          0.021
```

# 3D Gravity Inversion

## Selecting Inversion Type

Select the gravity data set to subject to inversion in the **Database** dialog and click the **3D Inversion** button 

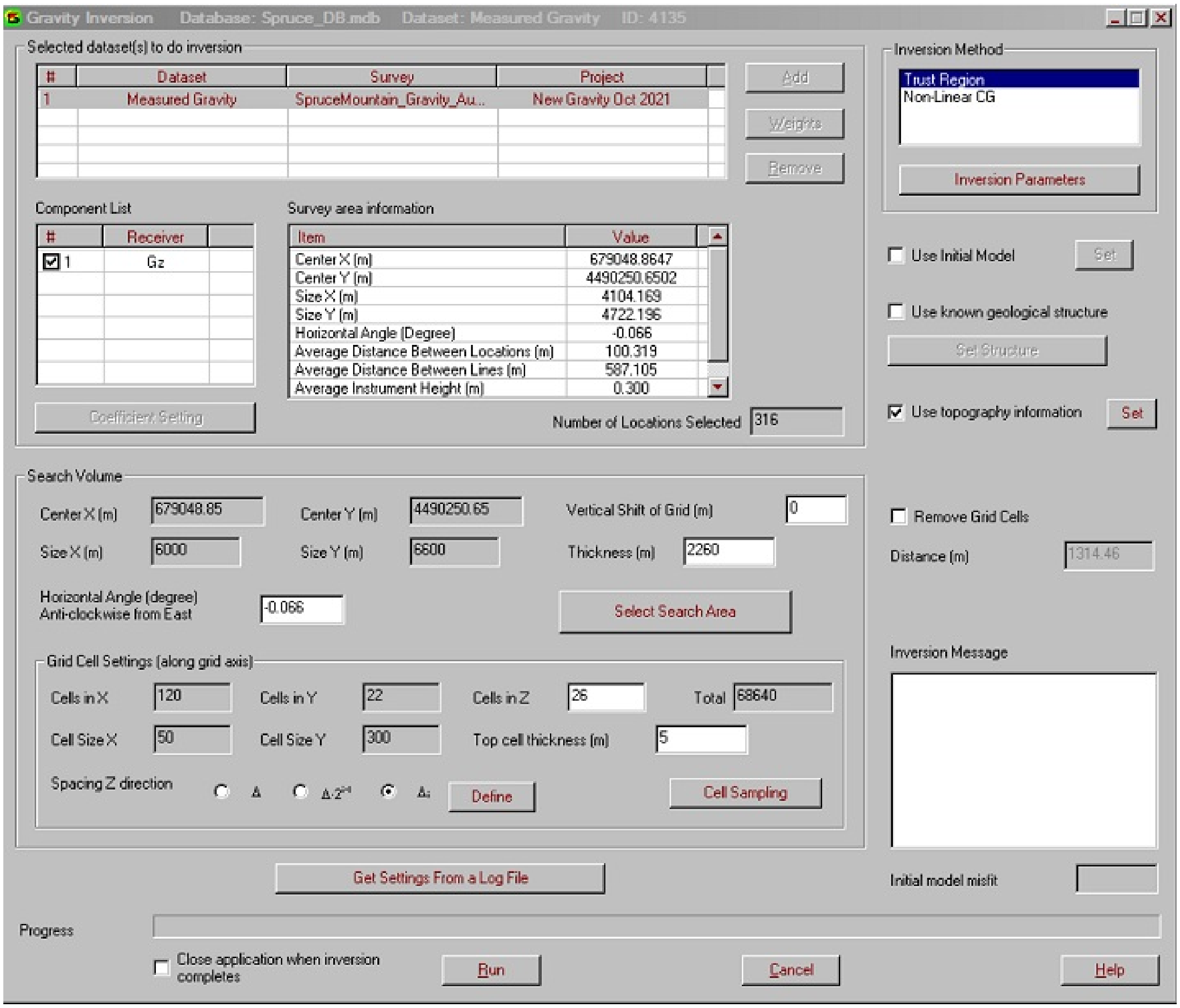
on the EMIGMA toolbar.




Select **3D Gravity Inversion** and click **OK**

The **3D Euler** inversion is a different style of inversion, inverting for discrete objects utilizing the main field and its derivative.

# Performing Inversion




When you have completed adjusting the settings, click the **Run** button to begin processing. When complete, a new data set will be created. Its name will be based on the inversion method you chose. The inversion results can be viewed graphically using EMIGMA's Visualizer or 3D contour tool.

**Component List** Components that will be used in the inversion are displayed here.

**Survey Area**
Click the Select search area button to launch the graphical tool which

enables you to specify the data points that will be used in the inversion calculations.

**Search Volume**

The default parameters in the **Search Volume** section will create a grid that covers the entire survey. You can modify the search area parameters by entering new values or by using the graphical tool available by clicking the button labelled **Select search area**.

**Cell Sampling**

Grid cells defined in **Search Volume** can be divided into smaller units when calculating the simulated data by clicking **Cell Sampling**. Type your values in the X, Y and Z boxes to specify the number of samples in the X, Y and Z directions.

**Grid Settings**

Confirm the number and layout of grid points to be used in the inversion in the **Grid Cell Settings** area. The points will be evenly spaced in the x and y directions. Choose **Δ** for evenly spaced points in the z direction or $\mathbf{\Delta \cdot 2^{i-1}}$ for exponentially spaced points. You may specify a custom spacing by selecting $\mathbf{\Delta_i}$. Your custom settings can be later modified by clicking **Define**.

**Log File**

A log file is created each time an inversion is run. Click **Get Settings From a Log File** to use the settings from a previous inversion.

**Inversion Methods**

There are two inversion methods to choose from. Set parameters for your chosen technique by clicking the **Inversion Parameters** button.

- **Trust Region** - Faster than Non-Linear CG and has better handling of model constraints.
- **Non-Linear CG** - General concept is to start with an initial guess and go looking for the best fitting model by minimizing a given function using an iteration process.

**Initial Model**

Click the checkbox labelled **Use Initial Model** to specify an initial model.

Return to the initial model window by clicking the nearby **Set** button.

**Use topography information**

This option will be enabled if you imported your data with a gps z channel or if you want to import a topographic polyhedra file constructed from downloaded DEM data. Select this option and the gps z values will be used when performing the inversion. When loading inversion results to the visualizer, a window will appear asking to display the survey according to z or gps z. Select gps z to see the inversion results with topography.

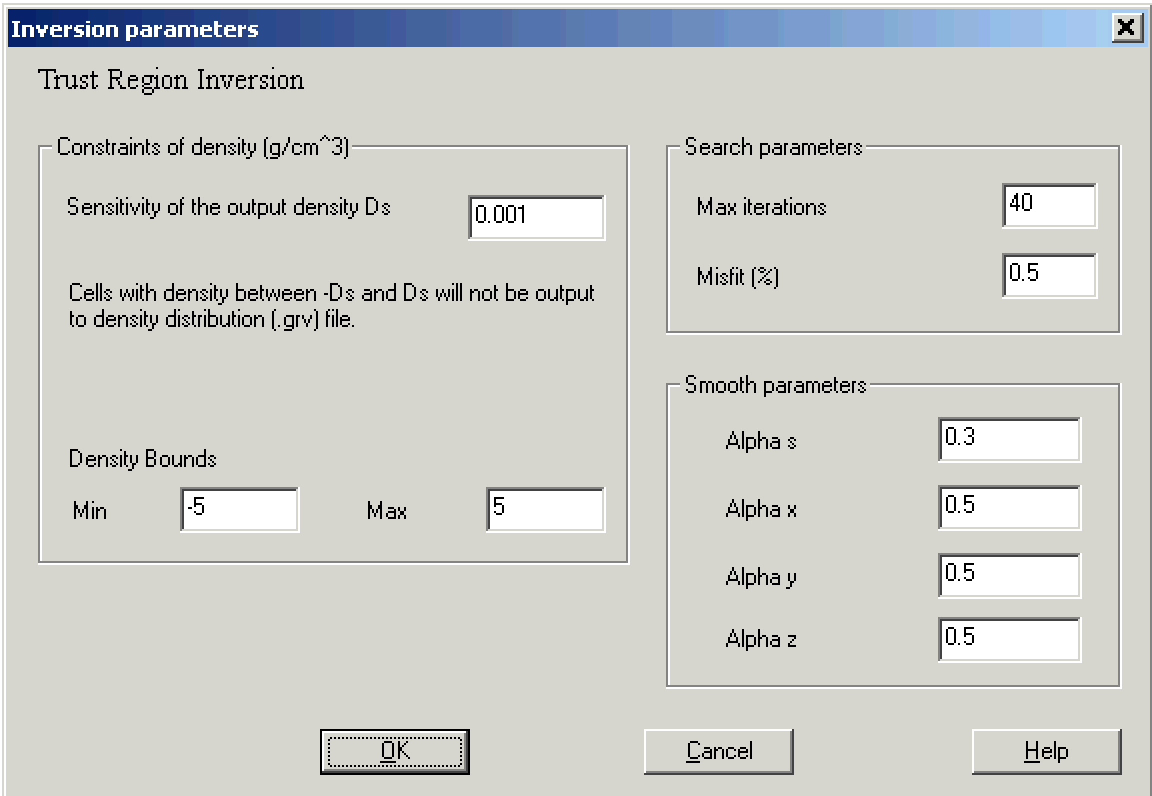
**Remove Grid Cells**

Any cells that are beyond the specified **Distance** from the closest data point will be removed from the inversion result.

**Initial model misfit**

Defines how close the initial model fits the data. The closer the value is to 0, the better the fit.

## Inversion Parameters.




**Maximum iterations** The number of iterations the program will run to generate the final solution. In general, the default value is sufficient for the inversion.

**Misfit**
The inversion solution will be considered found when the weighted difference between the measured and simulated scattered field falls below this value.

**Sensitivity of the output density Ds**
Cells with susceptibility between –Ds and Ds will be constrained or deleted after each iteration and will not be output to the density distribution (.grv) file.

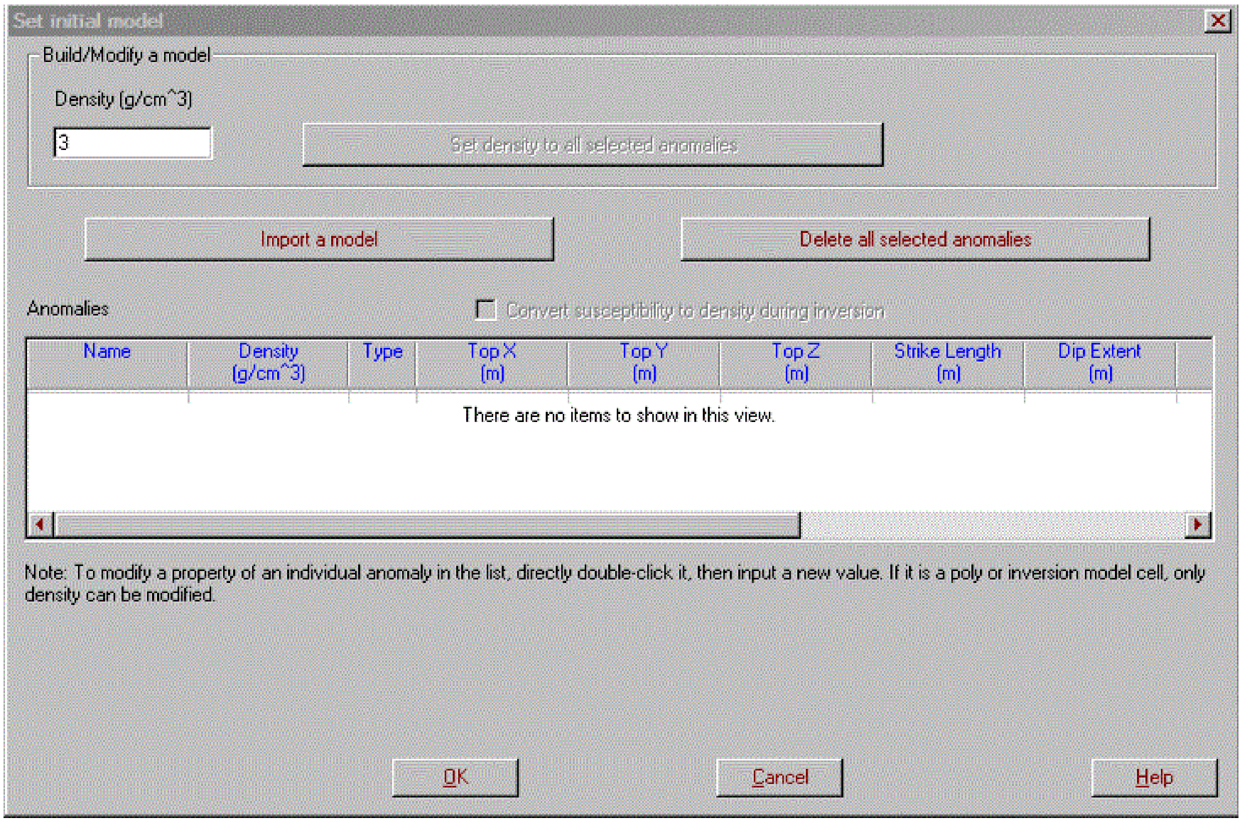
**Density Bounds**
Upon completion of each iteration, a density smaller than **Min** will be set to

**Min** and a density greater than **Max** will be set to **Max**.

**Smooth parameters**
Larger values will increase the smoothness of the inversion result. **Alpha s** decreases the range of all the susceptibility values. **Alpha x, y and z** decreases the difference between the susceptibility of two neighbouring cells in the x, y and z directions respectively. **Alpha s** can be as large as 30. Small values of **Alpha s** will allow a rough inversion.

## Specifying an initial model

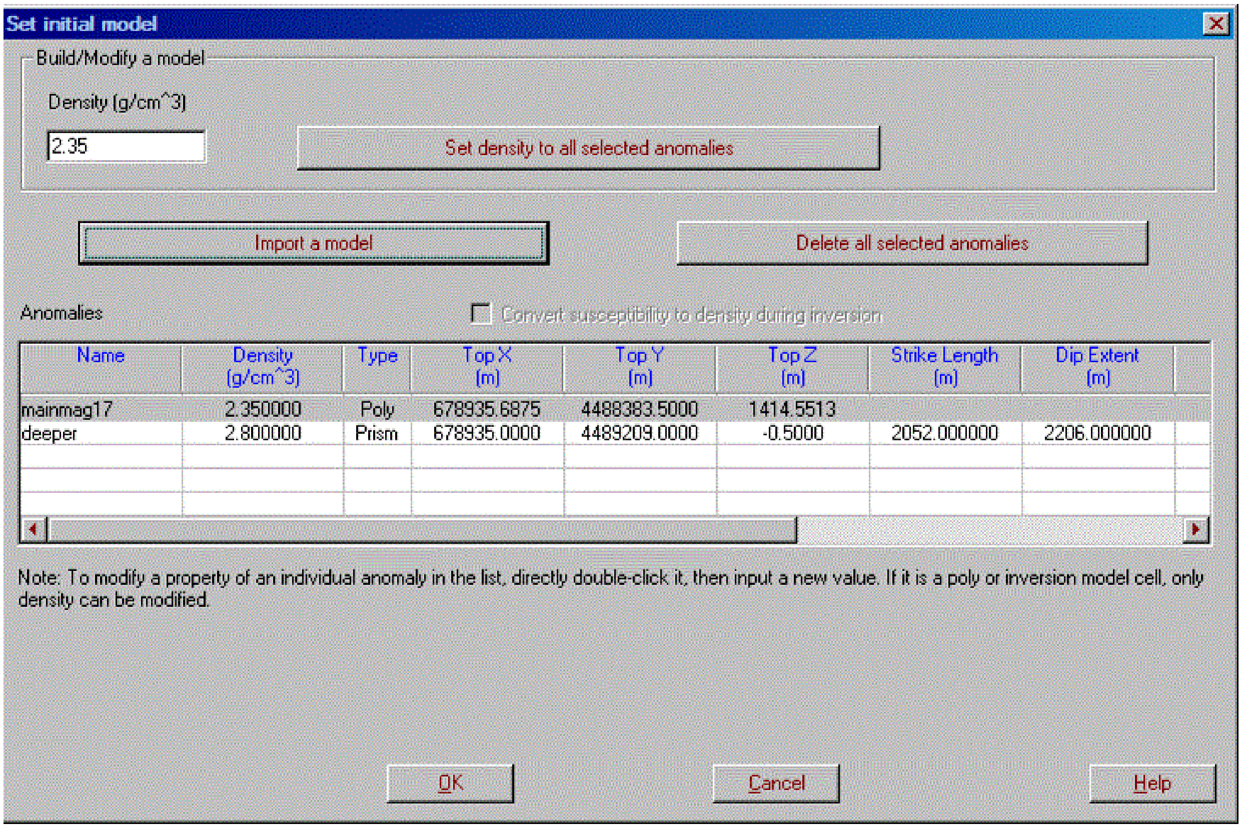
Selecting the **Use Initial Model** checkbox or clicking the **Set Initial Model** button will launch the following window:




The initial or starting model is either a prism, polyhedra or set of prisms and polyhedra developed from forward modeling or it can be a previous inversion, normally with a coarser grid. The starting model is described by a list of prisms with various properties in the area labelled **Anomalies**.

**To add an initial model to the model list:**

- Click the **Import a model** button.




The interface will appear as above. In this case, the first element is a polyhedra and the second a prism.

**To modify an existing model element in the model list:**

- Double click the cell of the property you would like to edit in the Initial Model list.
- Input the new value and press Enter.

**To delete elements from the model list:**

- Select the elements to be deleted in the anomaly list.
- Click **Delete all selected anomalies**

**To apply the same values for a group of selected elements:**

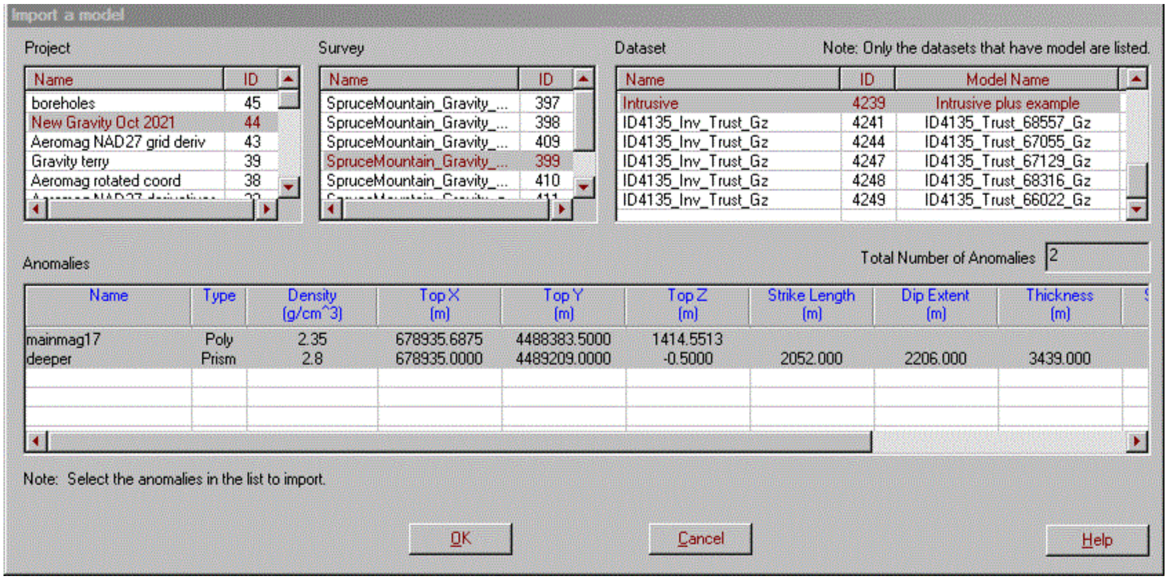
- Click the **Set density to all selected anomalies** button to modify the conductivity.

**To import a model from another data set in the current database:**

- Click Import a model.

- Select the project, survey, and data set with the desired model
- Click **OK** and the model will appear in the **Initial Model** list

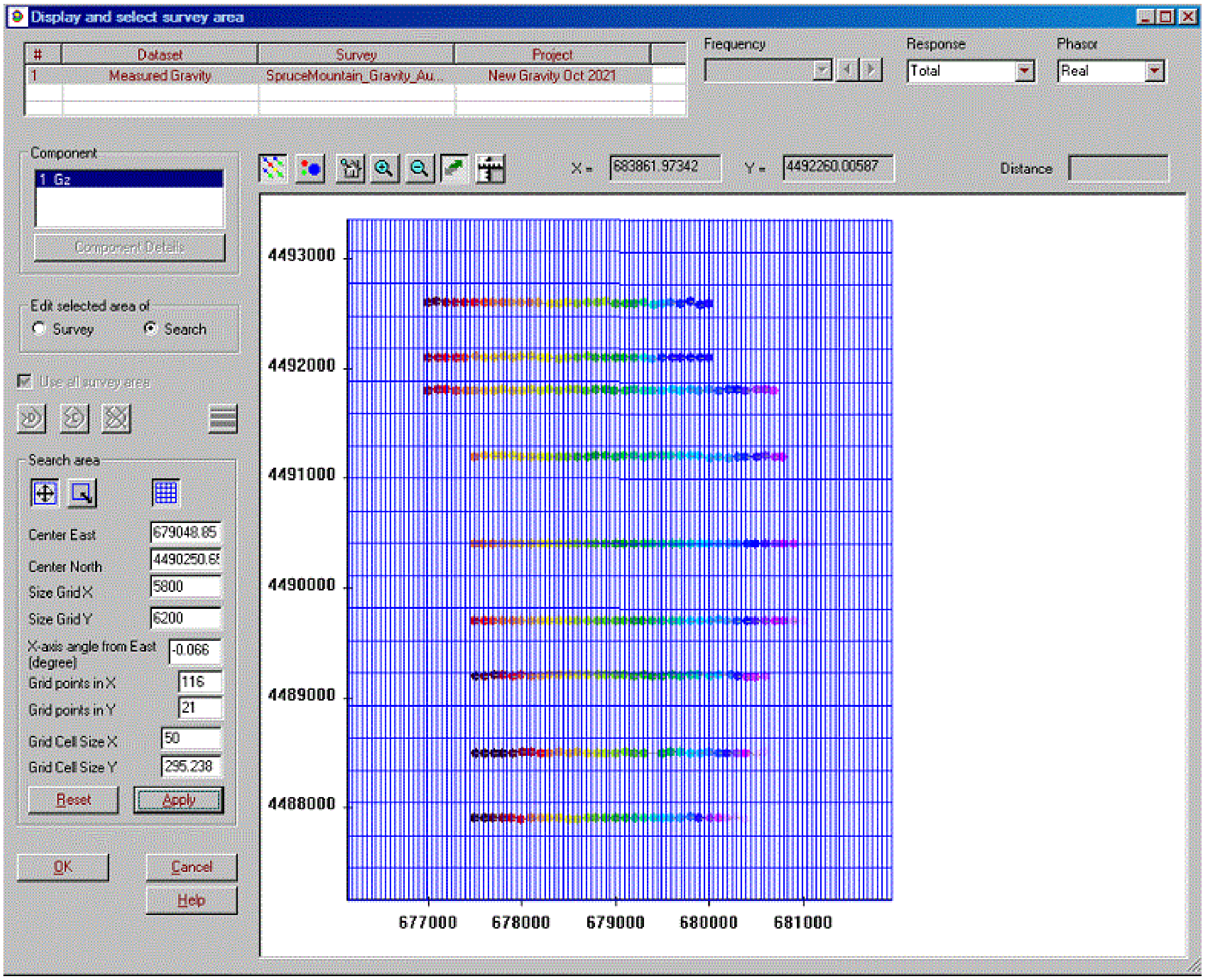
## Importing a model



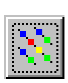

- All eligible projects are listed. Choose the project, survey and data set that has the model you want to use. A prism or polyhedral model may be used

# Selecting Survey and Search Area

Clicking the **Select Search Area** button launches the following window:



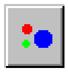

To toggle display of data values:

- Click the  button.

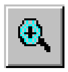
To change display of component or response:

- Make a selection from the **Component** and **Response** list.

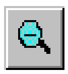
To change the size of the symbols for each data value:

- Click the  button. Use the slider on the window to appear to change the symbol size. Select **Show change immediately** to see the symbols change size as you move the slider.

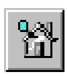
To zoom in on a section of the survey:

- Click the  button. Drag the mouse on the survey display to draw a rectangle describing area you would like to zoom in on.

To zoom out to the previous zoom level:

- Click the  button.

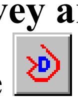
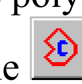
To zoom out to the original zoom level:

- Click the  button.

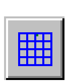
To find the coordinates of a location on the survey display:

- Move the mouse to the location and the coordinates will be displayed at the top of the window in the boxes labelled **X=** and **Y=**

To select only a section of the survey:

- Deselect the checkbox labelled **Use all survey area**. This will enable the three buttons underneath. Make sure the  button is pushed in. Click on the display of the survey at the vertices of the polygon which describes the area you would like to select. Click on the  button to close the polygon and complete your selection. Click on the  button to delete the polygon if you are unsatisfied with it.

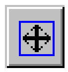
To toggle the display of the search area grid:

- Click the  button.

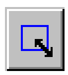
To select a search area:

- Make sure **Search** is selected in the **Edit selected area of** section. The parameters in the **Search Area** pane can be modified and the effect of any changes can be seen by clicking **Apply**.

To move the search area:

- Click the  button and drag the search area box with the mouse.

To resize the search area:

- Click the  button and resize the search area box with the mouse.

## Defining a Geological Structure

Clicking the **Use known geological structure** checkbox or the **Set Structure** button will launch the following window:

**To add a structure to the list:**
- Specify the density, position and radius of the spherical structure in the **Build Geological Structure** section.
- Click the **Add Structure** button.

**To modify an existing entry in the list:**
- Click the line of the structure you would like to edit in the **Geological Structure List**.
- Enter the new susceptibility, position or radius values in the **Build Geological Structure** section.
- Click the **Modify Structure** button.

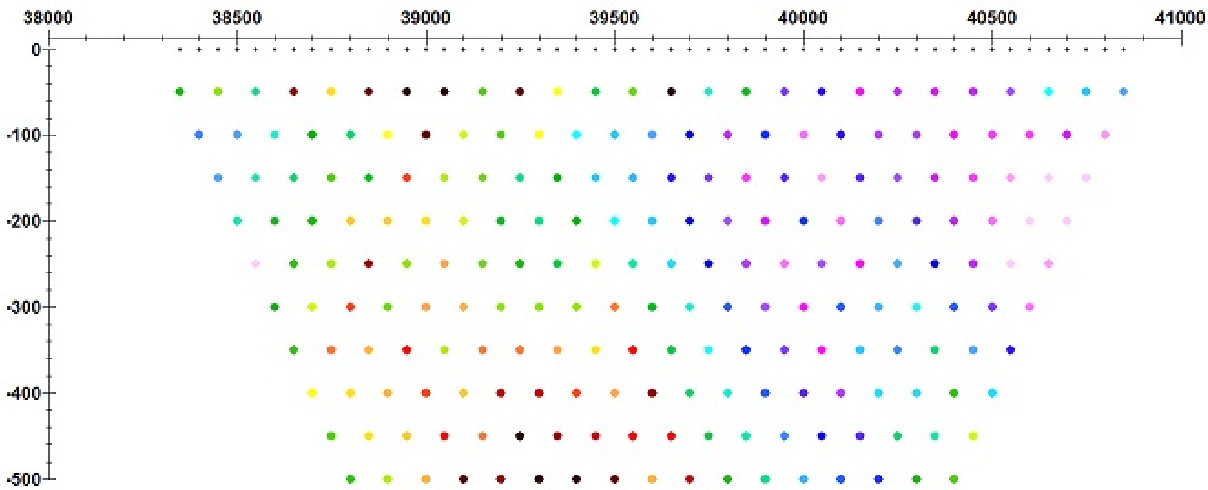
**To delete from the list:**
- Select the entry to be deleted in the list.
- Click **Delete the selected structure**

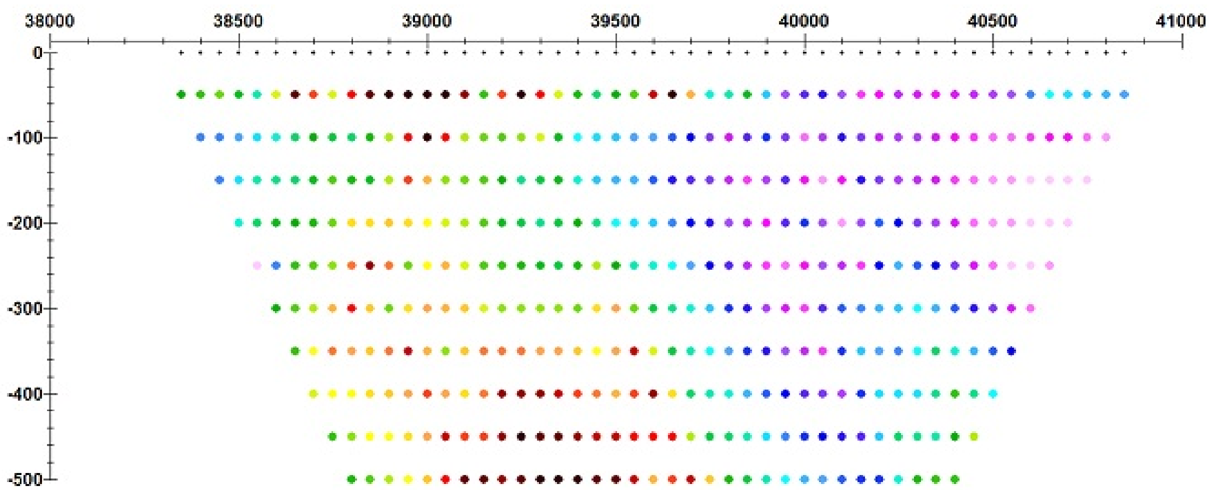
**To save the list to an ascii file:**
- Click the **Save structures to a file** button.
- Specify a location and filename for the file.

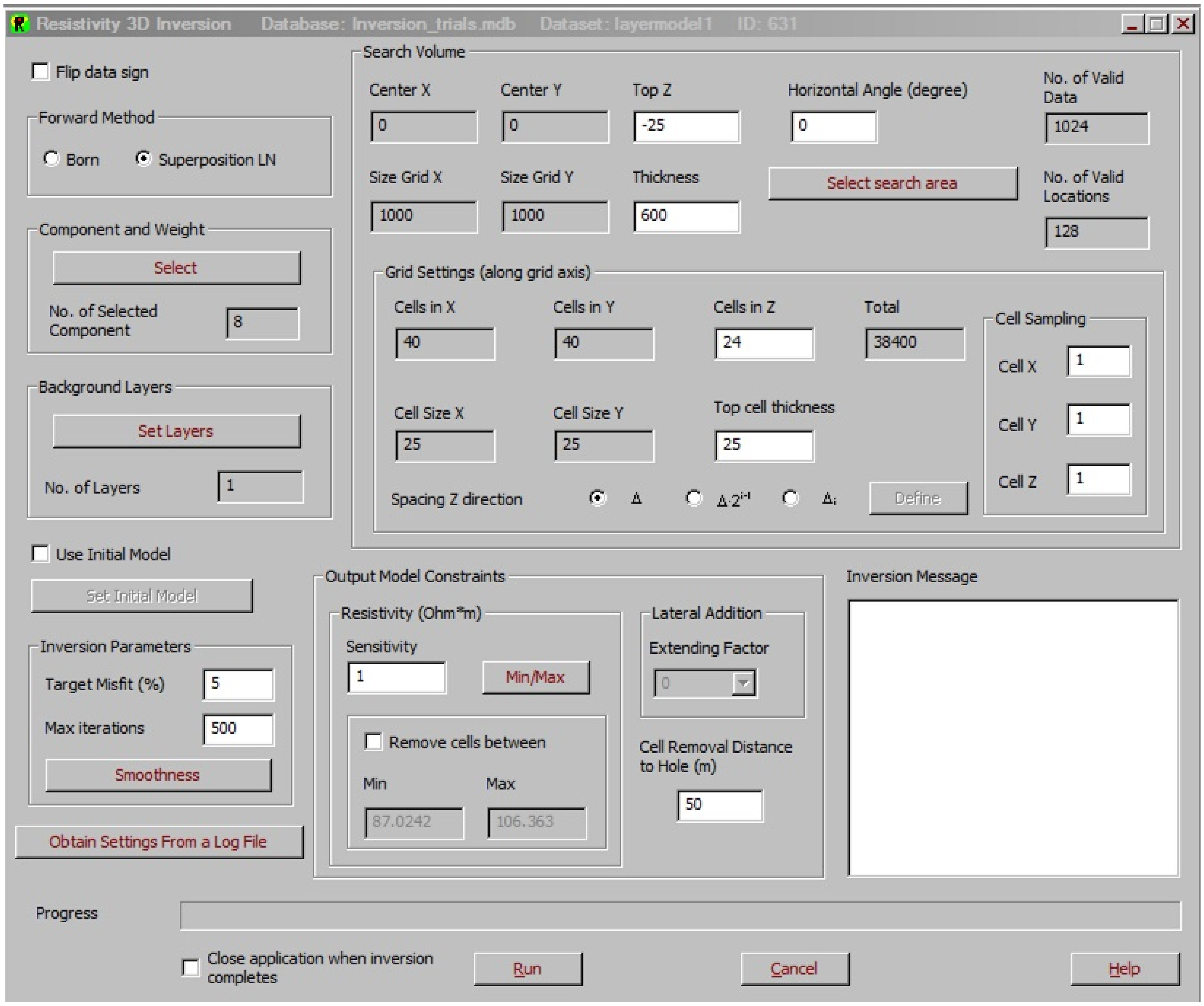
**To import a previously saved list of structures:**
- Click **Load structures from a file**.
- Select the file and the **Geological Structure List** will be populated with the structures in the file.

## Performing Inversion

This inversion is designed to invert DC resistivity data of several survey styles: 1. Surface transmitters and receivers, 2. Surface Tx's, borehole Rx's, 3. Surface to borehole Tx, Surface or Borehole Rx's. It is designed for both moving systems and fixed transmitter surveys. Here, we will illustrate with a standard pole-dipole survey. The figure below shows the data in a pseudosection display both the original data as well as the interpolated data to fill in the gaps since survey procedures to improve production always leaves gaps in the data sampling.






When you have completed adjusting the settings, click the **Run** button to begin processing. When complete, a new data set will be created. The inversion results can be viewed graphically using EMIGMA's Visualizer or 3D contour tool.

**Forward Model** The majority of resistivity inversions use approaches in their forward routine which do not maintain interaction between cells nor account for the bending of currents. This approach is termed a Born approximation. Our forward techniques also provide the full scattering effect of a cell but do not include interactions between cells. This approximation is termed a Superposition LN approximation.

## Component and Weight

Click the **Select** button to launch the interface that will allow you to choose to invert on any or all of the components in the dataset and assign weights to each one.

**Set Layers** The forward technique and thus the inversion is an integral equation technique and thus looks for anomalous structure in user prescribed layered background. This approach has numerous numerical as well as theoretical advantanges. User layered earth modeling is required to derive a reasonable background model. Or EMIGMA's 1D inversion can be utilized using all of the data to find the one best layered model.

**Survey Area**
Click the **Select search area** button to launch the graphical tool which enables you to specify the data points that will be used in the inversion calculations.

**Search Volume**
The default parameters in the **Search Volume** section will create a grid that is slightly smaller than the entire survey. You can modify the search area parameters by entering new values or by using the graphical tool available by clicking the button labelled **Select search area**.

**Cell Sampling**
Grid cells defined in **Search Volume** can be divided into smaller units when calculating the simulated data by clicking **Cell Sampling**. Type your values in the X, Y and Z boxes to specify the number of samples in the X, Y and Z directions.

**Grid Settings**
Confirm the number and layout of grid points to be used in the inversion in the **Grid Settings** area. The points will be evenly spaced in the x and y directions. You may choose between **Evenly** or **Exponentially** spaced points.

**Log File**
A log file is created each time an inversion is run. Click **Get Settings From a Log File** to use the settings from a previous inversion.

**Background Layers**
Select the **Set Layer Model** checkbox to specify a layer model to be used during the inversion. Modify the layer model by clicking the **Set Layers** button.

**Initial Model**
Click the checkbox labelled **Use Initial Model** to specify an initial model. Return to the initial model window by clicking the **Set Initial Model** button.

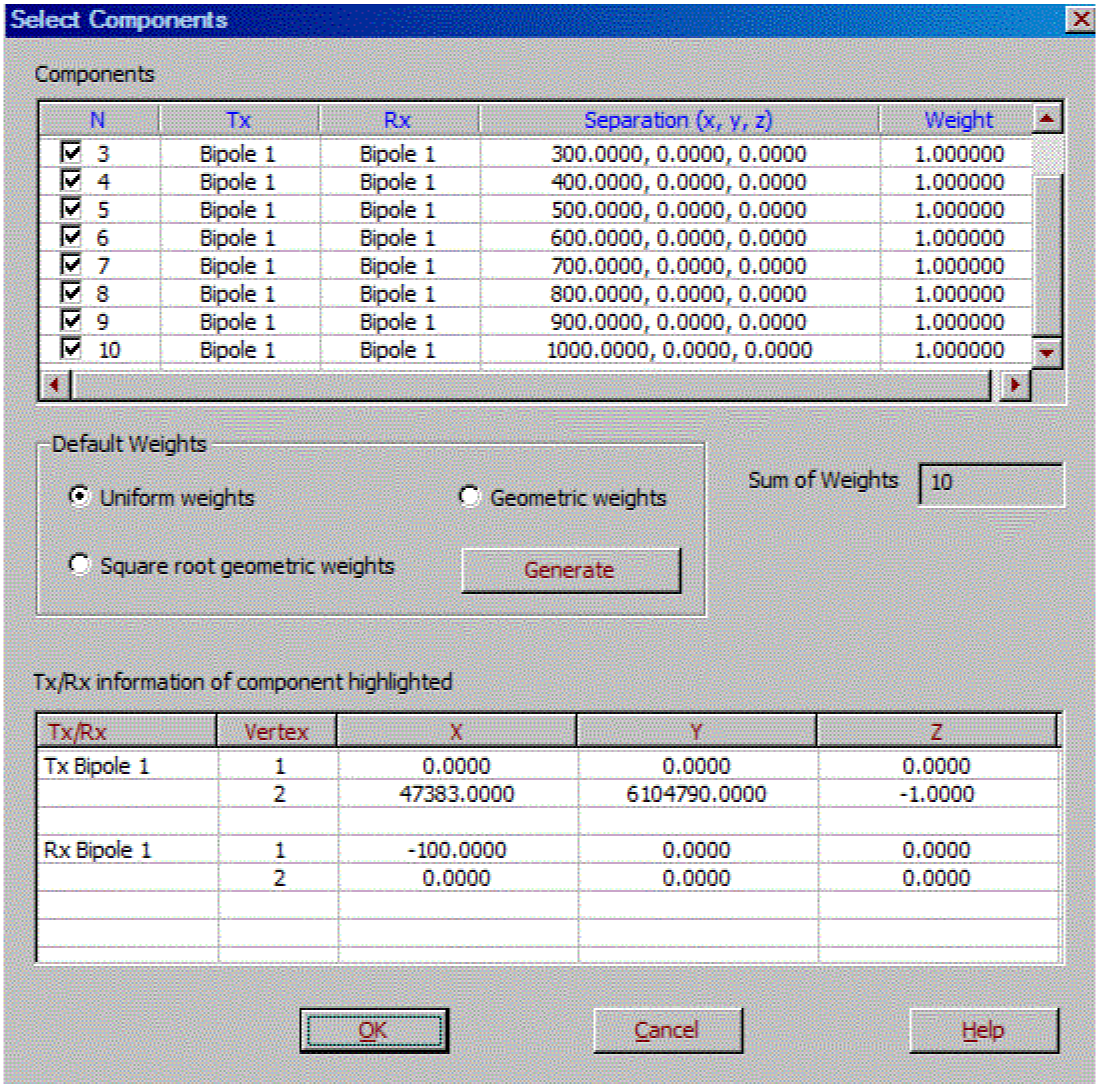
**Output Model Resistivity Constraint**
Cells with an absolute resistivity less than the **Sensitivity** will be constrained or thrown out after each iteration and will not be output to the resistivity distribution (.res) file.
Upon completion of each iteration, a resistivity smaller than **Min** will be set to **Min** and a resistivity greater than **Max** will be set to **Max**.

**Remove cells between**
Any cells with resistivity values that are between the specified **Min** and **Max** will be removed from the inversion result.

**Inversion Parameters**
**Max iterations** is the number of iterations the program will run to generate the final solution. In general, the default value is sufficient for the inversion.
A larger **Smoothness** value will increase the smoothness of the inversion result by decreasing the range of the resistivity values.
The inversion solution will be considered found when the difference between the measured and simulated data falls below the **Target Misfit**.

## Specifying Components

Click **Select** in the **Component and Weight** section and the following window appears:




Select the components to be used in the inversion by clicking the checkboxes in the **N** column of the **Components** list. In this case there are 2 Tx electrodes, one local and one distance and the receiver dipoles are offset from the current injection electrode with 10 separations from 100m to 1000m

### Default Weights

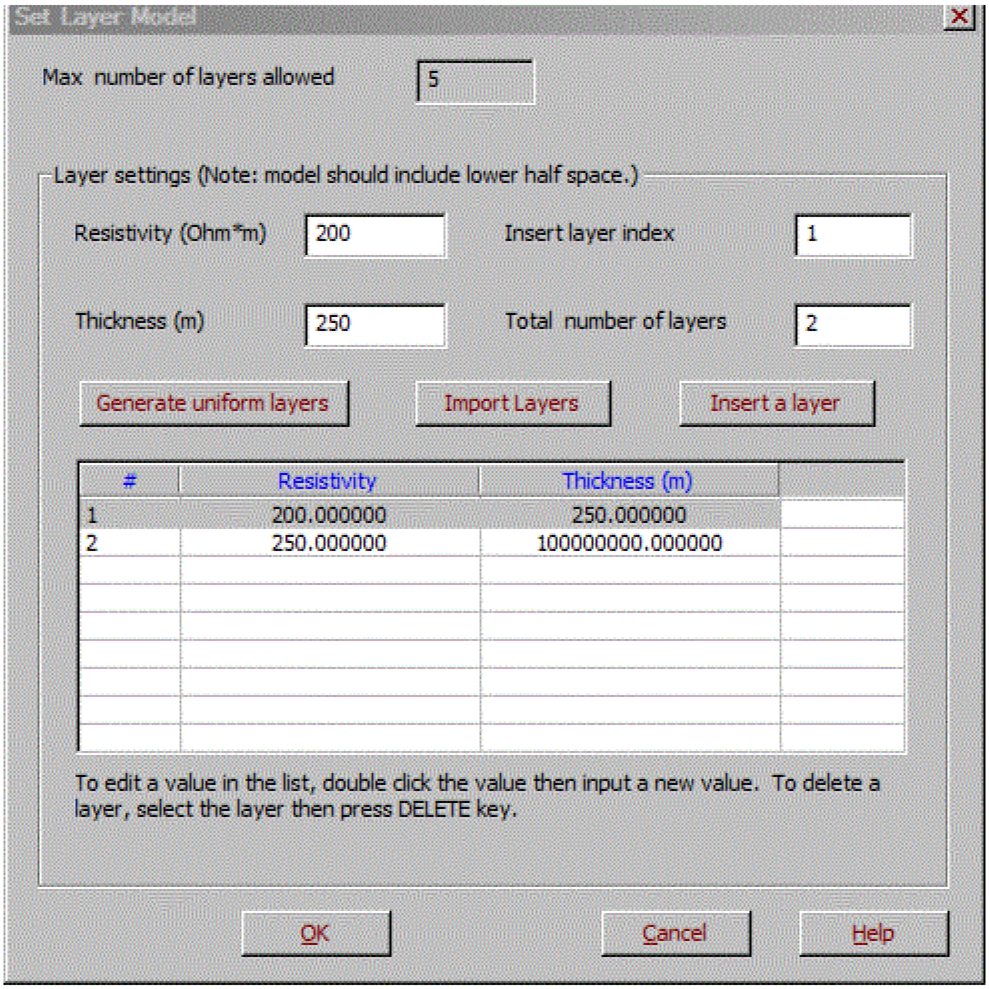
- **Uniform Weights** - each component will be given equal weighting
- **Geometric Weights** - weighting for each component will be calculated using the tx-rx separation
- **Square root geometric weights** - weight values are the square roots of the **Geometric Weights**

Edit a weight by double clicking the value you would like to change. Enter a new value and press Enter.

Information about the vertices of the transmitter and receiver can be found in the **Tx/Rx information** section.

## Specifying a Layer Model

Select **Set Layer Model** or click the **Set Layers** button and the following window appears:




To load a layer model from another data set:

- Click **Import layers** to load an entire or part of a layer model from another data set.

To create a set of identical layers:

- Specify the layer resisitivity and thickness as well as the number of desired layers then click **Generate uniform layers**. The deepest layer generated will be the basement and will have a thickness of 1e+8.

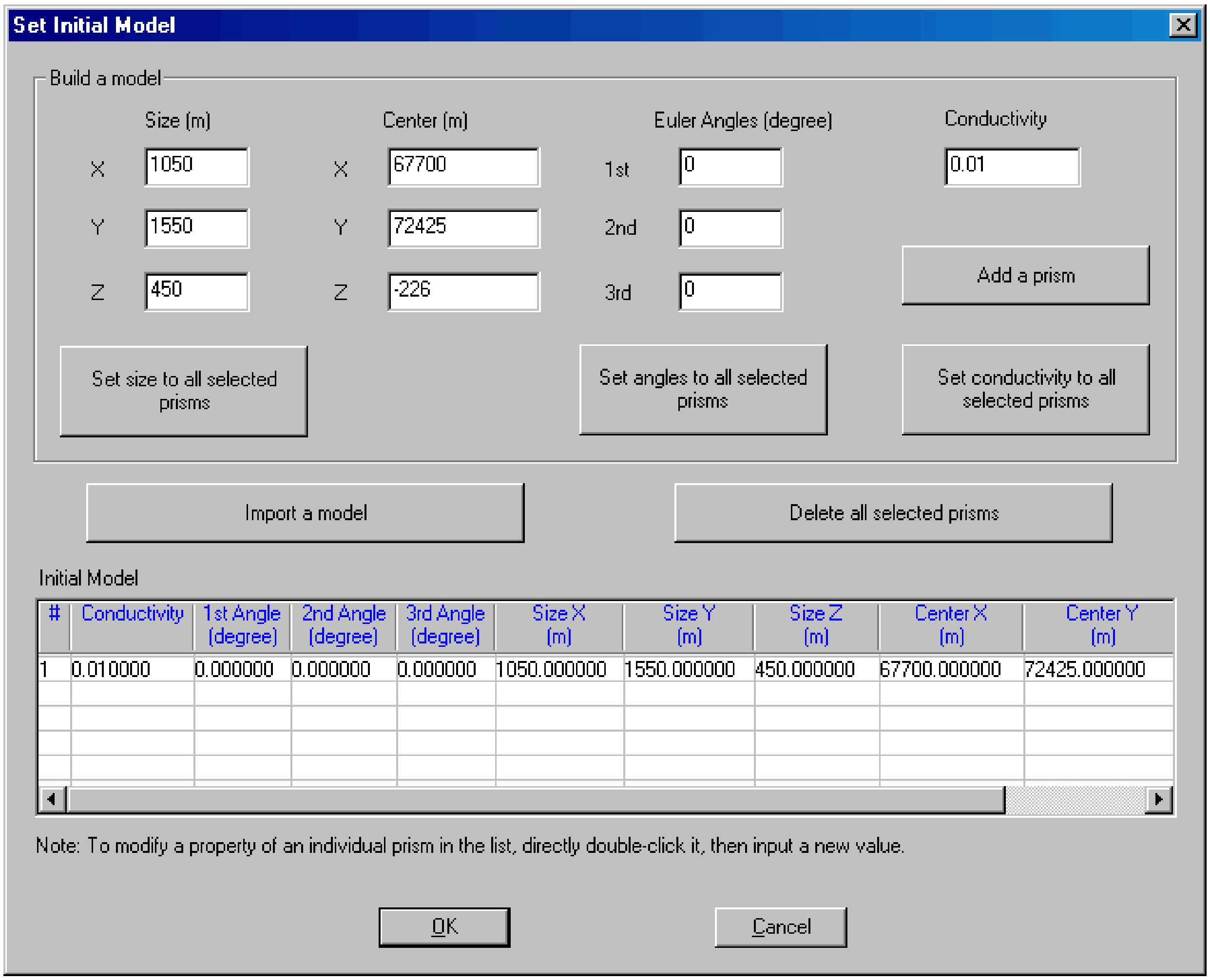
To insert one layer:
- Specify the layer resistivity and thickness. Insure the Total number of layers displayed is the same as in the list.
- Click **Insert a layer** and a new layer will be inserted at the location specified by the **Insert layer index**.

To delete a layer:
- Select the desired from the list and delete with the DELETE key.

To edit a value in the list:
- Double click the value then input a new value.

## Specifying an initial model

Selecting the **Use Initial Model** checkbox or clicking the **Set Initial Model** button will launch the following window:




The starting model is described by a list of prisms with various properties in the box labelled **Initial Model**

### To add a prism to the model list:

- Specify the conductivity, size, position and orientation of the new prism in the **Build/Modify a prism** section.
- Click the **Add a prism** button.

**To modify an existing prism in the model list:**

- Double click the cell of the property you would like to edit in the Initial Model list.
- Input the new value and press Enter.

**To delete prisms from the model list:**

- Select the prisms to be deleted in the anomaly list.
- Click **Delete all selected prisms**

**To apply the same values for a group of selected prisms:**

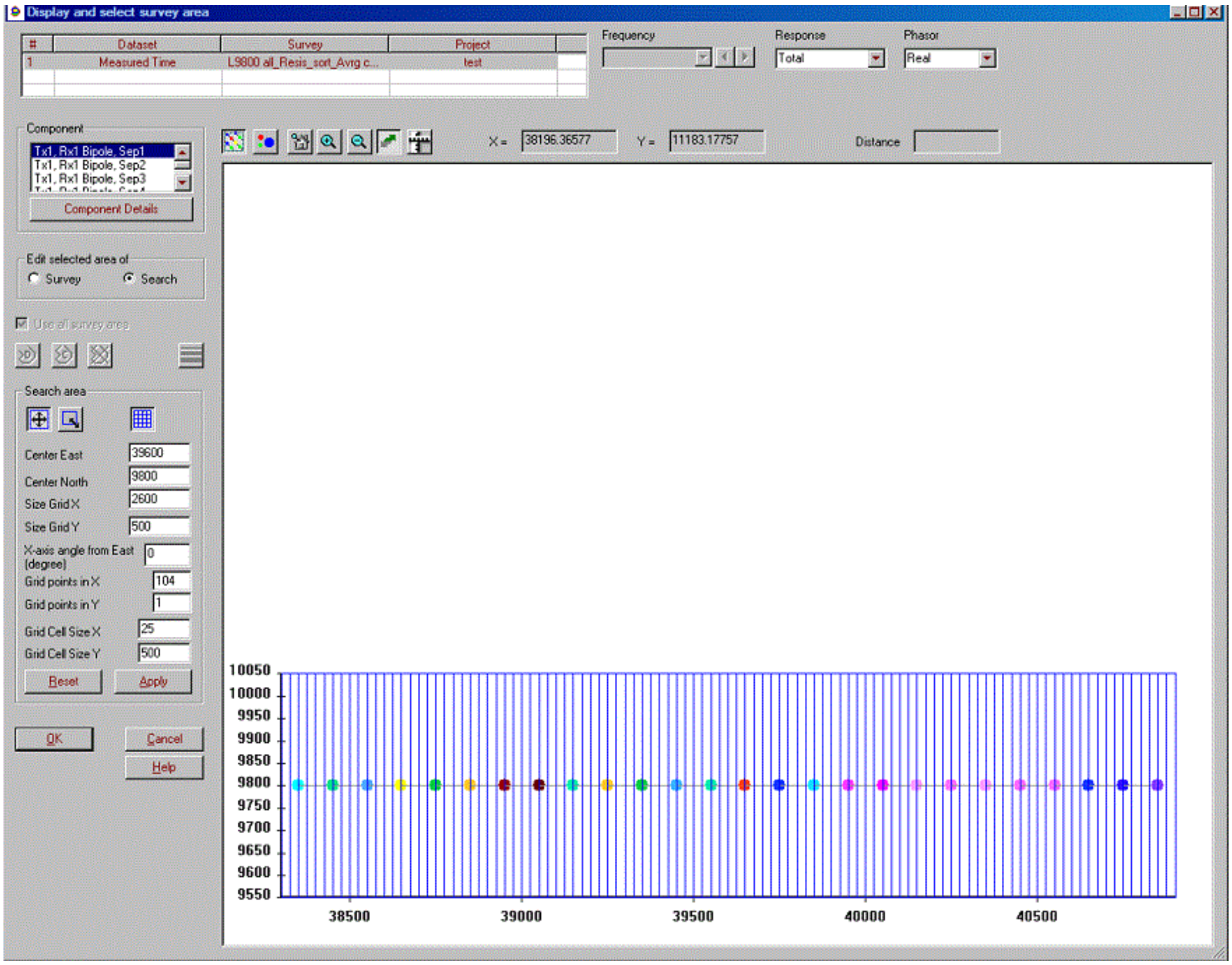
- Click the **Set conductivity to all selected prisms** button to modify the conductivity.
- Click the **Set angles to all selected prisms** button to modify the angles.
- Click the **Set size to all selected prisms** button to modify the size.

**To import a model from another data set in the current database:**

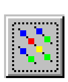
- Click **Import a model**.
- Select the project, survey, and data set with the desired model
- Click **OK** and the model will appear in the **Initial Model** list

## Selecting Survey and Search Area

Clicking the **Select Search Area** button launches the following window:



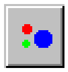

To toggle display of data values:

- Click the  button.

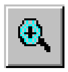
To change display of component or response:

- Make a selection from the **Component** and **Response** list.

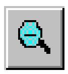
To change the size of the symbols for each data value:

- Click the  button. Use the slider on the window to appear to change the symbol size. Select **Show change immediately** to see the symbols change size as you move the slider.

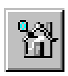
To zoom in on a section of the survey:

- Click the  button. Drag the mouse on the survey display to draw a rectangle describing area you would like to zoom in on.

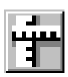
To zoom out to the previous zoom level:

- Click the  button.

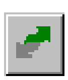
To zoom out to the original zoom level:

- Click the  button.

To find the distance between two points on the survey display:

- Click the  button. Drag the mouse between the two points on the survey display and the distance will be displayed at the top of the window in the box labelled **Distance**.

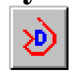
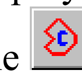
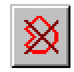
To display the survey in a proportional view:

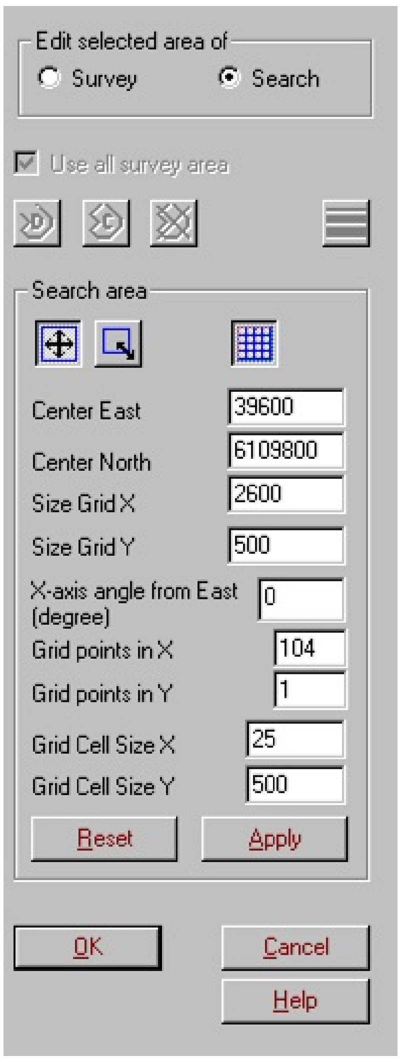
- Click the  button.

To find the coordinates of a location on the survey display:

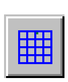
- Move the mouse to the location and the coordinates will be displayed at the top of the window in the boxes labelled **X=** and **Y=**

To select only a section of the survey:

- Deselect the checkbox labelled **Use all survey area**. This will enable the three buttons underneath. Make sure the  button is pushed in. Click on the display of the survey at the vertices of the polygon which describes the area you would like to select. Click on the  button to close the polygon and complete your selection. Click on the  button to delete the polygon if you are unsatisfied with it.



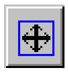

To toggle the display of the search area grid:

- Click the  button.

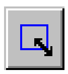
To select a search area:

- Make sure **Search** is selected in the **Edit selected area of** section. The parameters in the **Search Area** pane can be modified and the effect of any changes can be seen by clicking **Apply**.

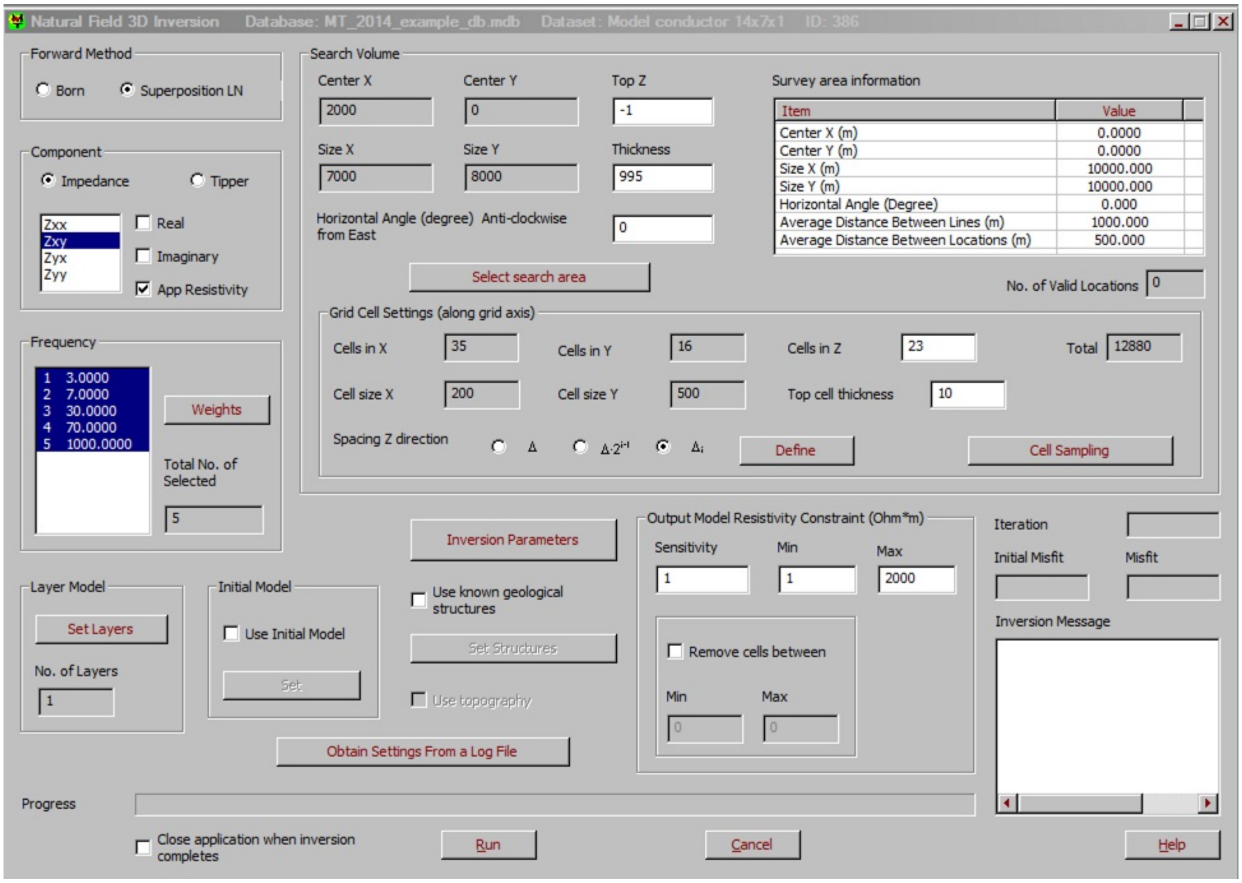
To move the search area:

- Click the  button and drag the search area box with the mouse.

To resize the search area:

- Click the  button and resize the search area box with the mouse.

# Performing MT/AFMAG/ZTEM Inversion




When you have completed adjusting the settings, click the **Run** button to begin processing. When complete, a new data set will be created. The inversion model results can be viewed graphically using EMIGMA's visualizer or contour tool. The data can be viewed in the plotter. Cell removal post-processing may be done by selecting **Data Processing** from the main toolbar then **Inversion Model Processing** and **Cell Removal** from the windows that appear.

**Forward Method** There are two forward modeling technique options: **Born** (weak scattering) and **Superposition LN** (strong scattering)

**Component**

You can choose to invert on the real and imaginary tipper or impedance components. Or Apparent Resistivity for the impedance components.

**Frequency**
You may select the frequencies that will be used when performing inversion by clicking the entries in the list box. By default, each frequency is given equal weight. This can be changed by clicking **Weights**.

**Survey Area**
Click the **Select search area** button to launch the graphical tool which enables you to specify the data points that will be used in the inversion calculations.

**Search Volume**
The default parameters in the **Search Volume** section will create a grid that almost covers the entire survey. You can modify the search area parameters by entering new values or by using the graphical tool available by clicking the button labelled **Select search area**.

**Cell Sampling**
Grid cells defined in **Search Volume** can be divided into smaller units when calculating the simulated data by clicking **Cell Sampling**. Type your values in the X, Y and Z boxes to specify the number of samples in the X, Y and Z directions.

**Grid Settings**
Confirm the number and layout of grid points to be used in the inversion in the **Grid Settings** area. The points will be evenly spaced in the x and y directions. Choose $\mathbf{\Delta}$ for evenly spaced points in the z direction or $\mathbf{\Delta \cdot 2^{i-1}}$ for exponentially spaced points. You may specify a custom spacing by selecting $\mathbf{\Delta_i}$. Your custom settings can be later modified by clicking **Define**.

**Log File**
A log file is created each time an inversion is run. Click **Get Settings From a Log File** to use the settings from a previous inversion.

**Set Layers**
This technique is an integral equation technique and thus a background layered model is required. You can determine this via inverting multiple stations in the 1D inversion.

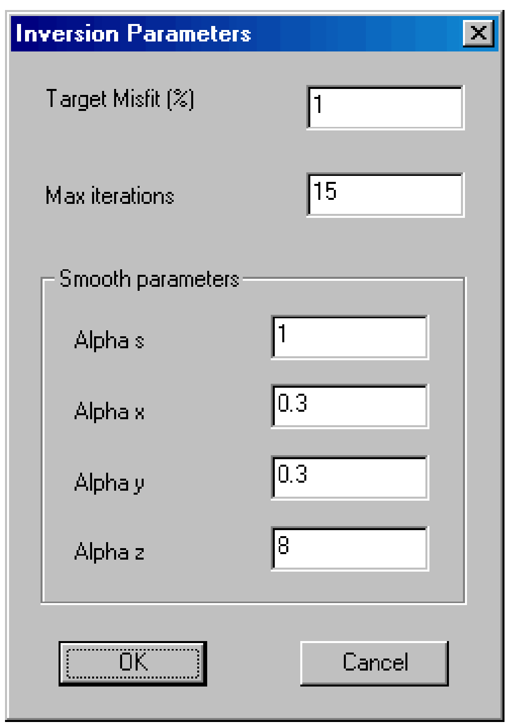
**Initial Model**
Click the checkbox labelled **Use Initial Model** to specify an initial model.
Return to the initial model window by clicking the **Set** button.

**Geological Structure**
Click Use known geological structure to define a structure that will apply constraints to the inversion result. Your settings can be later modified by clicking **Set Structure**.

**Output Model Resistivity Constraint**
Cells with an absolute resistivity less than the **Sensitivity** will not be output to the resistivity distribution (.res) file.
Resistivity values are constrained to be between **Min** and **Max**

**Remove cells between**
Any cells with resistivity values that are between the specified **Min** and **Max** will be removed from the inversion result.

**Inversion Parameters**
Clicking this button will display the following window:



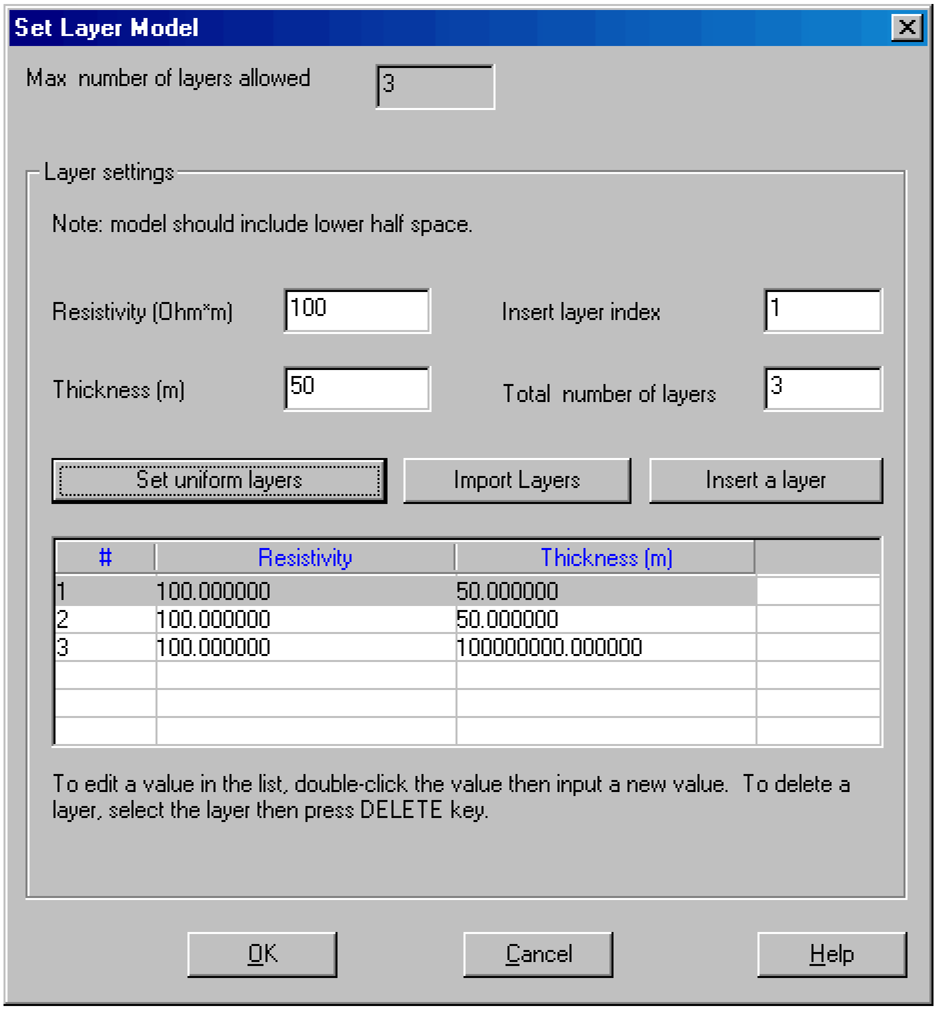
The inversion solution will be considered found when the difference between the measured and simulated data falls below the **Target Misfit**.

**Max iterations** is number of iterations the program will run to generate the final solution. In general, the default value is sufficient for the inversion.

The coefficient **Alpha s** controls the overall property variation of the inverted model. A increased value of this coefficient leads to a smoother inversion model with small variations while a decreased value gives a model with relatively sharp variations. The 3 coefficients **Alpha x**, **Alpha y** and **Alpha z** specify the relative measure of the model variation in 3 spatial directions. By varying the relative magnitudes of these coefficients, the inversion can construct models that are smoother in the preferential spatial directions. For instance, choosing **Alpha x** to be bigger than the other two coefficients will produce a model with an enhanced smoothness in the X direction.

## Specifying a Layer Model

Select **Set Layer Model** or click the **Set Layers** button and the following window appears:




To load a layer model from another data set:

- Click **Import layers** to load an entire or part of a layer model from another data set.

To create a set of identical layers:

- Specify the layer resisitivity and thickness as well as the number of desired layers then click **Set uniform layers**. The deepest layer generated will be the basement and will have a thickness of 1e+8.

To insert one layer:

- Specify the layer resistivity and thickness. Insure the Total number of layers displayed is the same as in the list.
- Click **Insert a layer** and a new layer will be inserted at the location specified by the **Insert layer index**.

To delete a layer:

- Select the desired from the list and delete with the DELETE key.

To edit a value in the list:

- Double click the value then input a new value.

## Editing the Initial Model.

Clicking the **Use Initial Model** button will launch the following window:

The starting model is described by a list of prisms with various properties in the box labelled **Initial Model**

**To add a prism to the model list:**

- Specify the resistivity, size, position and orientation of the new prism in the **Build a model** section.
- Click the **Add a prism** button.

**To modify an existing prism in the model list:**

- Double click the cell of the property you would like to edit in the Initial Model list.
- Input the new value and press Enter.

**To delete prisms from the model list:**

- Select the prisms to be deleted in the list.
- Click **Delete all selected prisms**

**To apply the same values for a group of selected prisms:**

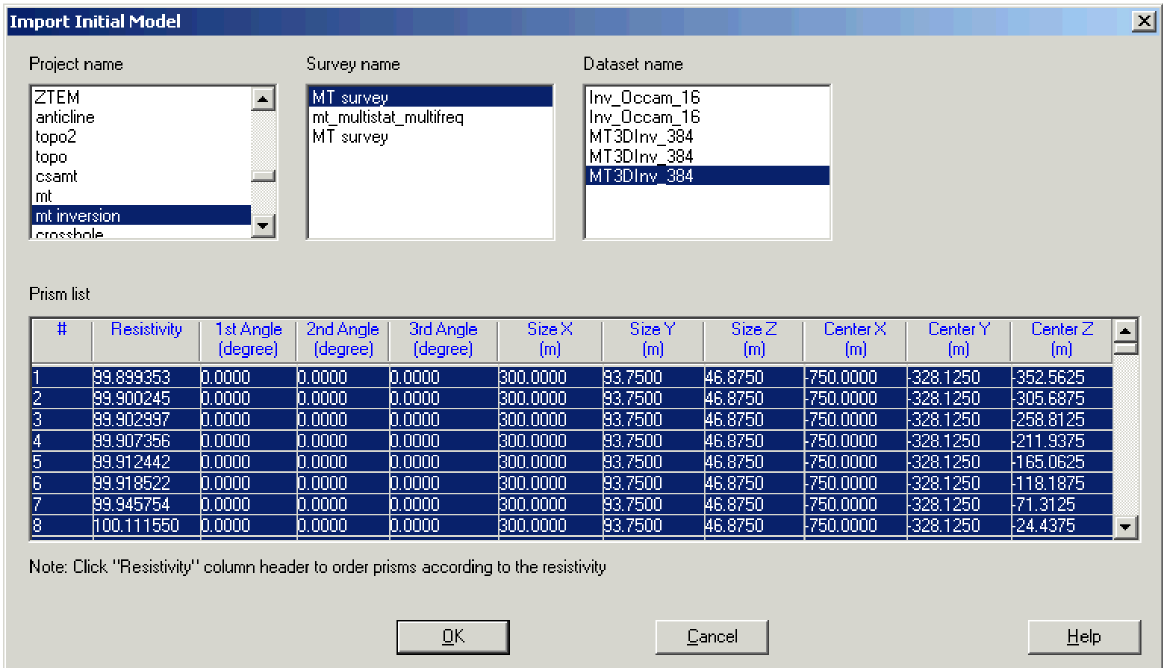
- Click the **Set resistivity to all selected prisms** button to modify the resistivity.
- Click the **Set angles to all selected prisms** button to modify the angles.
- Click the **Set size to all selected prisms** button to modify the size.

**To import a model from another data set in the current database:**

- Click Import a model.
- Select the project, survey, and data set with the desired model

- Click **OK** and the model will appear in the **Initial Model** list

## Importing a model




- All eligible projects are listed. Choose the project, survey and data set that has the model you want to use. The **Prism** list can be sorted according to resistivity by clicking the **Resistivity** column header. Click **OK** to import the model.

## Editing the Grid Cell Thickness.

Selecting $\mathbf{\Delta_i}$ or clicking the **Define** button in the **Grid Settings** section will launch the following window:

The interface displays the total thicknesses before and after editing as well as the topmost z value. The cell sizes are listed in the **Search grid cell thickness** section.

**To modify an existing entry:**

- Specify the **Thickness** and select the entries to which you would like to assign this thickness.
- Click **Modify the selected**.

**To insert a new entry:**

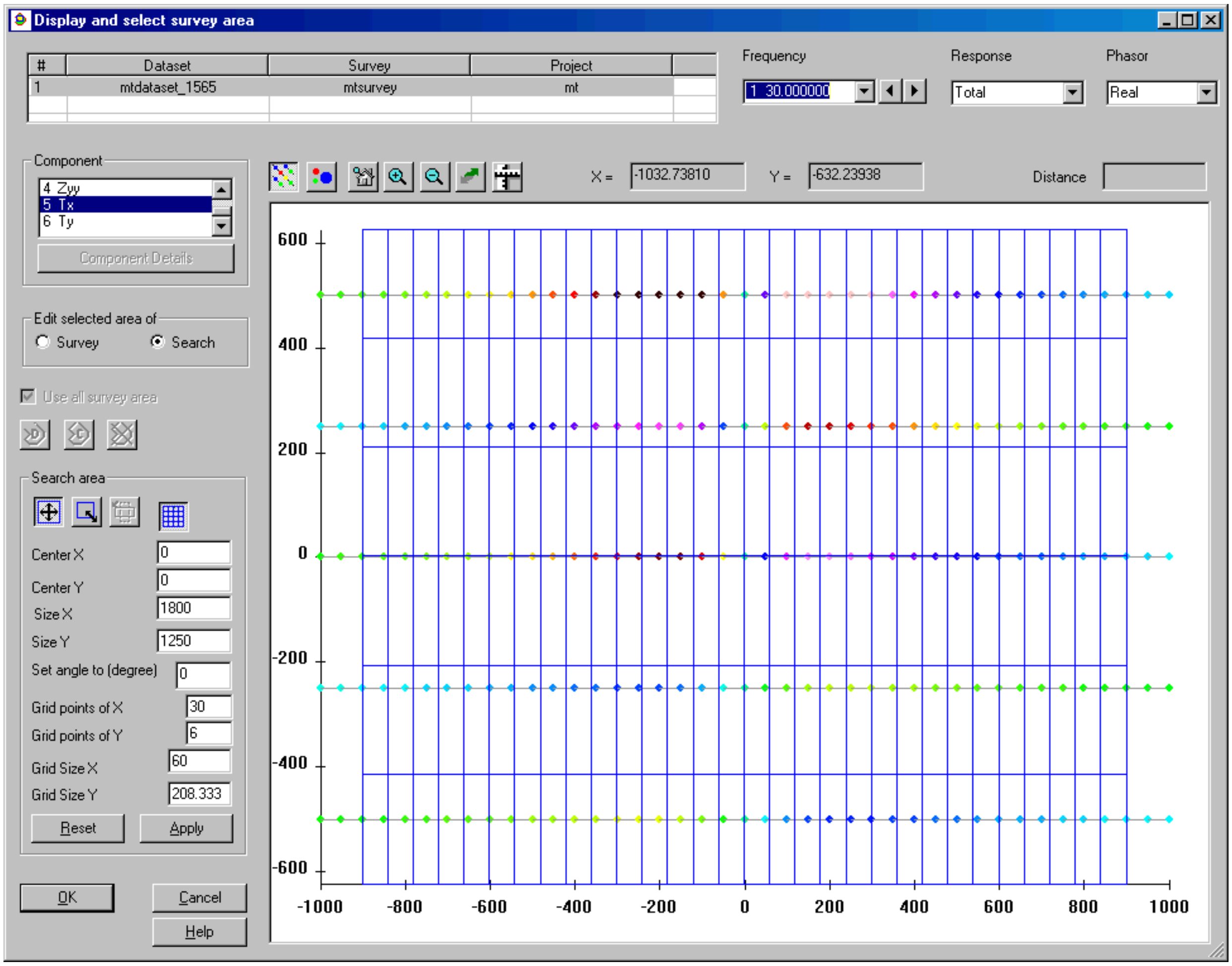
- Specify the **Thickness**.
- Specify the line where the new entry should be inserted in the **Insert Index** box.
- Click the **Insert a thickness** button.

**To delete an entry:**

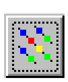
- Select the entries to be deleted.
- Click **Delete the selected**.

## Selecting Survey and Search Area

Clicking the **Select Search Area** button launches the following window:



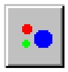

To toggle display of data values:

- Click the  button.

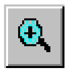
To change display of component or response:

- Make a selection from the **Component** and **Response** list.

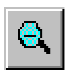
To change the size of the symbols for each data value:

- Click the  button. Use the slider on the window to appear to change the symbol size. Select **Show change immediately** to see the symbols change size as you move the slider.

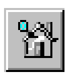
To zoom in on a section of the survey:

- Click the  button. Drag the mouse on the survey display to draw a rectangle describing area you would like to zoom in on.

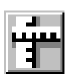
To zoom out to the previous zoom level:

- Click the  button.

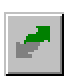
To zoom out to the original zoom level:

- Click the  button.

To find the distance between two points on the survey display:

- Click the  button. Drag the mouse between the two points on the survey display and the distance will be displayed at the top of the window in the box labelled **Distance**.

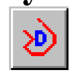
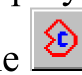
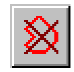
To display the survey in a proportional view:

- Click the  button.

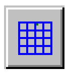
To find the coordinates of a location on the survey display:

- Move the mouse to the location and the coordinates will be displayed at the top of the window in the boxes labelled **X=** and **Y=**

To select only a section of the survey:

- Deselect the checkbox labelled **Use all survey area**. This will enable the three buttons underneath. Make sure the  button is pushed in. Click on the display of the survey at the vertices of the polygon which describes the area you would like to select. Click on the  button to close the polygon and complete your selection. Click on the  button to delete the polygon if you are unsatisfied with it.

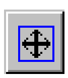
To toggle the display of the search area grid:

- Click the  button.

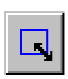
To select a search area:

- Make sure **Search** is selected in the **Edit selected area of** section. The parameters in the **Search Area** pane can be modified and the effect of any changes can be seen by clicking **Apply**.

To move the search area:

- Click the  button and drag the search area box with the mouse.

To resize the search area:

- Click the  button and resize the search area box with the mouse.

## Defining a Geological Structure

Clicking the **Use known geological structure** checkbox or the **Set Structure** button will launch the following window:

Various propeties of the structures are listed in the box labelled **Geological Structure List**

**To add a structure to the list:**

- Specify the resistivity, position and radius of the spherical structure in the **Build or Edit a Geological Structure** section.
- Click the **Add to List** button.

**To modify an existing entry in the list:**

- Click the line of the structure you would like to edit in the **Geology Structure List**.
- Enter the new resistivity, position or radius values in the **Build or Edit a Geological Structure** section.
- Click the **Modify** button.

**To delete from the list:**

- Select the entry to be deleted in the list.
- Click **Delete the selected structure**
  OR
- Click **Delete all structures** to clear the entire list.

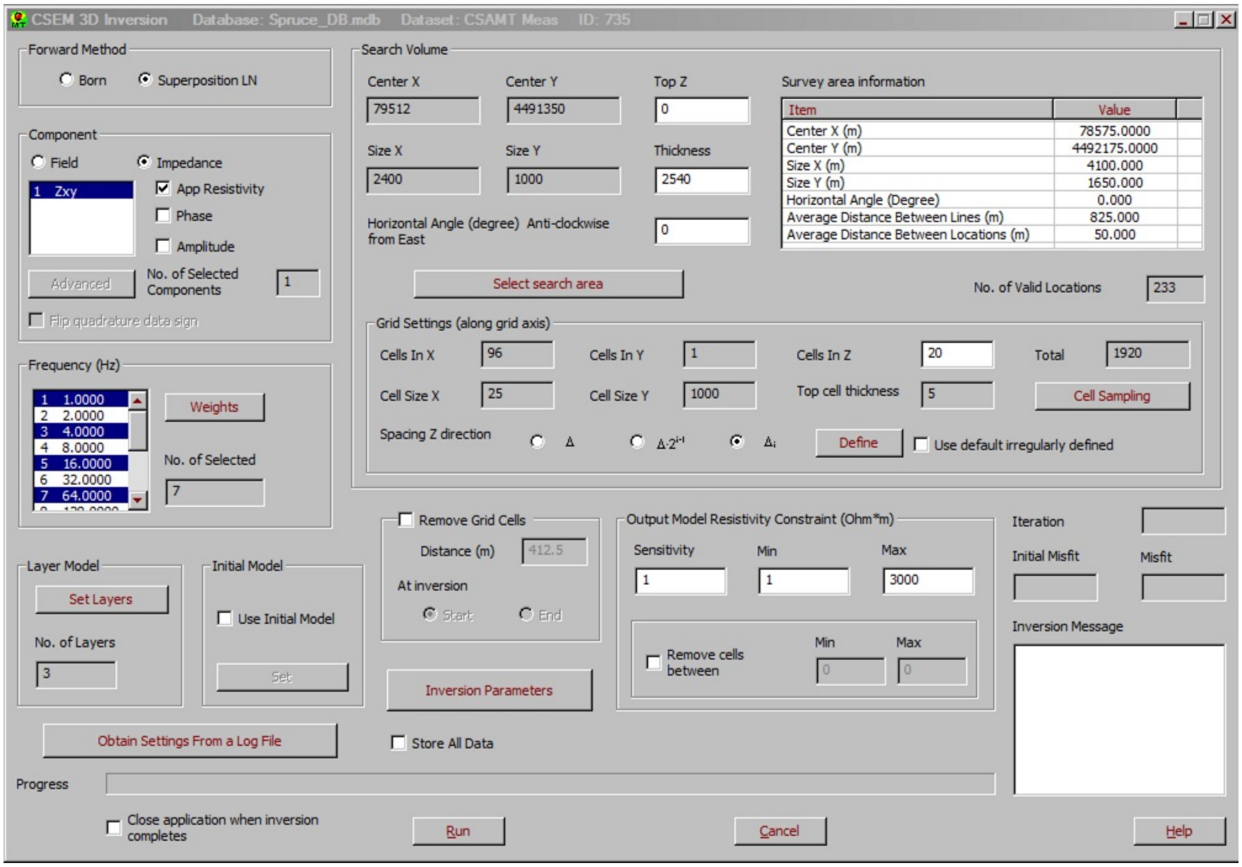
**To save the list to an ascii file:**

- Click the **Save structures to a file** button.
- Specify a location and filename for the file.

**To import a previously saved list of structures:**

- Click **Load structures from a file**.
- Select the file and the **Geology Structure List** will be populated with the structures in the file.

# Performing Inversion




**Overview** When you have completed adjusting the settings, click the **Run** button to begin processing. When complete, a new data set containing the synthetic data of the inversion model will be created. The inversion model results can be viewed graphically using EMIGMA's 3D Visualizer or 3D Contour tool. The data can be viewed in the plotter with comparisons to your data and other inversion and forward models. Cell removal post-processing may be done by selecting **Data Processing** from the main toolbar (gear icon) then **Inversion Model Processing** and **Cell Removal** from the windows that appear. Also, export of the inversions to other formats is done through these tools.

**Forward Method**

There are two forward modeling technique options: **Born** (weak scattering) and **Superposition LN** (strong scattering) which is a specific EMIGMA capability. However, the latter is much slower than the former.

**Component**
You can choose to invert on the real and imaginary phase of the selected fields. For impedance components, you can select apparent resistivity, phase or amplitude. If the data set has multiple transmitters, the **Advanced** button for advanced component selection will be enabled.

**Frequency**
You may select the frequencies that will be used when performing inversion by clicking the entries in the list box. By default, each frequency is given equal weight. This can be changed by clicking **Weights**.

**Survey Area**
Click the **Select search area** button to launch the graphical tool which enables you to specify the data points that will be used in the inversion calculations. This is not normally used as you can easily create a subset of your data and save to your database.

**Search Volume**
The Search Volume is the volume in 3D space inside which the inversion will try to find anomalous resistivities. The default parameters in the **Search Volume** section will create a grid that almost covers the entire survey. You can modify the search area parameters by entering new values or by using the graphical tool available by clicking the button labelled **Select search area**. Thus, you may select the size of the horizontal rectangle defining the volume, the azimuth of this inversion grid and the deltaX, deltaY sampling of the grid.

**Cell Sampling**
Grid cells defined in **Search Volume** can be divided into smaller units when calculating the simulated data by clicking **Cell Sampling**. Type your values in the X, Y and Z boxes to specify the number of samples in the X, Y and Z directions. In techniques such as FD and FE, each cell has only one value for the internal electric fields which form the sources for the secondary measured field. In this technique, you may set the internal current sampling as desired. For Example, if a quasi-2D, then the cell dimension in the strike direction may be large and thus you should increase sampling in that direction.

**Grid Settings**
Confirm the number and layout of horizontal grid points to be used in the inversion in the **Grid Settings** area. The points will be evenly spaced in the x and y directions. Choose **Δ** for evenly spaced points in the z direction or $\mathbf{\Delta \cdot 2^{i-1}}$ for exponentially spaced points. You may specify a custom spacing by selecting $\mathbf{\Delta_i}$. Your custom settings can be later modified by clicking **Define**. Select the checkbox labelled **Use default irregularly defined** to use default custom spacing.

**Log File**
A log file is created each time an inversion is run. Click **Get Settings From a Log File** to use the settings from a previous inversion.

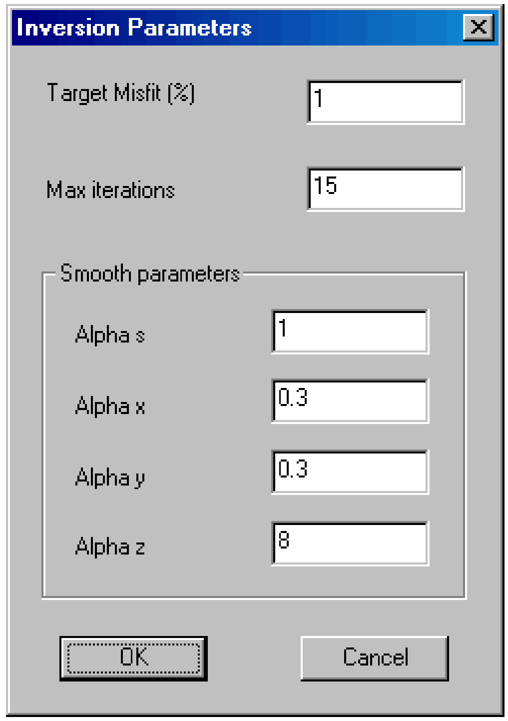
**Initial Model**
Click the checkbox labelled **Use Initial Model** to specify an initial model. Return to the initial model window by clicking the **Set** button.

**Output Model Resistivity Constraint**
Resistivity values are constrained to be between **Min** and **Max** Cells with an absolute resistivity which have a difference between the cell resistivity and the background resistivity less than the **Sensitivity** will not be output to the resistivity distribution (.res) file.

**Remove cells between**
Any cells with resistivity values that are between the specified **Min** and **Max** will be removed from the inversion result. This is designed for limiting the high resistivity and low resistivity range.

**Store All Data**
Certain data are chosen to be used in the inversion. However, it can be useful to see how the inversion models fits the data that was not utilized. This selection simulates the final inversion model for all components and frequencies.

**Inversion Parameters**
Clicking this button will display the following window:




The inversion solution will be considered found when the difference between the measured and simulated data falls below the **Target Misfit** or the target misfit can no longer be reduced.

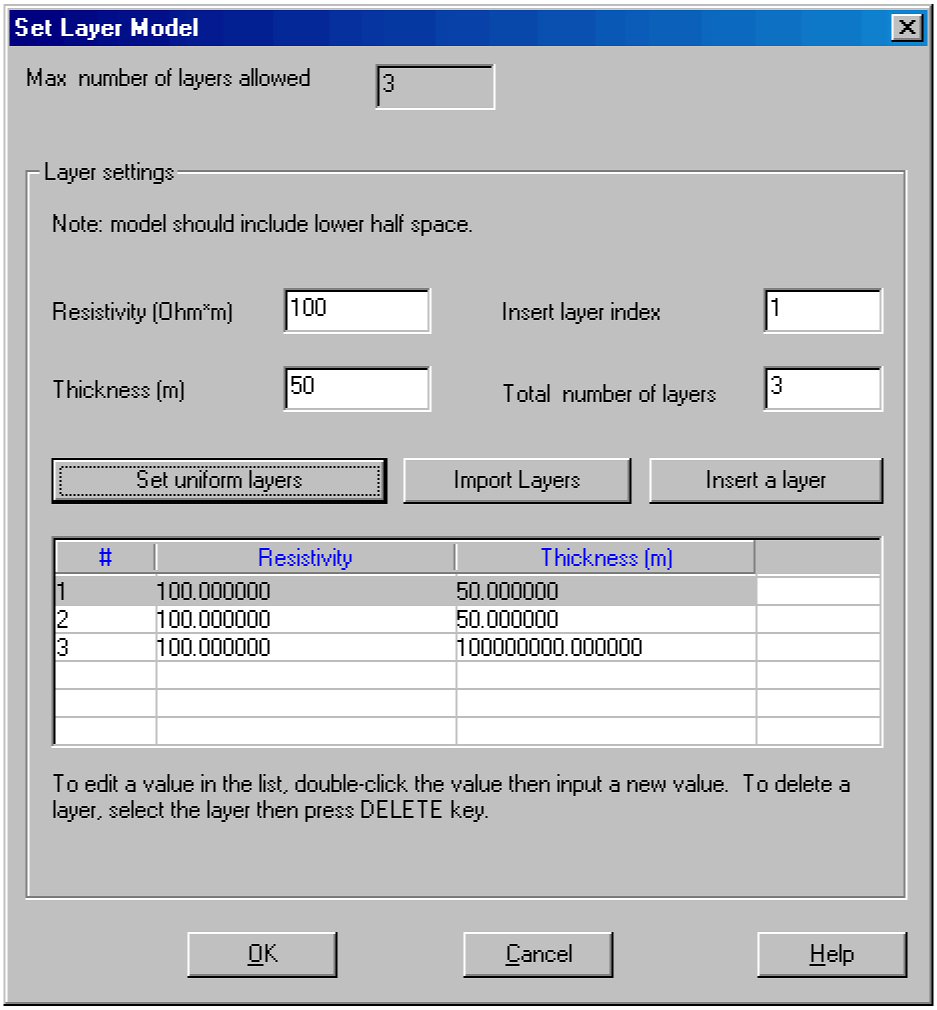
**Max iterations** is number of iterations the program will run to generate the final solution. In general, the default value is sufficient for the inversion.

The coefficient **Alpha s** controls the overall property variation of the inverted model. A increased value of this coefficient leads to a smoother inversion model with small variations while a decreased value gives a model with relatively sharp variations. The 3 coefficients **Alpha x**, **Alpha y** and **Alpha z** specify the relative measure of the model variation in 3 spatial directions. By varying the relative magnitudes of these coefficients, the inversion can construct models that are smoother in the preferential spatial directions. For instance, choosing **Alpha x** to be bigger than the other two coefficients will produce a model with an enhanced smoothness in the X direction.

## Specifying a Layer Model

For this inversion, the cells represent variations from a background layered model. The primary response to a ground current source is from the average resistivity variation with depth. By utilizing this model as the host model, we eliminate two import issues in geophysical inversion for resistivity variations and these issues are both related to model convergence. We no longer need to be concerned if our grid is taken sufficiently far away from the survey area to insure numerical convergence and we place much better controls on the ability to obtain and test convergence via the density of the cells. The issue of cell density is no longer an issue of numerical convergence of the forward algorithm but the issue of most importance to the user which is resolution.

Select **Set Layer Model** or click the **Set Layers** button and the following window appears:




To load a layer model from another data set:

- Click **Import layers** to load an entire or part of a layer model from another data set.

To create a set of identical layers:

- Specify the layer resisitivity and thickness as well as the number of desired layers then click **Set uniform layers**. The deepest layer generated will be the basement and will have a thickness of 1e+8.

To insert one layer:

- Specify the layer resistivity and thickness. Insure the Total number of layers displayed is the same as in the list.

- Click **Insert a layer** and a new layer will be inserted at the location specified by the **Insert layer index**.

To delete a layer:

- Select the desired layer from the list and delete with the DELETE key.

To edit a value in the list:

- Double click the value then input a new value.

## Editing the Initial Model.

One means to help reduce non-uniqueness of the inversion results while enabling known geological information is to use previous to the inversion, forward model simulations. Very often, the user has geological knowledge, resistivity information and possibility other geophysical data. By incorporating such information into geophysical models and simulating the response, one can often provide preliminary models conforming to known information. This is the idea here with this aspect of the inversion. These models may be either a set of prisms or polyhedra. Thus, while you can construct an approximate model at this stage, you can import a previously derived model.

Clicking the **Use Initial Model** button will launch the window below. You may build and insert a prism model from this interface or use the Import Initial Model to insert a previous model which is in your database.

The starting model is described by a list of prisms with various properties in the box labelled **Initial Model**

**To add a prism to the model list:**

- Specify the resistivity, size, position and orientation of the new prism in the **Build a model** section.
- Click the **Add a prism** button.

**To modify an existing prism in the model list:**

- Double click the cell of the property you would like to edit in the Initial Model list.
- Input the new value and press Enter.

**To delete prisms from the model list:**

- Select the prisms to be deleted in the list.
- Click **Delete all selected prisms**

**To apply the same values for a group of selected prisms:**

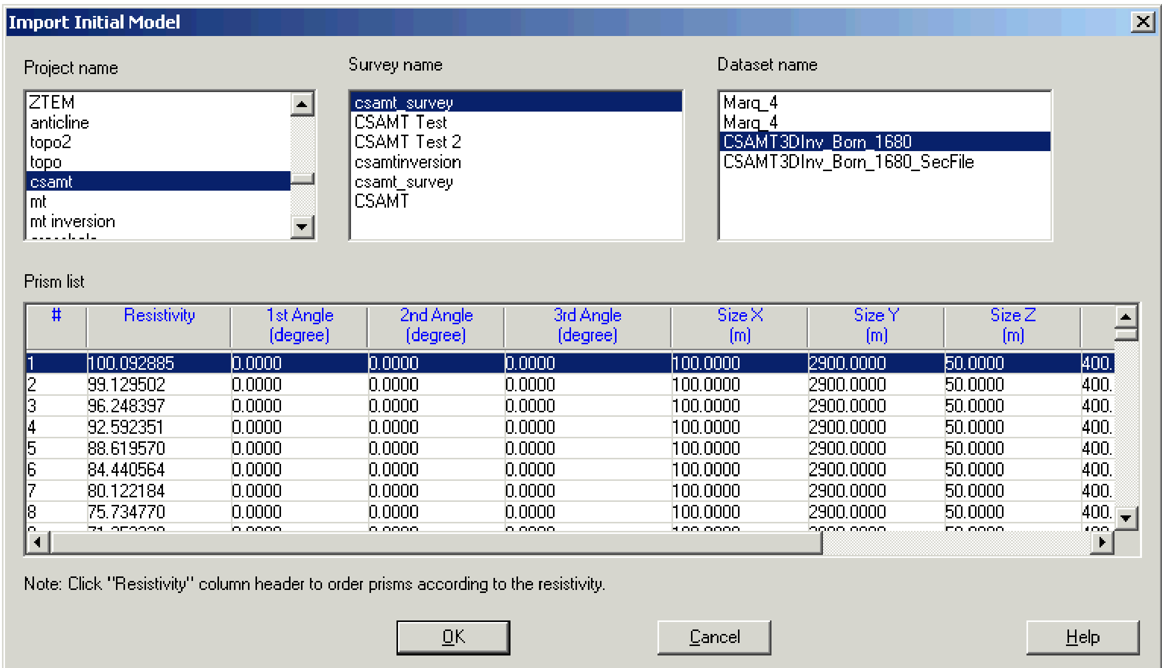
- Click the **Set resistivity to all selected prisms** button to modify the resistivity.
- Click the **Set angles to all selected prisms** button to modify the angles.
- Click the **Set size to all selected prisms** button to modify the size.

**To import a model from another data set in the current database:**

- Click Import a model.
- Select the project, survey, and data set with the desired model
- Click **OK** and the model will appear in the **Initial Model** list

# Importing a model

This functionality allows the user to import as a starting model either a forward 3D model or a previous 3D inversion.




- All eligible projects are listed. Choose the project, survey and data set that has the model you want to use. The **Prism** list can be sorted according to resistivity by clicking the **Resistivity** column header. Click **OK** to import the model.

## Editing the Grid Cell Thickness.

The inversion grid need not have a regular vertical spacing. As the inversion looks deeper it is less sensitive to small volumes and thus an increased vertical grid spacing is sometimes desirable. Clicking the **Define** button beside **Spacing in Z direction** will launch the following window:

The interface displays the total thicknesses before and after editing as well as the topmost z value. The cell sizes are listed in the **Search grid cell thickness** section.

**To modify an existing entry:**

- Specify the **Thickness** and select the entries to which you would like to assign this thickness.
- Click **Replace the selected**.

**To insert a new entry:**

- Specify the **Thickness**.
- Specify the line where the new entry should be inserted in the **Insert Index** box.
- Click the **Insert a thickness** button.

**To delete an entry:**

- Select the entries to be deleted.
- Click **Delete the selected**.

**To calculate new cell thicknesses:**

- Select the entries to be modified.
- Enter a value in the **Edit thickness** section.
- Clicking **Add(+)** will increase all the selected thicknesses by the entered value.

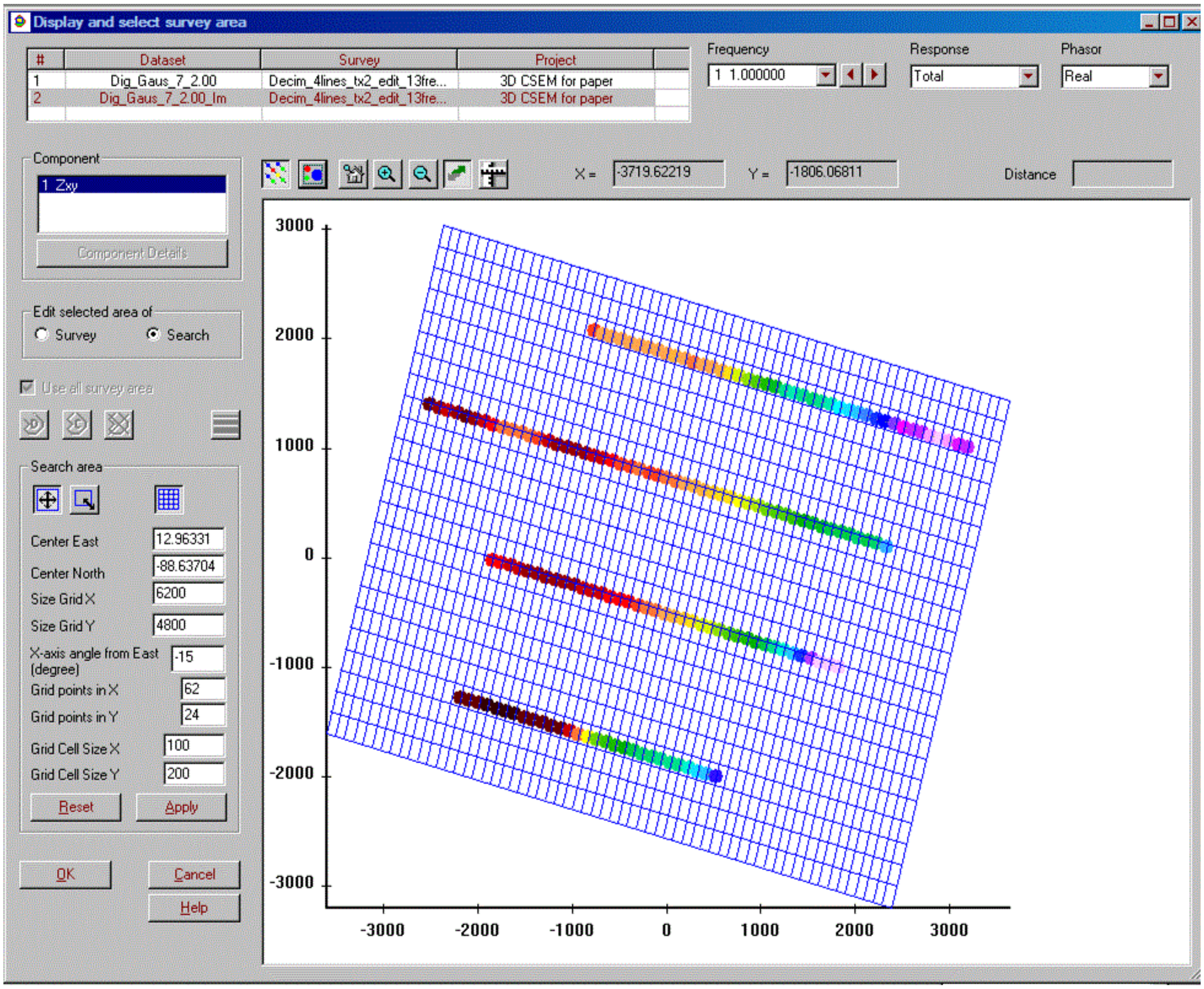
- Clicking **Multiply(x)** will multiply all the selected thicknesses by the entered value.

**To undo all changes:**

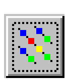
- Click the **Generate Default** button.

## Selecting Survey and Search Area

Clicking the **Select Search Area** button launches the following window:



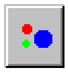

To toggle display of data values:

- Click the  button.

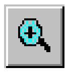
To change display of component or response:

- Make a selection from the **Component** and **Response** list.

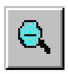
To change the size of the symbols for each data value:

- Click the  button. Use the slider on the window to appear to change the symbol size. Select **Show change immediately** to see the symbols change size as you move the slider.

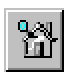
To zoom in on a section of the survey:

- Click the  button. Drag the mouse on the survey display to draw a rectangle describing area you would like to zoom in on.

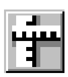
To zoom out to the previous zoom level:

- Click the  button.

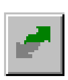
To zoom out to the original zoom level:

- Click the  button.

To find the distance between two points on the survey display:

- Click the  button. Drag the mouse between the two points on the survey display and the distance will be displayed at the top of the window in the box labelled **Distance**.

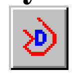
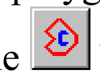
To display the survey in a proportional view:

- Click the  button.

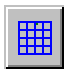
To find the coordinates of a location on the survey display:

- Move the mouse to the location and the coordinates will be displayed at the top of the window in the boxes labelled **X=** and **Y=**

To select only a section of the survey:

- Deselect the checkbox labelled **Use all survey area**. This will enable the three buttons underneath. Make sure the  button is pushed in. Click on the display of the survey at the vertices of the polygon which describes the area you would like to select. Click on the  button to close the polygon and complete your selection. Click on the  button to delete the polygon if you are unsatisfied with it.

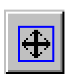
To toggle the display of the search area grid:

- Click the  button.

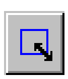
To select a search area:

- Make sure **Search** is selected in the **Edit selected area of** section. The parameters in the **Search Area** pane can be modified and the effect of any changes can be seen by clicking **Apply**.

To move the search area:

- Click the  button and drag the search area box with the mouse.

To resize the search area:

- Click the  button and resize the search area box with the mouse.

# Advanced Component Selection

**When using multiple transmitters, this functionality allows the user to make more advanced component selection** Clicking the **Advanced** button will launch the following window:

The available transmitters in the data set are listed in the box labelled **Transmitters**
The current number of selected components is displayed to the right of the **Receivers** list.

**To select a transmitter for inversion:**

- Click the square in the first column of the transmitter list and a check mark will appear

**To view the selected receivers related to a transmitter:**

- Click the transmitter in the **Transmitters** list.
- The related receivers in the **Recievers** list will be highlighted.

**To select the same receivers for each transmitter**

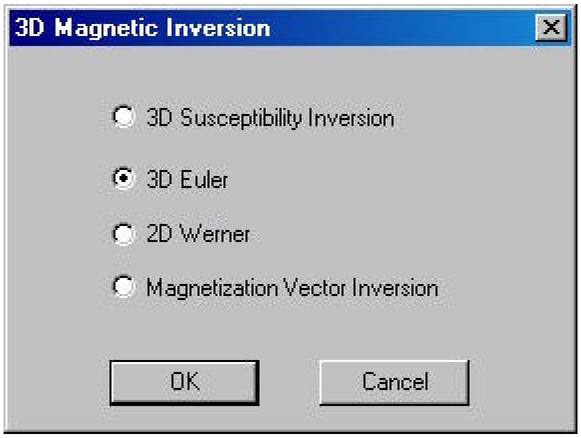
- Select the checkbox labelled **Set the selected Rx to all Tx**

# 3D Euler Deconvolution

## Selecting Inversion Type

Select the magnetic or gravity data set to subject to euler deconvolution in the **Database** dialog and click the **3D Mag/Grav Inversion** button  on the EMIGMA toolbar.

The following window will appear:

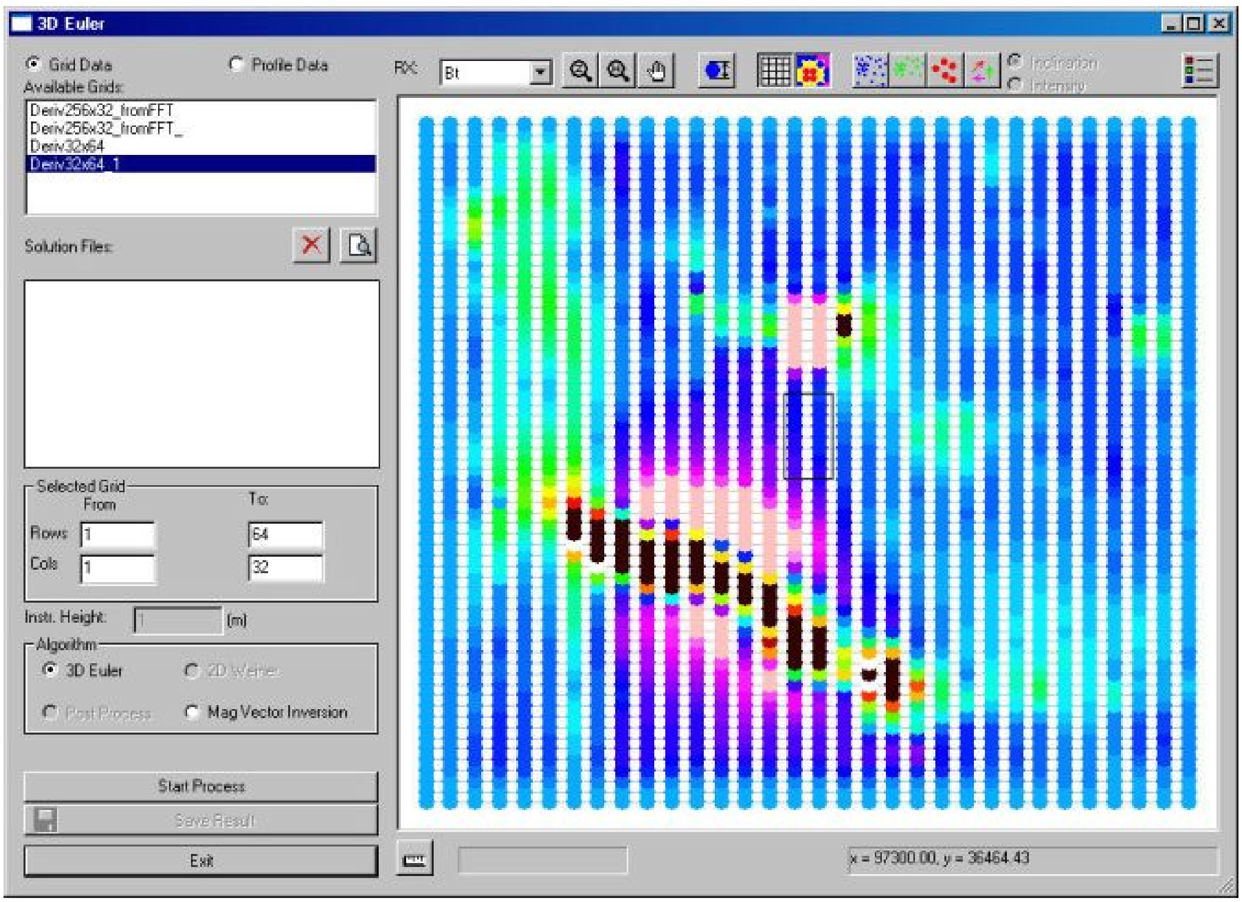



- 3D Euler - requires the selected data set to have a grid with derivatives attached. See Gridding and FFT.
- 2D Werner - for magnetic data only.
- Magnetization Vector Inversion - requires the selected data set to have an euler solution attached.

Select the desired option and click **OK**

# 3D Euler Deconvolution

The 3D Euler Deconvolution tool can be used to provide initial estimates of the location and depth of magnetic or gravity sources. Structural index and error estimates are determined as well. Once the initial result has been calculated, further post-processing may be applied. The post-processed solution can be used to calculate a vector inversion for magnetic data. These solutions may be viewed in more detail using the Visualizer and Grid Presentation tools.

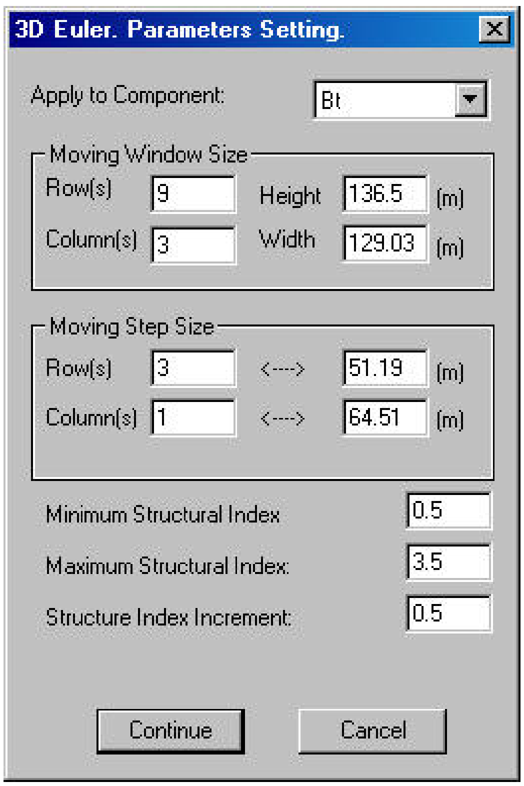



To perform Euler deconvolution:

- Select **3D Euler** in the **Algorithms** section.
- Select **Profile Data** to use your profile data
  OR
  Select **Grid Data** and select a grid data set from **Available Grids**. If

there are no grid data sets listed, you should return to the main database dialog and create a grid with derivatives for this data set. See FFT. The grid data set that you selected will appear in the large window to the right.

- You may limit the part of the grid which will be processed in the **Selected Grid** section. The default values process the entire grid.
- The height that the data was measured at can be specified in the **Instr. Height** box.
- Click **Start Process** to launch the **Parameters** dialog:




- The **Window size** specifies the section of the grid data which will be used to search for a solution. Values can be entered in either metres or grid cells. The window can be viewed graphically in the center of the grid display. In general, the window size will be approximately half of the maximum depth of the solutions.
- The **Step Size** by default is set to have the Euler processing performed at every grid point. Decrease the number resulting solutions by increasing the step size.
- The range of structural indices to search for can be specified in the boxes labelled **Minimum** and **Maximum Structural Index**. The

number of structural indices to search for can be specified by entering a value for the **Structural Index Increment**.

- When the parameters are correct, click **Continue**.
- Click **Save Result** if you wish to save the Euler solution to the database. The Euler solution must be saved in order to do any post-processing.

## 2D Werner Algorithm

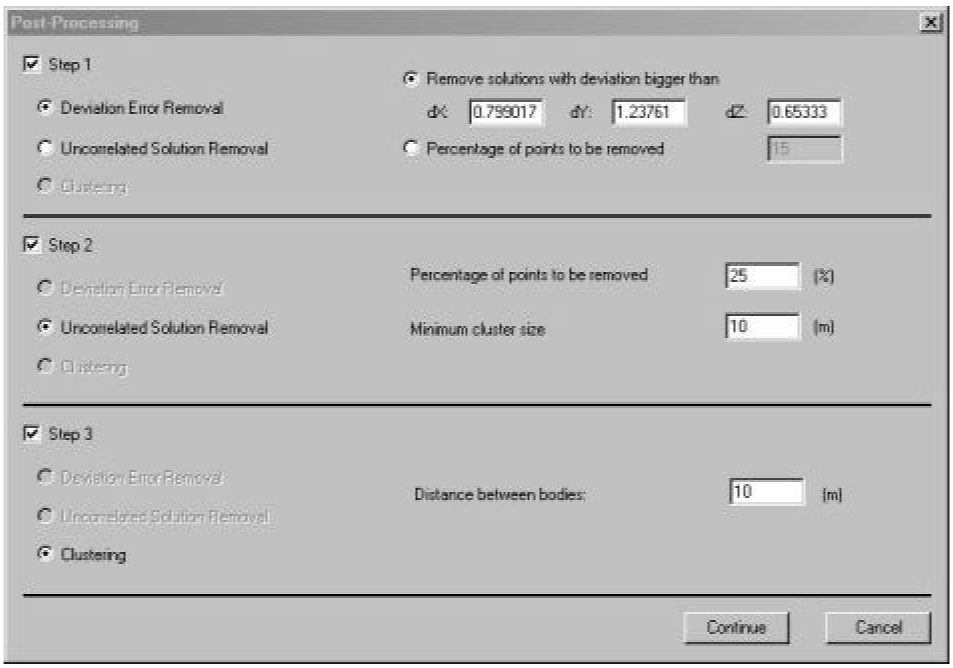
To perform the 2D Werner Algorithm on profile data:

- Select **Profile Data**.
- Select **2D Werner** from the **Algorithm box**.
- The height that the data was measured at can be specified in the **Instr. Height** box.
- Click the **Start Process** button and the **2D Werner Parameters** window appears.
- Enter the number of profile data values to use for each solution search. The value must be at least 9. Click **Continue**.
- Click the **Save Results** button to save the solution to the database and then be able to do further processing on the solution.

## Post-processing

To perform post-processing and isolate the more reliable solutions:

- Select an euler solution from the **Solution Files** list.
- Select **Post Process** from the **Algorithm** box. The **Post-Processing** window will appear:




- There are three available operations.
- It is possible to do all operations at once or one at a time. Activate the **Step** checkboxes corresponding to the number of operations you would like to do.
- **Deviation Error Removal** - Either enter a percentage of solutions to remove which have the highest deviation OR enter a deviation for which solutions will be removed if the specified deviation is exceeded.

  To create a Rodin solution:

- Select Uncorrelated Solution Removal
- Specify the percentage of solutions to be removed which are least densely distributed spatially. **Minimum cluster size** should be set to

the approximate size of a target body.

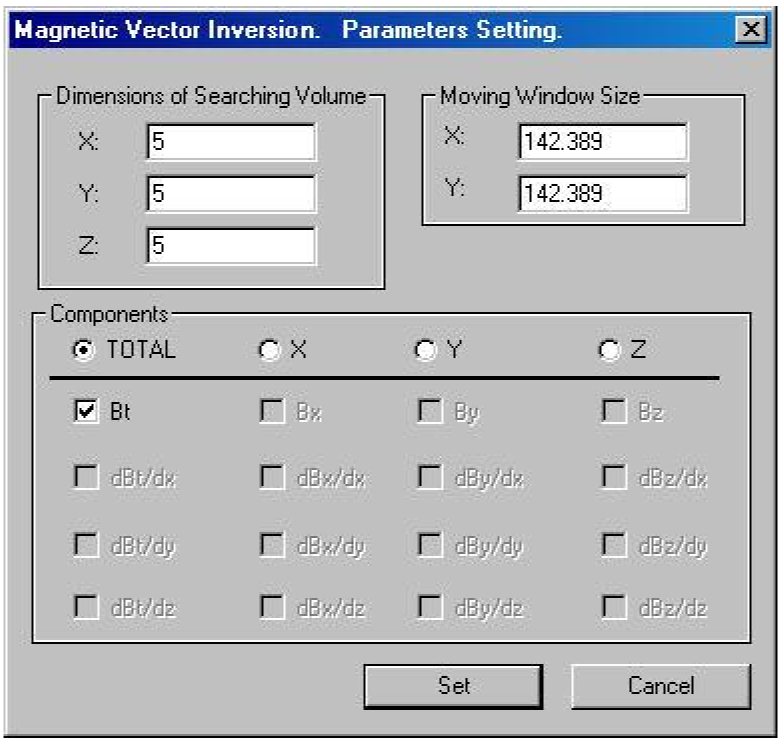
To create a final solution:

- Select a Rodin solution from the **Solution Files** list. The filename will have a green checkmark beside it. Select **Post Process**, activate the **Step**checkbox then select **Clustering**. Specify the **Distance between bodies**.
- Click **Continue** to perform the post-processing.

## 3D Mag Vector Inversion

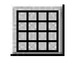
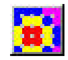
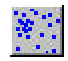
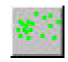
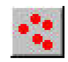
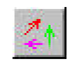
To perform vector inversion:

- Select a final euler solution from the **Solution Files** list. The filename will have a red checkmark beside it. See Post-Processing to learn how to generate this file.
- Select **Mag Vector Inversion** from the **Algorithm** box.
- Click **Start Process**. The **Parameters** window will appear:

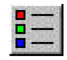


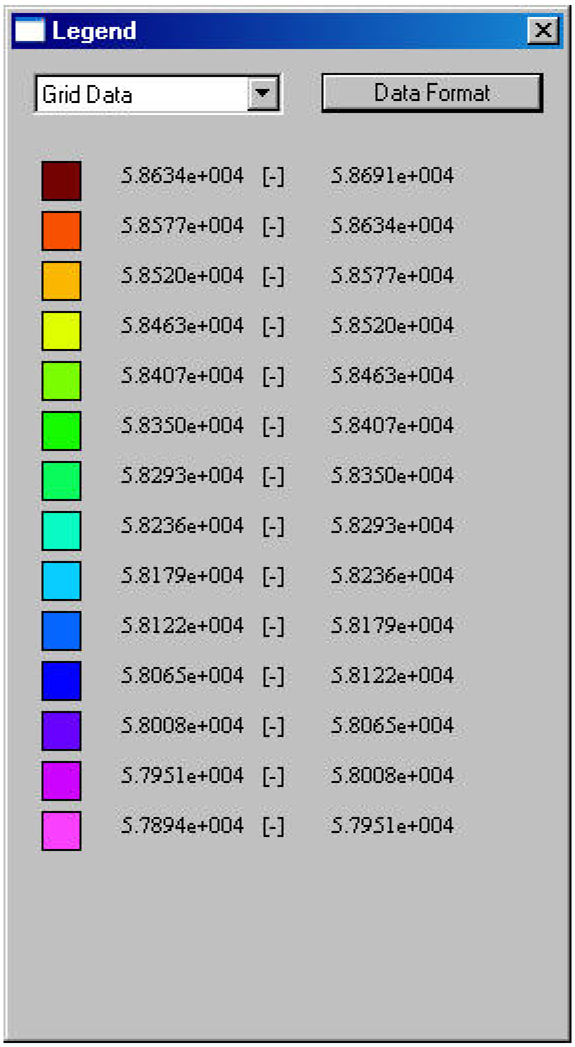

- **Moving Window Size** specifies the section of grid data surrounding each solution that will be used in the processing.
- **Dimensions of searching volume** should be set to a value inversely proportional to the certainty that the euler solutions are correct.
- Select the **Components** for which processing will be performed.
- Click **Save Result** to save the inversion to the database.

## Displaying Data

- Select the data for a different component using the **RX** combo box.
- Toggle the display of the Grid/Profile lines using the  button.
- Toggle the display of the Grid/Profile data using the  button.
- Toggle the display of the Euler solution using the  button.
- Toggle the display of the filtered solution using the  button.
- Toggle the display of the final solution using the  button.
- Toggle the display of the vector inversion results using the  button. The colours of the vector arrows can indicate the **Inclination** or **Intensity** depending on which has been selected.

## Displaying a Legend

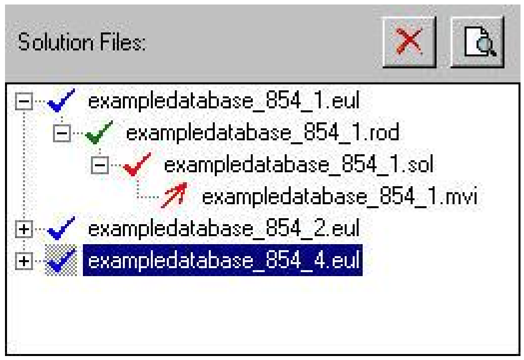
- Toggle the display of the data legend using the  button. This button will be available for grid/profile data and mag vector inversion data. The legend dialog will be launched:




- If both grid/profile and vector inversion data are being displayed, the combo box located at the top of the box can be used to choose which data the legend applies to.
- Change the format of the legend data values by clicking the **Data Format** button. The number decimal digits to display can be set here as well as whether the values will be in fixed or scientific notation.

## The Solution File List

Solutions which have been saved to the database are listed here.




Euler solutions are indicated by a blue checkmark.
Filtered solutions are indicated by a green checkmark.
Final solutions are indicated by a red checkmark.
Mag vector inversions are indicated by a red arrow.
A plus sign indicates there are other solutions hidden from view. Click on the plus sign beside the file to access the other solutions.

To view a solution file:

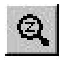
- Select the file to be viewed. Click the button or right click on the file and select **View**. The data contained in the file appears in its own window.
- The file can be exported to an xyz or qct file by clicking the appropriate **Export** button or right clicking on the item in the solution list.

To delete a solution file:

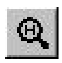
- Select the file to be deleted. Click the button or right click on the file and select **Delete**.

## Viewing Data in More Detail

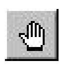
To zoom in on a section of the data:

- Click on the  button.
- Drag the mouse while holding the left mouse button to select the area you would like to zoom in on.

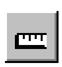
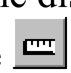
To return to the original zoom level:

- Click on the  button.

To move to a new area of the data:

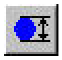
- Click on the  button.
- Drag the mouse while holding the left mouse button in the direction you would like to move the data.

To measure a distance on the grid:

- Click on the  button at the bottom of the window.
- Click on the first location and drag the mouse while holding the left mouse button to the second location. The distance between the two points will appear in the area beside the  button.

## Changing Data Symbol Size

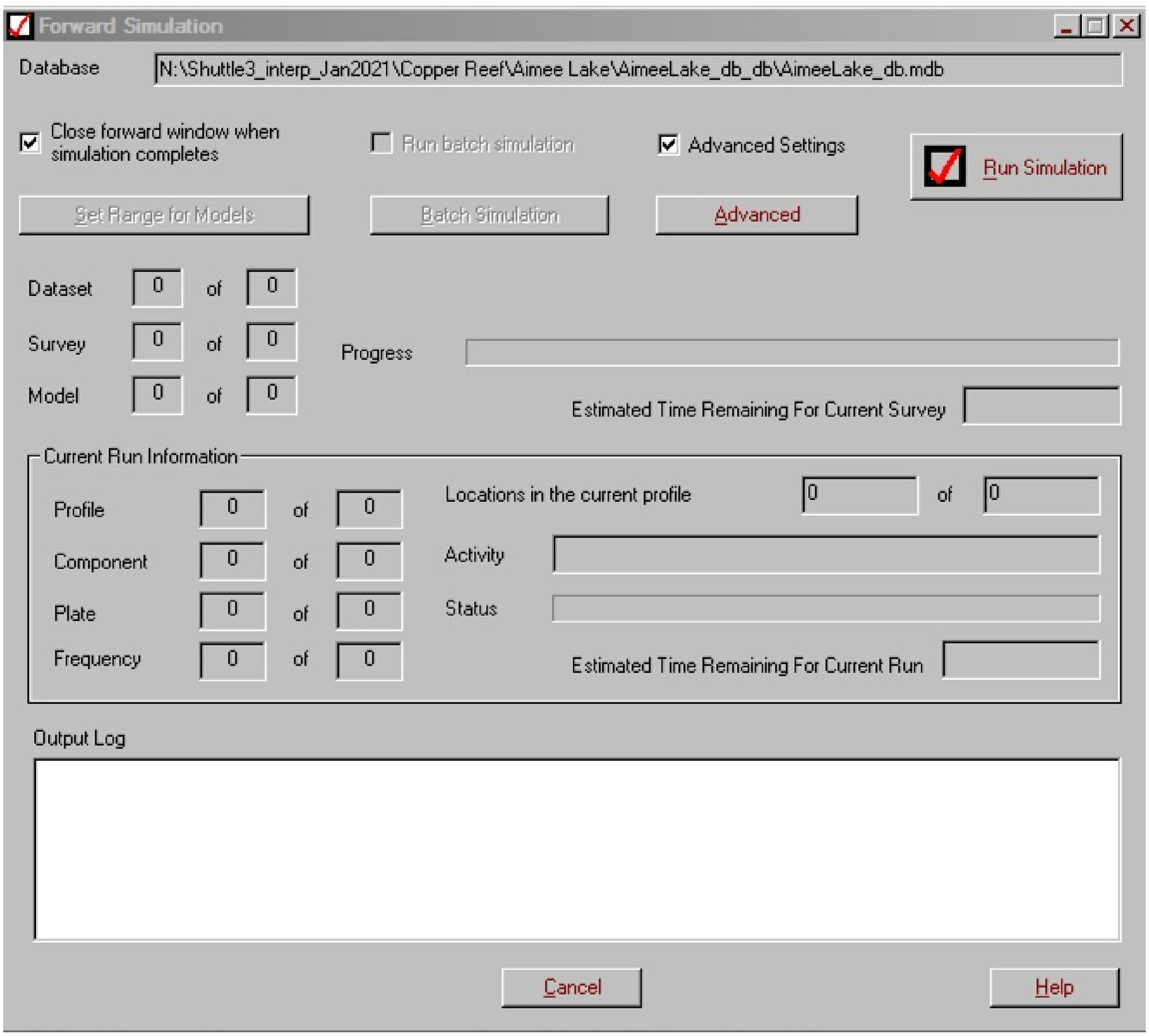
To change the size of the data symbols:

- Click the  button and the **Set Symbol Size** dialog appears.
- Move the appropriate slider to change the data symbol sizes and click **OK**.

# Forward Simulation

Select the data set to subject to simulation in the **Database** dialog and click the **Forward Simulation** button  on the EMIGMA toolbar. A message will ask you whether you want to overwrite the selected data set. Click **Yes** to overwrite it and **No** to create a new data set.

The **Forward Simulation** dialog will appear, with the upper field showing the path to your database:



You can run simulation as is or specify one of the four available modes (see **Related Topics** below). During simulation, which may take a long time, the **Forward Simulation** dialog will keep you updated on the progress, providing information on the status and estimated simulation time of a current survey on the whole and each current run in particular.

***Note***. *Y*ou can minimize the **Forward Simulation** dialog to do some other things maximizing it from time to time to check on the progress. The **Cancel** button stops the operation and closes the **Forward Simulation** dialog upon completion of a current run.

**Related Topics**

Forward Simulation As Is

Model Suite Generation - Setting Ranges for Models

Location Load Mode

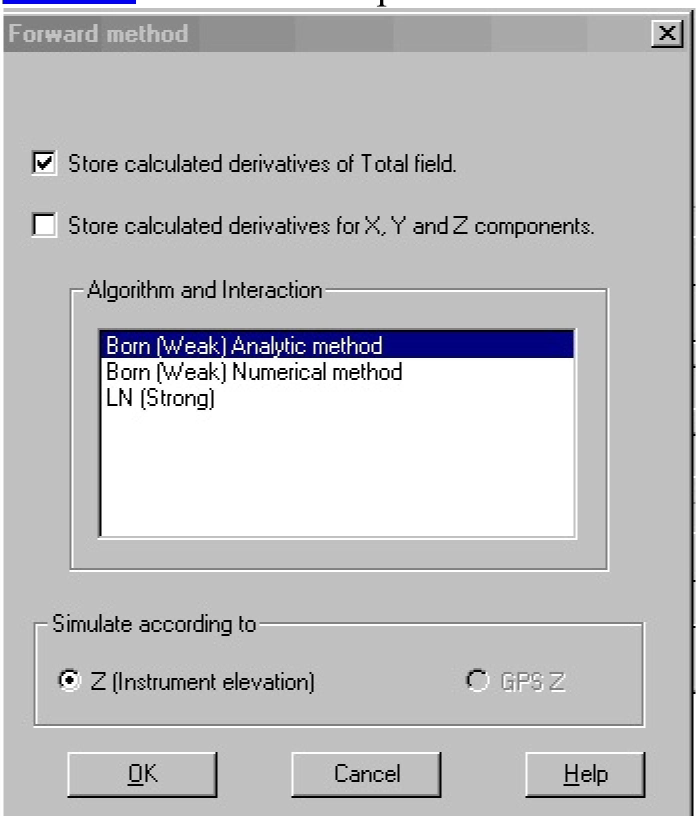
Batch Simulation

Green's Function

Advanced Settings

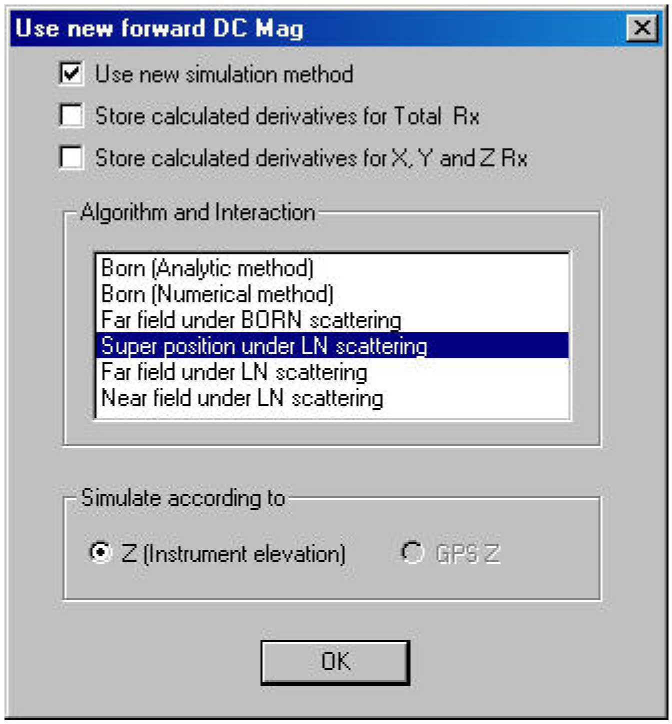
# Forward Simulation As Is

In the **Forward Simulation** window, click **Run Simulation** to perform simulation as is.

1. With FEM data, simulation will start right away.
2. In the case of magnetic, gravity or resistivity systems, a **Forward Method** window will open.
   
   
   This allows for specifying
    - Whether and what type of derivatives will be calculated
    - Which Algorithm will be utilized
    - Whether the elevation of the model is with respect to GPSZ (available if GPSZ of the stations exists or ground elevation
    - The type of interactions required
3. In the case of TEM, the **Run Simulation** button may bring up the **Spectral Sequence Configuration** window.

**Use Forward Window**

In the case of magnetic, gravity or resistivity system, the **Use new forward** window will open offering you to select the scattering algorithm and the type of interaction to be used in simulation:




- Check the **Use new simulation method** box in the upper part of the dialog to enable the list of methods below and select a method to apply. This option is available only for magnetic or resistivity data
- Check **Store calculated derivatives** to save the derivatives in the same data set. This option becomes active in case you are using a new simulation method.
- If the data set contains a GPS Z channel, these values can be used for the simulation by choosing the **Simulate according to GPS Z** option.
- Click **OK** to start simulation.

**Definitions:**

**Analytic** - the algorithm is closed form and can be computed analytically

**Numerical** - standard numerical integration over each model object

**Born** - weak scattering, internal field parallel to source field

**LN** - calculates full scattering

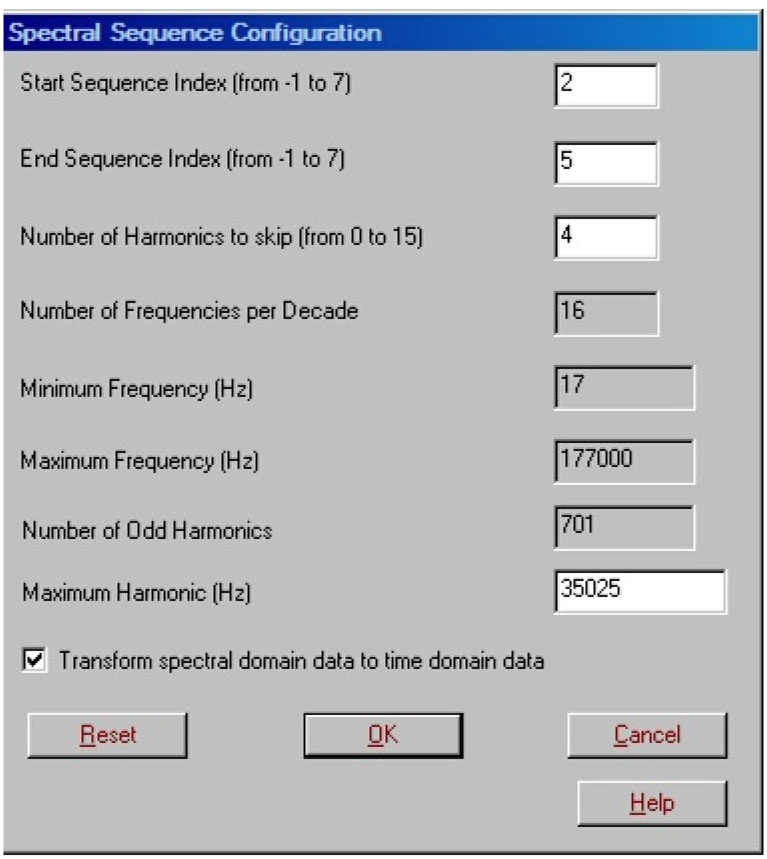
**Far Field** - calculates interaction but computing the effect of the field from one object on all others

**Near Field** - theoretical technique derived via LN approximation

**Spectral Sequence Configuration**

EMIGMA TEM simulations attempt to reproduce the instrument's proper system response. As the current is controlled in a repeating waveform and the data stacked over many periods of this waveform, the frequency to time transform is a Fourier transform. As such, all odd harmonics of the base frequency are required. In order to do this efficiently, a spectral sequence is generated.

In the case of TEM, the **Run Simulation** button will bring up the **Spectral Sequence configuration** window. If you wish to reselect your spectral settings click **Advanced Settings** and then **Display frequency to time domain transform settings**.




- Check or change, if necessary, the starting and ending frequency sequence indices and the number of harmonics to skip in the respective boxes.

- The values in the **Number of Frequencies per Decade** to be computed, and **Minimum Frequency** and **Maximum Frequency** to be computed boxes will change accordingly.
- Click the **Reset** button to retrieve the original values.
- Check the **Transform Spectral Domain Data to Time Domain Data** box to run this operation automatically otherwise a spectral dataset will be saved which can be transformed manually.

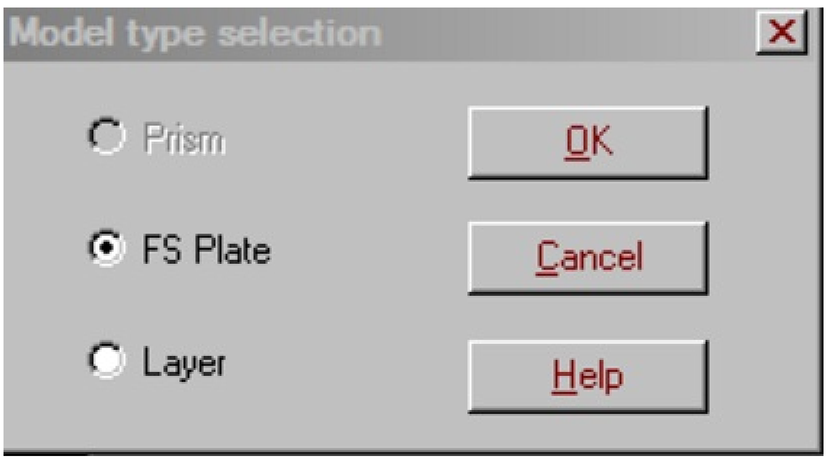
  This option is useful when you have modified an already available time-domain model and want to subject it to another forward simulation. In this case, you will not need to run the standalone transform to convert your spectral data into time-domain; this option will provide this operation automatically

- Click **OK** to apply changes begin the computation. Click **Cancel** to cancel the changes, if you have made any, and close the dialog.

## Model Suite Generation

Click **Set Range for Models** if you want to subdivide your model into parts differing from each other in a certain property, such as thickness, resistivity or susceptibility. The objective of this operation is to create a number of “models” to compare your measured data sets with and to select the one that complies best with your survey results

The **Model Type Selection** dialog appears offering you to select between **Plate**, **Prism** and **Layer.** As of now, only the **Plate** and **Layer** options are accessible, with the **Layer** option selected by default:



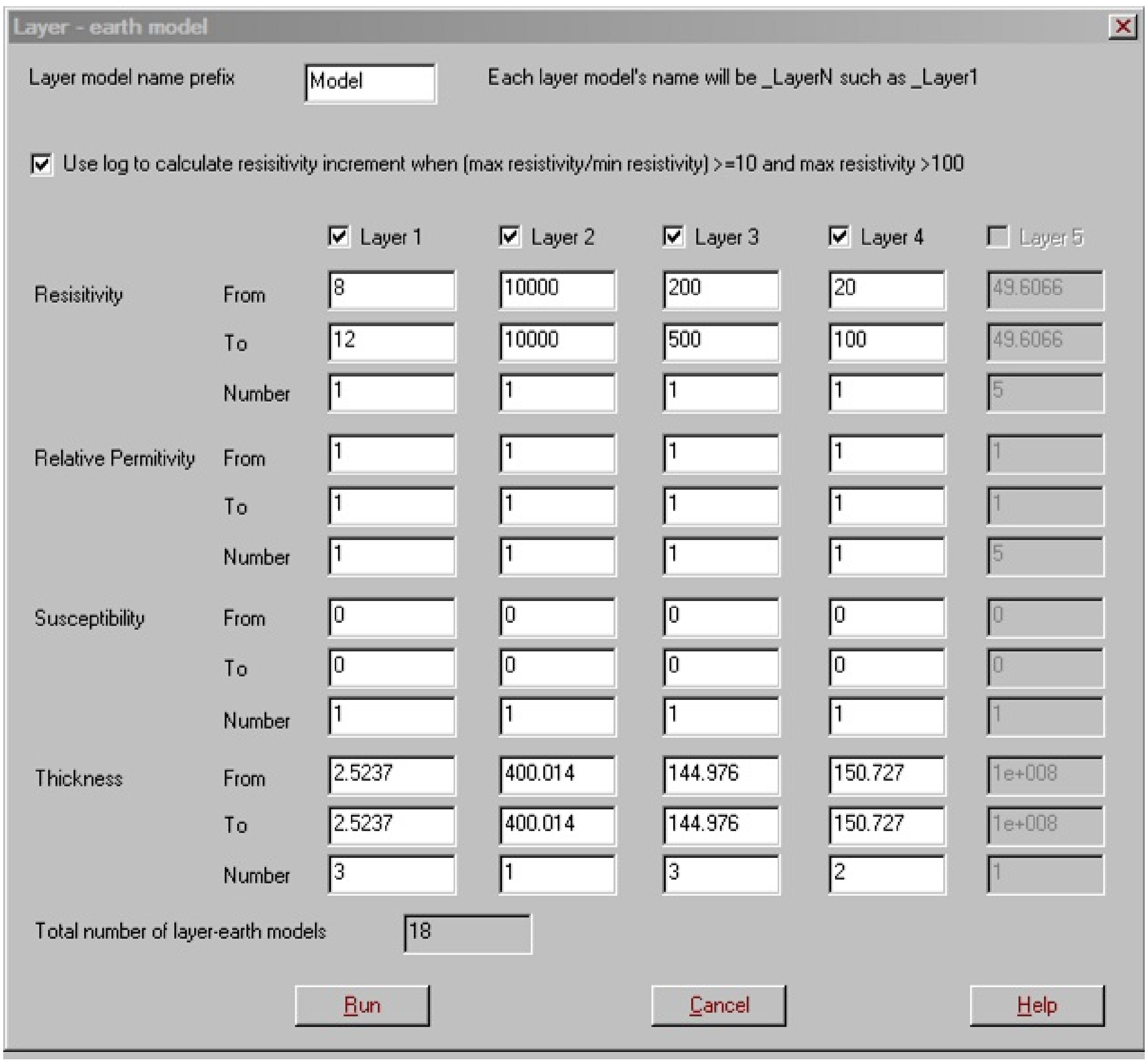

**See**

[Specify Ranges of a Layer-Earth Model]

[Specify Ranges of a Plate Model]

## Specify ranges of a layer-earth model

- This tool allows one to build a set of layered models which are variations of a useful model and then this computes in the background the response for all the models and stores each one individually in the database for later examination
- From the main Forward Simulation window, select **Set Range For Models** and then in the **Model Type Selection** window, leave the **Layer** option on and click **OK**. The **Layer - earth model** window will open:



- In the **Layer Model Name Prefix** field, type in the name of your model so that the resulting data sets ("models") could have the same name prefix followed by the underscore and a subsequent number (e.g. IP_Layer1, IP_Layer2 and so on)
- If max/min resistivity is equal to or more than 10 and maximum resistivity is more than 100, leave the box below the model name prefix checked to use logarithmic function and calculate resistivity increment
- Check the layer you want to subdivide into "models". The column below will become active
- Change any property range in the **From** and **To** boxes across **Resistivity**, **Susceptibility**, **Relative Permittivity** and **Thickness** and specify the number of "models" you want to simulate in the respective boxes

  You can select several properties or several layers at a time. In this case, the number of models to be simulated will be a multiplication product of all **Numbers**. For example, if, in the Layer 1 column, you have 3 in the **Number** field of one property and 2 in the **Number** field of another, the total number of the Layer 1 models will constitute 6. If you add 2 in the **Number** field of any property in the Layer 2 column, the number of models to be simulated will total 12. You can view the total number of layer-earth models in the respective box in the bottom part of the window

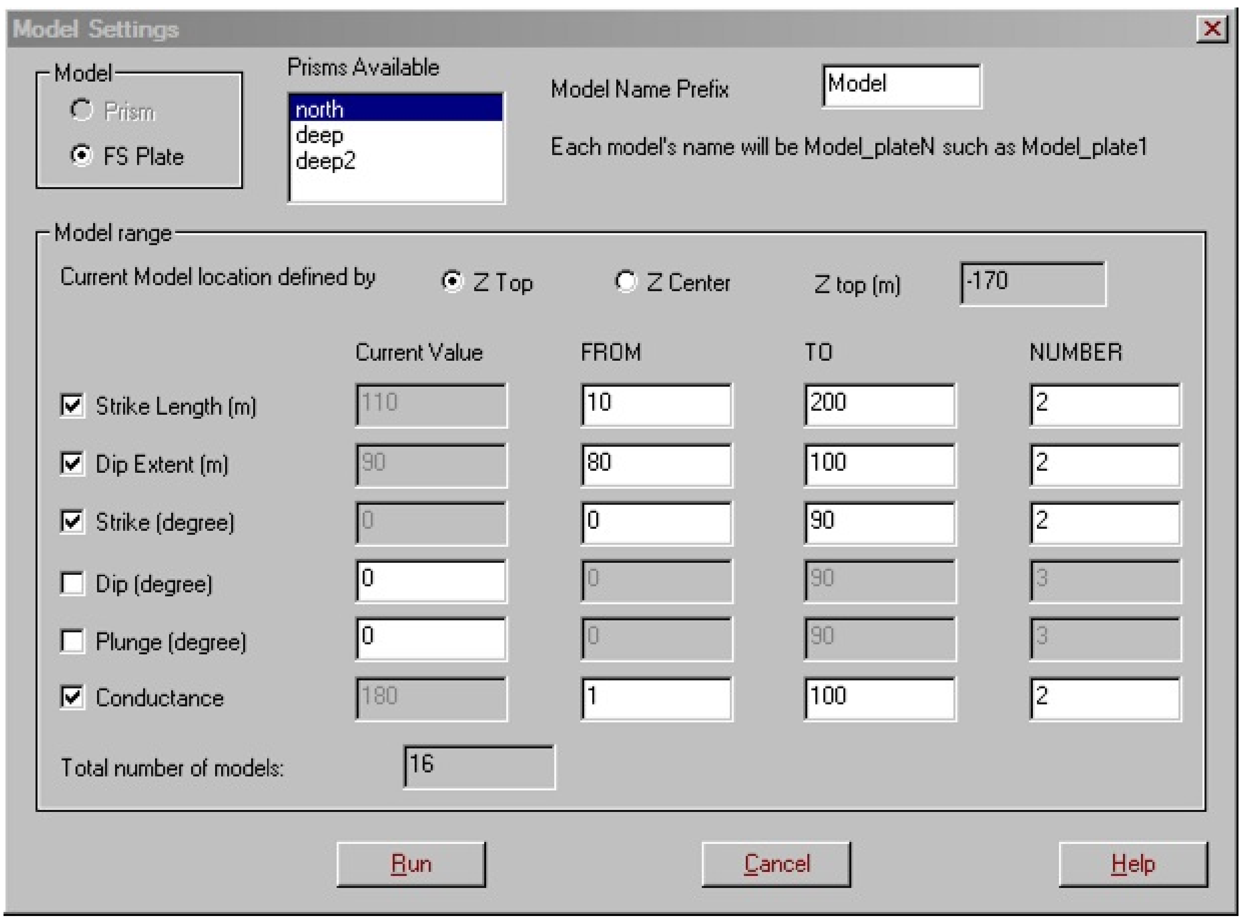
- All parameters specified, click **Run** to start simulation

# Specify the ranges of a plate model

The purpose of this tool is to setup a suite of plate models which will be run in the background and stored individually to the database for later examination. This tool is reached by selecting a dataset containing at least one plate model, then through the Forward Simulation window and then via **Set Range for Models**.

- Select the **Plate** option in the **Model Type Selection** window and click **OK**

The **Model Settings** dialog will open:



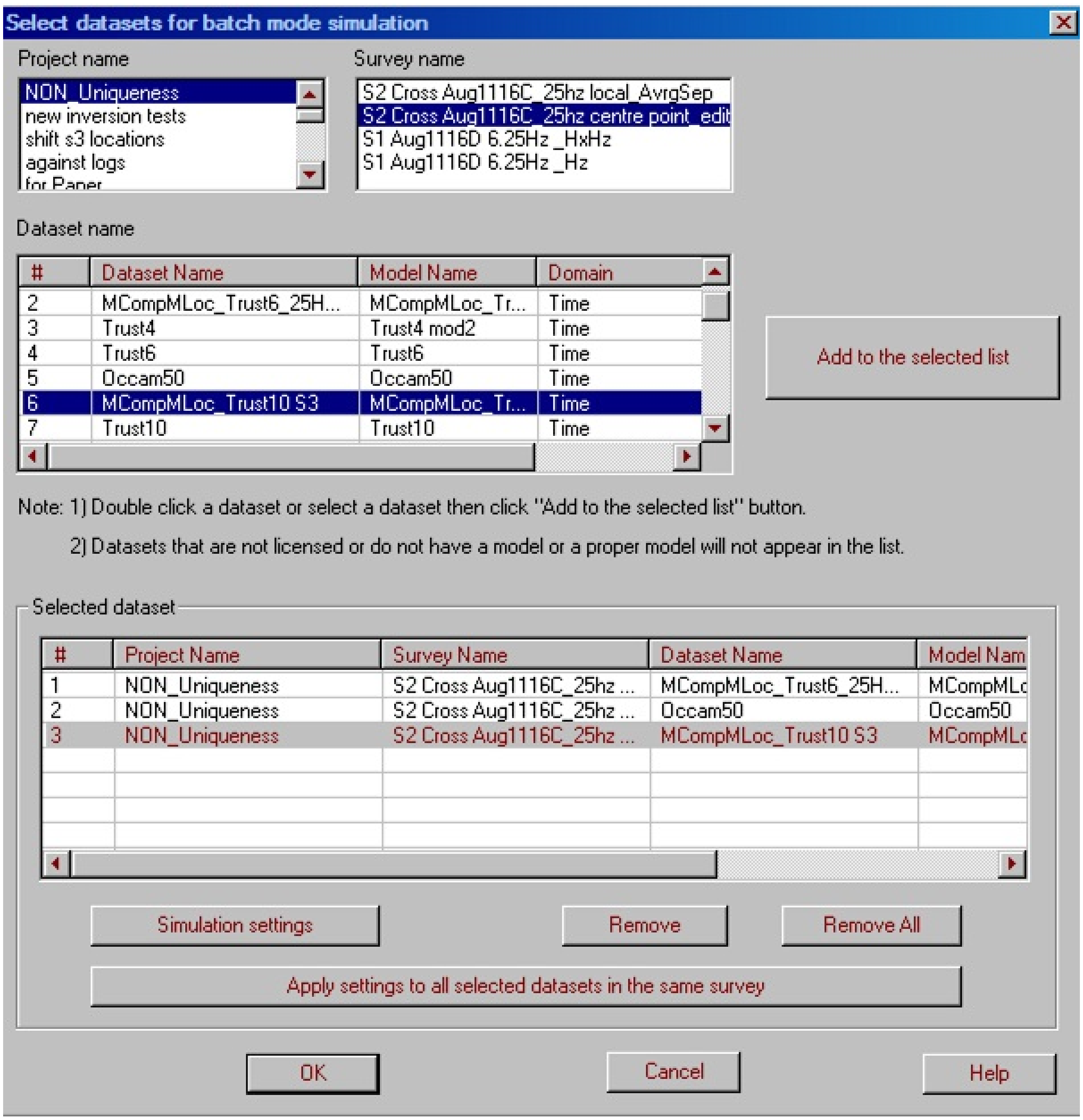

- Select the model to specify parameters from the **Prisms Available** field

- Type in the model name prefix in the respective box on the right. The resulting data sets ("models") will have the same name followed by the underscore and a subsequent number as shown in the example (Model_plate1)
- In the **Model Range** section, select between the center point and top center modes of the plate position in space
- Check the property you want to specify the ranges for. The **From**, **To** and **Number** fields will become active
- In these fields, specify the ranges and the number of models and click **Run**
- Note: the plate algorithm in the original dataset will be utilized for the plate suite

## Batch Simulation

The user may build models without computing the response. If the user constructs a set of models then these models may be run in the background in batch mode.

In the **Forward Simulation** window, check the **Run Batch Simulation** box. The **Select Data Set for Batch Mode Simulation** window will open:



In the **Project Name** and **Survey Name** lists, you will see selected the names of the current project and survey.

In the **Dataset Name** table below:

- Select the data set you want to subject to simulation and click **Add to the Selected List** button

  OR

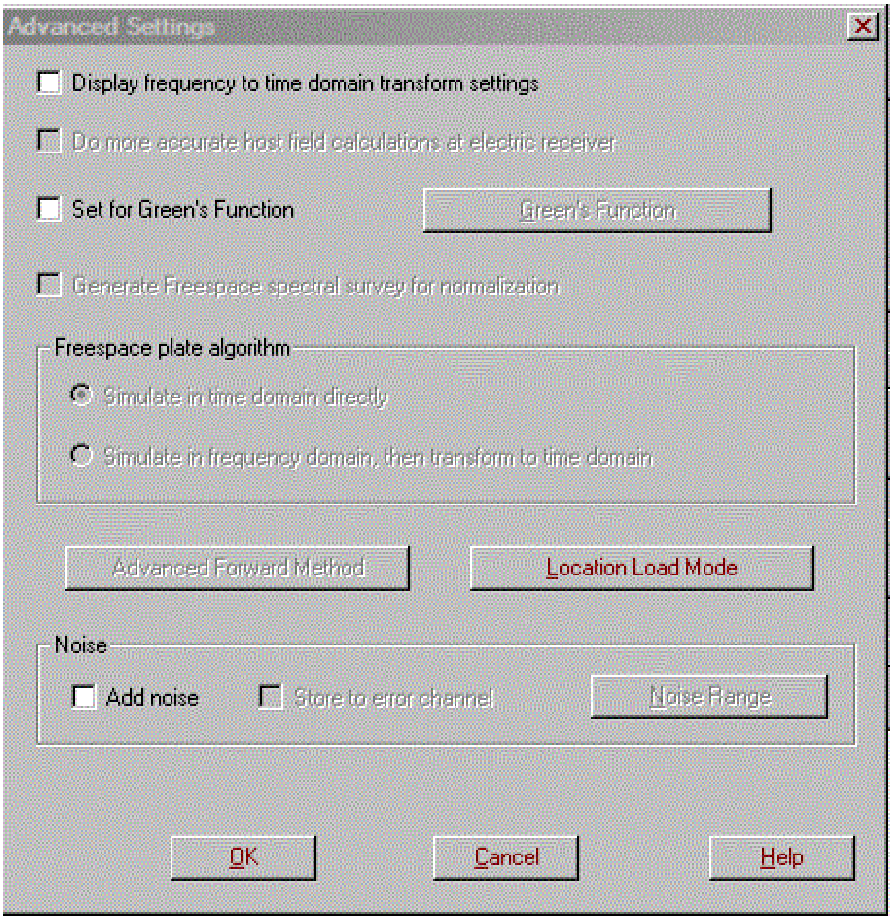
- Double-click on the required data set

  The data set will appear in the **Selected Dataset** table

- To remove a data set from this table, select it and click **Remove**; to remove all data sets from this table, click **Remove All**
- If your data are time domain, gravity, magnetic or resistivity, the **Simulation Settings** button becomes active. In the case of TEM it opens the **Spectral Waveform Configuration** window in which you can specify the frequency sequence indices, number of harmonics to skip, spectral-to-time-domain data transform. In the case of gravity, magnetics, resistivity, it opens the **Use New Forward**… window in which you can specify the method of simulation. For details, see Forward Simulation As Is
- Click **OK** in the **Select Data Set for Batch Mode Simulation** window to return to the **Forward Simulation** window and start simulation.

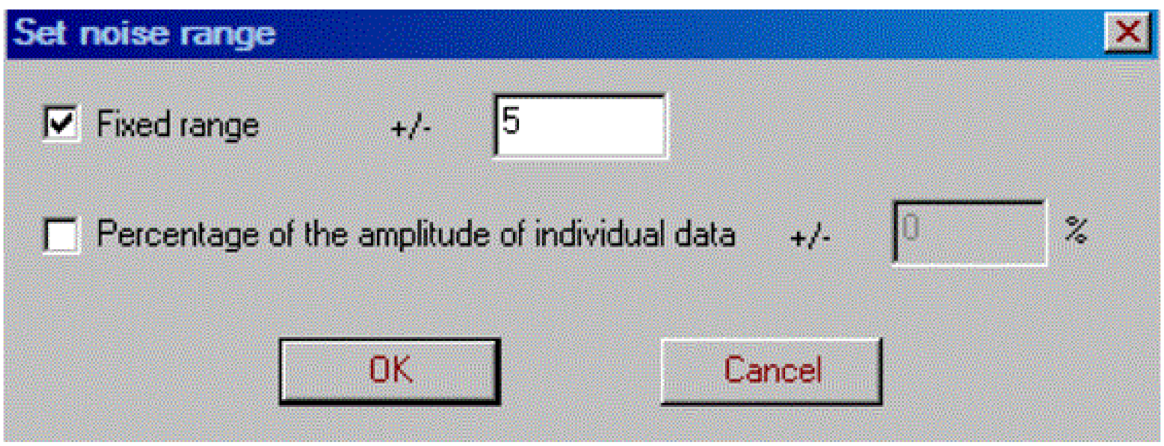
## Advanced Settings

In the [Forward Simulation]() window, click the **Advanced Settings** checkbox to access the **Advanced Settings** window:




- If you are simulating a time domain survey, the **Display the transform settings** option will be available. Uncheck the box if you do not want to specify settings for the transform tool. The transform settings window will not appear when simulation begins with this option off.
- If the survey is a simulated dataset and the first dataset in the survey and you require to normalize the data to freespace then you might need to compute a freespace spectral survey. This is the case, normally when the data consists of one component but normalization is to all 3 freespace components. Check the box labelled **Generate Freespace spectral survey for normalization** if normalization is desired.

- The option labelled **Do more accurate host field calculations at electric receiver** will be available for surveys with electric receivers. Check this option to increase the accuracy of the calculations. This option is off by default since it will increase the amount of time for the simulation and is usually not needed.
- For the freespace plate algorithm, there are two modes to simulate. The first and most accurate is to use **Simulate in the frequency domain, then transform to time domain**. This is because the correct bandwidth is utilized with appropriate low pass filter. If **Simulate in the time domain directly**, then this is similar to other plate modeling algorithms and assumes an infinite power spectrum.
- Select the Location Load Mode option by clicking its button.
- Access the Green's Function settings by checking the box labelled **Set for Green's Function**.
- To add noise to your data, click the box labelled **Add Noise** and the **Set Noise Range** window appears:




- The noise introduced to the data can be either a **Fixed range** or a **Percentage of the amplitude of individual data**. Check one or both of the available options and enter the value in the box labelled **+/-**. Selecting both options adds the two noise values together.

Click **OK** to save your settings and return to the **Forward Simulation** window.

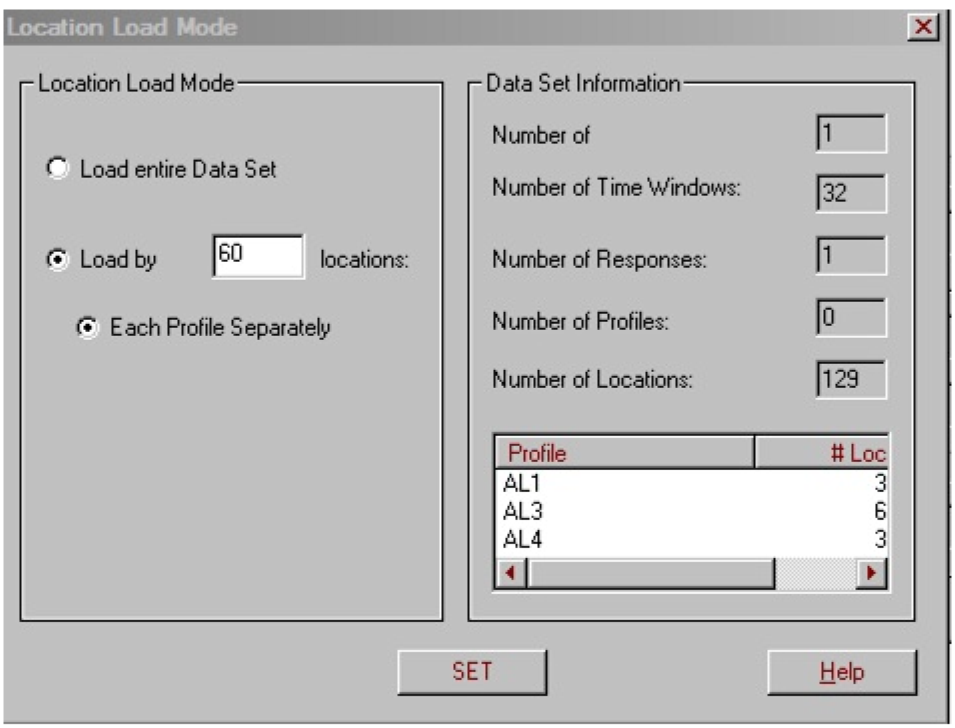
Click **Run Simulation** to launch forward simulation.

### Location Load Mode

This option may be required when computing a very complex model and utilizing some algorithms which require a great deal of memory. EMIGMA generally computes a response by loading only a subset of the data, saving the response for that subset and then loading the next subset and then so on. But, sometimes the subset is too large and has to be decreased

In the **Forward Simulation** window, click the **Advanced Settings** checkbox. Next, click the **Location Load Mode** button in the **Advanced Settings** window to specify the order for the algorithm to pick up locations during simulation. This option is not needed if you are applying a new simulation method to your magnetic or resistivity data as well as in the case of gravity systems.

The **Location Load Mode** window opens:




In the right-hand section, you can see the general information on your data set, the list of profiles and the number of locations per profile.

In the left-hand section:

- Select **Load Entire Data Set** if your data set contains not too many locations and you want to subject them to simulation all at once
- Select the **Load by** option if your data set is too big. Click **SET** to save your settings, click OK in the **Advanced Settings** window and return to the **Forward Simulation** window.

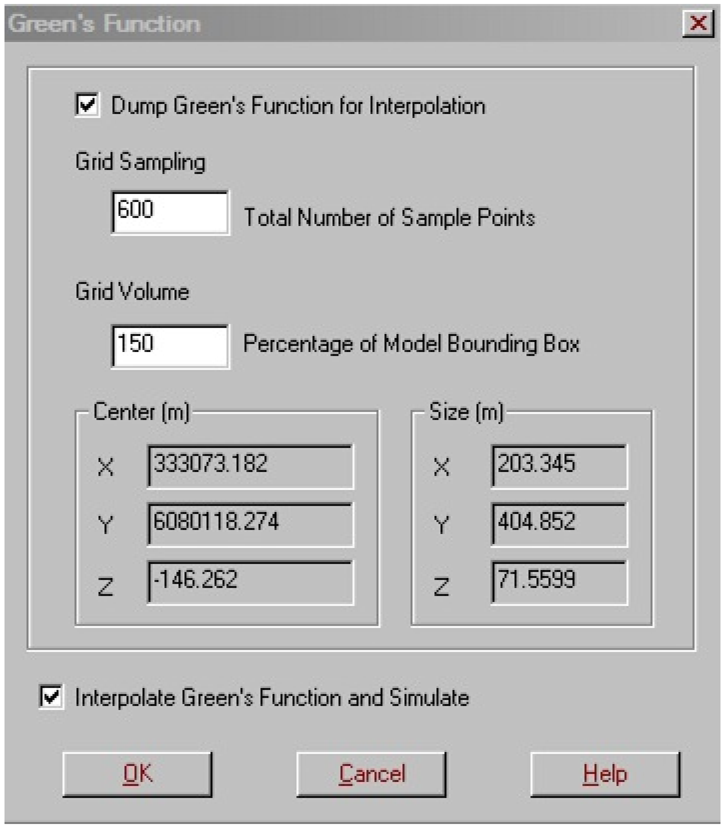
  Click **Run Simulation** to launch forward simulation.

## Green's function

Green's function calculations are an integral part of the forward simulation process. The respective calculation is run automatically as a routine program based on certain default settings. However, in some cases, it may be useful to save the result of this calculation for future simulations, especially if your models represent “slight” variations of each other differing, for example, in the dip of structures or their electrical properties. It is also necessary to use the Green's function for non-level transmitters.

The **Set for Green's Function** option will be accessible for prism or poly models with either the LN or ILN scattering algorithm. In layered earth models, i.e. in the absence of scatterers, it will be inapplicable. It also does not apply for static systems.

- Check the **Advanced Settings** box in the **Forward Simulation** window. Next, check the **Set for Green's Function** box in the **Advanced Settings** window. This will bring up the **Green's Function** window:




- Click **OK** to launch simulation using the settings offered by default and displayed in the **Dump Green's Function for Interpolation** section.

  Or, you can replace the default values in the **Grid Sampling** and **Grid Volume** fields:

  **Grid Sampling** allows up to 1000 points within the **Grid Volume**, a model region, which may be any multiple of the bounding box around the model, e.g. 150%. The larger the model region, the more points will be required to maintain a fixed accuracy and the more complex (and time-consuming) the resulting interpolation will be.

- Selecting only **Dump Green's Function for Interpolation** will just compute and save the Green's function. To use the computed Green's function to calculate the response, **Interpolate Green's Function** must be selected.

- The files are stored in the database directory in Simul--> Green, in separate folders for each survey. Only one set of .dmp files can be saved for a given survey. Any further Green's function calculations for the survey will overwrite these files.

- Once the Green's function has been saved for a survey, it may be used in subsequent forward simulations for models with the same layered model and within the same grid volume. If the stored .dmp files are not appropriate for the model, the simulation will not run.

- To use a previously calculated Green's function for simulation, select only **Interpolate Green's Function** in the **Green's Function** window. Using a stored Green's function will increase the speed of the calculation. This is useful for running a series of models with similar dimensions but different conductances, for example, as it decreases the computation time.

# FSEMTRS – Frequency to Time Domain Transform

In EMIGMA, there are two ways to carry out frequency to time-domain transform. The first is to do it during the forward simulation procedure. It applies when you have changed an already available time-domain model or modeled an actual field dataset and now are going to subject it to forward simulation. In this case, the simulation procedure, which is launched from the **Forward Simulation** window, will be preceded by the **Spectral Waveform configuration** window offering you to perform the fast transform and adjust the basic transform settings (for more details, see **Forward Simulation As Is**).

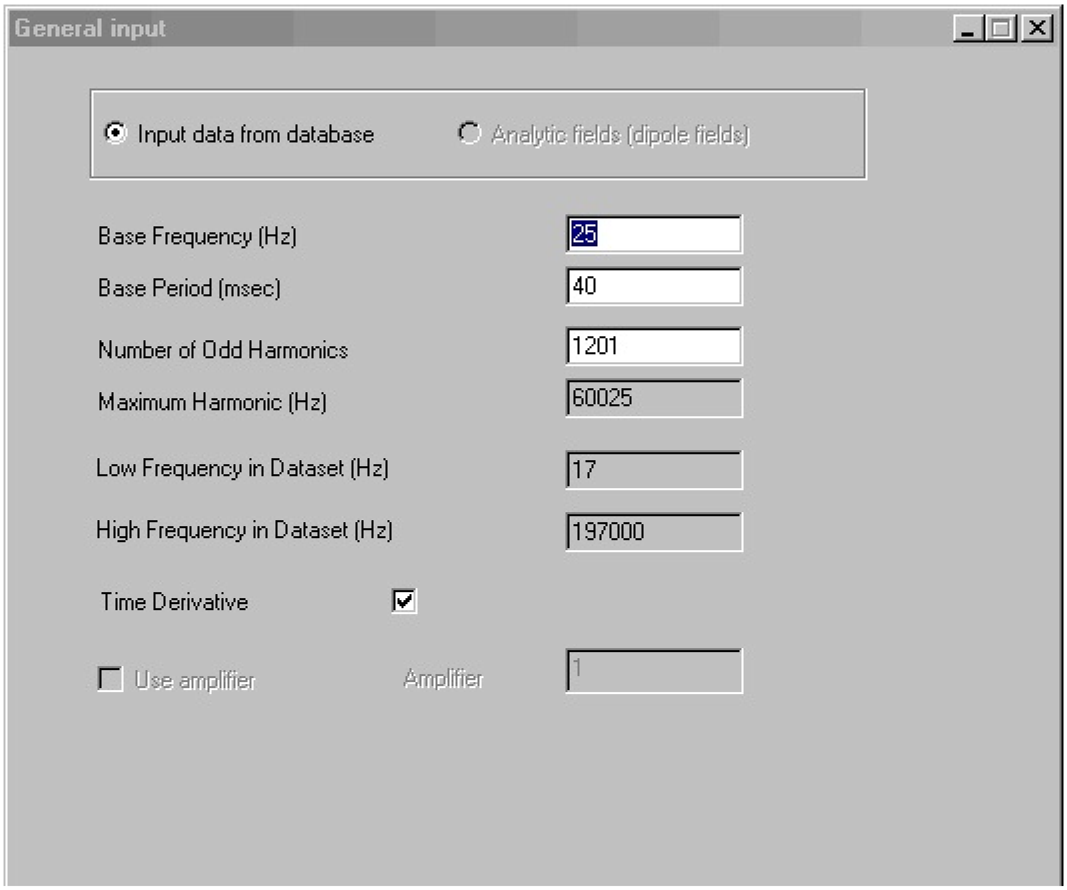
The second way is to use the standalone frequency to time domain transform. It applies when you create your survey from scratch or refuse the fast transform during the forward simulation of your new time-domain model and obtain a spectral data set as a result. You can now subject this data set to the standalone transform which allows adjustment of all possible transform settings throughout a number of dialogs to appear.

In the main **Database** window, select a spectral data set to transform and click **FSEMTRS** button ωT on the EMIGMA toolbar. If your dataset is inherited from a previous model and already contains a time-domain data set, the following message will appear:


Click **OK** to proceed to the first page of the FSEMTRS wizard and you may adjust your settings. Otherwise, you will find default settings have been made for you./span>

## General Input

In the **General Input** page:




The **Input Data from database** button will be turned on, since your data values are stored in EMIGMA's database

- Edit base frequency or base period in the respective boxes; if your time-domain data values are imported, these two settings will be detected and displayed automatically. If you change the value of one of the settings and click in the box of the other, the latter will update accordingly as well as the value of maximum harmonic in the box below

- Adjust the **Number of Odd Harmonics** value if required. The number 4096 displayed as a reasonable default is not a maximum. If you change it, click in the **Maximum Harmonic** box to update the value in it

- Low Frequency and High Frequency indicate the spectral limits of your data set

- De-select, if necessary, the **Time Derivative** box which is always checked except for the cases when your survey contains electric field data. This would be done, for example, if you wanted to create outputs for the B-field or H-field rather than their time derivatives.

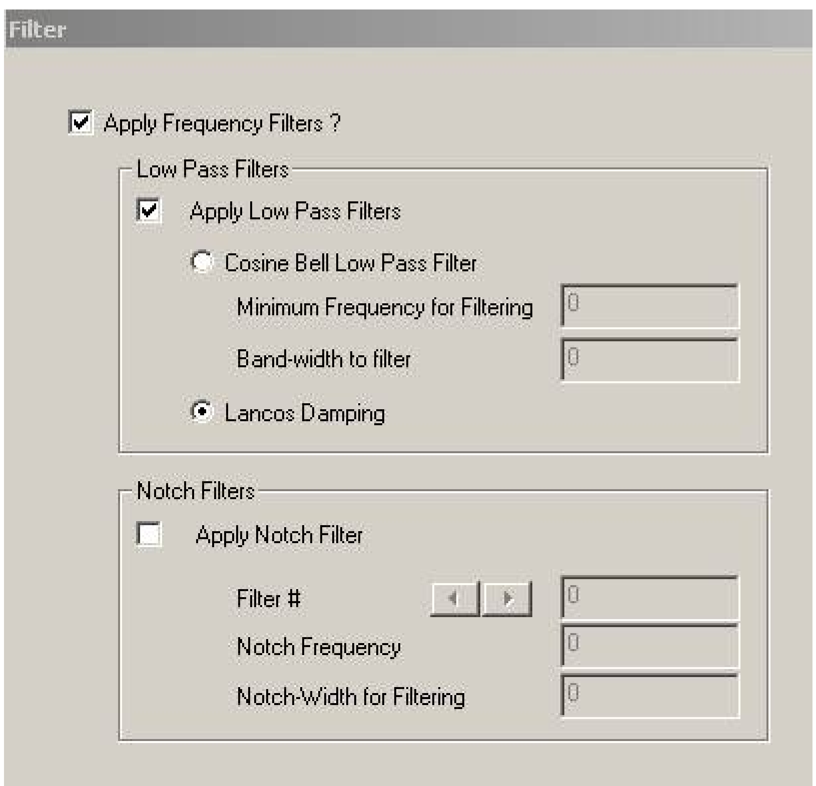
- Click **Next** to proceed to the **Filter** dialog.

***Note***. The **Analytic fields** button, **Use amplifier** and **Amplifier fields** are for our testing purposes and are not accessible to the user.

## Filter page

The frequency sampling scheme produced with the Spectral option in EMIGMA (see **Specify Spectral Mode**) results in 9 sets of frequencies, with sequence indices from –1 to 7 and maximum frequency being 17.10 MHz. Time-domain systems are band limited with low frequency filters. For magnetic data, usually with a cutoff ranging between 20 to 70 kHz. In other words, when using the sampling scheme and interpolation of FSEMTRS, frequencies up to sequence index 4 (1.7 to 17 kHz) are all that is required to provide a good first approximation to the response. To obtain better accuracy, use up to set # 5. There are no advantages in using higher frequencies, since only in a very few cases do time-domain systems measure this part of the response. The exception is newer systems with air-filled coils that appear to have higher bandwidths.

In the **Filter** page:



- If the **Apply Frequency Filter** box is de-selected, check it to activate the available options

- Check the **Apply Low Pass Filter** box to allow selection of one of the two types of low pass filters.

  By default, the Lancos filter will be on. This filter dampens higher frequencies much like real systems and provides a simple smooth low pass. To apply a cosine bell filter, select the respective option. In this case, you will be offered to specify the minimum frequency and the bandwidth to filter

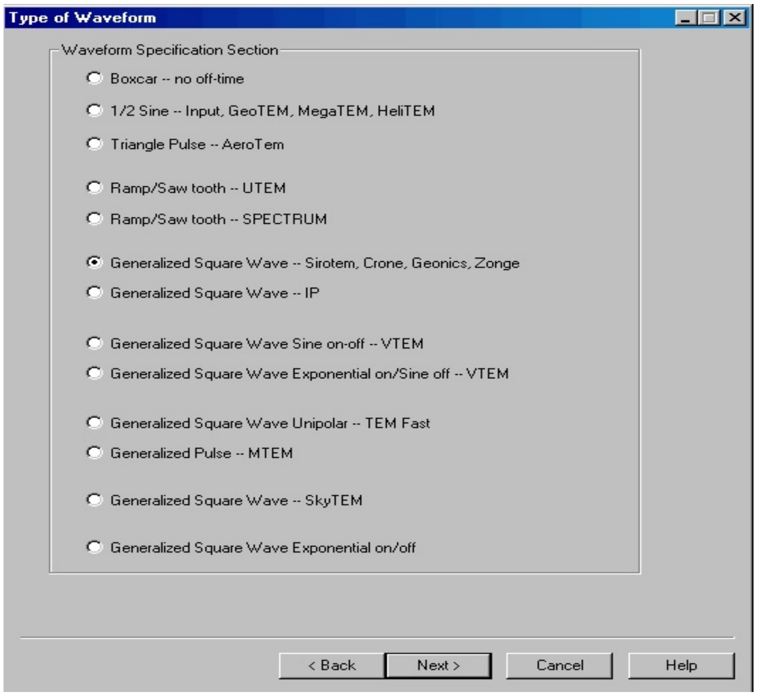
- Check the **Apply Notch Filter** box to allow selection of a notch filter

  In the boxes below you are offered to specify the notch frequency (center point) and width to filter. Currently only one notch is allowed

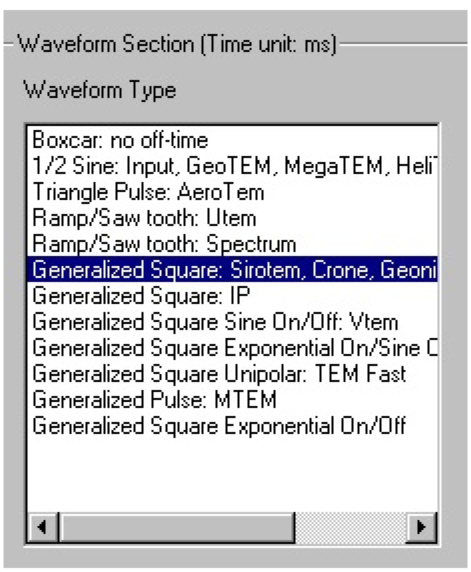
- Click **Next** to proceed to the **Type of Waveform** dialog.

# Types of Waveform

Select the type of waveform to be used by EMIGMA to simulate time-domain response. All of the waveforms are characterized by dual polarity:




And this is what is seen in the **Waveform** tab of **Configuration** under **Waveform Settings**



See:

BoxCar

1/2 Sine Wave

Triangle Wave

Ramp/Sawtooth

Generalized Square Wave

**BoxCar**

- Select **BoxCar** in the **Type of Waveform** dialog

  The BoxCar waveform is a positive ON for a half period followed by a negative ON for the other half-period. When the time derivative is taken, it creates an impulse once every half-period. No off-time is allowed for this waveform. No additional waveform characteristics need to be specified

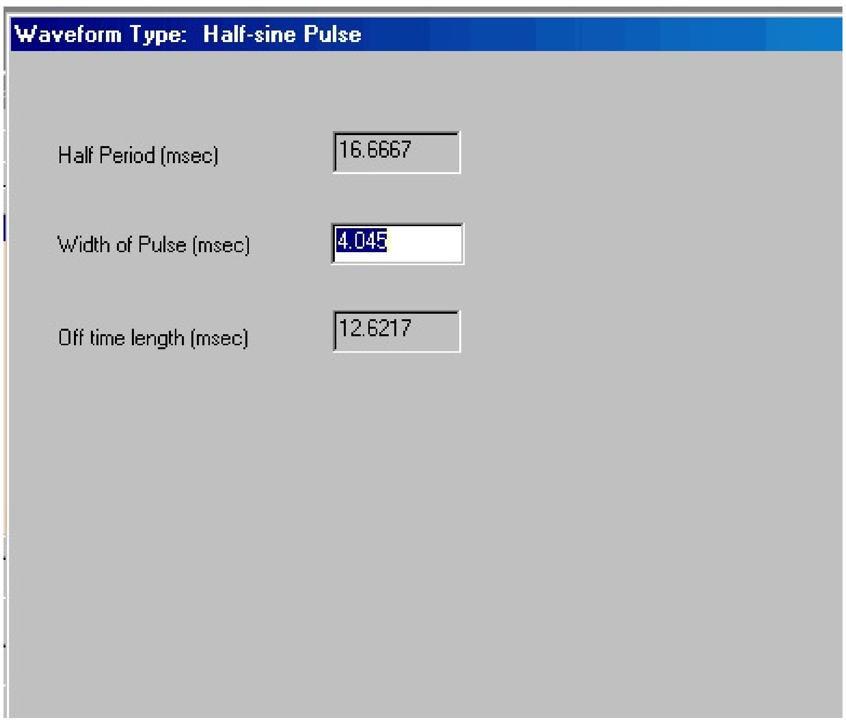
This is a theoretical waveform rather than an actually realized waveform. To realize this waveform then the power spectrum of the data must contain energy to an infinite frequency and cannot represent turn-off circuits in the transmitter nor low pass filters nor non-linear effects in the receiver.

- Click **Next** to directly proceed to the digitizing step of the FSEMTRS procedure.

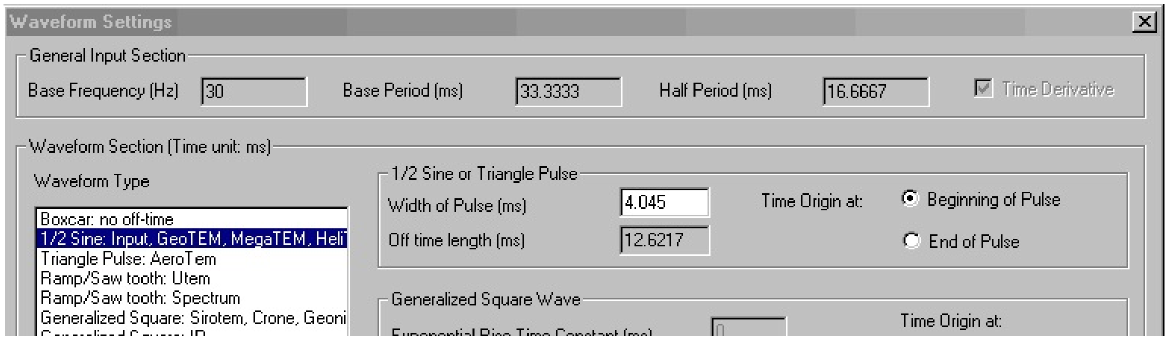
### 1/2 Sine wave

The ½ **Sine** is an on-pulse which approximates a half period of a sine function followed by an off-time and then a negative repetition to remove any DC offset. It may suit INPUT, new and old GEOTEM, MegaTEM or Questem data or merely any waveform of the ½ sine type. This waveform requires further specification:

- In the **Type of Waveform** window, select **½ Sine** and click **Next**. The **Waveform Type: Half-Sine Pulse** window appears:



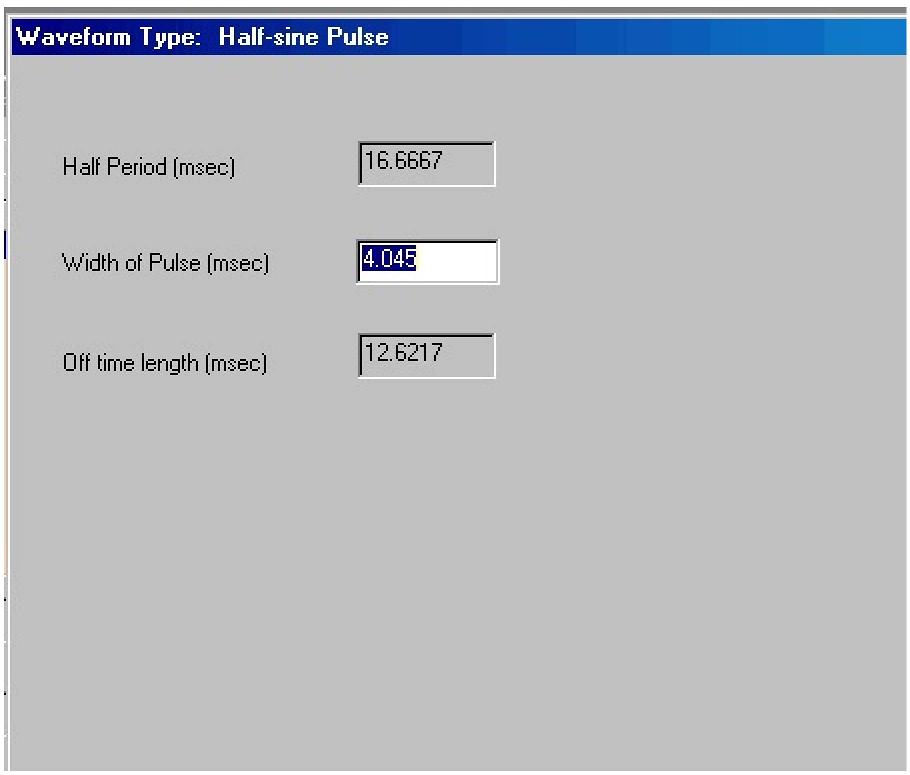

- Change the pulse width in the respective box and click in the **Off time length (msec)** box to update the value in it
- Click **Next** to proceed to the digitizing step of the FSEMTRS procedure and set windows and start time for the window
- In direct transform, the following interface appears under **Waveform Settings**


Waveform Settings
General Input Section
Base Frequency (Hz)
30
Base Period (ms)
33.3333
Half Period (ms)
16.6667
Time Derivative
Waveform Section (Time unit: ms)
Waveform Type
Boxcar: no off-time
1/2 Sine: Input, GeoTEM, MegaTEM, HeliT
Triangle Pulse: AeroTem
Ramp/Saw tooth: Utem
Ramp/Saw tooth: Spectrum
Generalized Square: Sirotem, Crone, Geoni
1/2 Sine or Triangle Pulse
Width of Pulse (ms)
4.045
Off time length (ms)
12.6217
Time Origin at:
Beginning of Pulse
End of Pulse
Generalized Square Wave
Time Origin at:

**Triangle wave**

The triangle waveform is a kind of a half-sine wave having two linear slopes, one up and one down, equivalent in time and followed by an off-time. This waveform suits AeroTem data. This waveform requires further specification

- Select **Triangle Pulse** in the [Type of Waveform]() window and click **Next**. The **Waveform Type: Triangular Pulse** window will open:




- Edit the pulse width as required in the respective box and click in the **Off time length (msec)** box to update the value in it
- Click **Next** to proceed to the digitizing step of the FSEMTRS procedure.

**Ramp**

The ramp has a linear rise followed by a linear decay, with the slope of the triangle being $\pm$4/T. No off-time is allowed. The two ramp options available are suitable for UTEM and SPECTRUM data, respectively. No additional details need to be specified.

Click **Next** to directly proceed to the digitizing step of the FSEMTRS procedure and to normalization procedures.

**Generalized square wave**

The generalized square wave (the first two options) is characterized by an exponential rise, linear ramp and an off-time followed by a negative repetition. The first option can be suited to simulate many commercial waveforms (Sirotem, Crone, Zonge, Geonics). The second, IP, can only be selected for electric field data.

The third, **Sine On-Off**, is characterized by a quarter-sine rise and fall, with all other conditions being the same, and is suitable for VTEM data. This is the type of waveform utilized in the earlier years of VTEM.

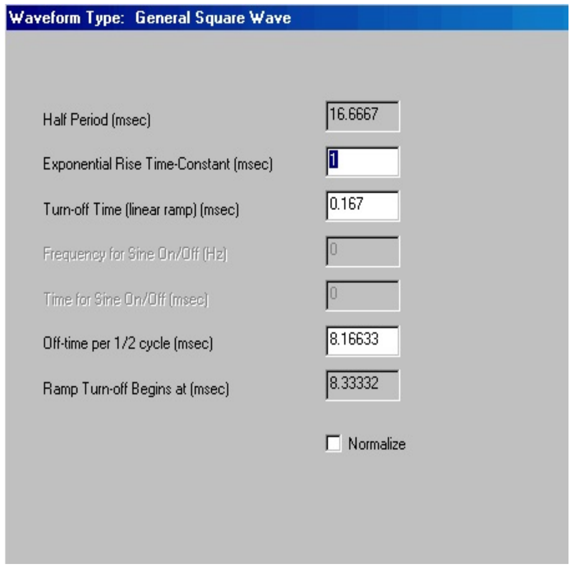
The fourth, **Exponential on/Sine off** is a modification of the third which more closely matches the actual VTEM waveform. It has an exponential rise. The turn-off is some fraction of a sine function. Unlike the previous option, it need not be a full quarter-sine. Examination of the waveform file will allow determination of the specific characteristics of the VTEM survey.

The fifth, **SKYTEM**, allows settings specific to SKYTEM data

The sixth is for an exponential Turn OFF. This, for example, is suitable for the TerraTEM system. The current is turned off but there is no control of the turn off and thus the loop decays with an exponential. This can be exacerbated by a conducting cover which causes a back EMF in the wire and extends the turn off.

All six options require further specification

- Select a required option in the **Type of Waveform** window and click **Next**. This will bring up the **Waveform Type: General Square Wave** window:




In the case of the first and second options:

- Specify the exponential rise time-constant and turn-off time (linear ramp) in the respective boxes. This is not critical but should not be too slow.

  For IP data, generally, the turn-off time is not known but is generally not relevant as the first window is long after turn-off

- Click in the **Off-time per ½ cycle (msec)** box to update the respective value in it. The latter depends on the ramp time.

- To normalize your output, check the **Normalize** box. In the case of IP, this box is checked by default and set to an ON Time measurement as traditionally IP on time is considered resistivity data

  And off-time data is normally is the voltage near the end of the on-time. However, you may uncheck and now the IP data consists of ON and OFF time data is units of Volts or mVolts and is examined as

  with any other type of time domain data. If importing your data, then you will be asked how you wish to normalize your data

If you selected the third, **Sine On-Off** option:

- Specify the frequency for sine On or Off in the respective box and click in the **Time of Sine On or Off** box to update the value in it.

- The **Normalize** box will be inaccessible, since VTEM data values are absolute.

For the fourth option:

- Specify the exponential rise time-constant.

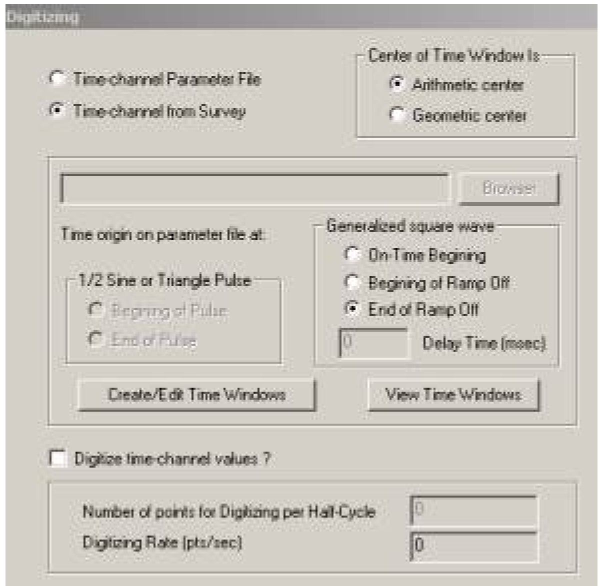
- Specify the frequency for sine-off and the time for sine-off in the respective boxes. Unlike the **Sine On-Off** option, these values are not linked, enabling the turn-off to be set as less than a quarter period of the sine function.

- The **Normalize** box will be inaccessible, since VTEM data are absolute

- Click **Next** to proceed to the digitizing step of the FSEMTRS procedure.

## Digitizing

After waveform selection, the **Digitizing** page will appear which is to determine the time windows for the data




In the upper left-hand corner, select between an already available parameter file and a survey to take a desired time-window array from. In both cases you can edit this array and save it as a new parameter file. Or create time windows from scratch.

See

Digitize Actual Measured or Simulated Time-Domain Data

Digitize Data from Scratch Using a Time-Channel Parameter File

Calculate Several Measurements in a Time Window

**Digitize actual measured or simulated time-domain data**

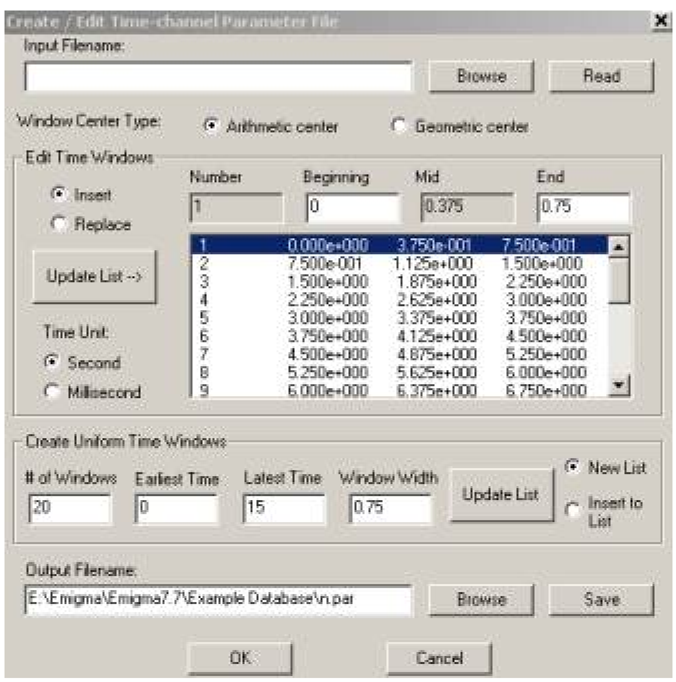
If your survey contains actual measured or simulated time-domain data, the **Time-channel from Survey** option in the [Digitizing]() page will be on by default

- Select between **Arithmetic Center** and **Geometric Center** in the **Center of Time Window Is** section

- Click the **View Time Windows** buttons. The **Time Windows** page will open showing the start, end and mean (dependently of the center type selected) values of each of the windows. You can choose the sec or msec representation of the data in this dialog

To edit your time window settings:

- Click the **Create/Edit Time Windows** button to open the respective window




The table in the **Edit Time Windows** section of the page will display all available time windows.

To edit any of the windows:

- Select a required window from the table and edit the beginning and the end of the window in the respective boxes above. The **Mid** box will show the mean of the two values dependently of the window center type you selected in the **Digitizing** page. However, if necessary, you can change the window center type right here, in the respective section on top
- Select **Replace** if you want to replace the former window with the adjusted and **Insert** if you want to add this new window to the existing list of windows
- To delete a time window, select it and press Delete
- Click the **Update List** button and view the results.

The time values can be represented both in seconds and milliseconds. Select the time units in the lower left-hand corner of the section

To create uniform time windows:

- Specify the number of windows you want to create in the **# of Windows** box

- Specify the earliest and latest times to define a required time interval and click in the **Window Width** box to calculate the respective value

- In the left part of the section, select the **New List** option to replace the list in the table above or **Insert** to add your new array of windows to this list

- Click **Update List**

- To save your new list of windows, click **Browse** to the right of the **Output Filename** field and choose the directory to save it in as a parameter (*.par) file. You need not save the windows as they will be stored to the new dataset

- Click **OK** to return to the **Digitizing** dialog

**Digitize data from scratch using a time-channel parameter file**

If you created your survey from scratch and want to use an already available time-channel parameter file:

- Select the **Time-Channel Parameter File** option on the **Digitizing** page. The filename field below will become active

- Click **Browse** to open the standard Windows-style Open dialog and find the required parameter file. There is a wide selection available in the EMIGMA\TimeChannelFiles directory.

- Create/Edit allows you to create your own windows

- To view this file, click the **View Time Windows** button in the **Edit Time Windows** section

- Specify the time origin the time gates are to be relative to.

  If the waveform selected is ½ sine, the **½ Sine or Triangle Wave** section will be active. Select between the **Beginning of Pulse** and **End of Pulse** options

  If the waveform selected is generalized square, the **Generalized Square Wave** section will be active. Select between **On-Time Beginning**, **Beginning of Ramp-Off** and **End of Ramp-Off**

To change the parameter file

- Click the **Create/Edit Time Windows** button. The respective window will open, with the **Input Filename** field containing your parameter file

  If you decide to use a different parameter file at this point, you can either click the **Browse** button to the right of the **Input Filename** field to browse for another file or type the name of this other file directly in the **Input Filename** field. Click the **Read** button to update the list of time windows in the table below

- Edit your time channels as required, see Digitize Actual Measured and Simulated Time Domain Data. Click **Save** to overwrite the parameter file or **Browse** to save it as a separate filename
- Click **OK** to return to the **Digitizing** page

**Calculate several measurement in one time window**

**Actual measurements are made more often than the number of time windows. The data within a window are “binned” (averaged) to create the data. If you wish to reproduce this aspect, it is done here.**

- On the **Digitizing** page, check the **Digitize Time-Channel Values?** box. The section below will become active
- Specify the number of points for digitizing in a half-cycle in the respective box. The digitizing rate below will update accordingly

## Normalization

In all cases, where normalization is required, the **Normalization** window will open after the **Digitizing** step. Its appearance will vary dependent upon the waveform. These specifications will be set by default for you if your survey contains measured or simulated time-domain data.

See

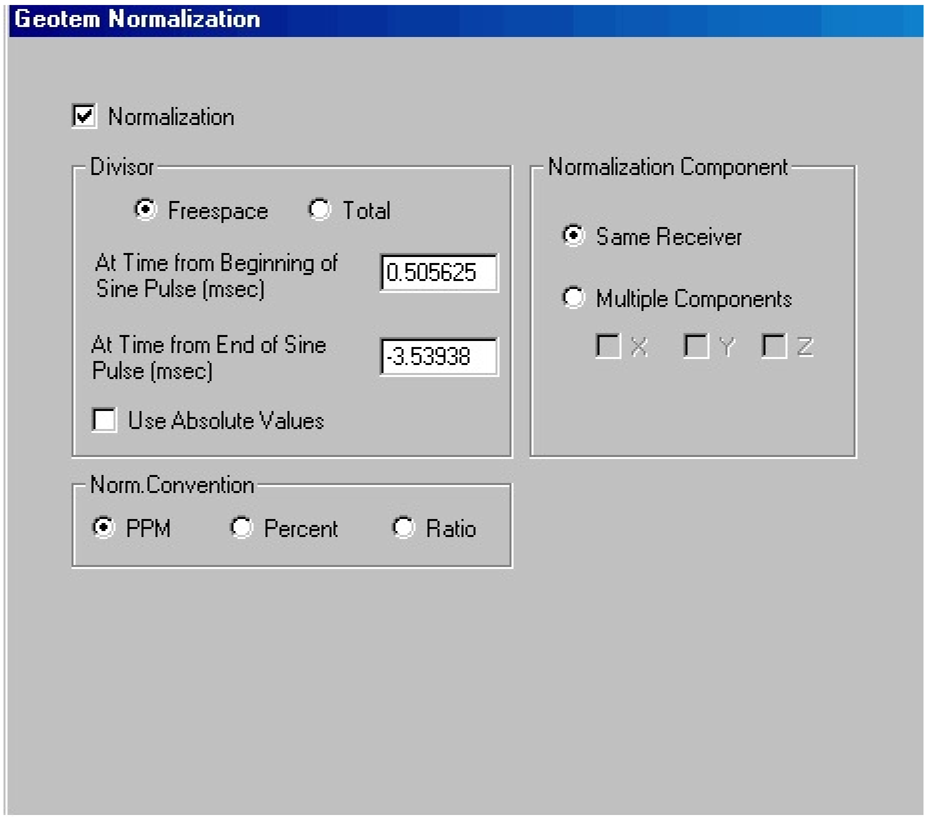
Half-Sine Wave

Ramp (UTEM and Spectrum)

General Square Wave IP

**Half-sine wave**

In the case of a ½ sine wave, the Digitizing step will be followed by the **Geotem Normalization** window:




- Check the **Normalization Check Box** to activate the sections below.

  In the **Divisor** section:

- Select between the **Freespace** and **Total** response to be used as a divisor. Normally freespace

- Specify the time channel to use for normalization: type a required value in one of the boxes, **At Time from the beginning of Sine Pulse** or **At Time from End of Sine Pulse**. Click in the other box to update the value accordingly

- Leave the **Use Absolute Values** box checked to utilize the absolute value of the divisor. Otherwise, de-select this box

In the **Norm Convention** section, select between the three output units: ppm, percent and ratio

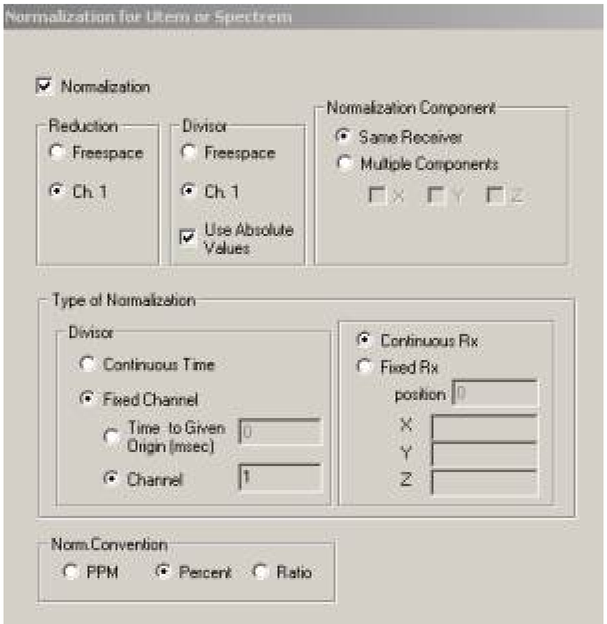
In the **Normalization Component** section:

- Select **Same Receiver** if you want to normalize your data to the same component as measured which is normal
- Select **Multiple Component** to specify one-, two- or three-component normalization. In the latter case, the freespace or total response from all components is used as the normalizing factor
- Click **Next** to proceed to the final **Output** window.

Previous/Next

**Ramp (UTEM or Spectrum)**

In the case of UTEM or Spectrum surveys, where ramp waveforms are used, the **Normalization** dialog to open will be as follows:




- Check the **Normalization** box to enable all the sections of the dialog

In the **Reduction** section:

- Select between **Freespace**(primary field) and **Ch. 1** for UTEM and **Ch N** (last channel) for SPECTRUM to be subtracted from data prior to division

In the **Divisor** section:

- Select **Freespace** or **Ch. 1** (UTEM)/**Ch N** (SPECTRUM) to divide your data by after the reduction

- Check the **Use Absolute Values** box to utilize the absolute value of the divisor

In the **Type of Normalization** section:

- Select between the **Continuous Time** and **Fixed Channel** options

  If you selected **Freespace** in the **Divisor** section above, the **Continuous Time** option will be on by default; it means that each time channel will be divided by its own freespace component. If needed, you can change this option to **Fixed Channel** and thus to divide by **Freespace** of only **Ch 1/Ch N** or specify a certain time within this channel in the **Time to Given Origin** box.

  If you selected **Ch 1/Ch N** option, the **Fixed Time** button will be on by default; it means that each channel will be divided by **Ch 1/Ch N** or by some concrete time within **Ch 1/Ch N**, if you specify it in the **Time to Given Origin** box.

- Select between continuous and fixed receiver

  The **Continuous Rx** option will provide normalization of data for all locations; the **Fixed Rx** option will normalize data only in a certain location. Currently, the latter option is not available.

In the **Normalization Component** section:

- Select **Same Receiver** if you want to normalize your data to the same component as measured

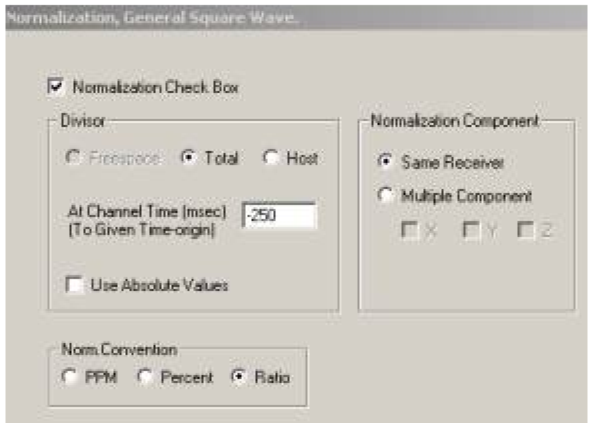
- Select **Multiple Components** to specify one-, two- or three-component normalization. In the latter case, the freespace or Ch 1/Ch N from all components is used as the normalizing factor

In the **Norm.Convention** section, select between the three output units available: ppm, percent and ratio

Click **Next** to proceed to the final **Output** dialog.

## General square wave IP

In the case of the General Square Wave IP, the **Normalization** dialog will be as follows:




- Check the **Normalization Check Box** to activate the sections below. But, the software supports unnormalized time domain IP data as well.

In the **Divisor** section:

- Select between the three options (**Freespace**, **Total** and **Host**) for your data to be normalized to

- Set the time to use for normalization. The default is an **On-Time** relative to the time origin selected in the previous window (**Digitizing**). Change it, if necessary in the **At Channel Time (msec)** box

- Check the **Use Absolute Values** box to utilize the absolute value of the divisor

In the **Normalization Component** section:

- Select **Same Receiver** if you want to normalize your data to the same component as measured

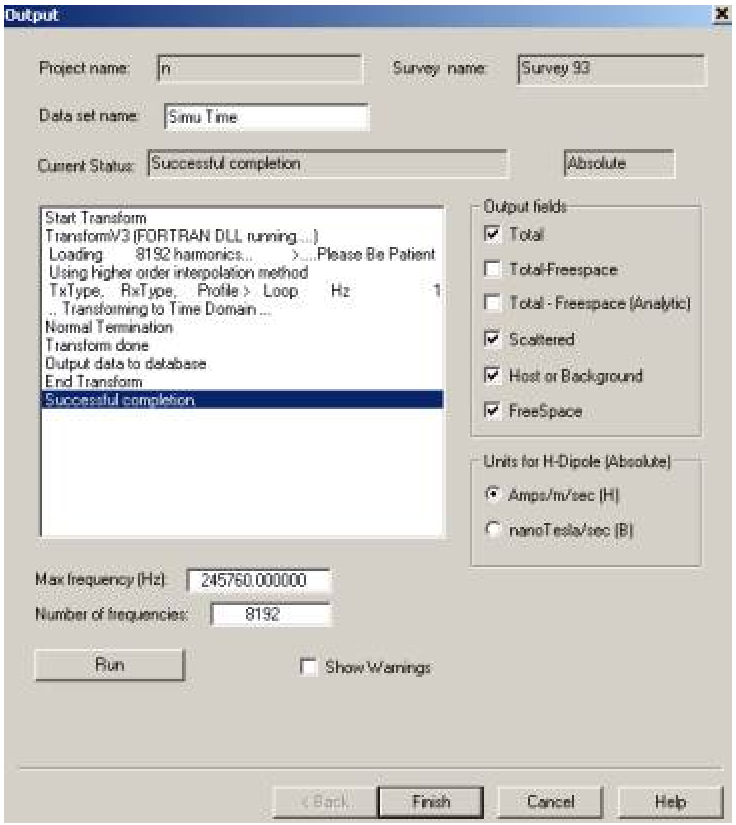
- Select **Multiple Component** to specify one-, two- or three-component normalization. In the latter case, the freespace, total or host response from all components is used as the normalizing factor

In the **Norm Convention** section, select between the three output units available: ppm, percent and ratio

Click **Next** to proceed to the final **[Output]** dialog

# Output

The **Output** dialog is the final step of the frequency to time domain transform wizard. From this dialog the transform is launched, with the results to be stored in the database:




In the upper part of this dialog, you will see the name of the project and the survey number. The name of the data set to be created is generated automatically in the respective box. The **Output Fields** section will show the available measured/simulated and calculated responses. The gray box above the **Output Fields** section indicates whether or not normalization is applied

- In the **Units for H-Dipole (Absolute)**, select between the two options offered – Amp/m and nanotesla/sec. In the case of normalized data, this section will be disabled

- Click **Run** to start the transform. You will be able to follow the main stages in the central field, whereas the **Current Status** box above will be updated accordingly

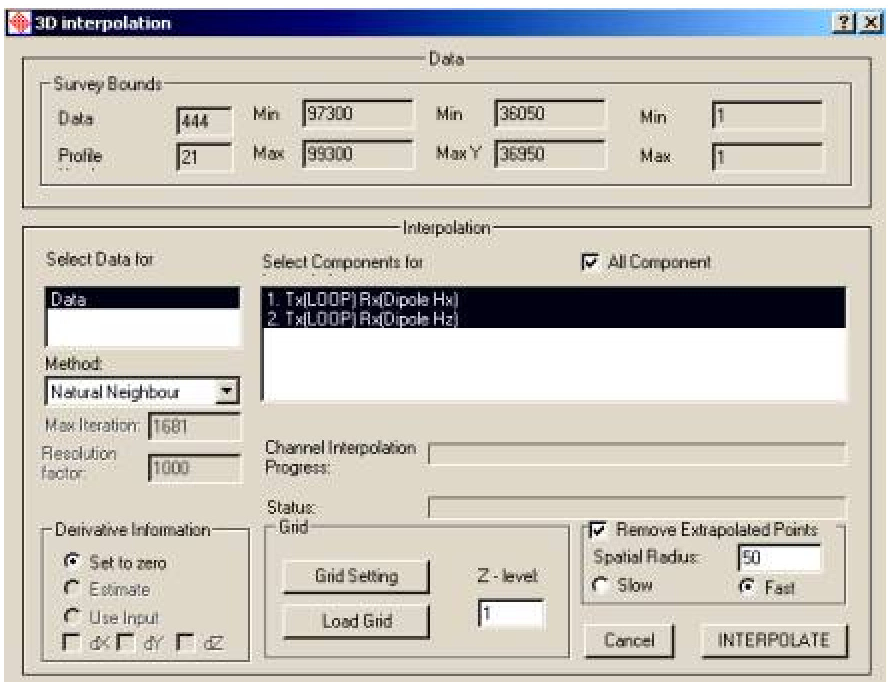
  ***Note***. *Before running the transform, you can check the **Show Warnings** box to have a better control over each step of the operation.*

The transform complete, the output file is written to the database. Select **Finish** to close the application.

# Gridding

EMIGMA offers 5 gridding algorithms and an easy-to-use and intuitive interface. You have to specify the terms of interpolation and to define the desired parameters of your grid, and your grid is ready. You can also load an existing grid from your database.

Select the data set to interpolate in the **Database** dialog and click the **Gridding** button  on the main toolbar of EMIGMA. The **3D Interpolation** dialog will open, with the top section (**Data**) showing the profile and coordinate information of your data set:




**3D Interpolation Dialog**

To create a grid:

Specify the Terms of Interpolation

Specify Grid Parameters

Load an Existing Grid

and click **INTERPOLATE** in the right-hand corner of the **3D Interpolation** dialog.

***Notes***. To view the interpolation results, click the **GridPresentation** button  on the main toolbar of EMIGMA

To view grid information, click the **Has Related Grid** button in the **Database** dialog.

## Specify the terms of interpolation

In the **Interpolation** section of the main [3D Interpolation] window:

- Select the type of data to interpolate in the **Select Data** field

  All data obtained by means of import or simulation in EMIGMA, subjected to normalization, etc., are considered as core data and are referred to as ***Data***; all the rest calculated through various algorithms are considered as optional and are referred to in accordance with their type, e.g. ***Apparent Resistivity***, ***Apparent Depth***, ***Voltage***, etc.

- Click on a component in the **Select Components** field to involve it into interpolation or check the **All Components** box to have all components participate in the interpolation process
- Select the method of interpolation in the respective dropdown list.

  There are five choices: **Natural Neighbour**, **Delauney Triangulation**, **Shepard** or True to data, **Thin Plate Spline** and **Minimum Curvature**, with the first being the most frequently used. If you select **Minimum Curvature**, type the maximum number of iterations to be performed and specify the resolution factor in the respective fields below, which in this case will become activated

- If your data contain derivative information, you can carry out interpolation based on all the three derivatives at a time. In this case, the result will be more accurate in comparison with what is obtained when you use data as is. Turn the **Use Input** button on in the **Derivative Information** section and select the derivatives to participate in interpolation
- Check the **Remove Extrapolated Points** box to activate the respective section and forbid extrapolation to the "no data" locations
- Set a required spatial radius to restrict the area of interpolation.

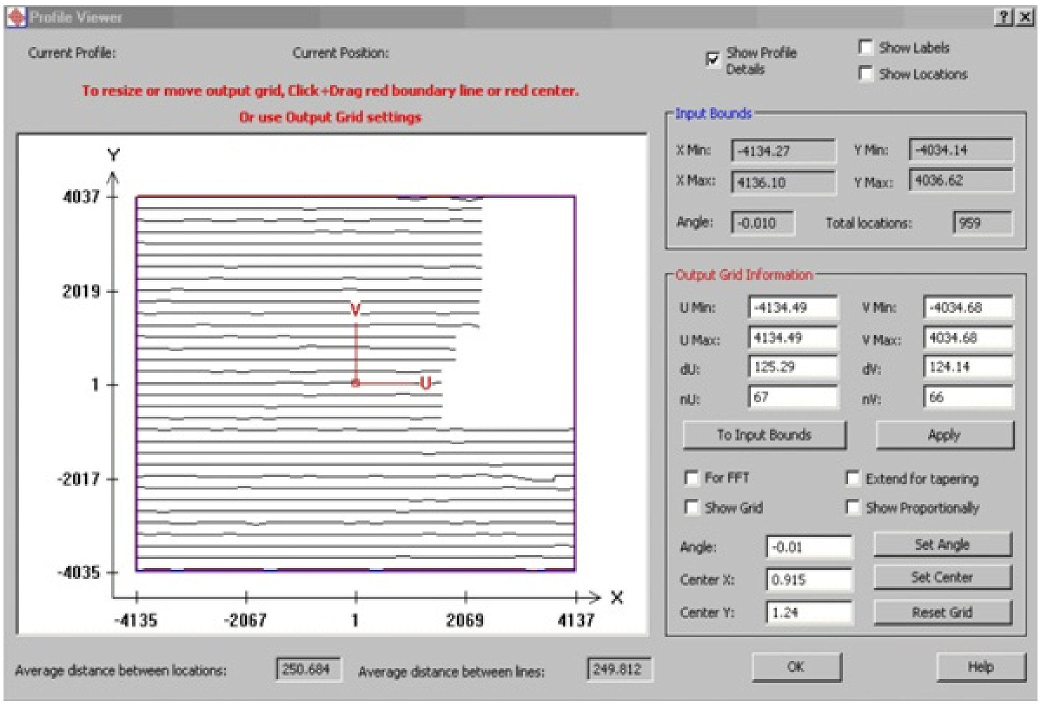
  In the present example: a spatial radius of 50 means that if there are no data in the radius of 50 m around a given point - a grid cell center - this cell will be removed from interpolation

- Select between the slow and fast interpolation algorithms (slow is more accurate, but fast is almost always sufficient)
- Click **OK**

## Specify grid parameters

Click the **Grid Setting** button in the **Grid** section of the **3D Interpolation** dialog

. The **Profile Viewer** dialog will open:




In this dialog:

- Customize the coordinate system by selecting required settings in the upper right-hand corner of the dialog
- Select between the **Show Profile Details** and **Show Locations** options and check the **Show Labels** box to display the profile numbers (names)
- Specify the input bounds, a blue line enclosing the data to be involved in interpolation, in the X and Y **Min** and **Max** boxes of the respective section. Or, you can simply click and drag the input bounds (blue) right in the grid view field of the dialog. Set the rotation angle of the grid about its local center and click the **To Input Bounds** button

This will automatically change the **U** and **V** coordinates of the grid in the **Output Grid Information** section, and the input bounds (blue) and output grid bounds (red) will coincide

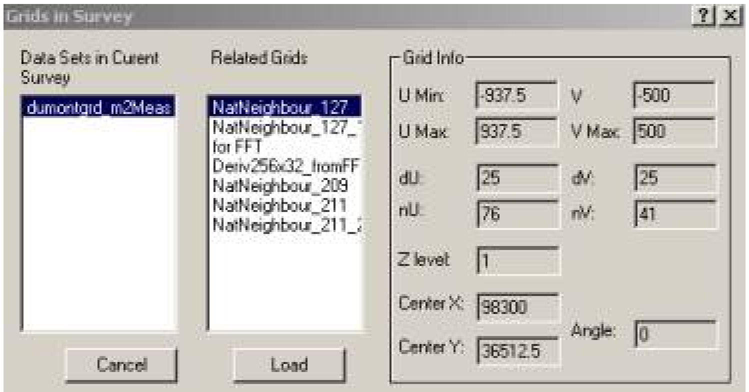
The gridding tool allows for an arbitrary azimuth for the output grid allowing, for example, to grid relative to a known or assumed structural strike.

- If you want the output bounds to cover a different area compared to the input grid, edit the **U** and **V** values and the angle of grid rotation in the **Output Grid Information** section and click **Apply**. The output grid (red) will be changed, while the input bounds (blue) will stay the same
- To adjust the grid density, increase or decrease **nU** and **nV** (number of points) or **dU** and **dV** (length of a grid cell side) in the **Output Grid Information** section. Note that **dU** need not equal **dV**
- Check the **For FFT** box, if you want to subject your data to FFT. This will automatically change the **nU** and **nV** values to the n power of 2
- To display the grid, check the **Show Grid** box; to provide its proportional view, check the **Show Proportionally** box to the right
- To edit the local center of the grid (U vs V), type your values in the **Center X** and **Center Y** boxes in the bottom of the **Output Grid Information** section and click **Set Center**
- To reset the boundaries of your grid to the ones determined by the initial Input Bounds coordinates, click **Reset Grid** in the **Output Grid Information** section.

- Click **OK** to return to the main **3D Interpolation** window

## Load an existing grid

- Click the **Load Grid** button in the **Grid** section of the **3D Interpolation** dialog. The **Grids in Survey** dialog will open:




- Select the data set containing the grid you want to load from the **Data Sets in Current Survey** list and the grid itself from the **Related Grids** list

  The settings of the grid to be loaded will be displayed in the **Grid Info** section on the right

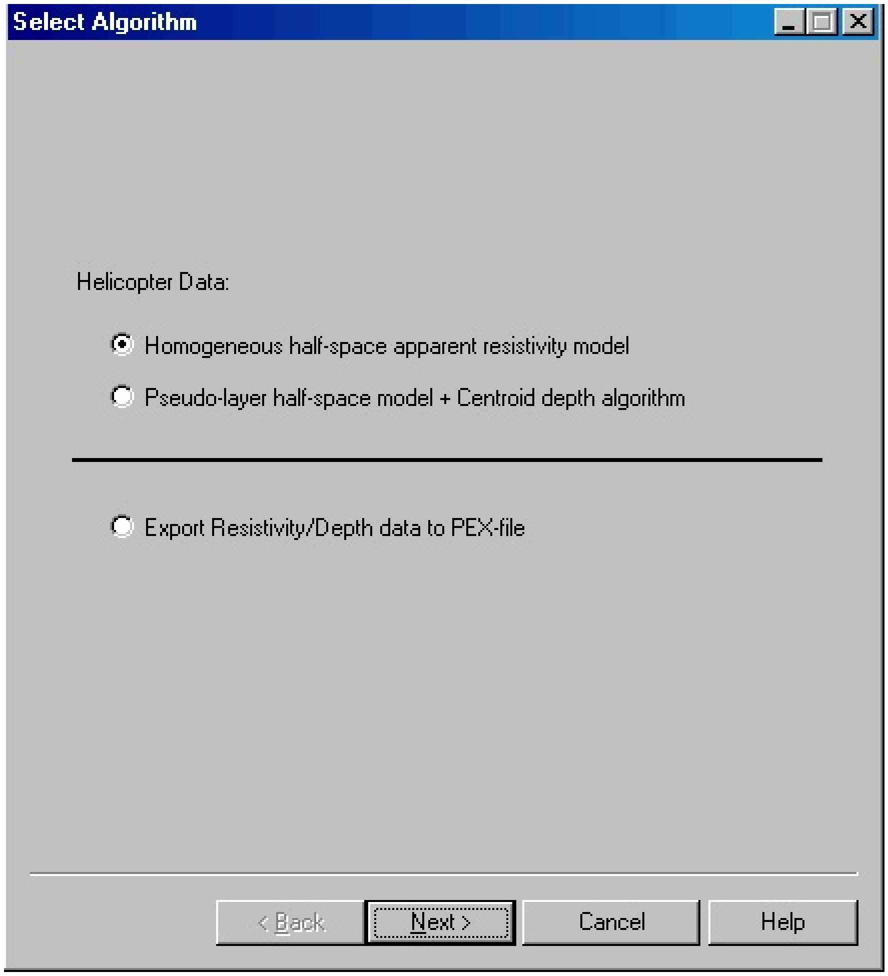
- Click **Load** to load the grid

# CDI (Conductivity Depth Imaging)

The CDI tool has two functions. First, to calculate a half-space apparent resistivity for each frequency and location for any type of dipole-dipole configuration which has a response to a halfspace. This calculation is generally performed on the Quadrature (imagingary) part of the data. It also allows you to calculate apparent resistivity and apparent depth of airborne dipole-dipole data. It was designed for the processing of airborne (mainly helicopter) FEM data but is also suitable for some fixed wing configurations.

Select a data set on the **Database** window and click the **Conductivity Depth Imaging** button 

on the main toolbar. The **Select Algorithm** window appears offering you to choose one of the two following models for dipole-dipole FEM data:




**Related Topics**

Calculate Apparent Resistivity (airborne and ground FEM data)

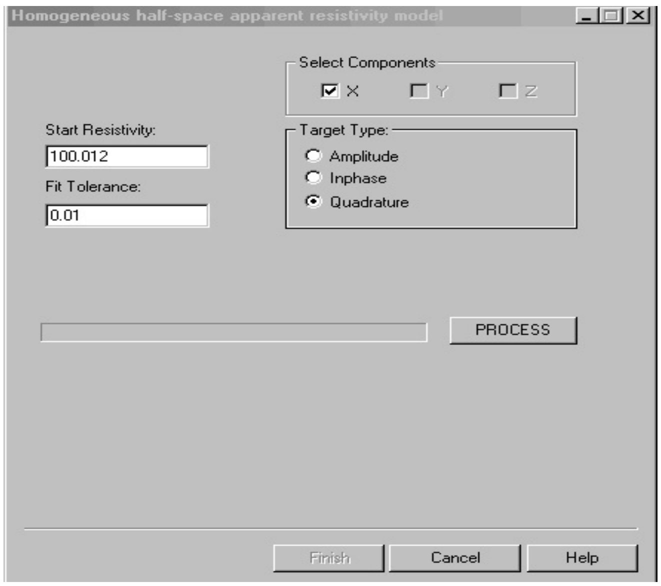
Calculate Both Apparent Resistivity and Depth (Sengpiel Section) (airborne FEM data)

Display Resistivity Processing Results

## Calculate apparent resistivity

- In the Select Algorithm window, choose **Homogeneous Half-Space Apparent Resistivity Model** and click **Select**.

  In the window to appear:




- Specify which receivers you wish to use in the calculation
- Specify start resistivity and fit tolerance in the respective boxes. The most appropriate settings depend on the sampling system being used.

  In the example above, the algorithm will search for matching solutions starting with resistivity 100. As soon as it finds a match for the first data within a difference less than the set threshold 0.01, it will stop and move over to the second data value(and so on). The number of iterations is not limited.

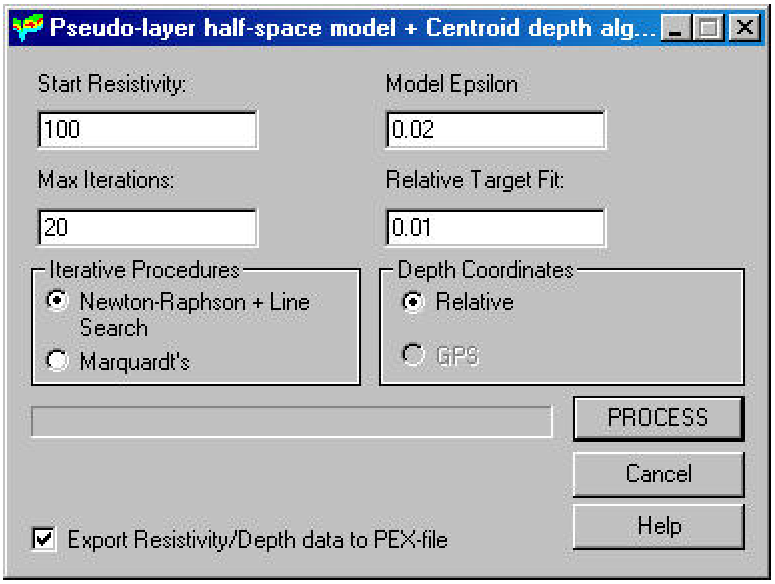
- In the **Target Type** section, select the phasor - Amplitude, Inphase or Quadrature - you are working with.

- Click **PROCESS**.

When finished, you will find a new "Halfspace Rho_" data set on the Database tab. This data can be then plotted or contoured, etc.

## Calculate both apparent resistivity and depth (Sengpiel section)

- In the Select Algorithm dialog, choose **Pseudo-Layer Half-Space Model + Centroid Depth Algorithm** and click **Select**. The following interface appears:




- Specify the starting resistivity, depending on your system, in the box labelled **Start Resistivity**.

  The algorithm used is similar to the one described in the **Homogeneous Half-Space Apparent Resistivity Model**, however, in this model, the number of iterations for the algorithm to search for a matching solution is limited.

- Specify the maximum number of iterations and select between the two kinds of iterative procedures in the respective section below

- Set the thresholds for the algorithm to stop searching in the **Model Epsilon** and **Relative Target Fit** boxes

- In the **Depth Coordinates** section, select **Relative** if you want to calculate depths relative to sea level and **GPS** if you want to take into account changes in topography

- Select the option labelled **Export Resistivity/Depth data to PEX-file** to save the processing results to a default PEX file automatically once complete. If you want to choose only some of the processing results for a PEX file then deselect this option and once the processing is complete, you will be able to select the data that will be exported to the PEX file.
- Click **PROCESS**.
- Next, if you chose not to export a default PEX file, a window will appear prompting you to perform the export. See Exporting to a PEX file for more details on how to export.

When finished, you will find a new Sengpiel Section_ data set on the Database tab of EMIGMA's main interface.

There will be a checkmark on the **Model** button for this data set to indicate a default PEX file was automatically saved. The contents of the PEX file can be viewed by clicking the **Model** button. See View Resistivity & Susceptibility Grid Data Files for more details.

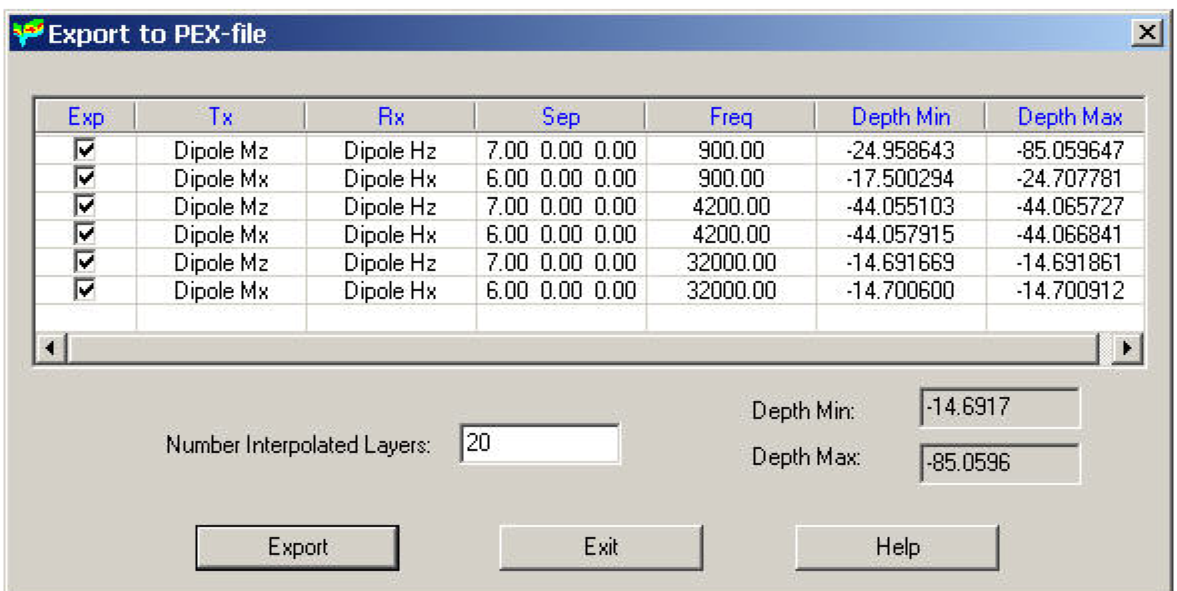
To view the results graphically, select this data set and click the **CDI Viewer** button  on EMIGMA's main toolbar (for further details see **CDI Viewer**).

A separate PEX_ data set will be generated if the PEX file was exported using the Export to PEX-file interface. This data set may also be viewed using the CDI Viewer tool.

## Exporting to a PEX file

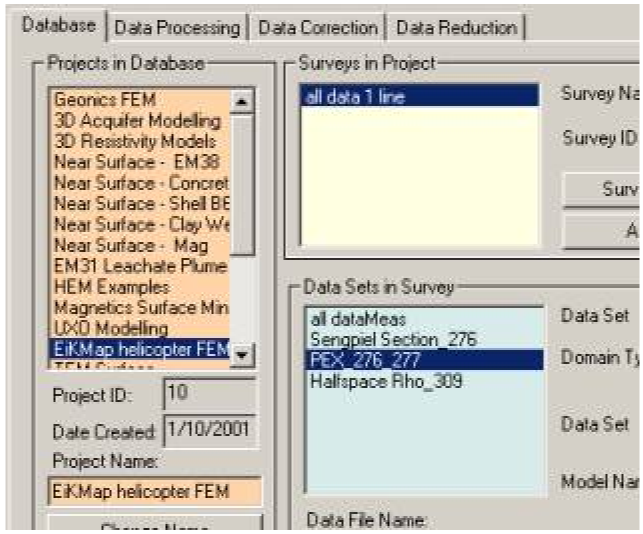
- Select a data set and click the **CDI** button  on EMIGMA's main toolbar to bring up the **Select Algorithm** dialog again
- Select the **Export Resistivity/Depth Data to PEX-file** option and click **Select**.
- The **Export to PEX-file** window will also be displayed after resistivity processing has been completed if a default PEX file was not saved. See Calculate both apparent resistivity and depth for details on creating a default PEX file.

If your data set contains no depth data (this refers only to the **Homogeneous Half-Space Apparent Resistivity Model**), a message will warn you that this export is not possible. Otherwise, the following window will open:




- In the table, de-select the components you do not want to export and specify the number of interpolated layers to divide the depth range into. In the example above, the depth range from 14.6917 to 85.0596 will be divided into 20 interpolated layers.

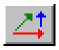
- Click **Export**. The PEX file will appear in the **Data Set** list of the **Database** tab:

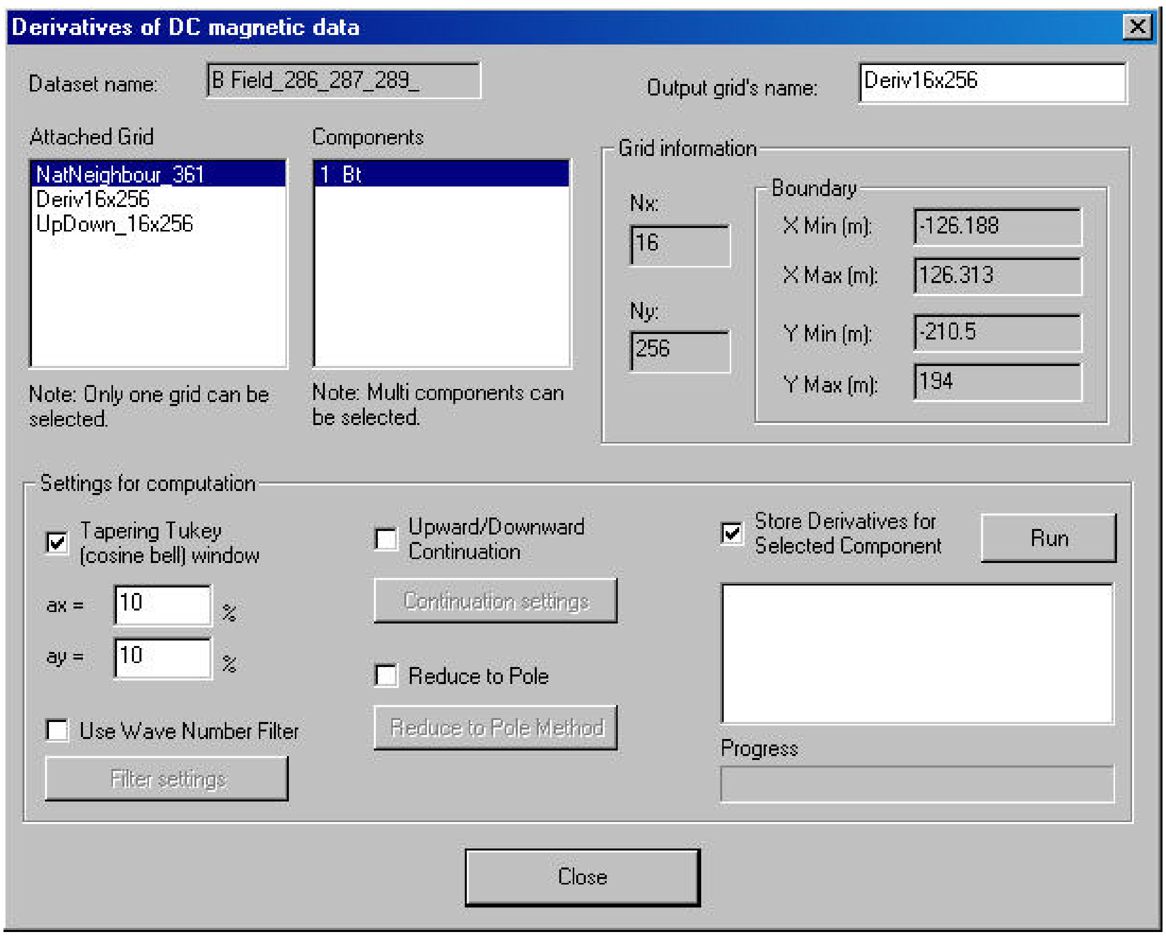



- Select this file and click the **CDI Viewer** button  on EMIGMA's main toolbar (for further details see **CDI Viewer**).

# Using the FFT Tool

To start the FFT Tool:

- Select the data set you wish to work with from the Database dialog. The data set must contain a grid which was created with the **For FFT** setting selected. See Gridding.
- Click the  button on the main toolbar and the following window appears:




- All the grids for the selected data set will be listed in the box labelled **Attached Grid**.
- Select a grid and its boundaries and number of points are displayed in the **Grid information** area.
- The components available for the grid are listed in the box labelled **Components**. Select the components that you want to perform processing

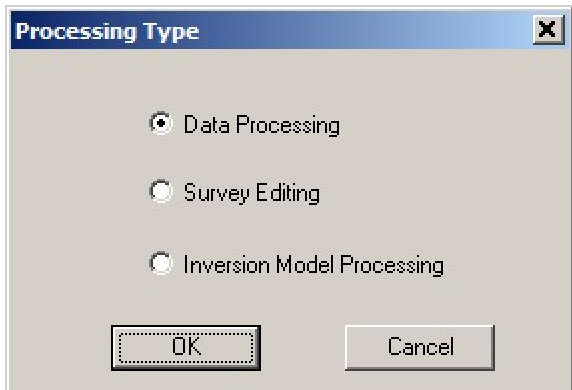
on.

- There are four groups of settings that can be specified: **Tapering Tukey window**, **Wave Number Filter**, **Upward/Downward Continuation** and **Reduce to Pole**. Check the appropriate checkbox to obtain access to the settings.
- To generate derivatives when processing, activate the checkbox labelled **Store derivatives for selected component**.
- Click **Run**.
- Once derivatives are calculated, the output grid may be selected to create higher order derivatives.

# Data Processing and Filtering

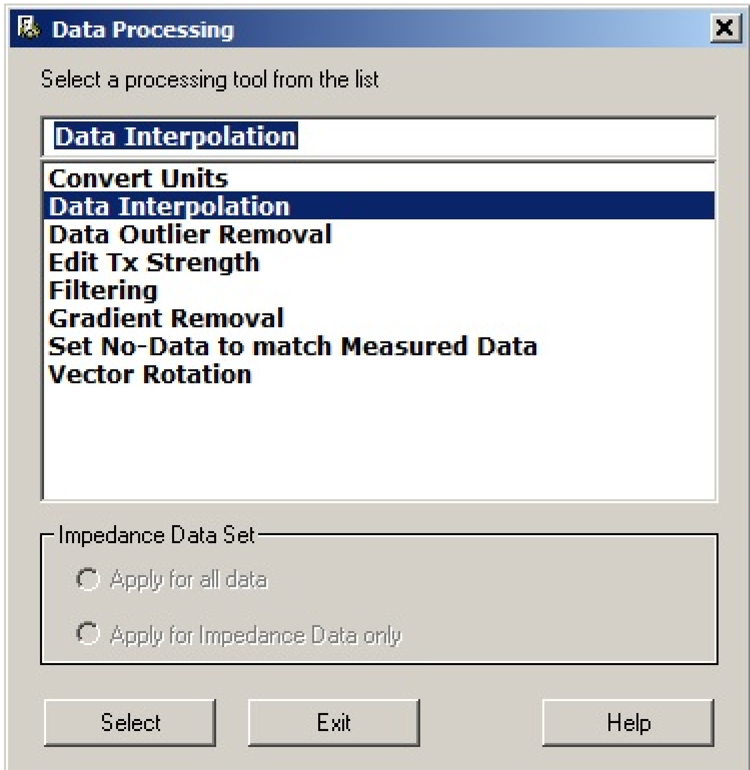
A number of processing operations are available to perform on your data. Select a data set from the main Database window and click the  button.

Select an item in the following dialog:




- Data processing
- Survey Editing
- Inversion model processing

# Data processing




- Convert units
- Data Interpolation
- Data Outlier Removal
- Edit Tx strength
- Filters
- Gradient Removal
- Set No-Data to match measured data
- Vector rotation
- Diurnal correction

- 

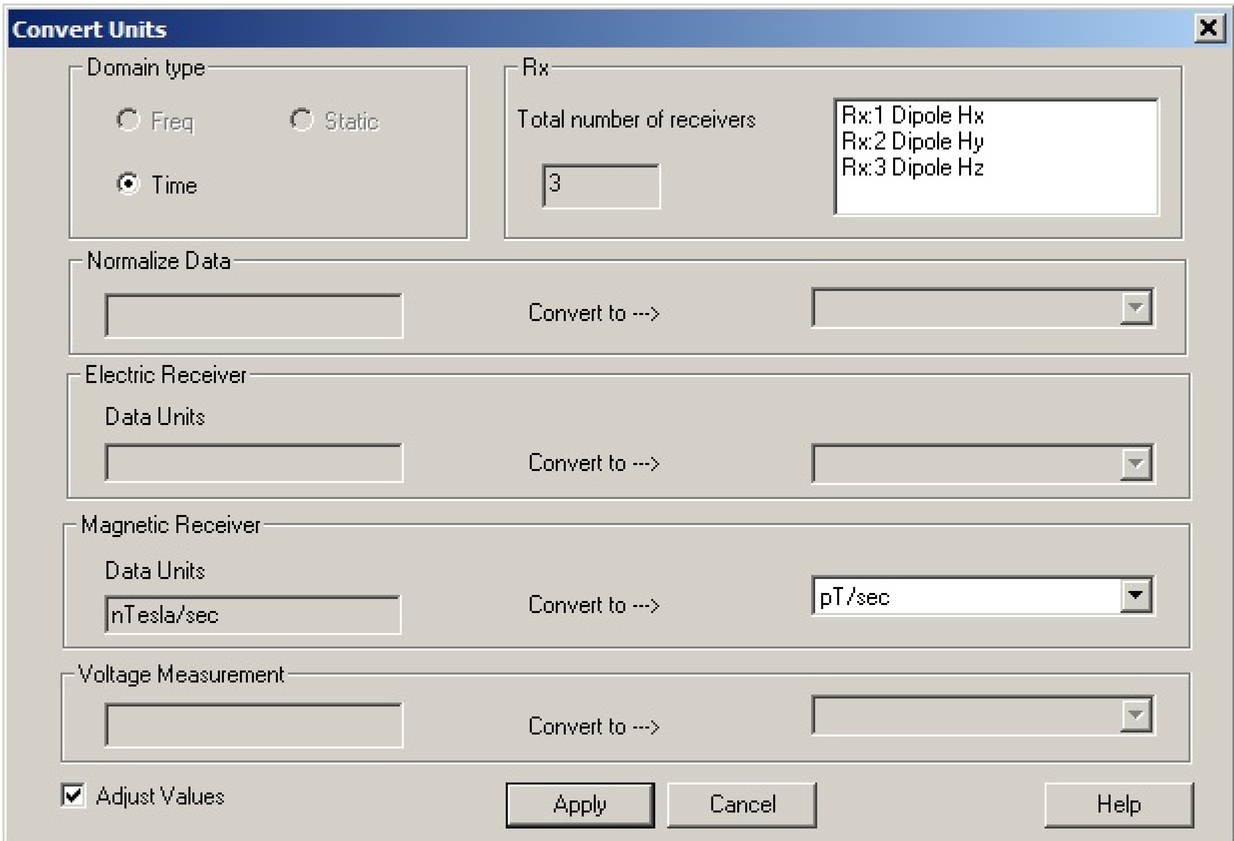
**Convert to standard IP units**
Normalizes data by voltage and presents them as a ratio.

- **MT data rotation**

- **Sort frequencies**

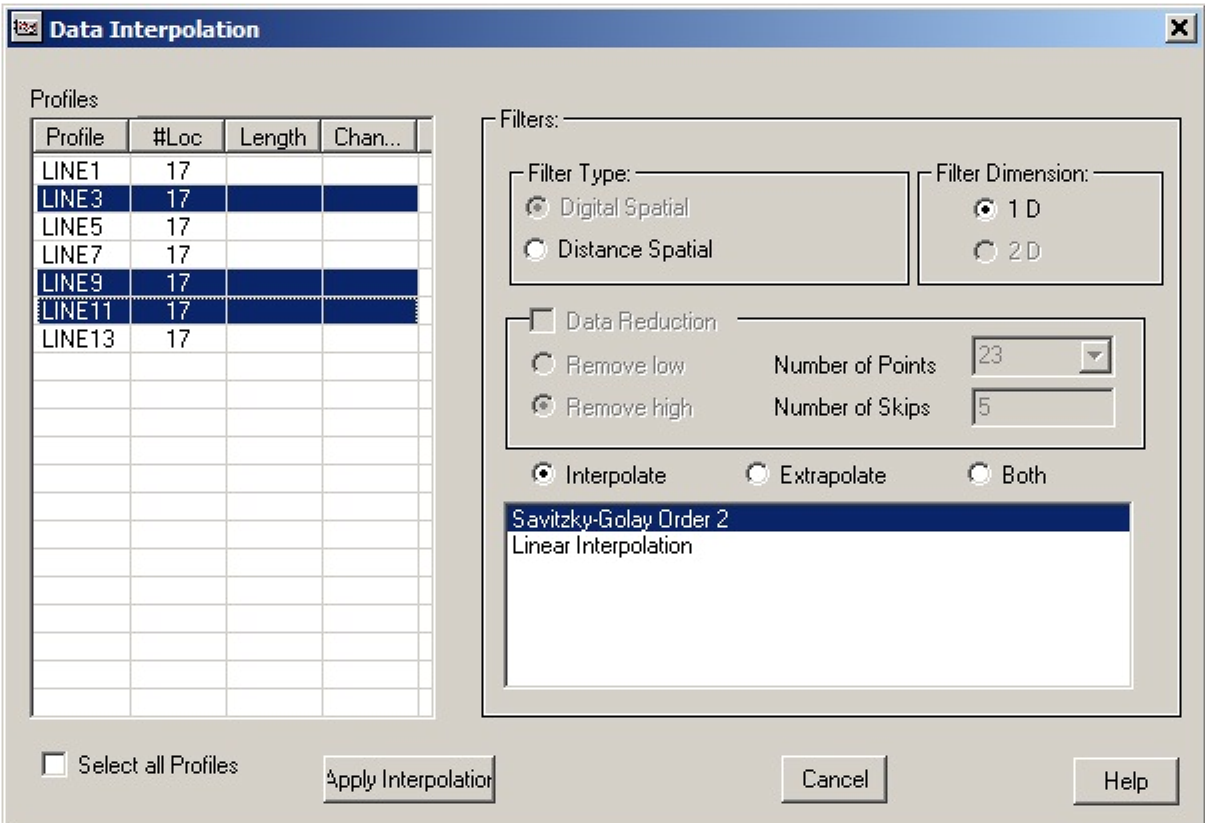
In this interface, you can also choose if selected function will be applied to impedance only (if it is present in the current data set), or to all data.

# Convert Units



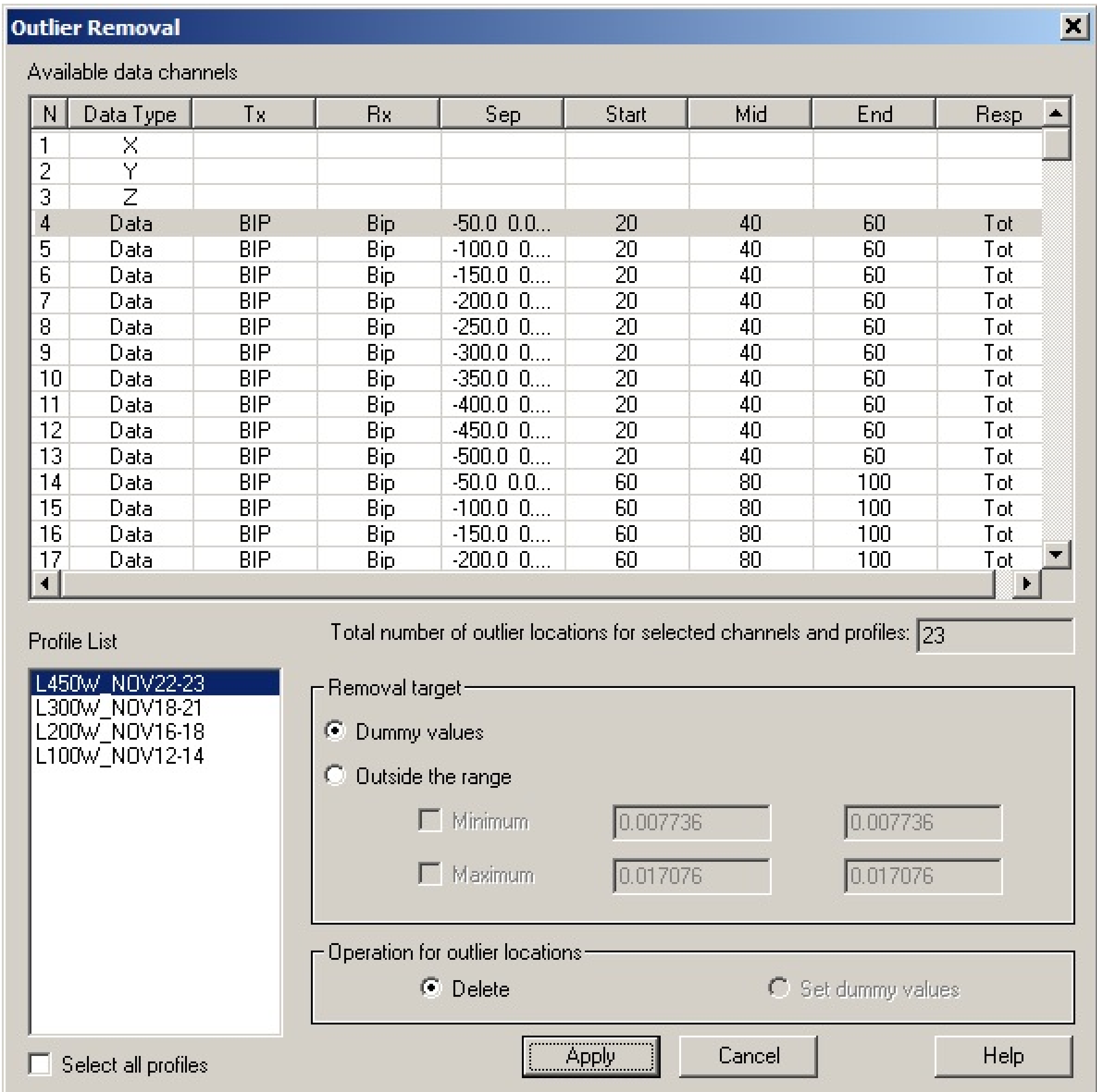

- This operation performs units conversion for certain data. Select desired units from "Convert to" drop down list.
- Click "Apply".

# Data interpolation




- Select one or several profiles, or click "Select all profiles" checkbox.
- Select the "Filter type", then a specific filter from filters list.
- Click "Apply interpolation" to save your data set.

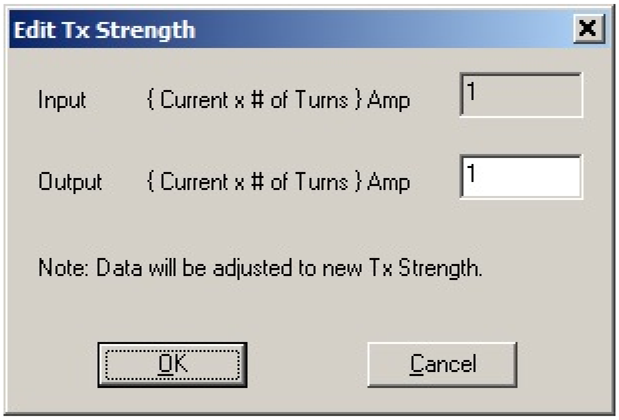
# Data Outlier Removal




- All the available channels are listed in the top window. Impedance data is loaded by default when it is available. Uncheck **Impedance Data Channels** if you do not want to process impedance data.
- Select the profiles as well as the channels to operate on from the two lists displayed.
- Set **Removal target** to **Dummy values** if you would like all non data values to be deleted.

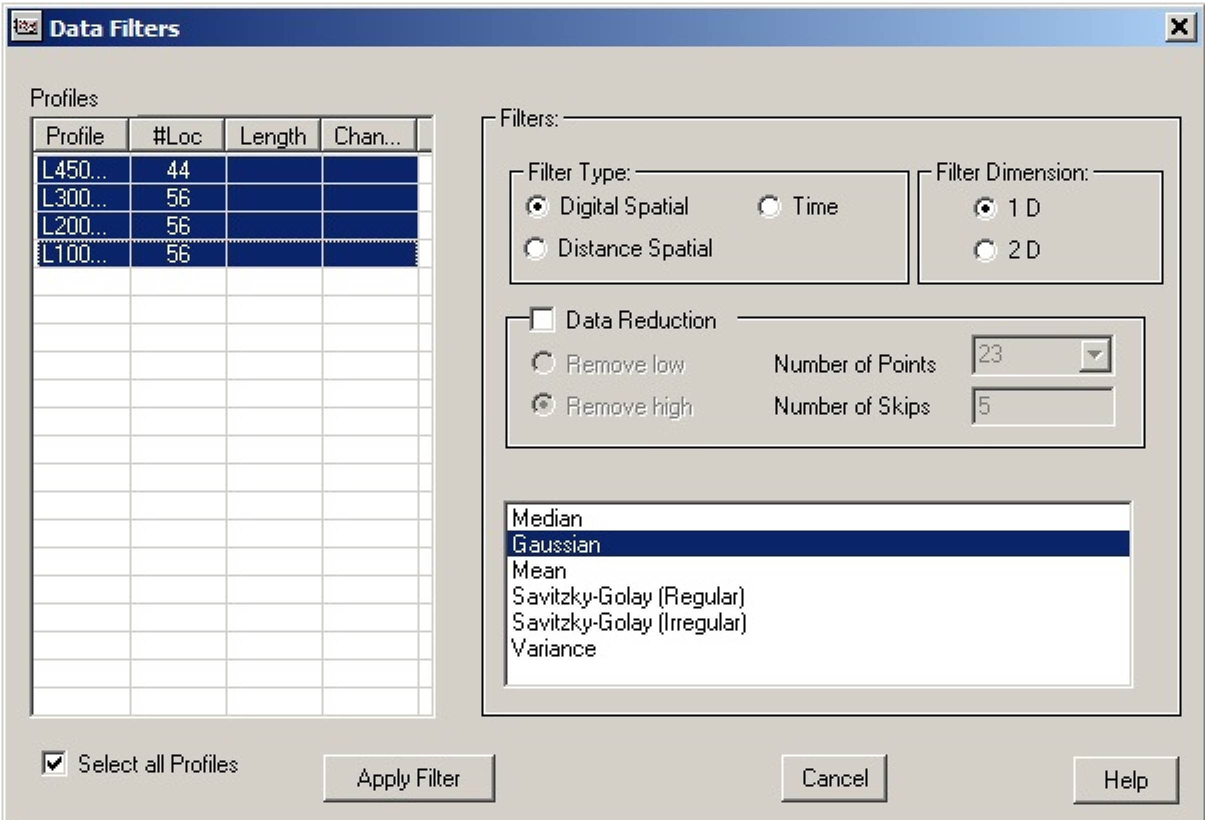
- Select **Outside the range** to specify limits for the data. You can choose from both minimum and maximum limits by activating the appropriate checkbox and then enter the new values. The minimum and maximum data values for the selected channels are displayed beside the user entered limits.
- The data values which fall outside the specified limits can either be deleted or set to dummy values. Choose which selection to perform on the data in the **Operation for outlier locations** section.

# Edit Tx strength




- Set multiplication coefficients for input and output, and click **OK**.

# Filters



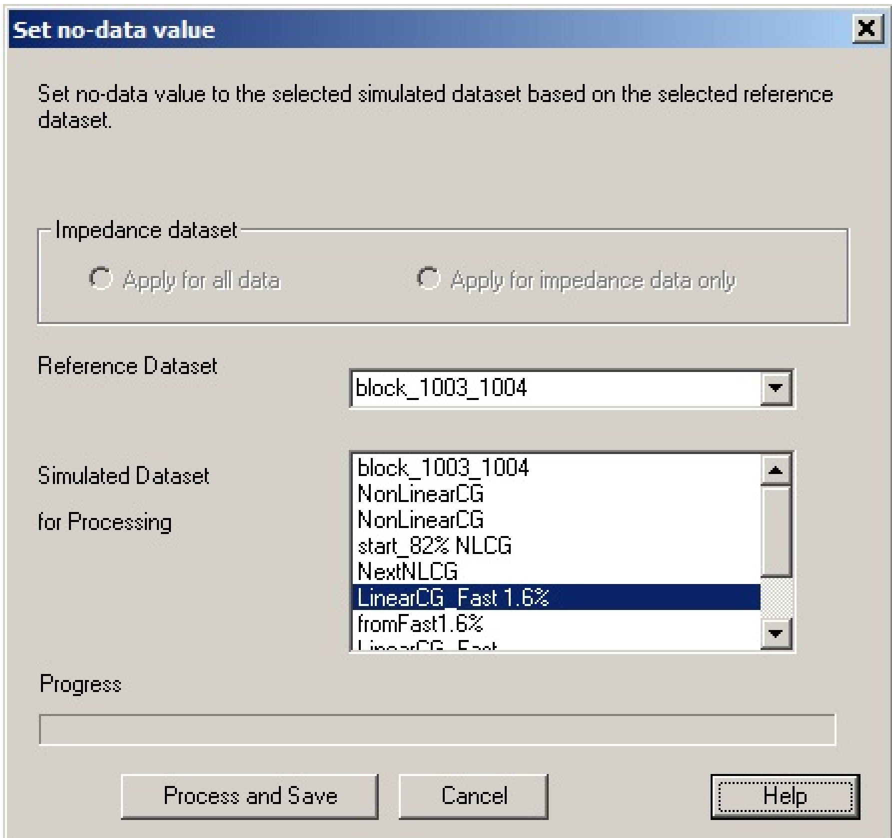

- Select the profiles to be filtered, or click "Select all profiles" checkbox.
- Select filter type using radiobuttons.
- Select specific filter from the filters list.
- Click "Apply filter".

# Gradient Removal

- Removes a 3D data trend. It will be applied to the entire data set.
  To apply to a section of the data, extract the section and create a new data set using the Survey Editor.

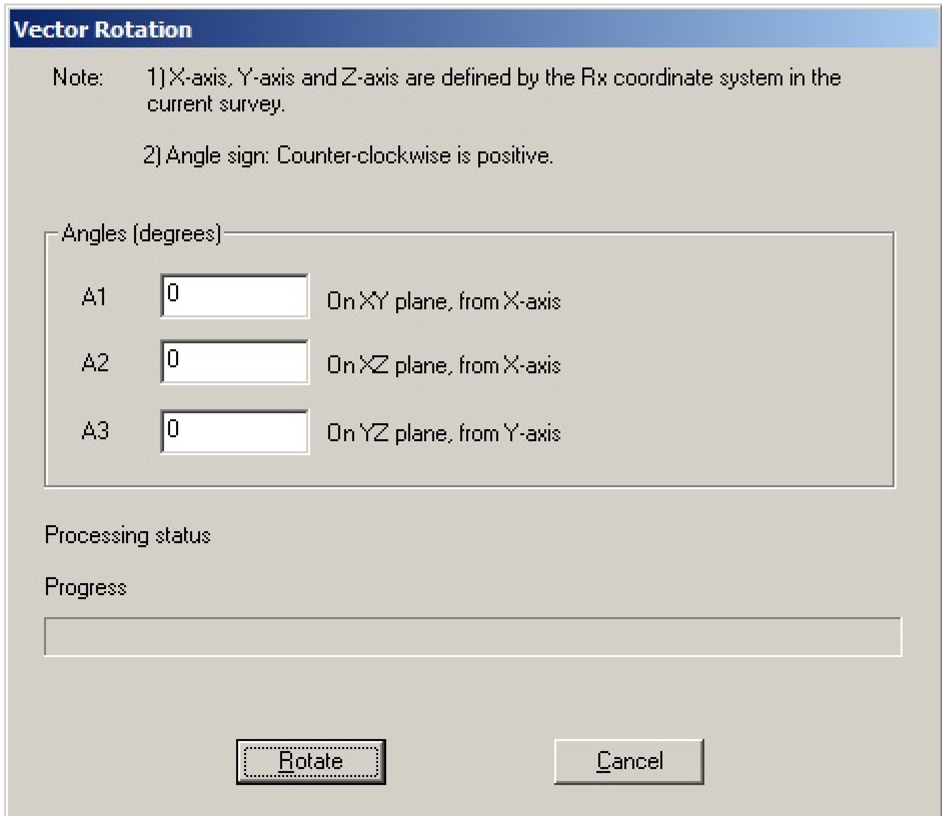
# Set No-Data to match measured data

This function takes 2 datasets and compares their coordinates. All points which have dummy values in the 1st set, will be assigned dummies in the 2nd. The purpose is to cast aside values of those points in simulated set which have no data in the measured source.




- Select 1st reference (measured) dataset from the top list.
- Select 2nd (simulated) dataset from the bottom list.
- If selected dataset contains impedance data, there is a choice: perform dummy replacing for all data, or for the impedance only.
- Click "Process and save".

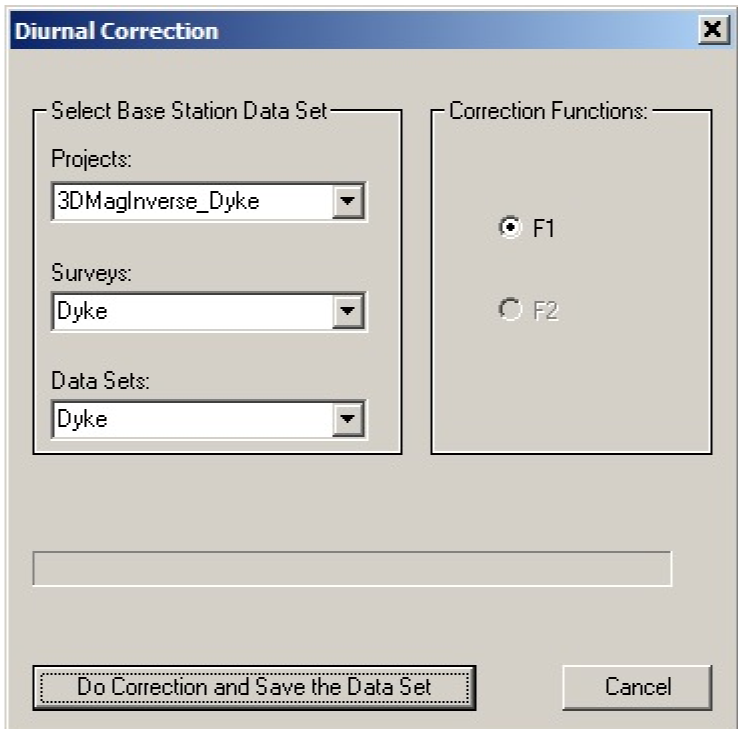
# Vector rotation




- Set rotation angles around 3 axes and click **"Rotate"**.

# Diurnal correction

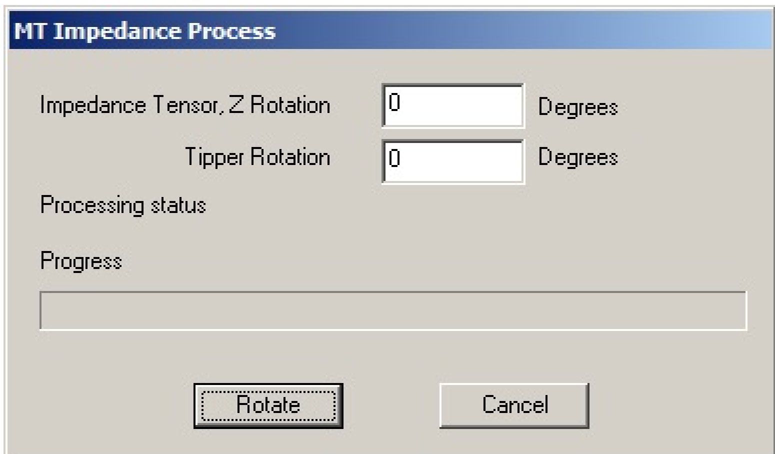
This function is available for magnetic data with FIDUCIAL channel present.




- Select Project, Survey and Data Set in the comboboxes, and click "Do correction and save dataset".

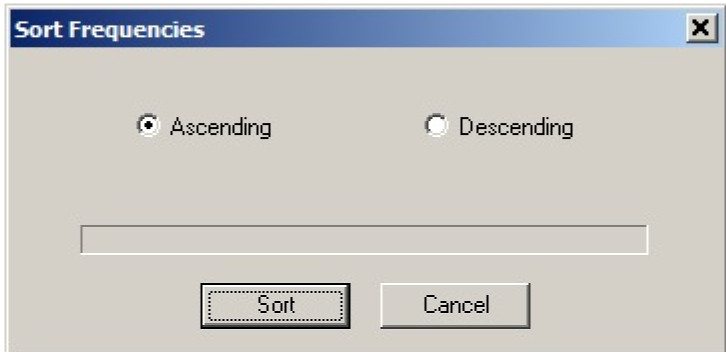
# MT data rotation

Insert values for rotation angles for tipper and impedance in the following dialog:




and click "Rotate".

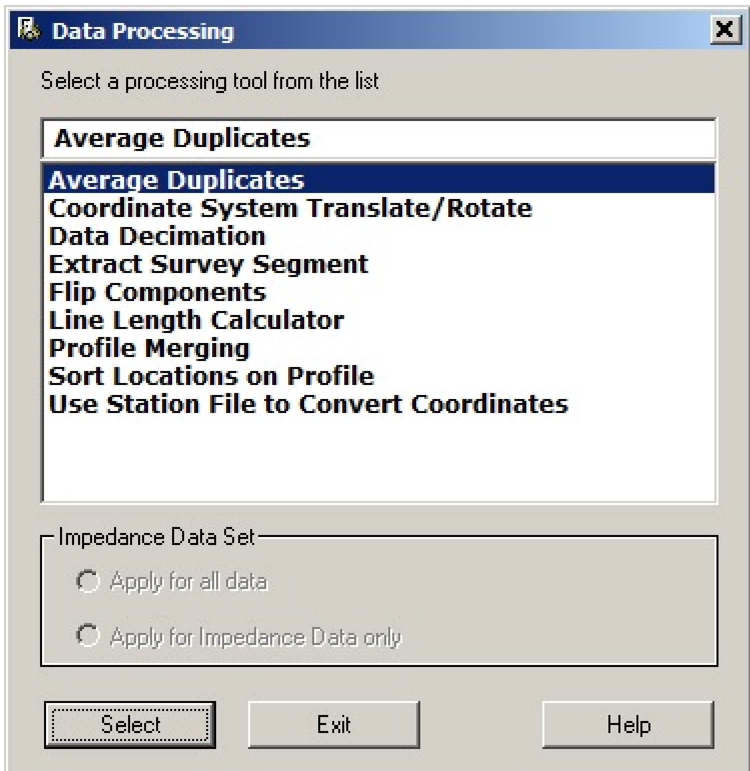
# Sort frequencies




Select "Ascending" or "Descending" sorting order and click "Sort" button.

# Survey editing

After selection "Survey editing", the following list of available functions appears:




- Average duplicates
- Coordinate Translation
- Data Decimation & Profile Merging
- Extract Survey Segment
- Line Length Calculator
- Sort Locations on Profile
-

[**Use station file to convert coordinates**]

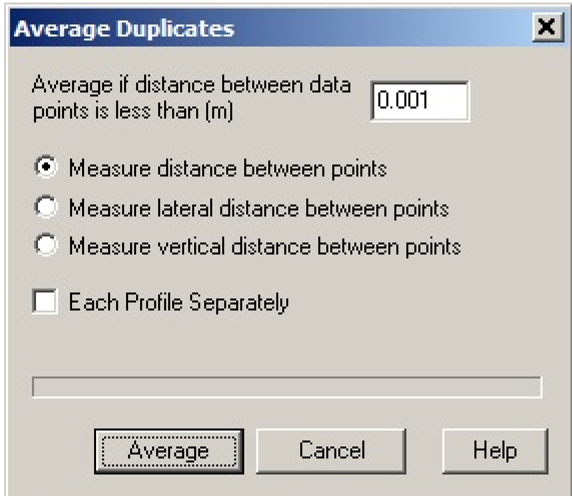
- 

**Convert to static resistivity/MMR**
Removes all time windows except on-time window. For frequencies, this function removes all frequencies except first one.

- 

**Flip components**
Switches the Y component data to X component data and vice versa.

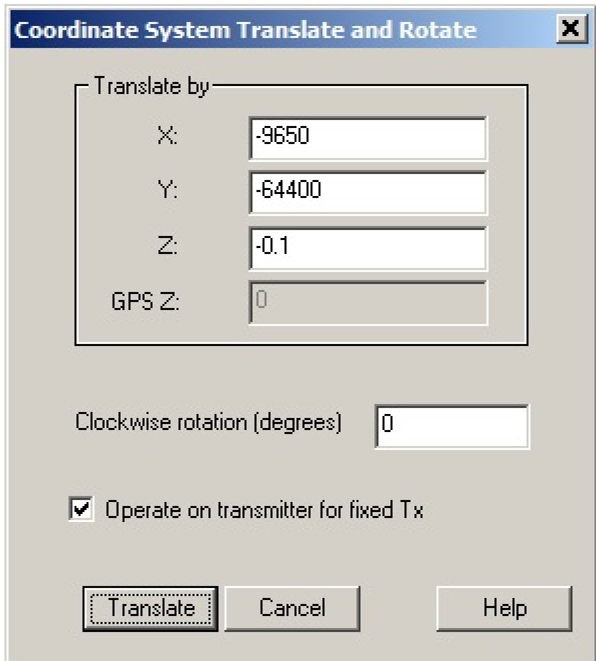
# Average Duplicates




- This operation searches for duplicated locations and replaces them with a location having an averaged data value.
- Specify how far apart locations need to be to be considered separate locations.
- Activate checkbox labelled **Each Profile Separately** to make sure data from two different profiles are not averaged.

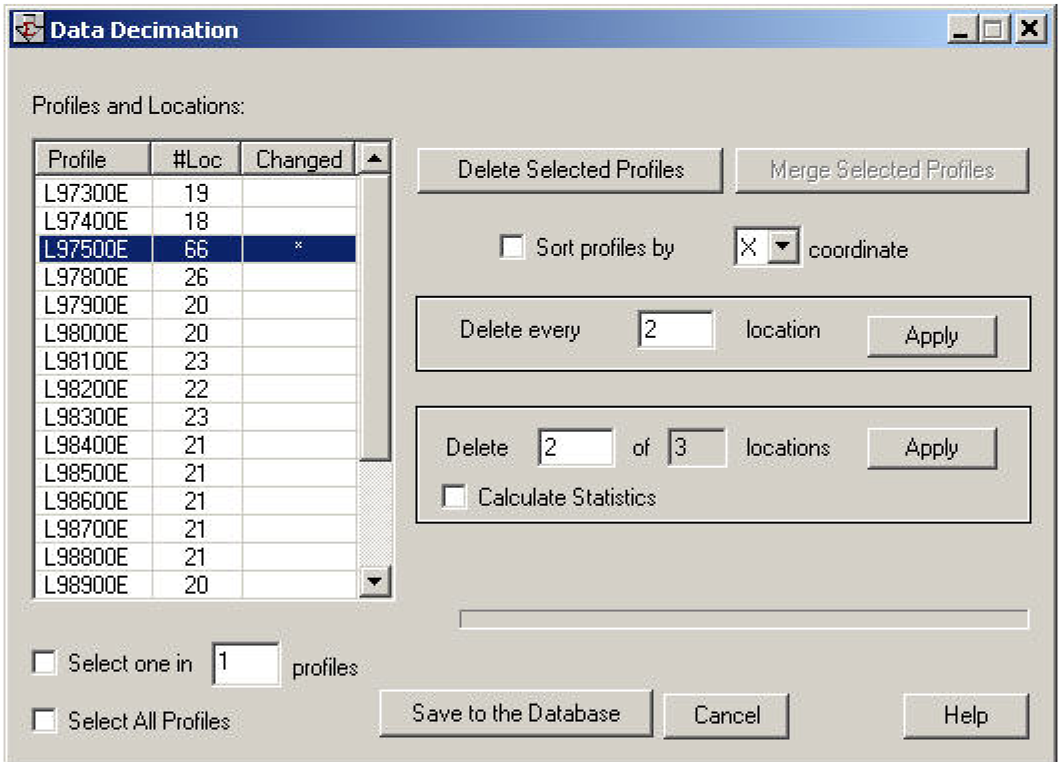
# Coordinate Translation

This function allows to perform coordinate system shift and rotation.




- The values entered for **Translation to new Origin** will be added to the current location coordinates (shift). The coordinate system can also be rotated by entering a value in degrees in the last box.
- If "Operate on fixed transmitters Tx" box is checked, the same coordinate transform will be applied to the transmitters.

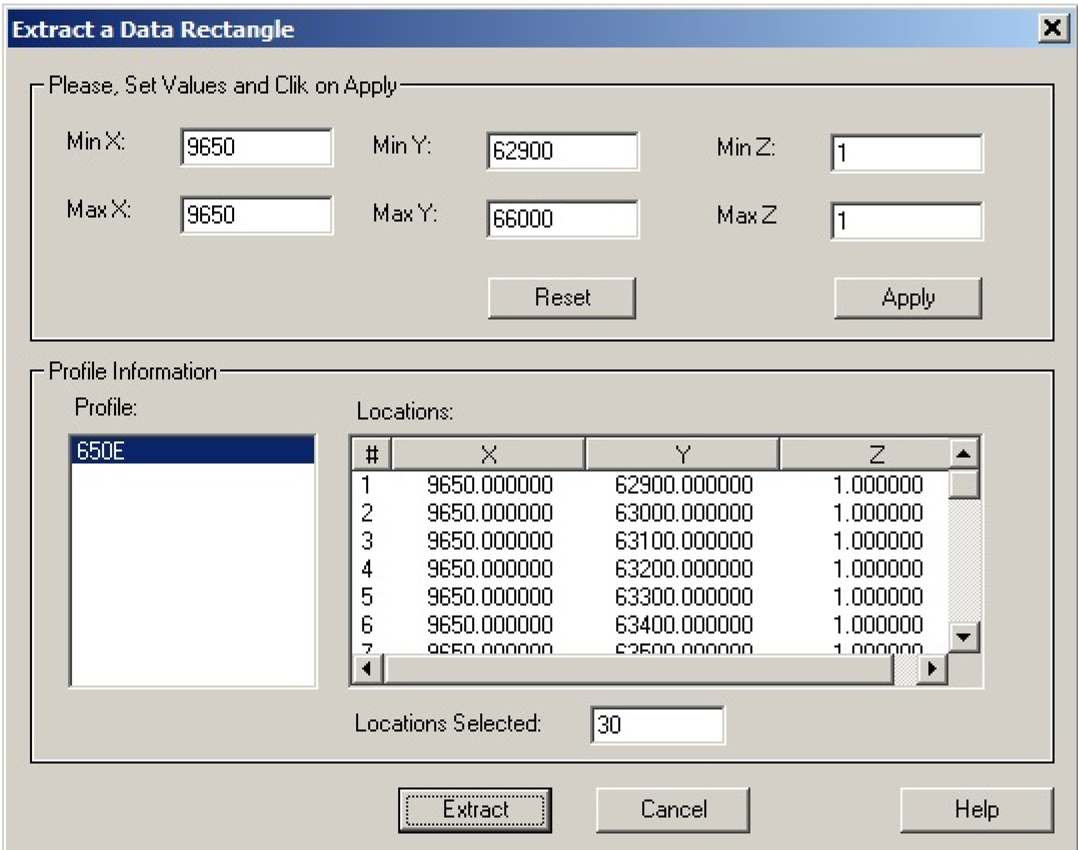
# Data Decimation & Profile Merging




- All profiles are selected by default. Click on the profile list to make a different selection or select the **Select one in ...** checkbox to select regularly spaced profiles
- If you have more than one profile selected, you may click **Merge Selected Profiles** to join them together. The name of the first selected profile will be used. The value in the **#Loc** will be updated and an asterisk will appear in the **Changed** column.
- Select a profile and click **Delete Selected Profiles** to delete it.
- Delete locations in the selected profiles by entering a value for **Delete Every _ Location** and click **Apply**. e.g. entering 3 will delete every third location.
- Delete locations the opposite way by entering a value for **Delete _ of _ location** and click **Apply**. e.g. entering 4 will keep every fifth location.
- Check off **Create Statistics** to add channels for estimated error. These added channels can be viewed in other tools such as EikPlot by switching the Data Type from Data to Estimated Error.

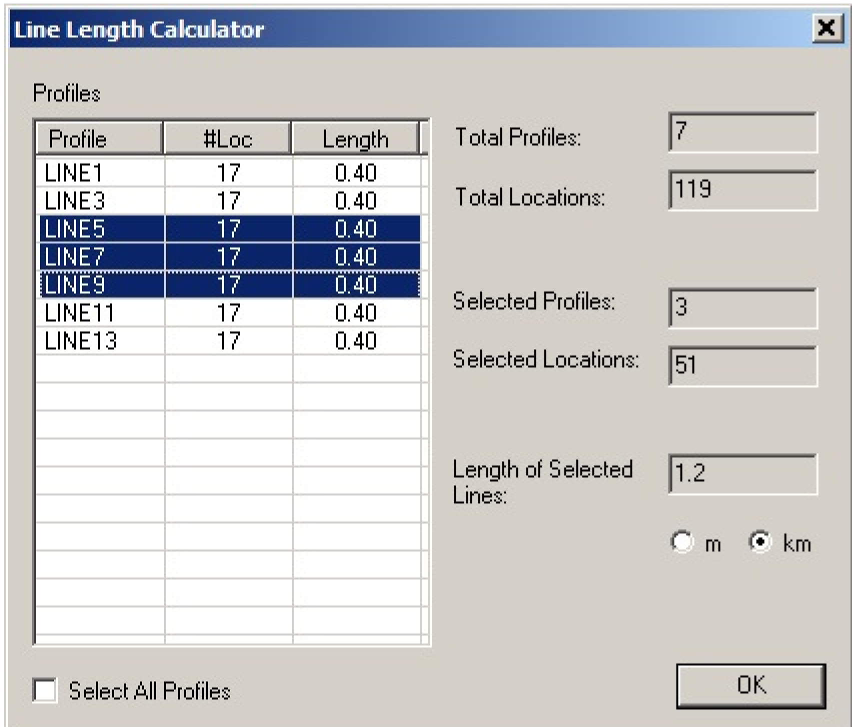
- Profiles can be sorted by x, y, or z coordinate by selecting the **Sort profiles by...** checkbox.
- When your profiles are set up the way you would like, you may save your changes by clicking the **Save to the Database** button.

# Extract Survey Segment



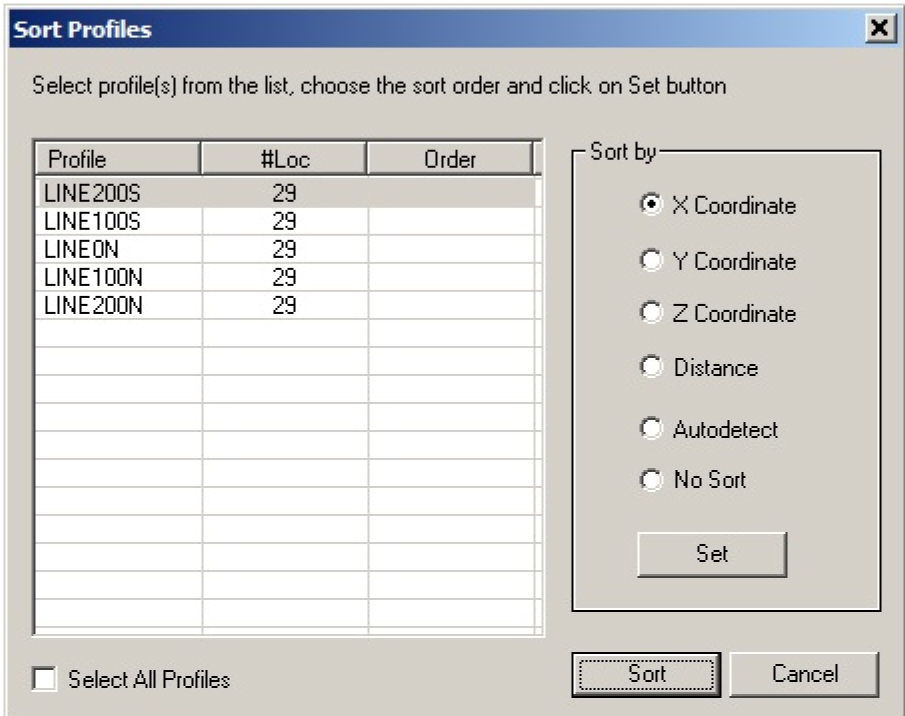

- A 3D section can be extracted from the selected data set by specifying the minimum and maximum x, y and z values. The original values can be restored by clicking **Reset**.
- The **Profile Information** section displays the locations that will be in the new data set.
- Click **Apply** to update the **Profile Information** then click **Extract** to create the new data set.

# Line Length Calculator




- Select one or several profiles, and their total length and number of locations will be displayed.

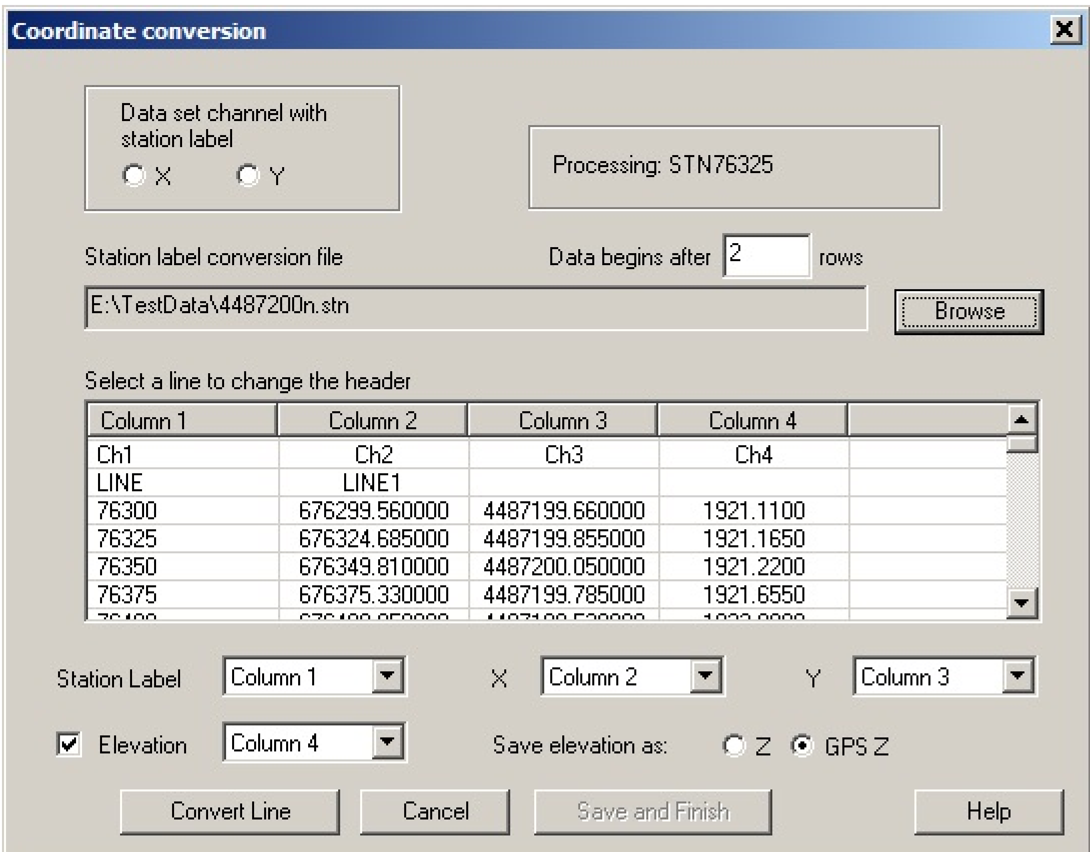
# Sort Locations on Profile




- Select the profiles to work on. Click **Set** to specify the parameter by which to sort. You may sort by a different parameter for each profile. Click **Sort** to create the new survey.

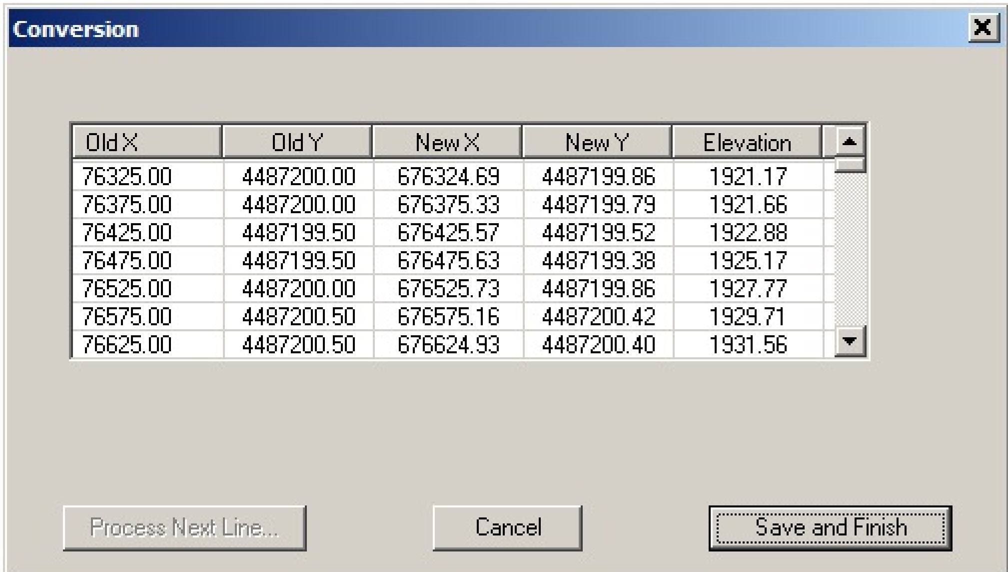
# Use station file to convert coordinates

Use this function to perform coordinate calculation based on recorded station locations.




- In the box labelled **Data set channel with station label**, choose which coordinate is used for the station label in the data set saved in the EMIGMA database. When the station label is found in the station file, the x and y values in the station file will replace the x and y values in the data set.
- Click "Browse" button and select station file (usually, it has .stn extension).
- If station file has a header, set how many lines to skip in **Data begins after...rows** field.
- In fields **X**, **Y**, **Elevation** select respective column numbers. In **Save elevation as**, select **Z** or **GPS Z** column header.

- Click **Convert line** button.
- Look at conversion results in the window displayed:

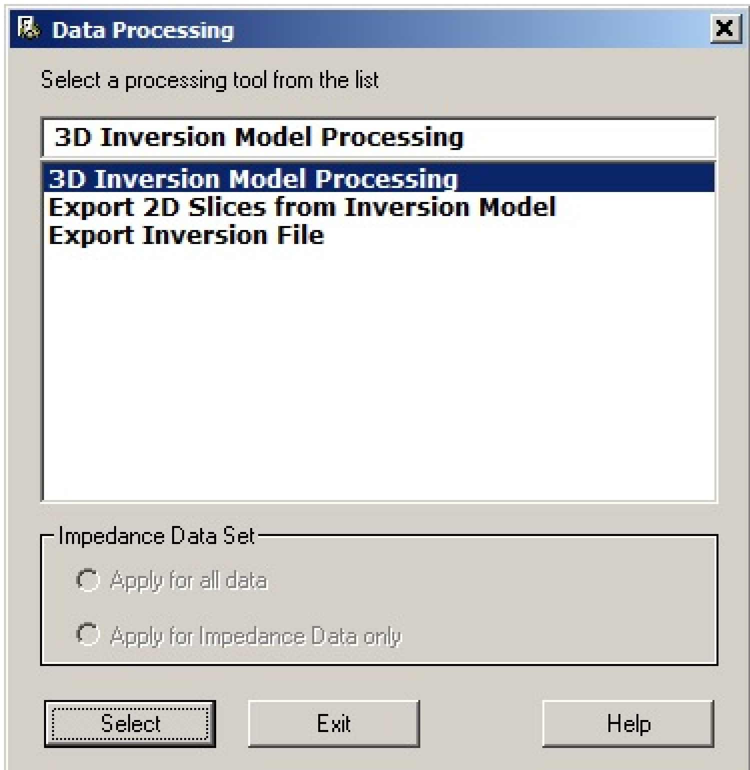



| Old X | Old Y | New X | New Y | Elevation |
|---|---|---|---|---|
| 76325.00 | 4487200.00 | 676324.69 | 4487199.86 | 1921.17 |
| 76375.00 | 4487200.00 | 676375.33 | 4487199.79 | 1921.66 |
| 76425.00 | 4487199.50 | 676425.57 | 4487199.52 | 1922.88 |
| 76475.00 | 4487199.50 | 676475.63 | 4487199.38 | 1925.17 |
| 76525.00 | 4487200.00 | 676525.73 | 4487199.86 | 1927.77 |
| 76575.00 | 4487200.50 | 676575.16 | 4487200.42 | 1929.71 |
| 76625.00 | 4487200.50 | 676624.93 | 4487200.40 | 1931.56 |

If results are satisfactory, click **Process next line** or **Save and finish**.

# Inversion model processing

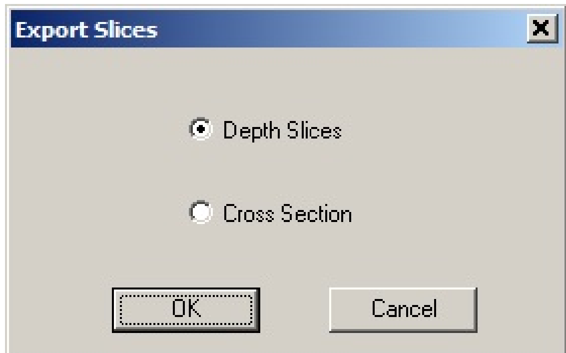
Select function from the following list:



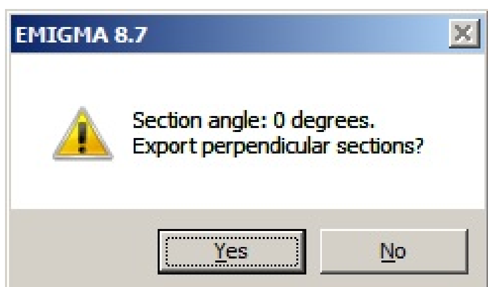

- [Export 2D slices from inversion model]
- [3D inversion model processing]
- [Export inversion file]

# Export 2D slices from inversion model

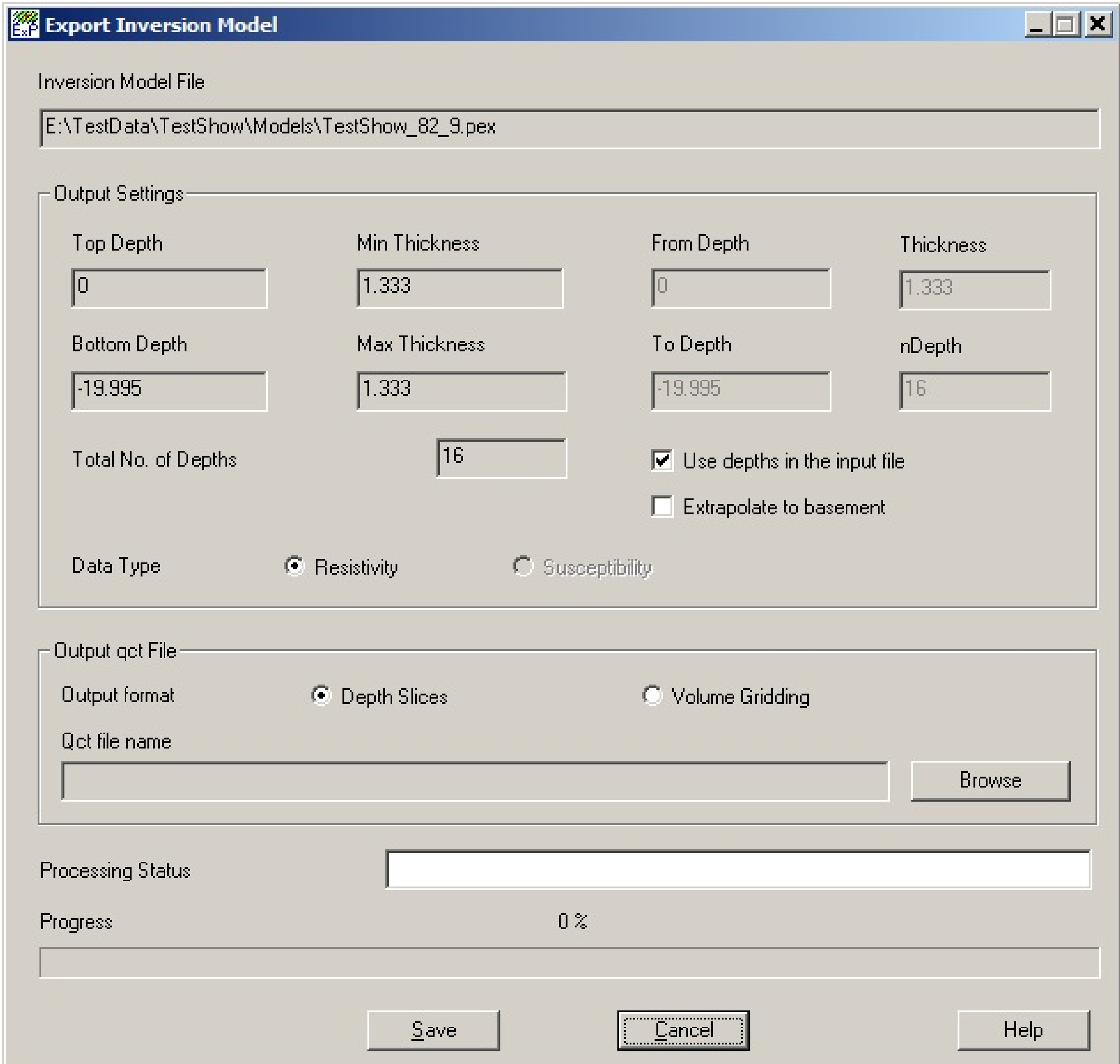
Select slice orientation in the initial window:




Vertical cross-section slices can go in the direction of survey lines, or perpendicular. Choose this option in the next window:




After a selection of "Depth slices", the following window appears:

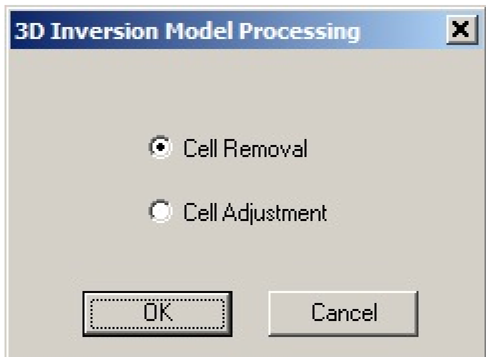



- If the "Use depth in the input file" checkbox is unchecked, slice definition fields "From depth", "To depth", "Thickness", "nDepth" become editable. The thickness of one slice and number of slices (nDepth) are calculated from each other, so the user can edit either of them.
- Select "Resistivity" or "Susceptibility" if both types of data are present.
- If "Extrapolate to basement" is checked, dummy data values at the lowest depth will be substituted with the closest real data.
- Select "Depth slices" or "Volume gridding". For "Depth slices" ,each horizontal level will be written as a separate channel, which name

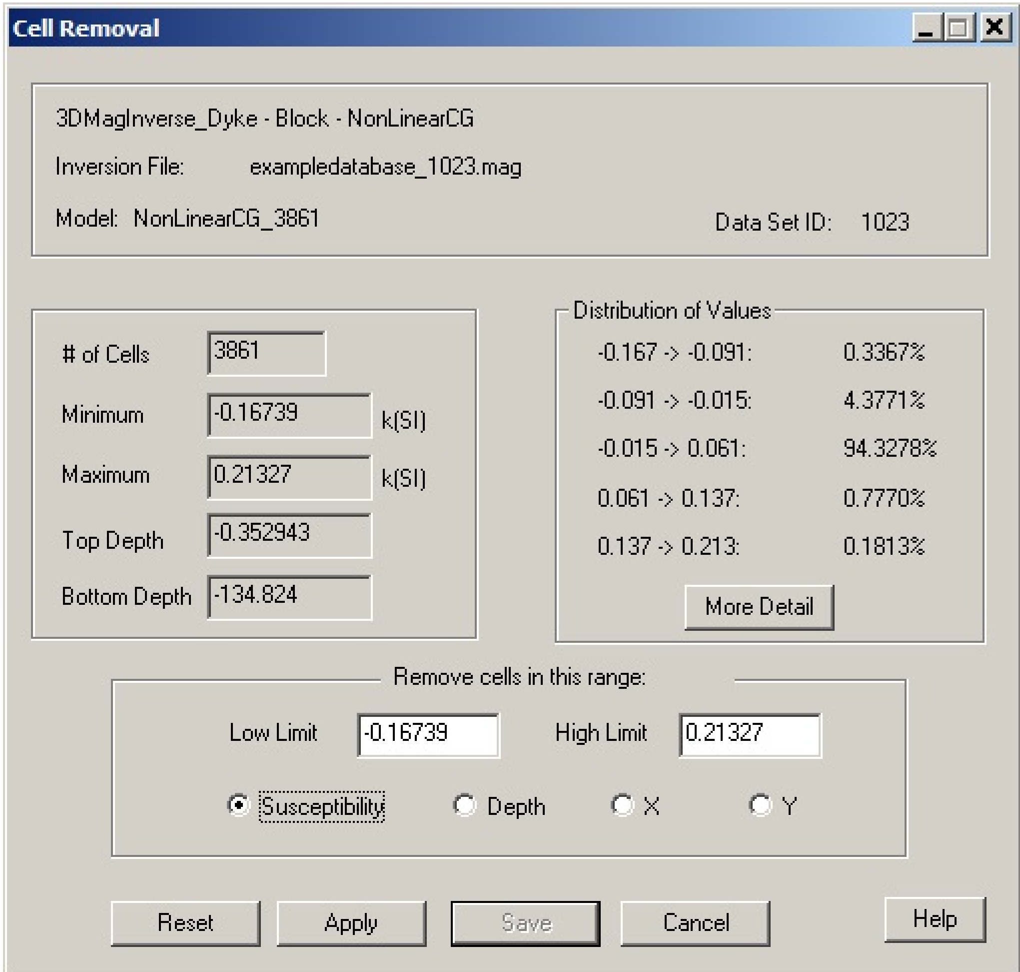
includes respective depth. For "Volume gridding", output.qct file contains "Depth" and "Resistivity" (Or "Susceptibility") channels.

# 3D inversion model processing

Select "Cell removal" or "Cell adjustment". "Cell removal" mode allows to filter out cells in certain data ranges. "Cell adjustment" mode reassigns different values to the cells in certain data ranges:

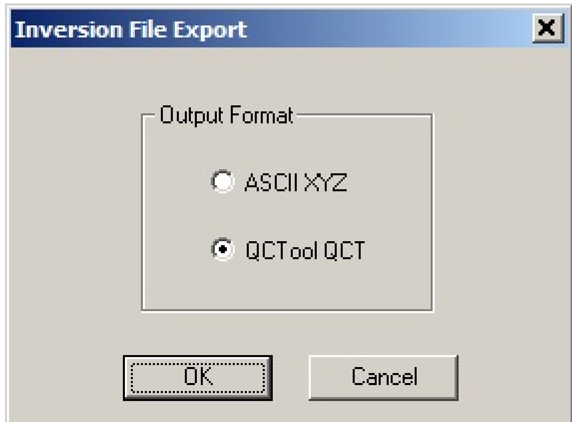



The following window then appears:



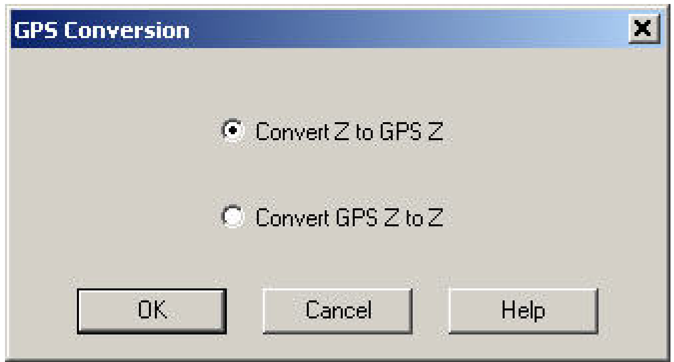

- You can work with different value ranges depending on whether "Susceptibility", "Depth", "X" or "Y" is selected.
- Clicking the "Apply" button will update the "Distribution of Values".
- By clicking "More detail", see an extended list of value distribution.
- Clicking "Reset" will discard all filtering.
- Click "Save" to create a new data set with your changes.

# Export inversion file




A 3d inversion result can be exported to xyz or qct file format.

# Adjusting to GPS elevations

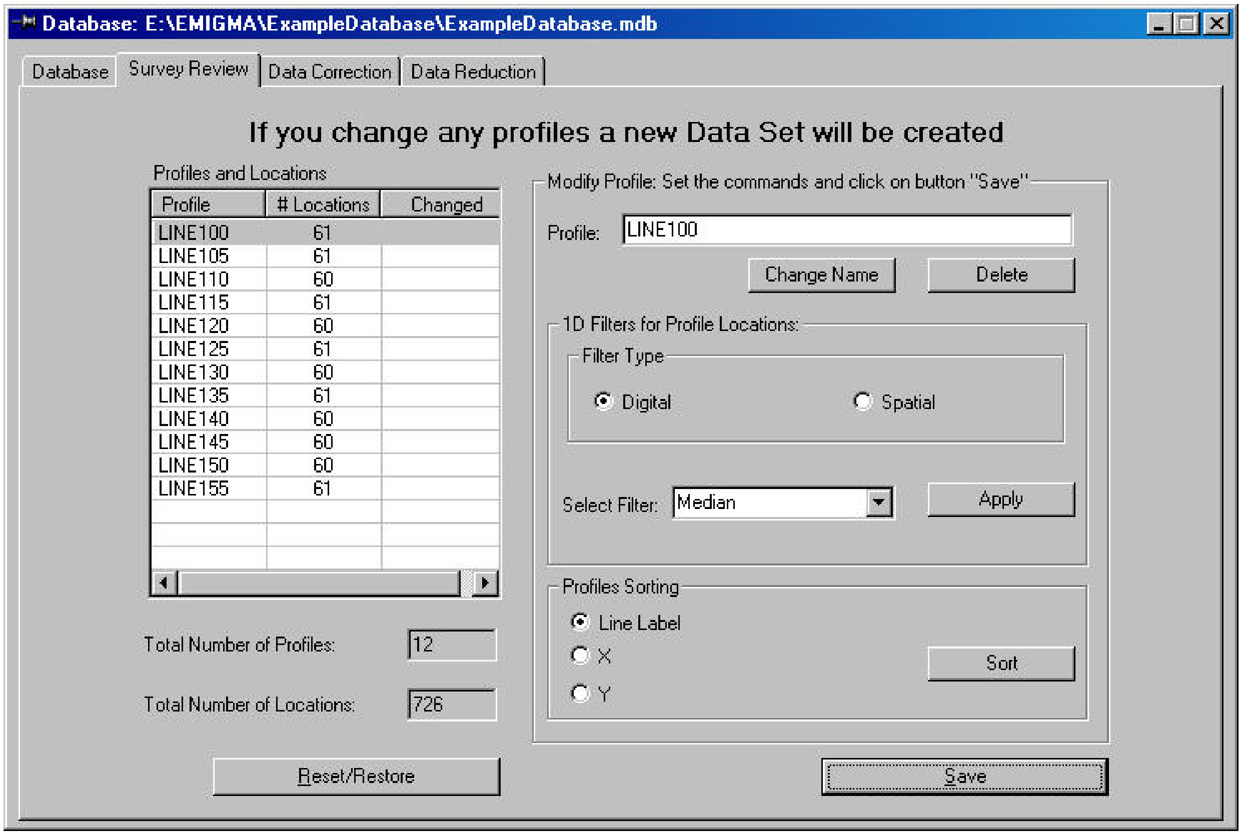



If the related data set for your 3D inversion result contains a gps z channel, you may convert the z values for the 3d inversion cell centres to be corresponding gps z values. You may also convert in the opposite direction from gps z to z by making the appropriate selection on the interface and clicking **OK**. You will see a new data set with the modified 3D inversion result added to the current survey.

# Database Tabs

## Survey Review

Click on the **Survey Review** tab on the Database dialog to reach this tool:




Use this tool to make changes to the profiles in a survey.

- Click on the profile you would like to modify in the Profiles and Locations list.

To change names:

- Edit the name that appears beside the Profile label and click Change Name. Click Save to keep your changes
- Click Delete to delete the profile. Click Save to keep your changes.

To sort profiles:

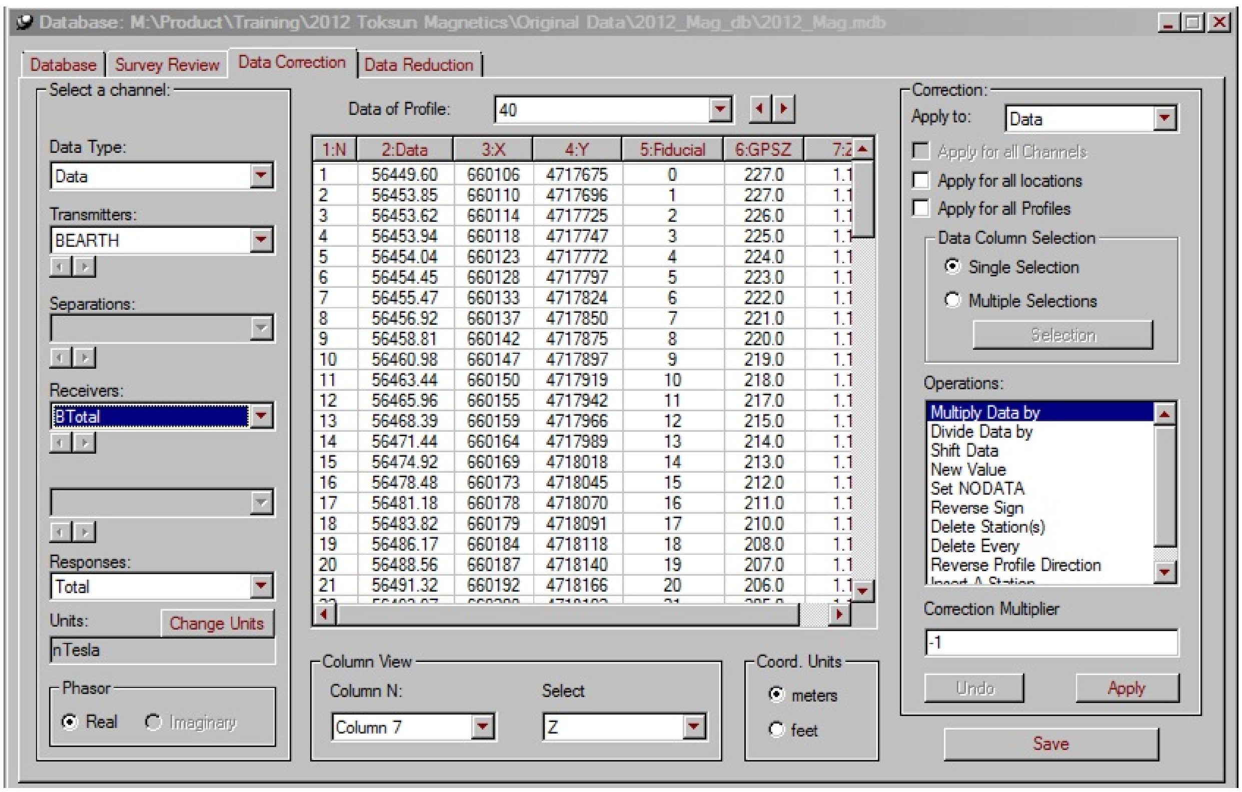
- Select the sorting index and click the Sort button.

To apply 1D Filters to the locations:

- Select either a digital or spatial filter type. Select the filter from the drop down list and click Apply.

## Data Correction

Click on the **Data Correction** tab on the main Database window to reach this tool. You may be asked if you wish to load all Profiles. If you wish to make your corrections to all profiles, then select ALL otherwise select the appropriate profiles. If you have a small survey then you can always select ALL but for very large airborne surveys, you may want to select the Profiles that you actually require.




Use this tool to make changes to the data values and coordinates in a data set. The various operations allowed are in the list box to the right.
When correcting data values:

- Select settings in the left column of this page to display the specific data channel you would like to view in the Data column.

In some cases, you may wish to eliminate the data without deleting the station. In this case, use **Set NODATA** for the stations and data channels that you wish.

In some cases, you may not have an adequate station sampling to fully understand the model response. To insert new stations which will have dummy measured data but for which the modeled response will appear use **Insert a Station**.

Transmitters, Receivers, Frequencies, Time Windows and Separations may be deleted on this page.

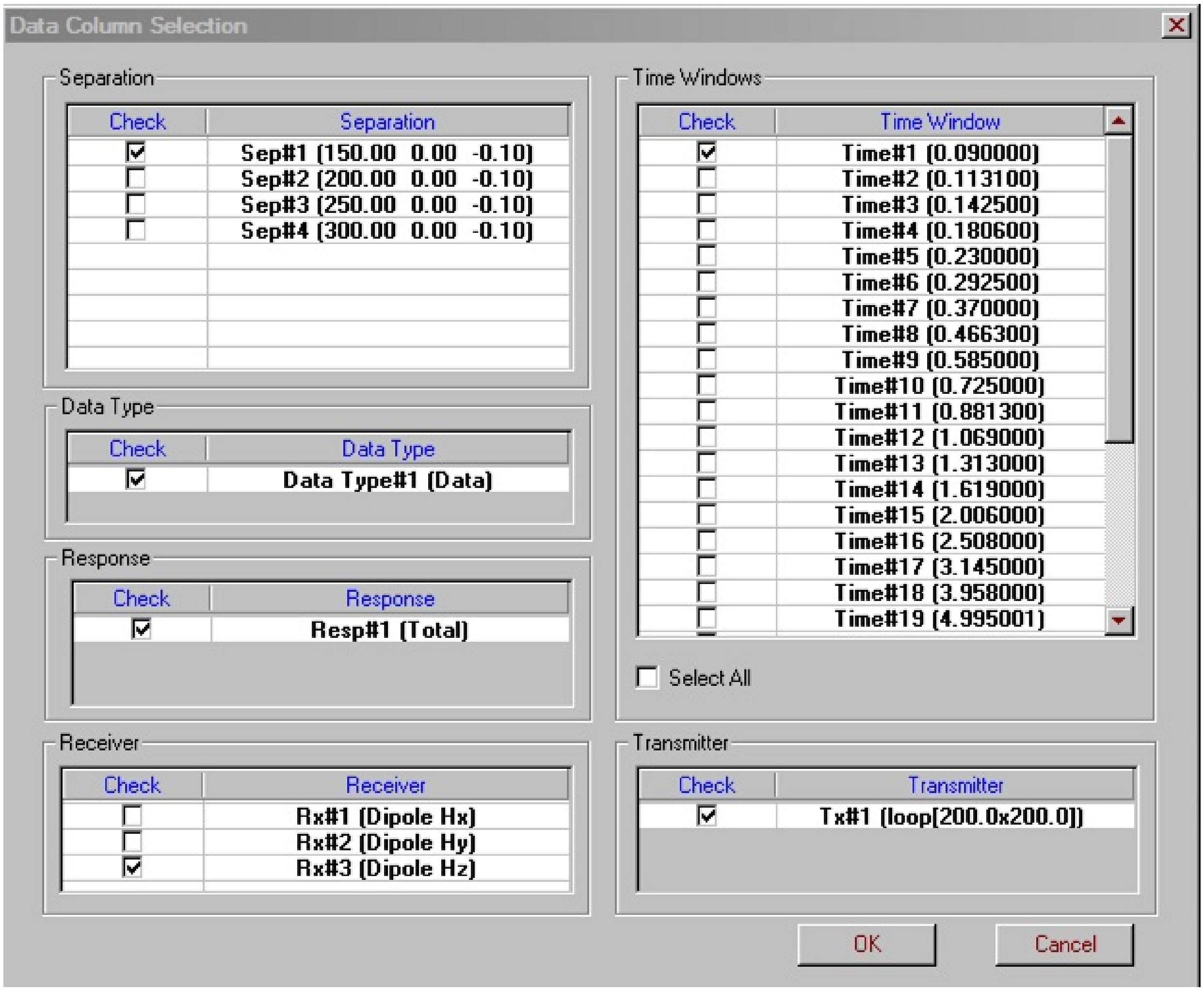
To reassign the column order:

- Select the column number from the Column N drop down list
- Assign a new set values to a column using the Select list to access other channels such as Fiducial or GPSZ

To correct data:

- Specify units as meters or feet in the Coord. Units box
- Choose which column you would like to work on in the Apply to box
- Specify whether the changes will be applied to a single column or multiple columns

You may apply changes to multiple columns by selecting the **Multiple Selections** option, then clicking the **Selection** button. The following window appears:

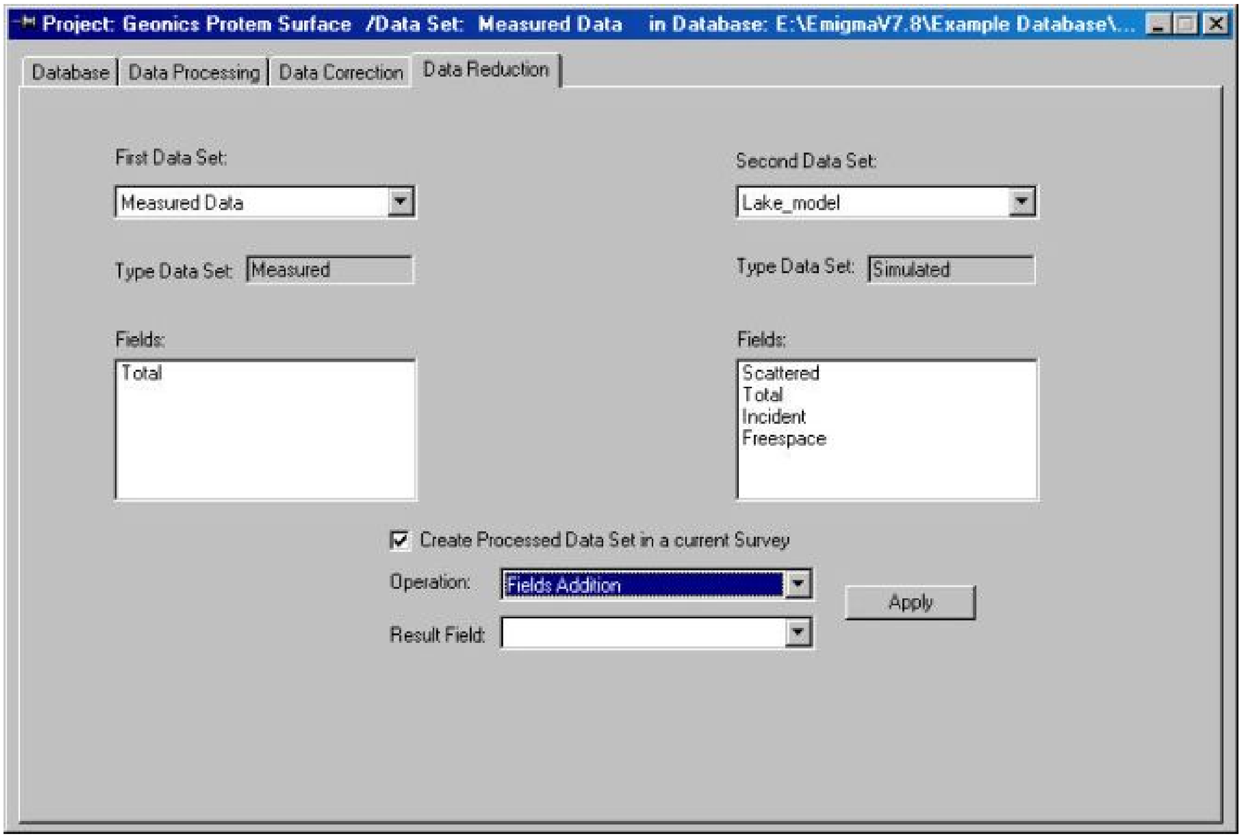



Select the data columns to which you would like apply changes then click **OK**

- Select if the operation will be applied to all locations or all profiles or a specific location.
- Select an operation from the Operations list to perform on the column you have chosen. Specify the Correction Multiplier if necessary. Click Apply to initiate the operation. The operation is reversible by using the Undo button.
- Click Save to apply these changes.

## Data Reduction

Click on the **Data Reduction** tab on the Database dialog to reach this tool:




Use this tool to perform operations using two data sets. All the data sets in the selected survey are available to be selected from in the two boxes labelled **First Data Set** and **Second Data Set**.

- Select the field to use for each data set in the boxes labelled **Fields**.

Define what the resulting field will be in the box labelled **Result Field**.

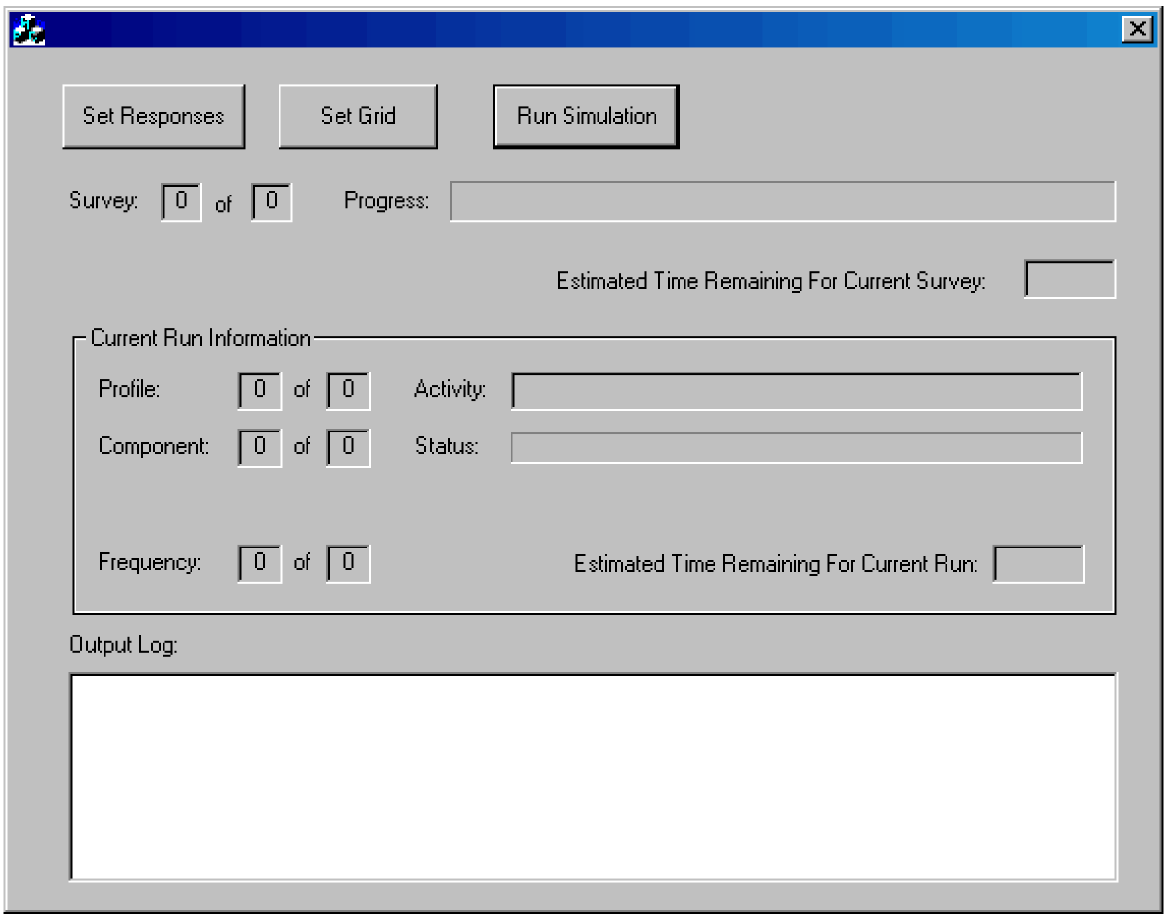
- Select the operation to perform on these two data sets from the Operation box.
- Uncheck **Create Processed Data Set in a current Survey** if you would like the new data set to created in a new survey.
- Click **Apply** to save the result in a new data set.

# Source Distribution

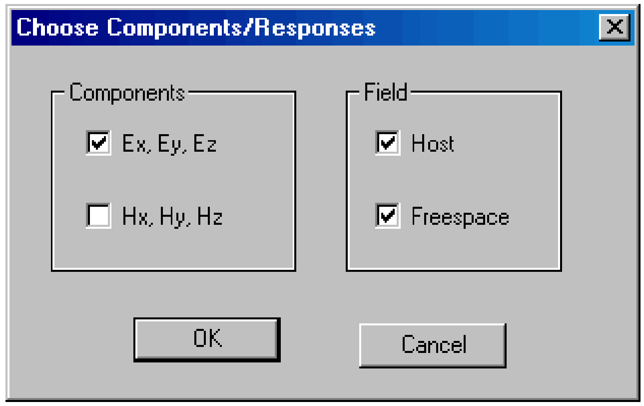
A 3D grid of vectors describing the em field associated with a survey's transmitter can be generated for viewing in the Visualizer.

These fields can be due to both the primary source or the secondary source (i.e. Background layers)

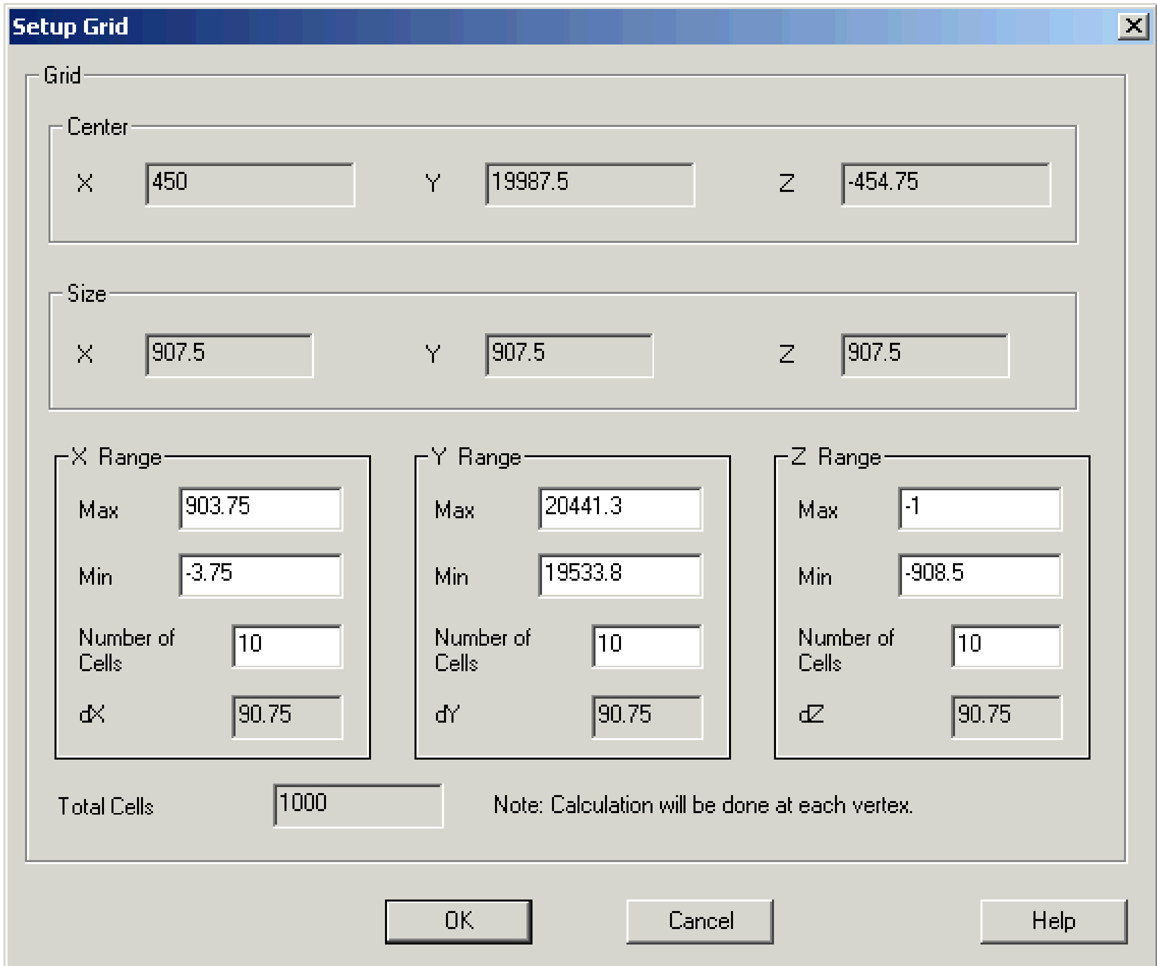
- Select a simulated data set from the main database window.
- Click the  on the main toolbar and the following window appears:




- Click the **Set Responses** button and the **Choose Components/Responses** window appears. Select from the available options of magnetic and electric components in the **Components** section. In the **Fields** section, choose from the **Host** and **Freespace** fields to be generated.

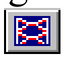


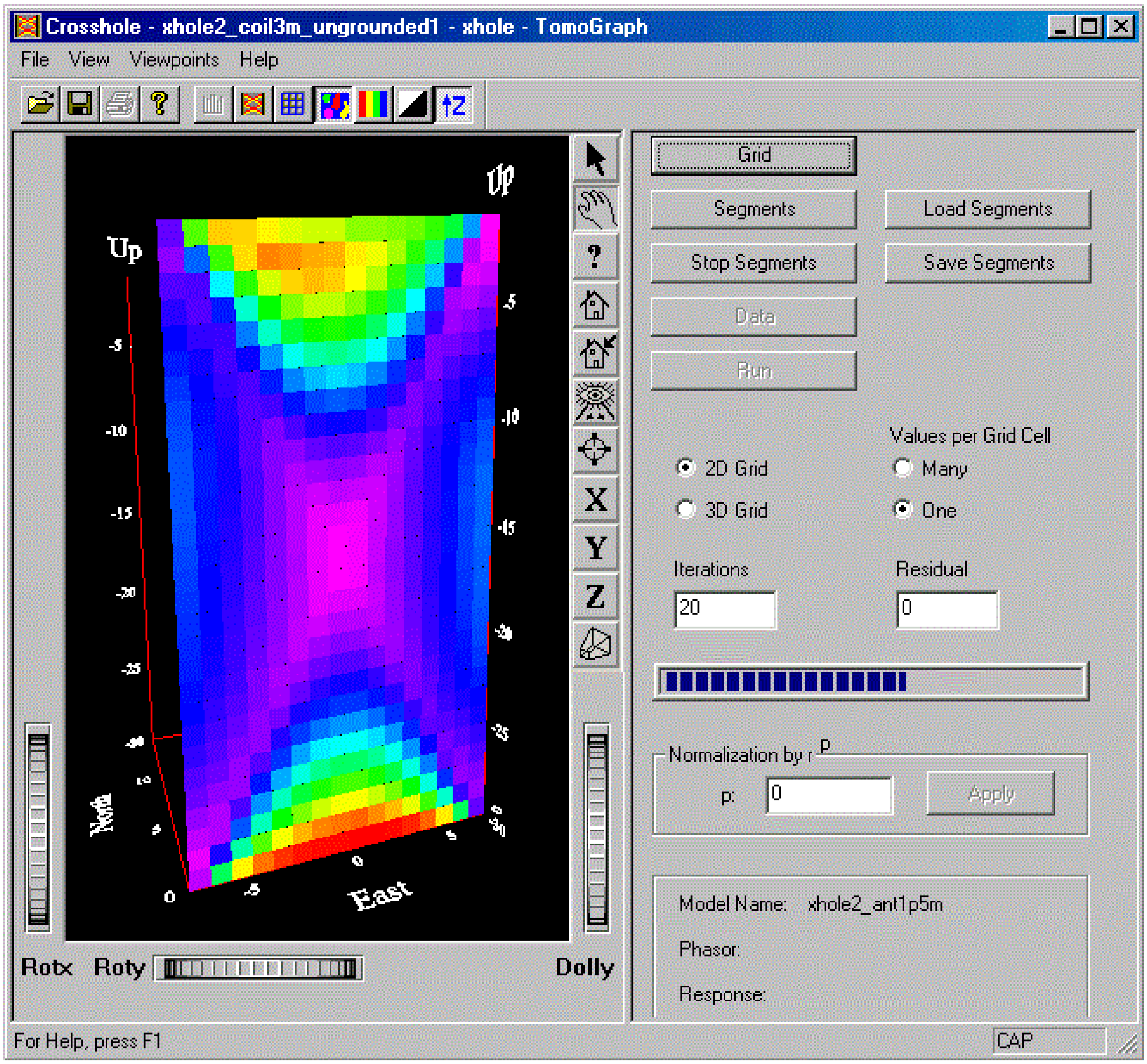

- If time domain data, you will first see the **Spectral Settings** window which by default will be inherited from your simulated dataset
- Click the **Set Grid** button to specify the size and location of the 3D grid and distribution of positions where the field vectors will be generated.




- Click **Run Simulation** and a new data set will be available in the main database window with a grid attached named "Learth Simulation".
- Select the new data set and launch the Visualizer from the main toolbar. Click the button on the Visualizer toolbar to view the result of the processing. Further information on the Visualizer interface can be viewed here.

# Tomography

- Select a cross borehole data set. This type of data set can be identified by viewing the Data Set Info window.
- Click the  button on the main toolbar and the following window appears:



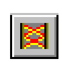
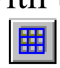

- Click the **Grid** button and specify the number of grid cells to be used for the tomography.

- Click the **Segments** button to calculate the length of the ray trace lines in each grid cell. This step can be interrupted by clicking the **Stop Segments** button.
- Click **Data** to specify the response, normalization type and channel which will be used.
- Specify the number of **Iterations** in its box and click **Run** to perform the tomography process. **Residual** will display the difference between the estimated and measured values as the processing is being performed.
- Segmentation can take a long time, so its result can be saved for repeated use. After segmentation is completed, click button **Save segments** and put a name for .seg file.
- Click **Load segments** to load previously created segmentation .seg file.
- If segmentation procedure became too long, it can be canceled by clicking **Stop segments** button.

### Additional Options

- Select the **3D Grid** option if the ray trace lines cannot be contained in a 2D plane.
- To disable the interpolation and display only one value per grid cell, select the **One** option under the heading **Values per Grid Cell** and click **Apply**.
- Normalization can be applied by entering a value for p in the box labelled **Normalization by r$^{p}$** and clicking **Apply**. (r is the length of the ray trace line segment.)

### Toolbar Buttons

- Remove the z axis with the  button.
- Toggle the background colour between black and white with the  button.
- Toggle the display of the ray tracing lines with the  button.
- Toggle the display of the grid lines with the  button.
- Toggle the display of the data with the  button.
- Choose the range of data values that should be assigned different colours by clicking the  button. The minimum and maximum values

can be specified by either using the sliders or entering values and clicking **Apply**. The percentage of values which fall between the minimum and maximum values can be viewed in the box labelled **Percentage of Data Displayed**.

- View the data values by clicking on the  button. Move the cursor to the grid cell you would like information on then the location and data value will appear at the bottom of the window.
- Save the data values to an xyz file using the  button.

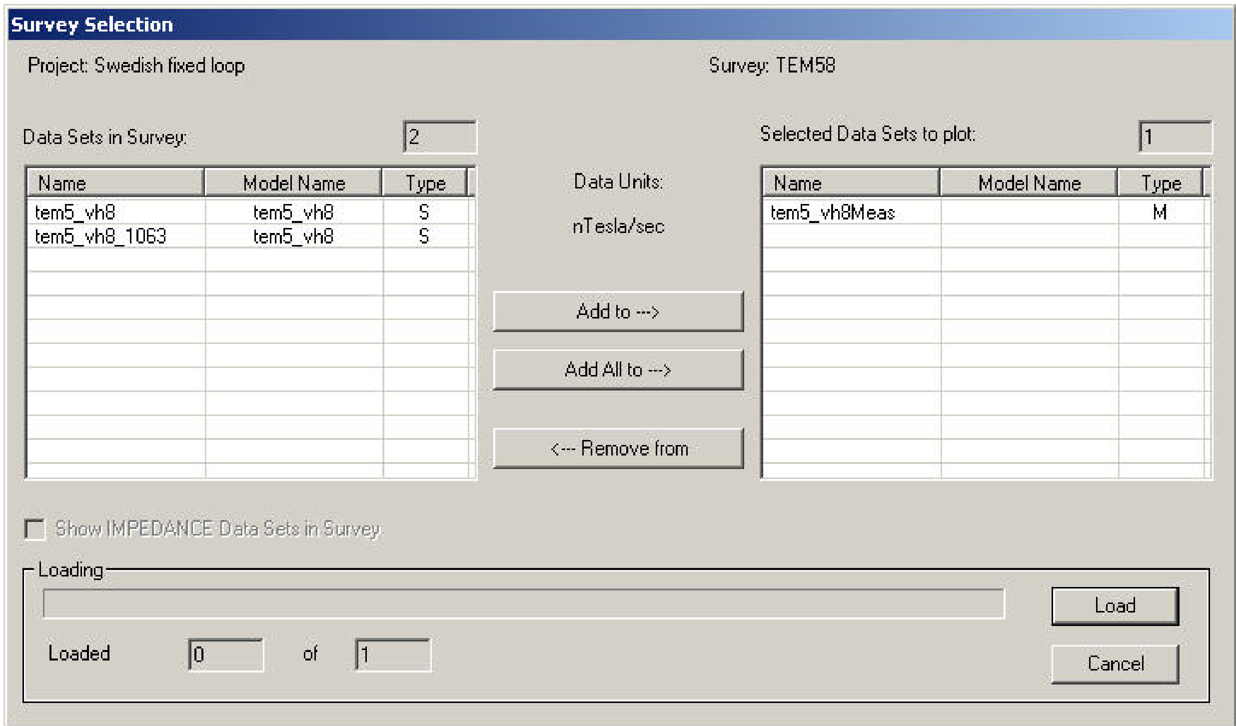
# Graphical Visualization Tools

## Plotting

### Select Data Sets to Plot

To plot your data, click the EiKPlot button  on the main toolbar. If your survey contains several data sets, a message will appear asking you whether you want to compare the current data set with the other data sets in the survey:


1. If you click **Yes**, the **Survey Selection** dialog will open offering you to choose data sets to be compared with your current data set:




- Select a data set from the list on the left and click **Add to**. In case you want to compare all the available data sets, click **Add All to**
- To remove a data set from the list on the right, select it and click **Remove from**.
- Click **Load**

2. If you click **No** in the message box, the initially selected data set will be plotted automatically.
3. To load the settings of a previously created plot, click the **Load Settings** button to open the **Get Settings** dialog, select a required file and settings therein and click **Load** (see **Saving Plot Settings**).

## Plot Static, Frequency- and Time-Domain Data

In the case of static, frequency- or time-domain systems, the first available channel of your data will be plotted automatically.

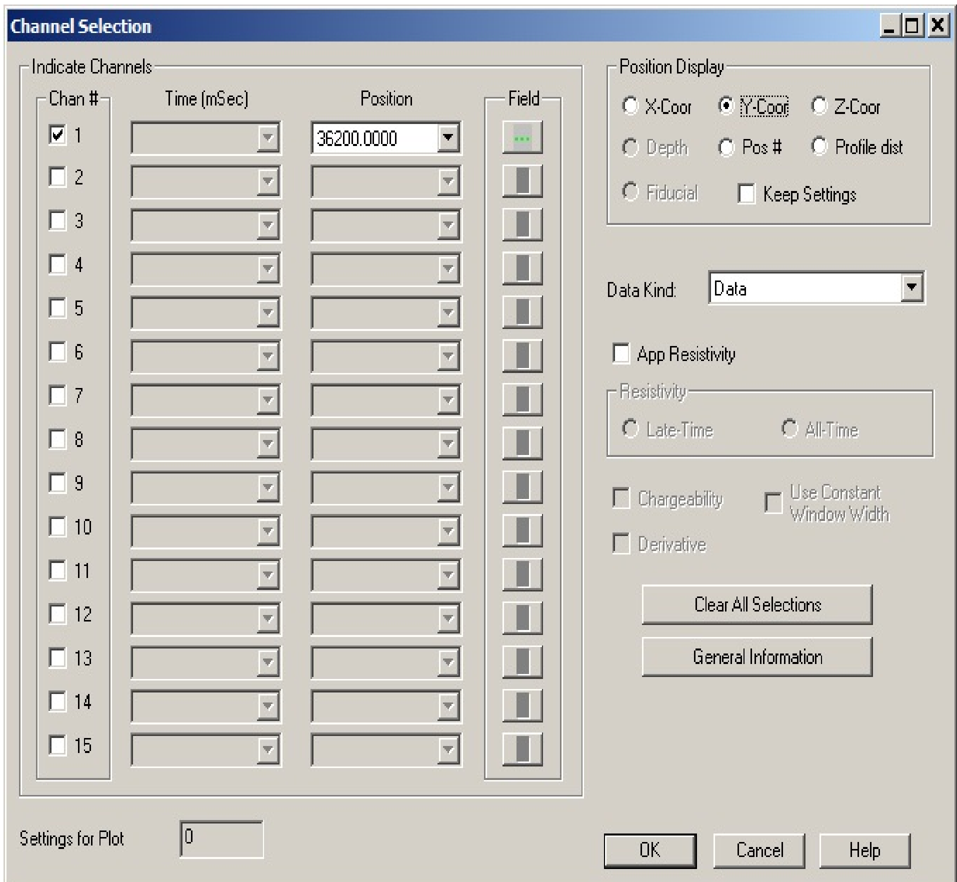
### Related Topics

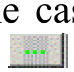
Edit Plot Settings

Switch between the Profile and Spectrum/Decay Display

## Edit plot settings

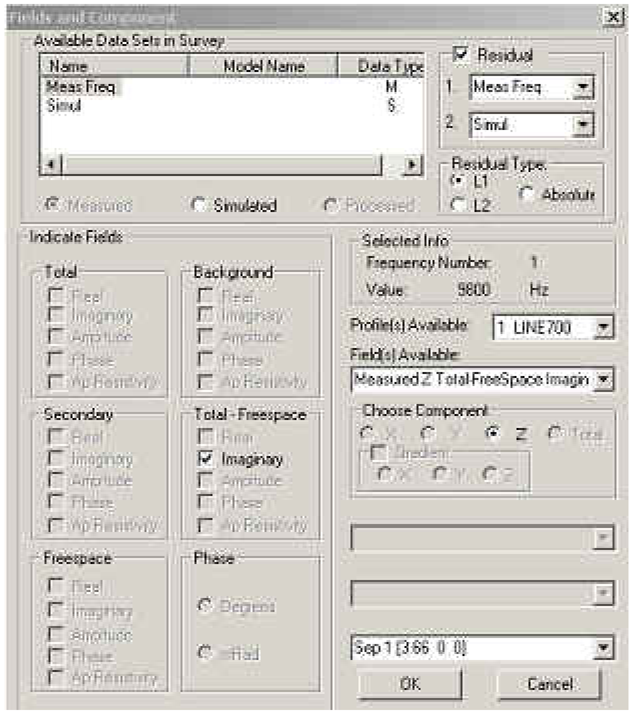
- Click the **Channels** button  on the EiKPlot toolbar or double-click anywhere in your plot. The **Channel Selection** dialog will open:



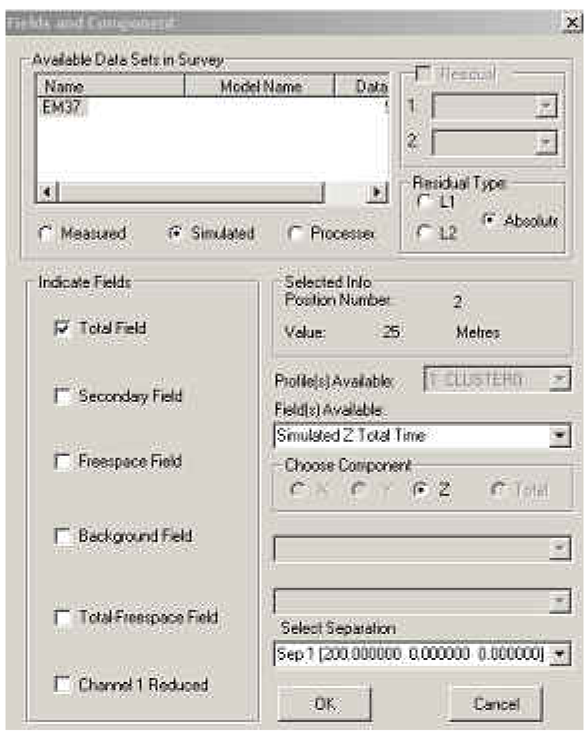

- Select the channel you want to display from the **Frequency** (**Time (msec)** in the case of time-domain data) dropdown list and click the **Field** button  to specify the fields and components

  If you want several channels to be displayed on the same plot, select as many **Plot** # boxes as you need. For example, if two channels are to be plotted at a time, check the second **Plot** # box and select the second channel from the respective dropdown list.

  In the case of static or frequency-domain systems, the **Fields and Component** dialog to appear will be as follows:




If your data is time-domain, the **Fields and Components** dialog to appear will differ only in the **Indicate Fields** section:




In both dialogs, the **Available Data Sets in Survey** section shows the name of the data sets and models you loaded and the type of data

(Measured, Simulated, Processed) in these data sets. You can see the number of channels you selected and their value in the **Selected Info** section.

- Select the data set(s) to plot if you have loaded several data sets
- Check the fields and phases (for frequency-domain and static) to plot in the **Indicate Fields** section
- Select the X, Y, Z or **Total** components in the **Choose Component** section if your receivers are dipoles
- Select the bipole from the respective dropdown list to become active in the bottom right-hand corner of the dialog above the **Select Separation** list if your receivers are bipoles. In this case, the **Choose Component** section will be disabled.

  ***Note.*** You can also select the field and component from the **Fields Available** dropdown list on the right. This automatically checks the required field and selects the respective component/bipole. *Checking more than one field will display the respective number of responses on the same plot*

- Select a transmitter from the **Select Transmitter** dropdown list to become active above the **Select Separation** list when multiple transmitters are used
- Select a separation to be used by the plotter from the **Select Separation** dropdown list, which becomes active in the case of a moving transmitter survey
- Provided you have both measured and simulated data, check the **Residual** box in the respective section in the upper right-hand corner of the dialog. The two dropdown lists below will be enabled. Select the data sets from these lists. In the **Residual Type** section, select an algorithm to be used for your data recalculation
- In case total derivatives have been measured or modeled (in magnetics and gravity surveys), select **Total** to enable the **Gradient** X, Y and Z buttons. Choose the gradient you want to plot
- Click **OK** to return to the **Channel Selection** dialog

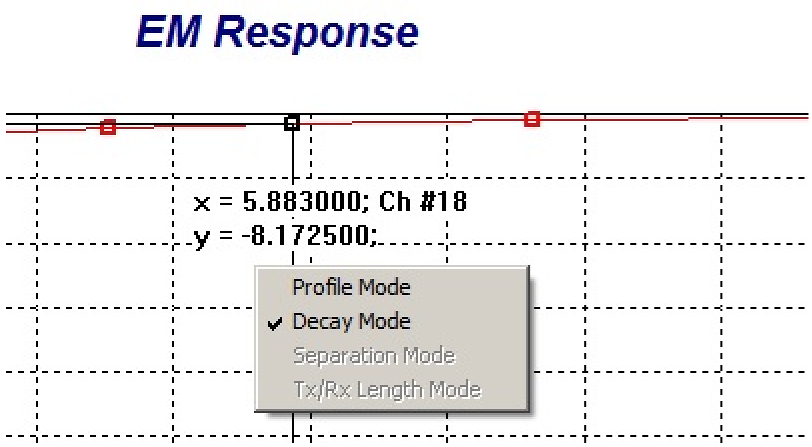
In the **Channel Selection** dialog, you can specify to plot the Apparent Resistivity response:

- Check the **Ap Resistivity** box. This will activate the **Resistivity** section. Select between **Real** and **Imaginary** for FEM and **Late-T** and **All-Time** for TEM
- Check the **Derivative** box to plot the derivative of the data selected. The derivative is calculated by a simple inline finite difference
- Click **OK** in the **Channel Selection** dialog to view the plot.

***Note***. *The warning "Curves containing non-data only are not plotted" means that your selection cannot be plotted, since it requests data that are not available.*

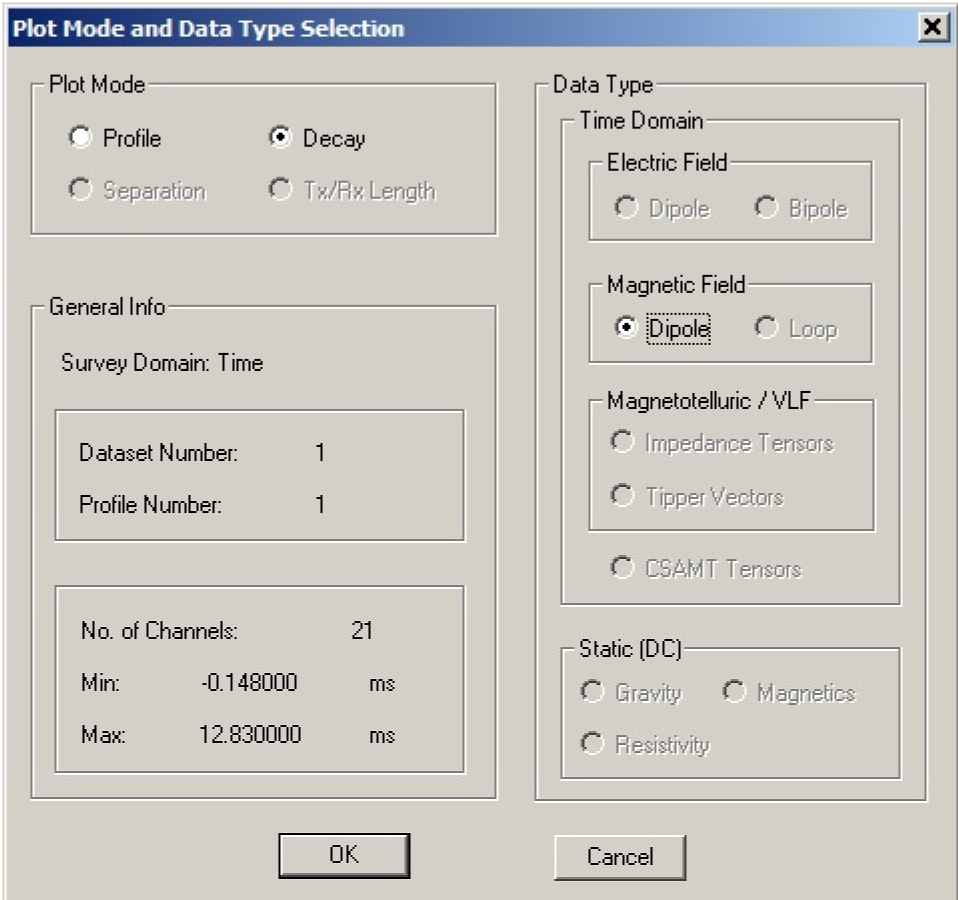
**Switch between the profile and spectrum/decay display**

Right click on the plot display and select one of available modes in the floating menu:



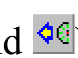

OR

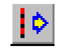
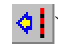
To specify in more detail the data to be displayed, select **Configure/Domain** or click the **Domain** button  on the EiKPlot toolbar. The **Plot mode and Data Type Selection** dialog opens:




- In the **Plot Mode** section of the dialog, select the **Spectrum** button if your data are frequency-domain or the **Decay** button if they are time-domain. In **Decay** mode, horizontal axis becomes "Time"; in **Profile** mode, it will be "Absolute X" or "Absolute Y" depending on profile direction; in **Spectrum** mode, it is "Frequency".

- Click **OK** to close the dialog and view the **Spectrum/Decay** response

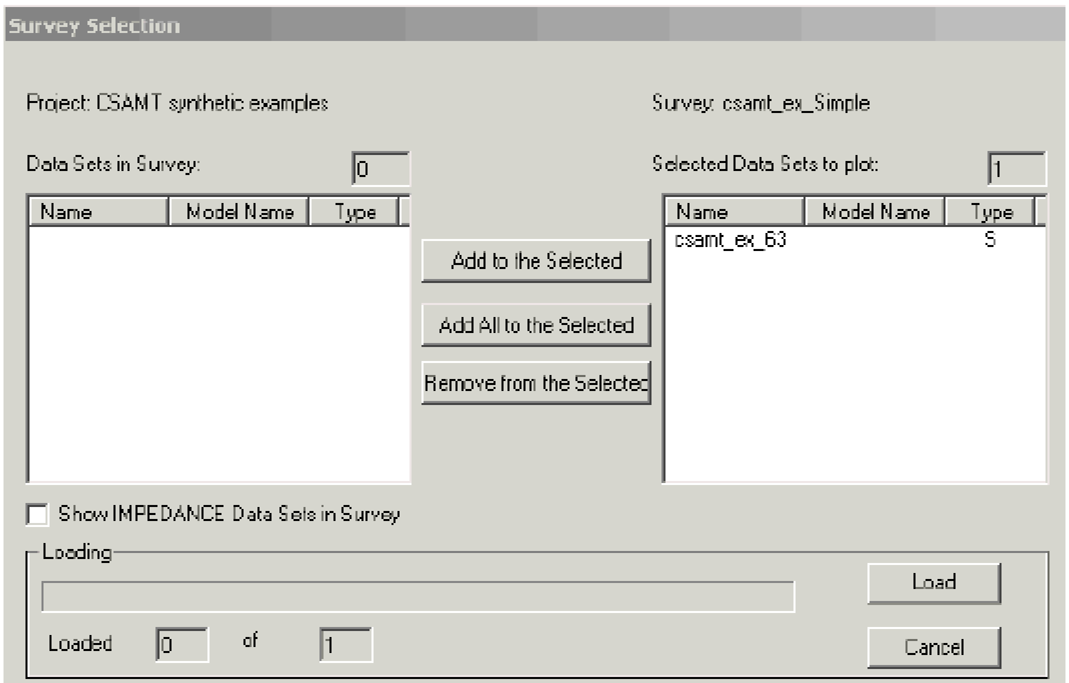
***Note***.
In **Decay** or **Spectrum** mode, use the **Next** and **Previous Position** buttons (  and ) on the EiKPlot toolbar to toggle forward and back through the available profile locations.
In **Profile** mode, use the **Next** and **Previous Channel** buttons (  and ) on the EiKPlot toolbar to toggle forward and back through the available profile locations.

Use the **Next** and **Previous Profile** buttons (  and ) to toggle forward and back through the available profiles.

## Plot MT and CSAMT Data

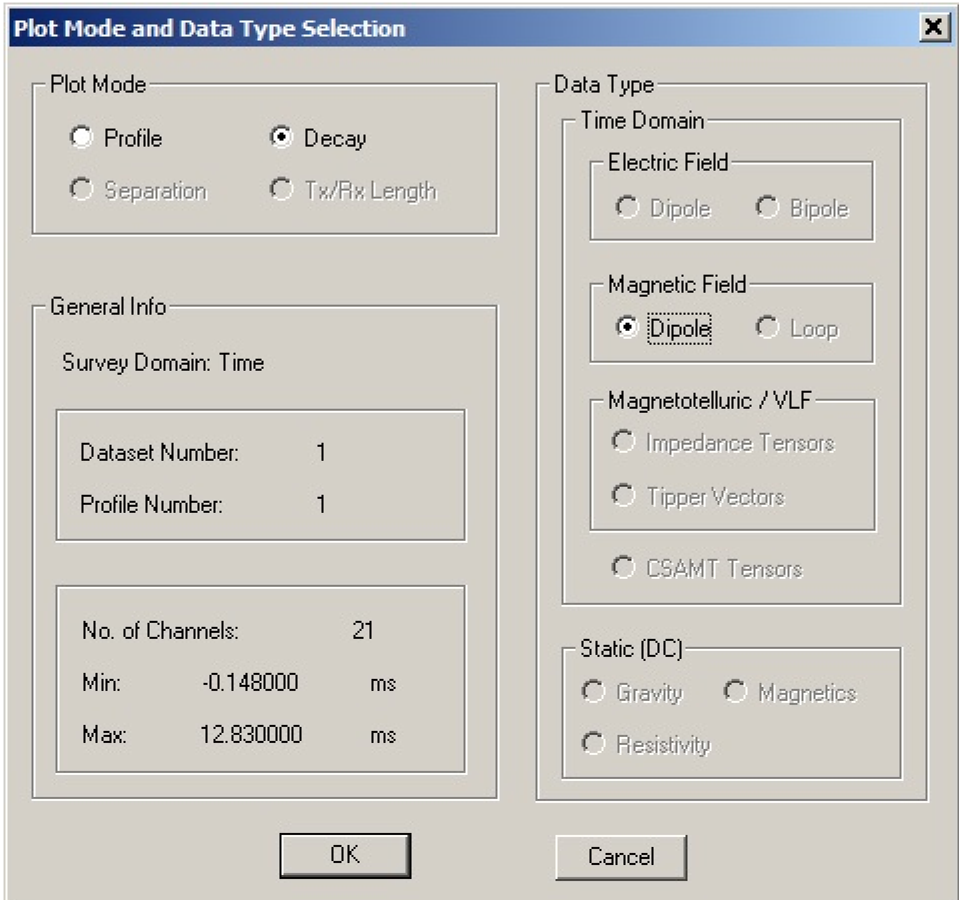
1. If your MT or CSAMT data contain impedance information, the plot will be generated automatically. However you can always view your initial data:

- Click the **Survey** button  on the EiKPlot toolbar. The **Survey Selection** dialog will appear:




- De-select the **Show IMPEDANCE Data Sets in Survey** box below the list of data sets on the left. All initial EM data sets will appear in the list of data sets

- Select a data set (or both data sets) to plot and click **Add to the Selected**. The **Domain and Data Type Selection** dialog will appear offering you to select between the electric and magnetic fields:



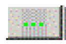

2. If your data set contains no impedance information, the **Domain and Data Type Selection** dialog will open prior to plot generation. Select between **Electric Field** and **Magnetic Field** and click **OK** to proceed.

**Related Topics**

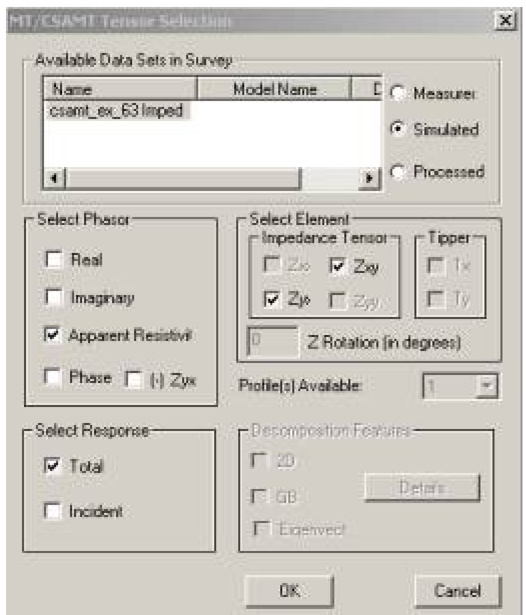
Edit Plot Settings

Switch from Impedance Tensors to Tipper Vectors

## Edit plot settings

- Click the  button on the EiKPlot toolbar or double-click anywhere in your plot. The **Channel Selection** dialog will appear. Select the channel you want to plot from the **Frequency** dropdown list and click the **Field** button 

  This will bring up the **MT/CSAMT Tensor Selection** dialog:




  In this dialog, the **Available Data Sets in Survey** section shows the name of the data sets and models you are currently in and the type of data in these data sets. You can see the number of channels you selected and their value in the **Selected Info** section.

- Select a required phasor in the respective section. Checking the **Real** and **Imaginary** phasors will display them both on the same plot. Selecting **Apparent Resistivity** or **Phase** will cancel all other selections

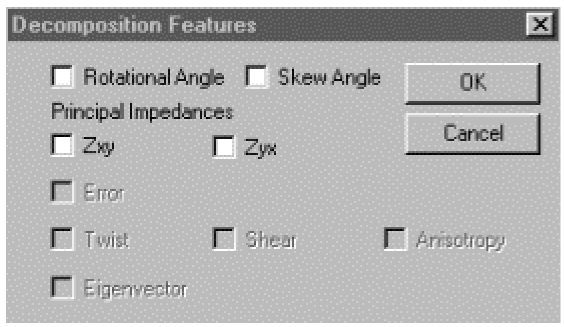
- Check the **(-) Zyx** box to de-select the respective element in the **Impedance Tensor** section

- Check a required response/responses in the **Select Response** section

- Select either one or more elements in the **Impedance Tensor** section, which is active by default

- Specify the rotation angle in the **Z Rotation (in degrees**) field. This option is active when all the **Impedance Tensor** elements are available

- Click in the **2D** box in the **Decomposition Features** section to bring up the respective dialog:




- Select **Rotational Angle** and **Skew Angle** or the principal impedances to calculate and plot

*Note*. *The* ***Decomposition Features*** *section is yet underway. As of now, only a standard Swift 2D decomposition is provided.*

**Related Topics**

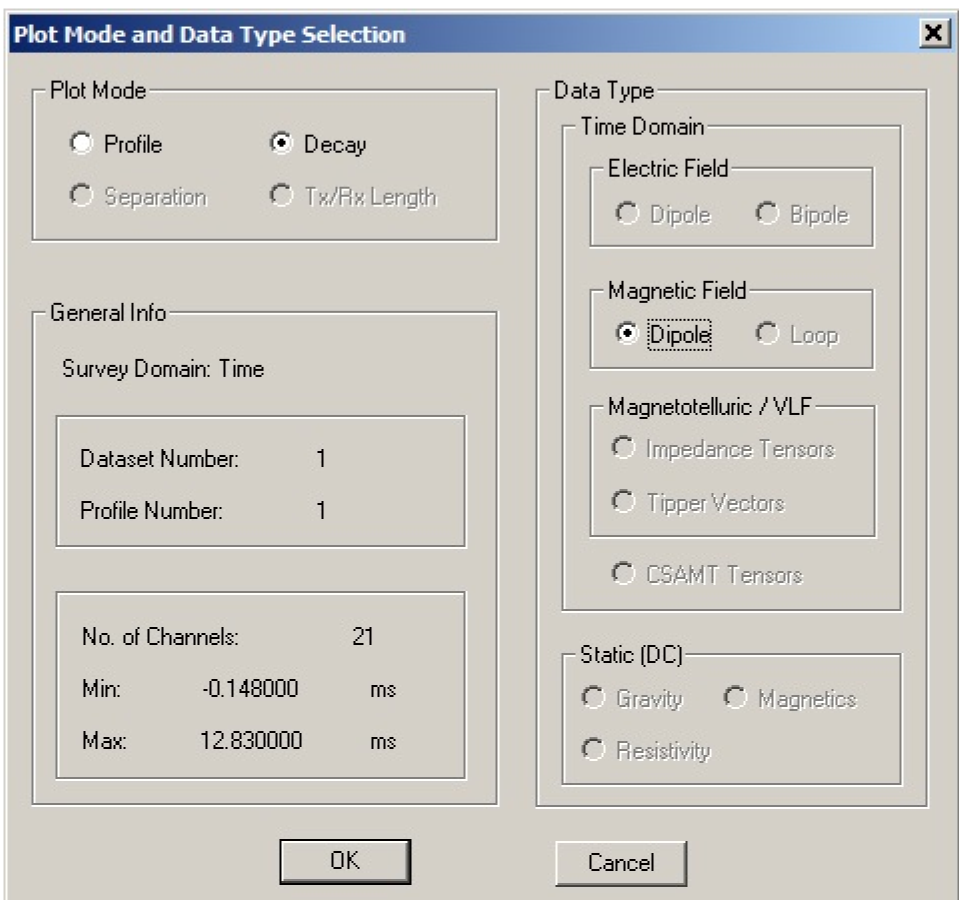
[Switch from Impedance Tensors to Tipper Vectors]

## Switch from Impedance Tensors to Tipper Vectors

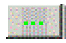
In MT, you may need to switch between **Impedance Tensors** and **Tipper Vectors**

- Click **OK** in the **MT/CSAMT Tensor Selection** dialog and then in the [Channel Selection]() dialog to close them both

- Select **Configure/Domain** or click the **Domain** button  on the EiKPlot toolbar. The **Domain and Data Type Selection** dialog opens:




- In the **Magnetotelluric** section of the dialog, select the **Tipper Vectors** button and click **OK**. The **Channel Selection** dialog reappears

- Click the **Field** button  to the right of the selected channel to reopen the **MT/CSAMT Tensor Selection** dialog

  You now have the **Tipper** section activated, whereas the **Impedance Tensor** boxes are disabled. The **Z Rotation** box will be disabled as well, however it may contain an angle setting if one was specified earlier

- Select the **Tipper** vector you want to plot

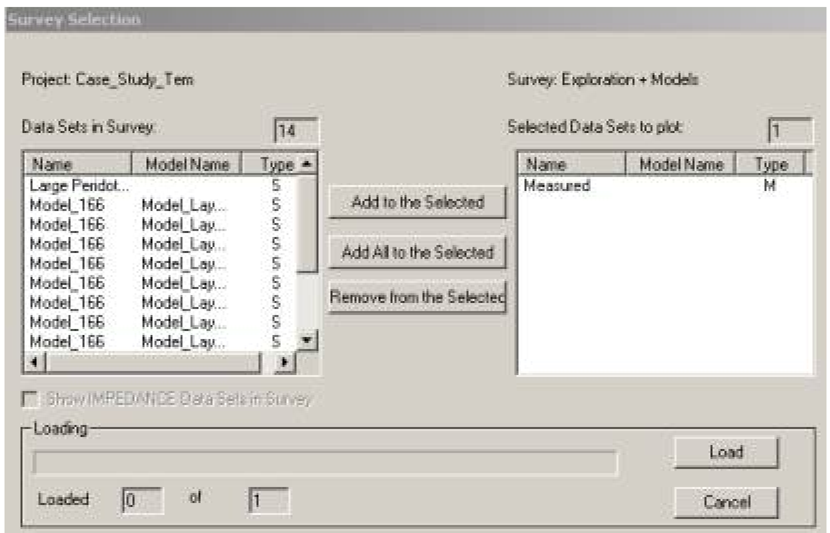
- Click **OK** to return to the **Channel Selection** dialog

- Click **OK** in the **Channel Selection** dialog to view the plot.

## Loading Additional Data Set(s)

If you have more than one data set in your survey, but you did not load them at the very beginning (see Select Data Set(s) to Plot), you can always do it from your plot display:

- Select **Configure/Survey** or click the **Survey** button on the EiKPlot toolbar. The **Survey Selection** dialog appears:

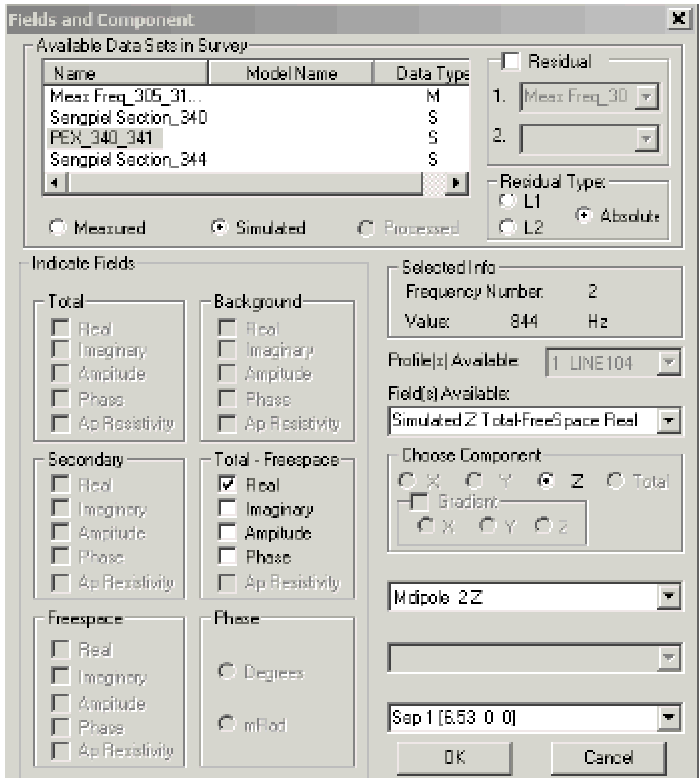



- Select the data set(s) from the list on the left and click **Add to the Selected**. The data sets selected will move to the list on the right

- Click **Load**. The **Channel Selection** dialog appears

- Select the channel(s) and click **Field** ( )

  In the **Fields and Components** dialog to appear:




- Select the data set from the respective list (now containing the data set(s) you added) in the upper left-hand corner of the dialog

- Specify fields and components as described in the **Edit Plot Settings** section in both Plot Static, Frequency- and Time-Domain Data and Plot MT and CSAMT Data

- Click **OK** in the **Fields and Components** and then in the **Channel Selection** dialogs to view the plot

***Note***. To plot several data sets at a time, check as many **Plot** # boxes in the **Channel Selection** dialog as the number of data sets you want to display. Select the channels from the activated **Frequency/Time/Static** dropdown lists. Click **Field** across each activated list and specify in each case the data set, field and components in the **Fields and Components** dialog to appear.

## Viewing Plots

Your data having been plotted, you get an easy access to different view options, such as switching between profiles, channels, separations, transmitters and receivers, customizing plot appearance, viewing model properties. All this is available from the EiKPlot menu and its toolbar offering a wide range of buttons. Or, you can also use the hot keys popping up when you hold your mouse cursor over the toolbar buttons.

To view the coordinates of any plot point, click it and hold the button down. You will see the X- and Y-axis values displayed over the cursor.

### Related Topics

Switch between Profiles

View Multiple Profiles on the Same Plot

Switch between Channels, Separations and Transmitters

View Multiple Plots at a Time

View Model Properties

Adjust the Scale of your Plot

Zoom in on a Fragment of your Plot

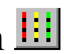
Toggle Grid On and Off

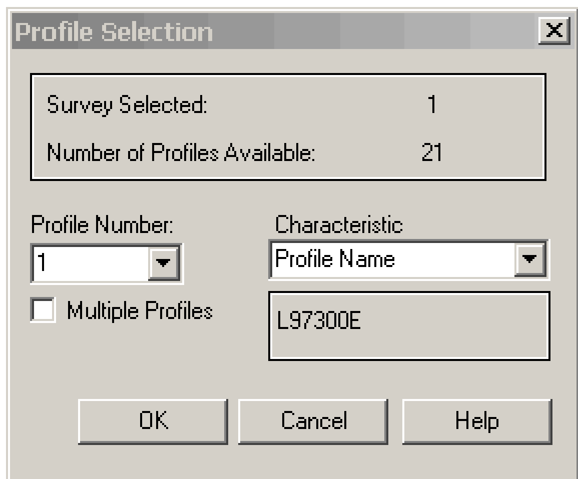
Customize Plot Appearance

## Switch between profiles

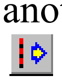
If you have more than one profile, you may need to switch from one profile to another or view multiple profiles at a time. Your first plotted profile is 1 by default.

To switch to another profile:

- Select **Configure/Profile** or click the **Profile** button  on the EiKPlot toolbar. The **Profile Selection** dialog opens:

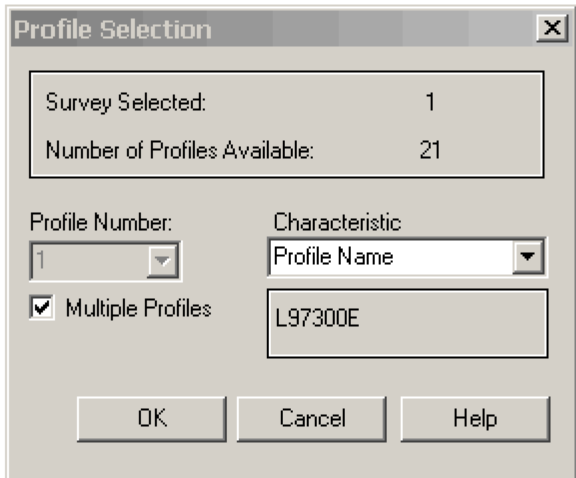



- Select another profile from the **Profile Number** dropdown list. In the **Characteristic** dropdown list on the right, you can select the profile name or the profile (start) X, Y and Z coordinates to view the respective information in the frame below

  ***Note***. You can also switch to another profile right from the EiKPlot toolbar. Click the **Next Profile**  and **Previous Profile**  buttons to toggle forward and back through all available profiles.

## View multiple profiles on the same plot

- Select **Configure/Profile** or click the **Profile** button on the EiKPlot toolbar. The **Profile Selection** dialog opens




- Select the **Multiple Profiles** box and click **OK** to close the dialog

- Double-click anywhere in the plot or click the **Channels** button on the EiKPlot toolbar to open the **Channel Selection** dialog

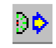
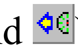
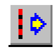
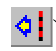
- In this dialog, check the next **Plot** # box to activate the **Frequency/Time/Static** dropdown list and select the channel

- Click the **Field** button to display the **Fields and Components** or **MT/CSAMT Tensor Selection** dialog. You now see the **Profile(s) Available** dropdown list enabled

- Select the number of the profile you want to add to your plot from this list

- Click **OK** to return to the **Channel Selection** dialog

- Click **OK** to close the dialog and view the plots.

***Note***.
In **Decay** or **Spectrum** mode, use the **Next** and **Previous Position** buttons (  and ) on the EiKPlot toolbar to toggle forward and back through the available profile locations.
In **Profile** mode, use the **Next** and **Previous Channel** buttons (  and ) on the EiKPlot toolbar to toggle forward and back through the available profile locations.
Use the **Next** and **Previous Profile** buttons (  and ) to toggle forward and back through the available profiles.

## Switch between channels, separations and transmitters

- In **Decay** or **Spectrum** mode, use the **Next** and **Previous Position** buttons ( and ) on the EiKPlot toolbar to toggle forward and back through the available profile locations.

- In **Profile** mode, use the **Next** and **Previous Channel** buttons ( and ) on the EiKPlot toolbar to toggle forward and back through the available profile locations.

- To switch between separations use the **Next** and **Previous Separation** buttons ( and ) on the EiKPlot toolbar.

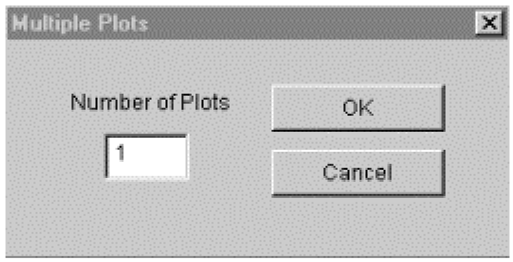
- To switch between transmitters, use the **Next** and **Previous Transmitter** buttons ( and ) on the EiKPlot toolbar.

## View multiple plots at a time

Viewing multiple plots at a time is especially convenient when you deal with responses that cannot be displayed on the same plot.

- Select **Settings/Custom/Number of Plots** from the EiKPlot menu

  The **Multiple Plots** dialog appears:




- Type the number of plots you want to display and click **OK**

  The window will be divided into the respective number of plots. The first plot will be the one you are currently in.

- If you want to check or edit the settings of your current plot, double-click anywhere in it to display the **Channel Selection** dialog. Make necessary changes as described in the Edit Plot Settings sections (see Plot Static, Frequency- and Time-Domain Data and Plot MT and CSAMT Data)

- Repeat this operation for all the other plots

To switch back to the single-plot display:

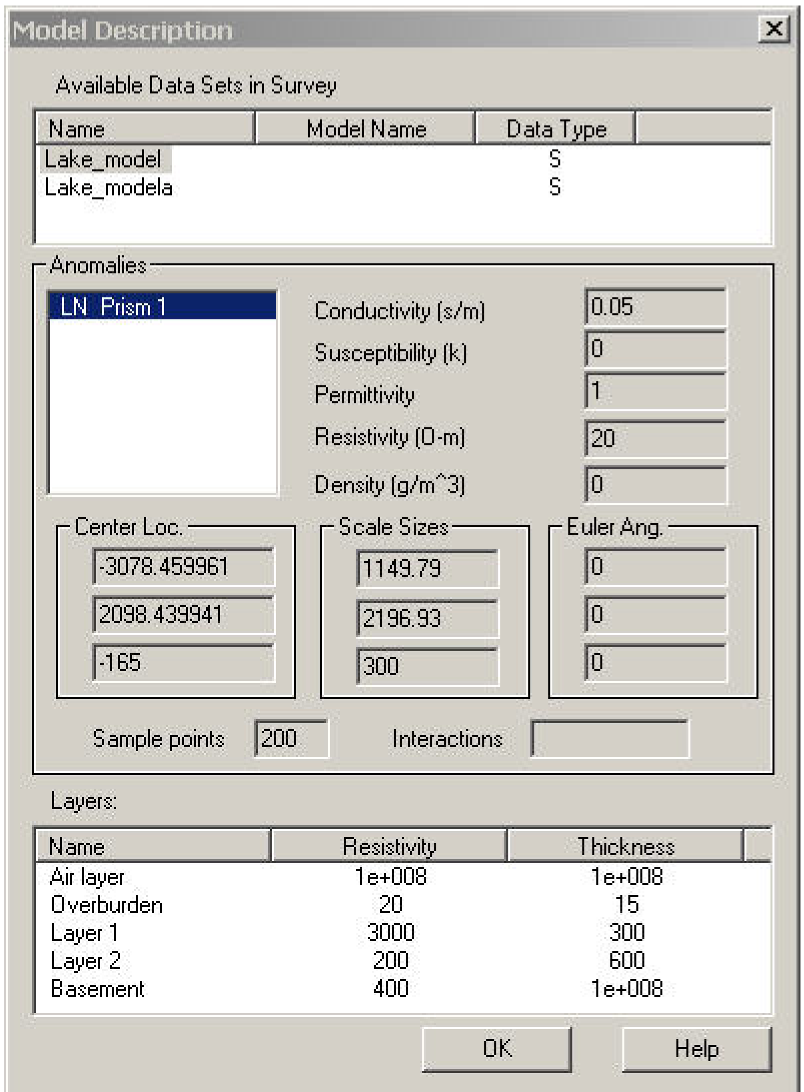
- Select **Settings/Custom/Number of Plots** from the EiKPlot menu to bring up the **Multiple Plots** dialog

- Change the number of plots to 1 and click **OK**

  This will switch you back to the full-screen single-plot view again.

## View model properties

To view model properties without leaving the plotter, click the **Model** button  on the EiKPlot toolbar. You will find the following information in the **Model Description** dialog to appear:




- List of data sets in your survey loaded in EiKPlot. You can select any data set to check the model information therein
- List of Anomalies. Select the target you are interested in to see its parameters (conductivity, permeability, Euler angles, scale sizes,

sample points and interactions)

- List of layers with resistivity and thickness information for each layer

If you have loaded several data sets, you can switch between your models using the **Previous**  and **Next**  Model buttons on the EiKPlot toolbar.

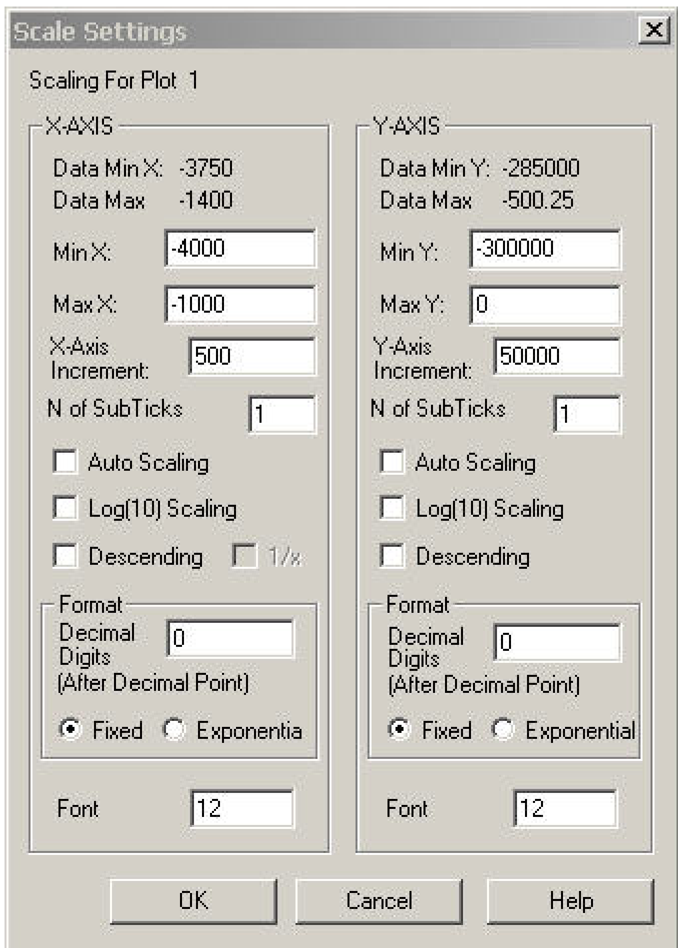
## Adjust the scale of your plot

EiKPlot allows you to customize the scale and to zoom in on a specific fragment of your plot.

To adjust the scale settings:

- Select **Settings/Scaling** or click the **Scaling** button  on the EiKPlot toolbar

  The **Scale Settings** dialog appears:




- The minimum and maximum X- and Y-axis settings and the X- and Y-axis increments are generated automatically from your input data. To change these settings, type your values in the respective fields

- To return to initial scaling, select the **Auto Scaling** checkbox. You can also do it later, right from the EiKPlot menu or toolbar. Select **Settings/To Initial Scale** or simply click the **Rescale** button .

- Change to the descending scale by selecting the respective checkbox. De-select to change back

- To customize the appearance of the axis labels, change the number of digits to be displayed after the decimal point, adjust units (fixed or exponential) and set a required font size in the respective fields

- To change to the logarithm scale, select the **Log(10) Scaling** checkbox.

  ***Note***. If your data contain negative or zero values, you will see a warning message indicating negatives plotted as positives and zeroes not plotted

- Click **OK** in the **Scale Setting** dialog to close the dialog and return to your plot.

## To zoom in on a fragment of your plot:

- Select **Settings/Zoom** or click the **Zoom** button on the EiKPlot toolbar

- Click in the area of your plot where you want the fragment to start and, without releasing the button, drag right/left and up/down until the required fragment is selected (outlined in green). Release the button

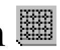
- To zoom in further on, repeat the operation.

  The two buttons on the EiKPlot toolbar, **Zoom Back** and **Zoom Forth** become active

- Use these buttons to toggle through all the fragments you have zoomed in

- Click the **Rescale** button to zoom out and return to the initial scaling.

## Toggle grid on and off

- To toggle the grid on and off, select or de-select **Settings/Custom/Grid** or click the **Set Grid** button  on the EiKPlot toolbar.

- To cancel the grid, you can also use the **Default** command from the **Settings** menu.

## Customize plot appearance

EiKPlot offers you a whole set of tools for customizing your plot appearance. You can change the color, style and label of the curve, adjust the appearance of the symbols, edit and move axis labels, mask any of your plots.

### Related Topics

Change the Color and Style of the Curve

Change the Curve Label

Mask a Curve

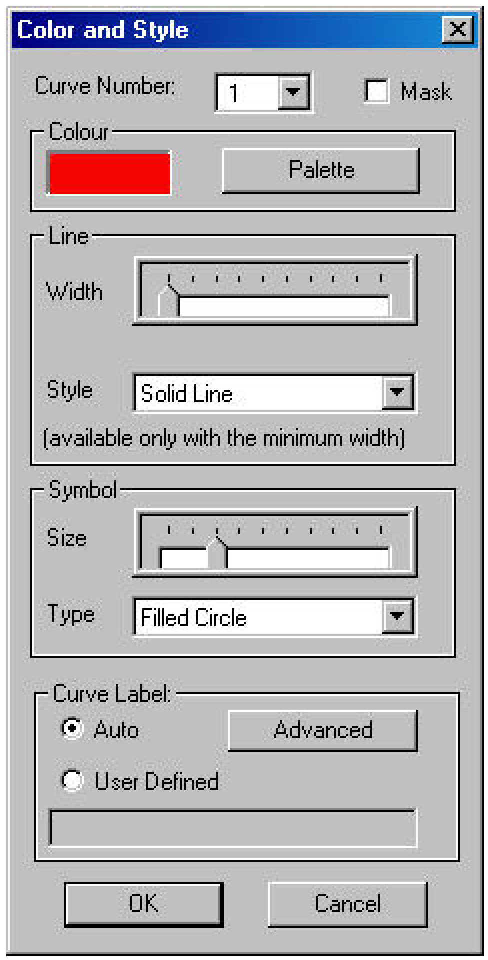
Adjust Axis Labels and their Font Size

## Change the color and style of the curve

- Select **Settings/Colour and Style** on the EiKPlot menu

OR

- Double-click the curve label (top left corner). The **Colour and Style** dialog appears




In the **Colour** section:

- Click the **Palette** button to open the standard palette of basic and custom colors. Change or add new colors and hues

In the **Line** section:

- Define the width of the line using the slider

  This option is applicable only to the **Solid Line** style. Other styles do not show on the plot unless the minimum line width has been selected

- Select the style of your curve from the respective dropdown list

In the **Symbol** section:

- Use the slider to increase or decrease the size of used symbols

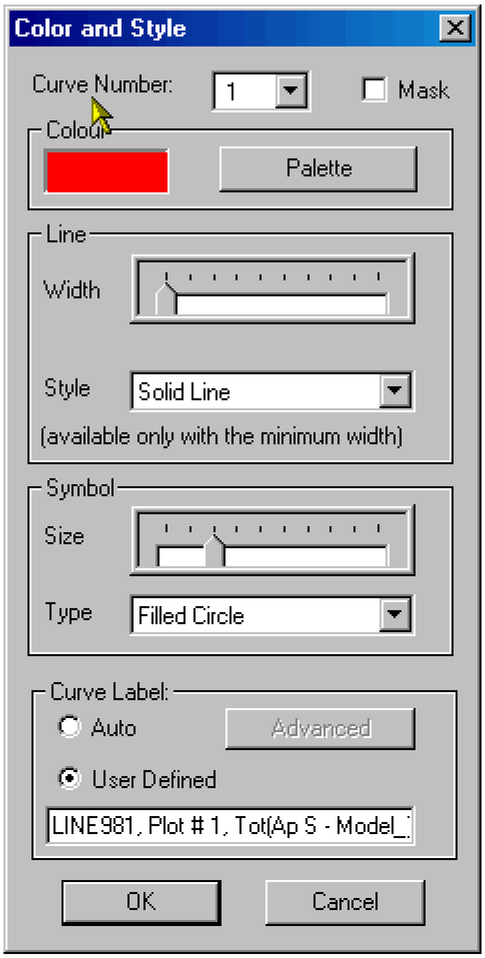
- Select the shape you want to apply from the **Type** dropdown list.

***Note***. To reset all your color and style changes, select **Settings/Default**.

## Change the curve label

- Double-click on the curve label to change

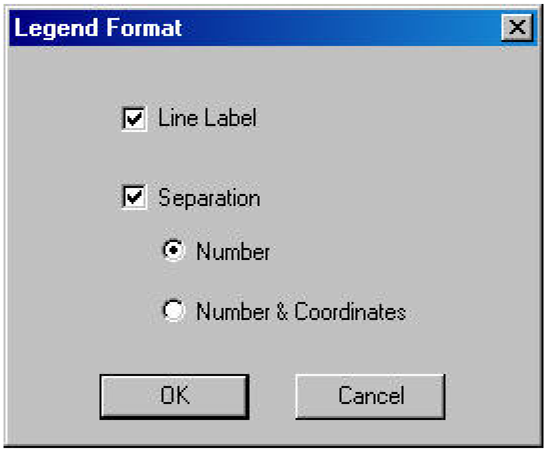
  The **Curve Label** field of the **Color and Style** dialog to open will contain this label




- Select **User Defined** and change the label name as desired and click **OK**

- To change back to the former label, open the **Color and Style** dialog again and delete the new curve label from the respective field. Click **OK**. The former label will reappear.

- To apply changes to all the labels, click on **Auto** then **Advanced** to reveal the **Legend Format** dialog:




It is possible to toggle the display of the **Line Label** and the **Separation** in the curve label. You have the option of showing the separation **Number & Coordinates** or just the separation **Number**.

Sample curve label: LINE9673050, Plot # 1, Tot(Ap S - Model_)B1 - S1(-50.00, 0.00, 0.00)

- Line Label - LINE9673050
- Separation Number - S1
- Separation Coordinates - (-50.00, 0.00, 0.00)

## Mask a curve

- Double-click on the label of the curve to mask
- Check the **Mask** box in the top right-hand corner of the **Color and Style** dialog to open and click **OK**

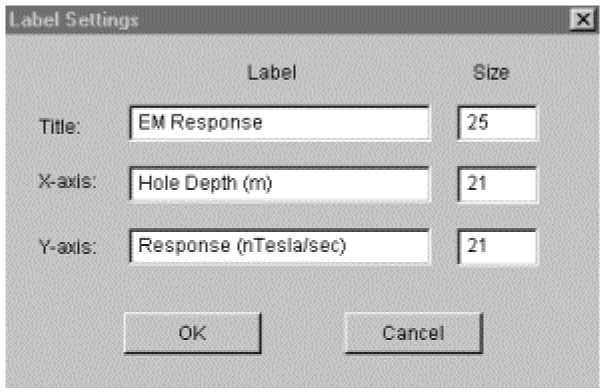
  The curve and its label will become gray

- To bring the masked curve back, de-select the **Mask** box

## Adjust axis labels and their font size

- Double-click on the axis label to change (or select **Settings/Labels** or click the **Change Labels** button on the EiKPlot toolbar)

  The **Label Settings** dialog appears:




- Make your adjustments and click **OK** to close the window and view the changes.

  **Note**. To change the position of an axis label on the plot, simply click and drag the label wherever you want.

## Converting to various displays

EiKPlot allows you to convert plots to various displays:

- Select **Settings/Custom/Flip Sim Data** to multiply all simulated data by –1 and thus to flip your plot. De-select this command to switch it back to the initial view.
  **Flip Measured Data** and **Flip Processed Data** perform this operation for measured and processed data.

- Select **Settings/Custom/Sim Quad Conv[+iwt]** and **Sim Quad Conv[-iwt]** to set "+" or "-" sign to the simulated quadrature data only.

- Select **Settings/Custom/Crone (X,Y)⇐PetRos EiKon (-Y,X)**, if you are the user of Crone systems and need to convert coordinates between Crone and PetRos EiKon formats.

- Select **Settings/Custom/Smooth Meas Data** to process your measured data and make the plot less jagged.

- Select or deselect **Settings/Custom/App Conductivity** to switch between the apparent resistivity and apparent conductivity displays.

- Select **Settings/Custom/Section ===> UHole** or **UHole ===> Section** to switch between X/Y/Z and S/N/W dipole receiver positioning. This feature is available only for borehole data. It has no effect if "Total" is selected in "Dipole receiver" section of "Fields and Components" dialog.

- To convert phase into required range, select one of **Settings/Custom/Phase** menu items : **Phase:-180 to 180 degrees, Phase: 0 to 360 degrees,** or **Phase: 0 to -360 degrees**.

- If currently plotted data have error field, you can select **Settings/Custom/Display Error Bar** to visualize it.

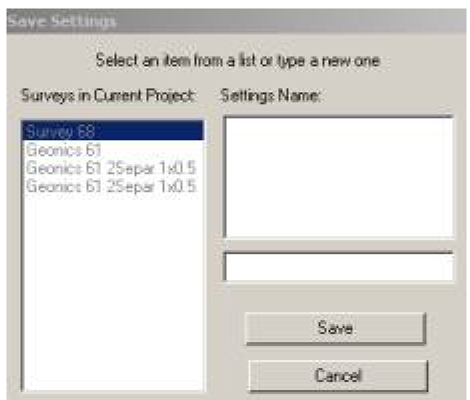
- Item **Settings/Custom/Number of positions to skip** is available in **Spectrum, Separation and Decay** mode. It allows to set how many locations are "jumped over" when buttons **Previous/Next Position** are clicked.

## Saving plot settings

This option allows for the rapid plotting of a number of models. For example, you could run a suite of models in batch mode, adjusting your layered earth, target positions, conductivity or size. Plot the first model and save the default settings as a .plt file. Then proceed to the next model and simply open the .plt file. The graph will be plotted for you automatically.

To save the settings of your current plot(s) as default:

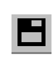
- Select **Default/Save As** or click the **Save Settings** button  on the EiKPlot toolbar. The **Save Settings** dialog appears




- Select the survey you want to save the settings file in

- In the field below the **Settings Name** box, type the name of your new .plt filename and click **Save**.

To save changes to an existing setting file

- Select **Defaults/Save** or click the **Save Settings** button  on the EikPlot toolbar

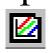
- In the **Save Settings** dialog, click **Save**, if you want to save the file under the same name, or type a new name if you want to save it as a separate file.

## Loading a default settings file

- Select **Defaults/Get** or click the **Get Settings** button  on the EiKPlot toolbar. The **Get Settings** dialog appears
- In the **Surveys in Current Project** list, choose the survey your settings file is in
- Select the **Settings Name** and click **Load**.

## Printing Plots

EiKPlot offers two output modes: Auto (Full-Screen) and Scaled Graphic. The former automatically makes your printed plot look as you see it on the screen. The latter enables you to change the scale of the plot to be printed and to add an information box.

***Note***. To print your plot in color, select **File/With Colour** or turn the **Colour Print** button  on before setting print properties.

### Related Topics

Print and Preview in Auto (Full-Screen) Mode

Print and Preview in Scaled Graphic Mode

Add an Information Box to your Plot

## Print and preview in Auto (Full-Screen) mode

To print the plot as you see it on the screen:

- Select **File/Print** to display the **Print** dialog
- In this dialog, specify the printer, the print range and the number of copies in the respective fields
- Click **Properties** to specify the document properties in the dialog to appear and click **OK**
- Click **OK** in the **Print** dialog to start printing

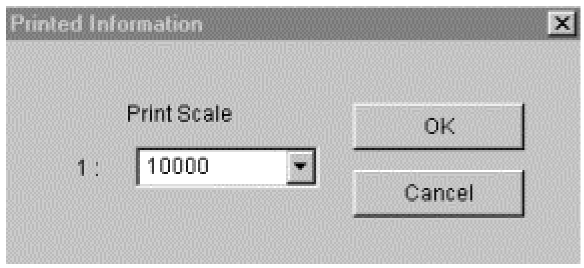
To preview your plot in the Auto (Full-Screen) mode:

- Select **File/Print Preview** (or **File/Output Settings and Preview/Auto**) or click the **Auto Preview** button  on the EiKPlot toolbar
- Use the **Next Page** and **Previous Page** buttons on the **Print Preview** toolbar to toggle through available pages
- Click the **Two Pages** button to preview two pages at a time, click it again to switch back to the **One Page** mode
- Click the **Zoom In** button to take a closer look at your plot and the **Zoom Out** button to move it away
- To close the **Preview** mode, click **Close**
- To print the plot, click **Print**. In the **Print** dialog to open, specify print properties and click **OK**.

## Print and preview in Scaled Graphic mode

To print your plot in the **Scaled Graphic** mode:

- Select **File/Output Settings and Preview/Scaled Graphic** or click the **Scaled Graphic** button on the EikPlot toolbar. The **Printed Information** dialog appears:



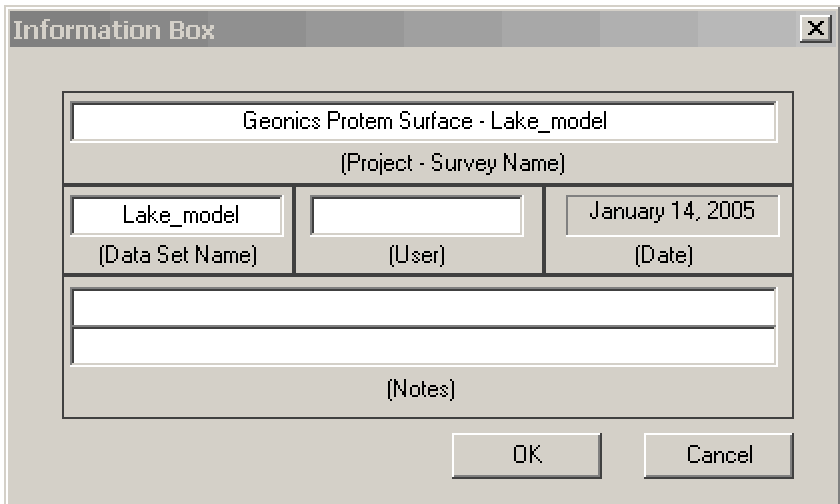

- Select the print scale from the respective list and click **OK**. This will display the **Print Preview** mode, with your plot changed to the scale you selected

  ***Note***. The **Print Preview** toolbar in the **Scaled Graphic** mode offers the same options as in the **Auto (Full-Screen**) mode.

## Add an information box to your plot

The **Scaled Graphic** mode allows you to add an information box to your plot:

- Click the **Print Information Box** button  on the EiKPlot toolbar. The respective dialog appears:



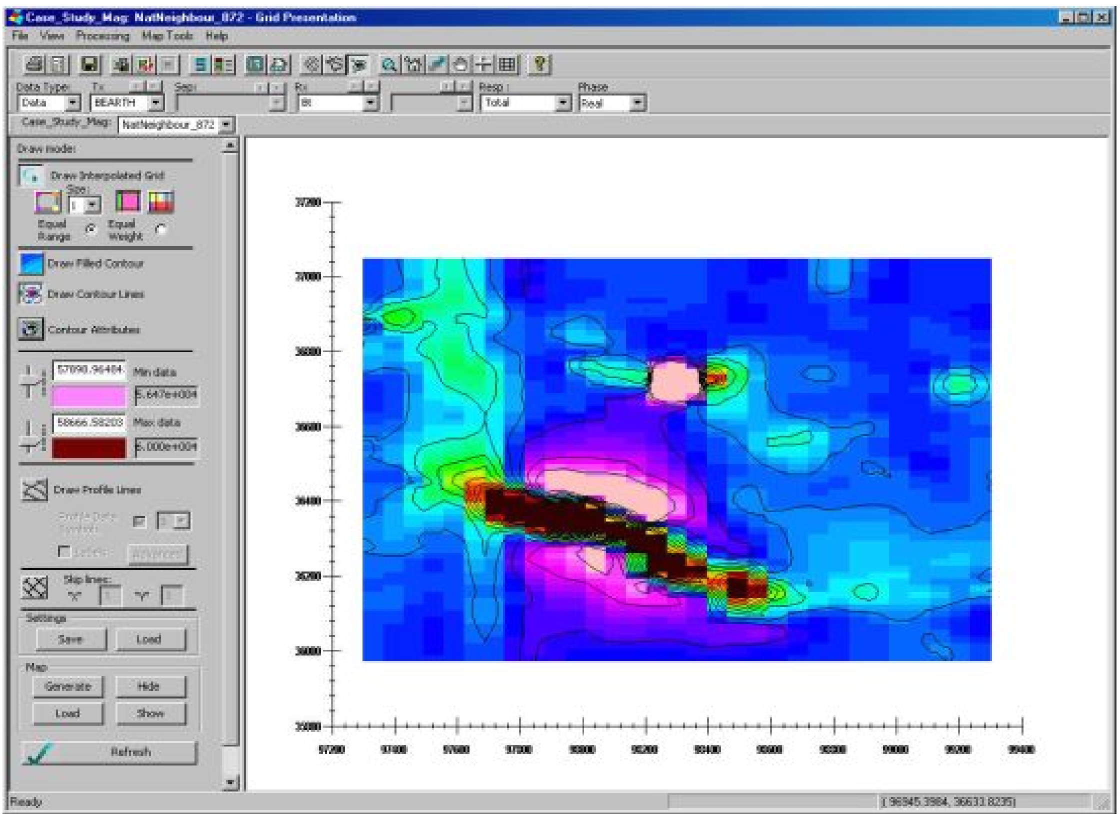

- Fill in the project name and user in the respective fields. The filename and the date are generated automatically

- Write your comments in the **Notes** field

- Click **OK**. The information box will be printed in the upper right-hand corner of the page.

# Grid Presentation

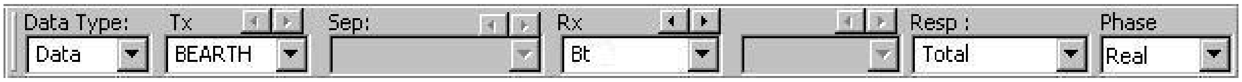
Provided your data set contains a grid (the **Grid(s)** button of the [Database]() dialog is checked), you can view and adjust it using the **GridPresentation** tool. Click the  button on the main toolbar to open the respective application:



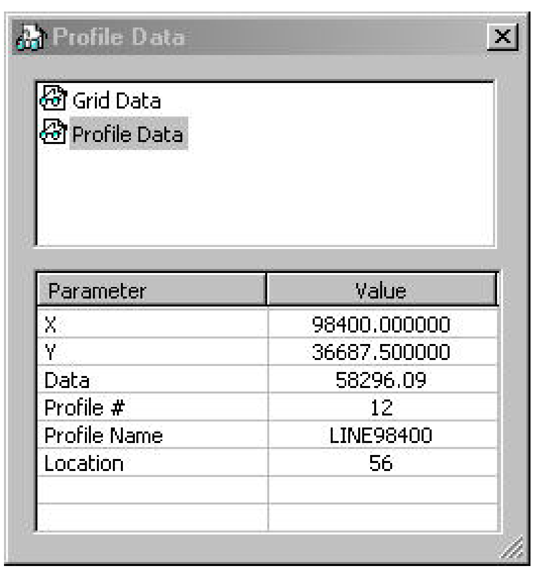

If your data set contains more than one grid, the grid to open will be the most recent one:

- To switch to another grid, select it from the dropdown list to the right of the name located on the bottom toolbar.
- An **Impedance** checkbox will be visible for the appropriate data to switch from grids with impedance data to grids without impedance data.

On the **Data Type** bar:




- If your data set contains different data types (data as is, recalculated data such as voltage, apparent resistivity, etc.), the **Data Type** dropdown list will contain the respective number of options, with **Data** selected by default. To switch to another option, select it from the list
- If there are several transmitters, receivers, separations or channels, select the required item from the respective dropdown list or use the arrow buttons above to switch between the items
- If both real and imaginary responses are available, turn one or the other button on to switch between the responses
- Check the **App Res** box to calculate apparent resistivity from data on the fly. Check **App Sigma** to switch to the respective grid. These boxes will only be visible when the data allows it.
- Click the  button to toggle the data parameters window:




This window will provide information on any point that is clicked on the survey image. It is possible to choose between Grid Data, Profile Data and Solution Data depending on what is available. The Profile Data option will be enabled when the Profile Lines are toggled on. The

Solution Data option will be enabled when an Euler solution is displayed(See Euler Overlays)

**Related Topics:**

Adjusting Grid Display

Creating Map Underlays

Saving Grids

Printing Grids

Analytic Signal (AS) and Horizontal AS

Decay Rate

Euler Overlays

### Adjusting the Grid Display

The left-hand panel of the **GridPresentation** dialog offers a number of tools to adjust your grid display.

Using this panel you can:

Change the Draw Mode of your Grid

Adjust the Range of Data to be Displayed

Adjust Contour Lines

Display Profiles

Display Grid Lines

Adjust Axes

Toggle the Coordinate Grid On and Off

Display the Legend

Provide the Full-Screen View of your Grid

Toggle off Bars and Panels

Zoom in on a Grid Fragment

## Change the draw mode of your grid

In the left-hand panel of the GridPresentation dialog:

- Click the **Draw Grid Symbols** button in the **Draw Mode** section to display each grid cell as a set of four data (vertices) and to assign a certain color to each data dependently of its value
- Click the **Draw Grid Cells** button to calculate the average of the data located in the vertices of a grid cell. The cell will be filled with a certain color assigned to the average value
- Click the **Draw Cells around Grid Vertices** button to display your grid as a set of cells drawn around each grid vertex and filled with a certain color assigned to the data value in the vertex
- Select **Equal Range** to assign different colors to equal ranges independently of the number of points in each range or **Equal Weight** to assign different colors to different ranges covering the same number of points

***Note***. To toggle your grid on and off, use the **Hide Grid Mesh** button .

## Adjust the range of data to be displayed

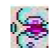
In the left-hand panel of the GridPresentation dialog:

- Insert the minimum and maximum values manually in the active **Min** and **Max** boxes or use the respective sliders on the left. The absolute minimum and maximum values are displayed in the disabled boxes on the right

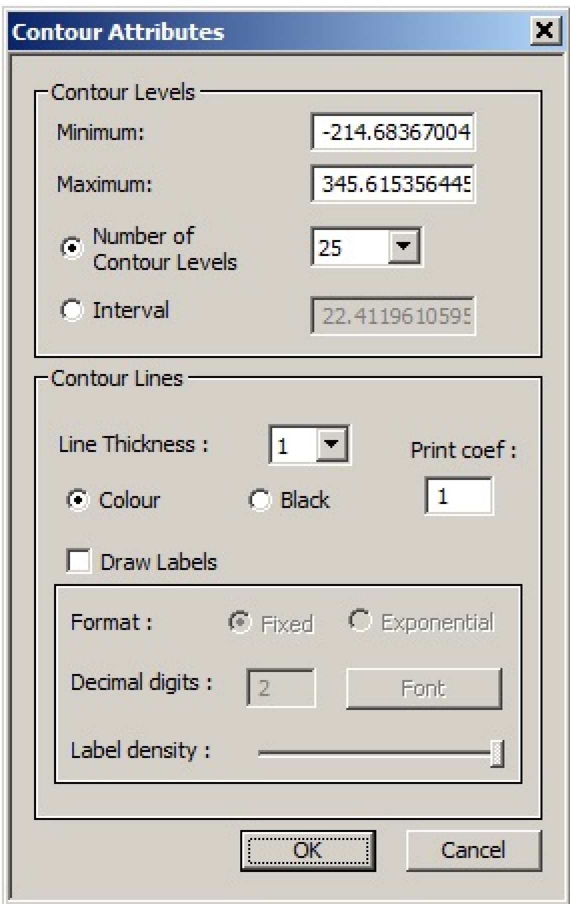
- Click either of the color palette boxes to bring up the palette and change, if required, the colors assigned to your data. To view the newly assigned colors, click the **Refresh** button in the bottom of the **GridPresentation** dialog
- To switch to the full data range view, click the  button on the **GridPresentation** toolbar

## Adjust contour lines

In the left-hand panel of the GridPresentation window:

- Click the **Draw Contour Lines** button  to toggle the contours on and off (you can also do it from the **GridPresentation** toolbar). Click the **Draw Filled Contour** button to fill your contours with color. When engaged, either button enables the **Contour Attributes** button.

  In the **Contour Attributes** dialog to open:




- Specify the limits of the data values in the **Minimum** and **Maximum**
- Select the contour density from the **Number of Contour Lines** dropdown list
  OR
  Select the **Interval** between each contour line.

- Select the thickness of lines to be drawn from the respective dropdown list
- If you plan to print the current image, also adjust "Printing coefficient", because contour line thicknesses looks differently on screen and paper.
- Select between the **Black** and **Color** options to draw your contours black or colored.
- Check the **Draw Labels** box to specify the format of the contour labels (fixed or exponential, number of decimal digits and font) and their density(number of labels to display).
- Click **OK**

## Display profiles

In the left-hand panel of the **GridPresentation** dialog:

- Click the **Draw Profile Lines** button to toggle profiles on and off. You can also do it from the **GridPresentation** toolbar. When engaged, this button enables the respective section of the dialog

In the **Draw Profile Lines** section:

To specify the same style and color for all of the profiles:

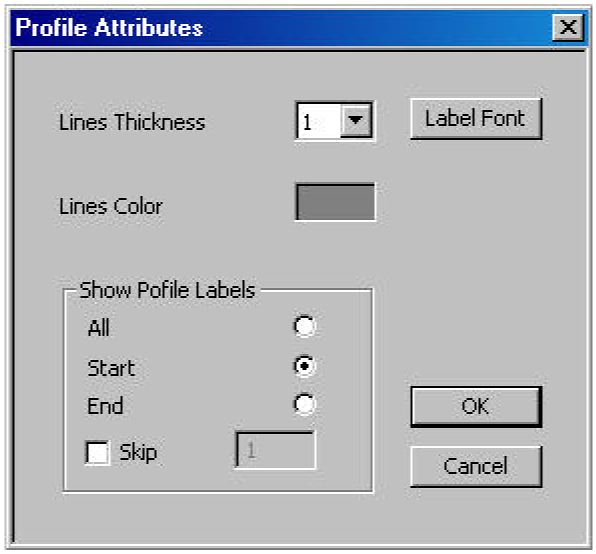
- Click the **Advanced** button and select the line thickness and color in the **Profile Attributes** dialog to appear (see below)

To display your profiles stacked, with different colors assigned to each location dependently of the data value in it:

- Check the **Profile Data Symbols** box and select the size of symbols from the dropdown list to the right. To better view the profiles and data distribution, click the **Hide Grid Mesh** button in the **Draw Mode** section to switch your grid off and leave only profiles on the screen

To display profile labels:

- Check the **Labels** box. Click the **Advanced** button to specify the format, density and location of the labels in the **Profile Attributes** dialog to open:




With the **Start** or the **End** option on, the profile labels will appear respectively at the beginning or the end of the profiles. The **All** option will show labels at both ends. The **Skip** option will skip the display of the number of labels specified.

## Display grid lines

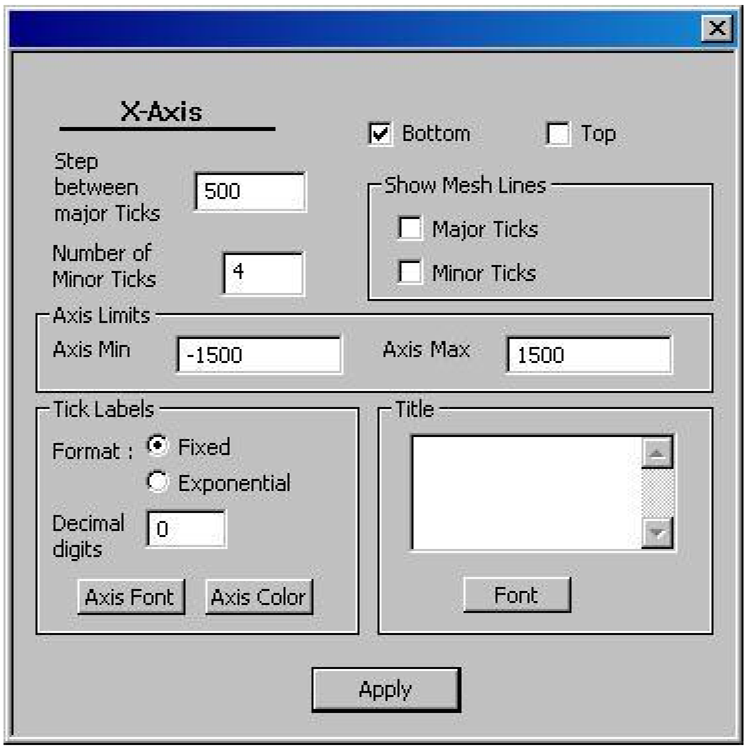
In the right-hand panel of the GridPresentation window:

- Click the **Show or Hide the Grid** button  on the left panel to toggle the grid lines on and off. You can also do it from the **GridPresentation** toolbar.
- If the grid is too dense, specify the number of grid lines to be skipped in the X and Y boxes to the right

## Adjust axes

In the image field of the GridPresentation window:

- Double-click in the region of the X axis to bring up the **X-Axis** dialog or in the region of the Y axis to display a similar **Y-Axis** dialog:




- Toggle the display of an axis on either side of the display using, in the case of the X-Axis, the **Bottom** and **Top** checkboxes.
- Edit the step between major ticks in the respective box.
- Check the **Major Tick** box and the **Minor Tick** box in the **Show Mesh Lines** section to display the coordinate grid.
- Edit the **Axis Min** and **Axis Max** values as desired in the **Axis Limits** section.
- Specify the format and color of the tick labels in the respective section.
- Type in the title of your X or Y axis in the respective field; use the **Font** button to specify the format of the title.
- Click **Apply**.

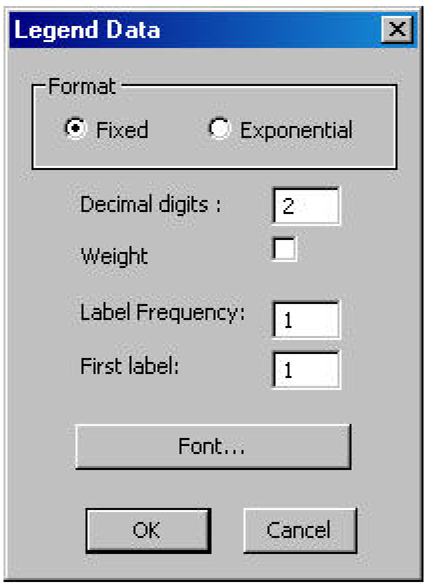
*Note*. To provide the proportional scaling of the axes, use the **Show Proportionally** button .

## Toggle the coordinate grid on and off

- Use the **Show Coordinate Grid** button  on the **GridPresentation** toolbar.

## Display the legend

- To display the legend, use the **Show Legend** button  on the **GridPresentation** toolbar. The legend can be moved to a new location by dragging it with the mouse.
- Click once on the legend and a resizing box will appear to allow the legend to be resized.
- Double click on the legend and the **Legend Data** window will appear:
- The format of the legend data values can be set here.




- The format of the legend data values can be set here.
- Check **Weight** to display how many data values fall into a certain interval
- Control the amount of intervals with **Label Frequency**.
- Leave a number of intervals without a label using the **First label** value.

## Switch to the full-screen view of your grid

Click the **Full Screen View** button  on the toolbar. You can also do it from the **View/Full Screen** menu

## Toggle off bars and panels

- De-select the respective item (**Toolbar**, **Data**, **Info** bars, **Left Pane**) from the **View** menu. To bring it back, select it again

## Zoom in on a grid fragment

- Engage the **Zoom In** button  on the **GridPresentation** toolbar
- Click in the grid and without releasing the mouse select the required fragment
- To return to the initial view, use the **Home** button  on the toolbar.

## Creating Map Underlays

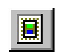
GridPresentation allows you to load an already available map or generate a map to be used as an underlay. Calibrated files must have a *.map extension.

See

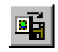
Saving To An Image File

Generate a Map

Load an Existing Map

To toggle the map underlay on and off, use the **Show** and **Hide** buttons in the **Map** section of the **GridPresentation** dialog or the  button on the toolbar.

## Save to an Image File

To save the image displayed to a file, click the  button on the toolbar. The **Generate** button in the **Map** section can also be used. The **Select Raster Type** window appears :

- **Raster with Georeferenced File** will create both an image and a map file. The available map file formats are for EMIGMA/QCTool, ArcView and MapInfo. If you do not desire a map file select **Raster Only**.

  Click **OK** and enter a name for the image file and select the file type before clicking **Save**. Available output file types are jpeg, bmp, gif, png, tiff, tga, pcx, wmf and emf. For jpeg and tiff formats, you will be asked for some compression settings. The default values work very well. The **Bitmap Export** window then appears:


  To modify the resolution:

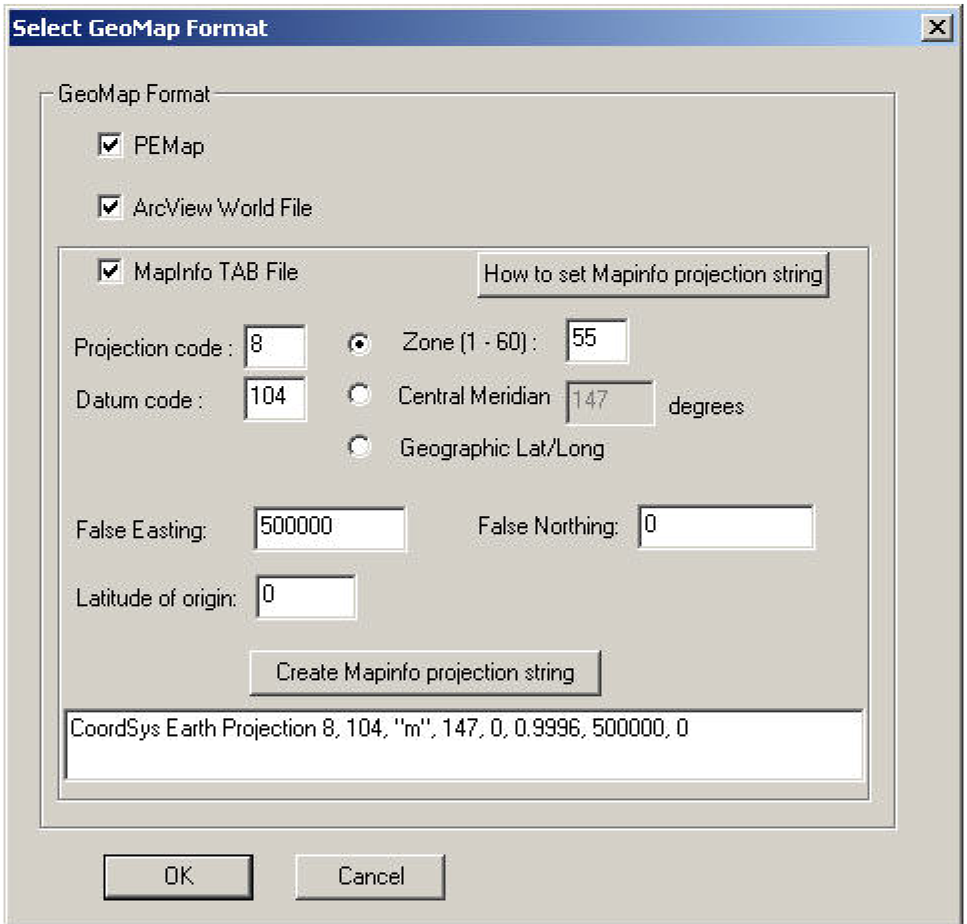
- Enter new values into the **Width** or **Height** boxes to modify the size of the image. Check the respective box to maintain aspect ratio.

- Enter new values into the **Horizontal** and **Vertical** boxes to modify the resolution. One value will be updated to be the same as the other if **Maintain aspect ratio** is selected.
- Select **Original** from the drop down box if it is displaying **Custom** to return to the original values.
- Click **OK**

If you had chosen to create map files, the **Select GeoMap Format** window will appear:


One or all of the options can be chosen. Select **PEMap** for use in EMIGMA, QCTool, and PEGeoMap. Select **ArcView World File** for use in ArcView. Select **MapInfo TAB file** for use in MapInfo. Click **OK** and a window will appear with information on the saved files.

Additional positioning information must be specified for the MapInfo file. A MapInfo projection string is required. One can be generated from the settings you have selected by clicking the **Create MapInfo projection string** button in the following dialog:




See PEGeoMap if you wish to customize your map file.

Also, you can save legend (colour value scale) separately, and then insert it into printed page if needed (see PEGeoMap PageLayout description). For this, right button click over the legend, popup menu string "Save legend bar as raster" will appear.

**NB:**
Note that when you save a grid which is not drawn as "proportional" (i.e., width and height ratio does not reflect real topographic sizes), saved raster also will not be proportional. However, obtained georeferenced image will be geographically correct. When read into mapping software (PEGeoMap, ArcMap, GlobalMapper, etc.), it will be positioned accordingly to its coordinates.

## Load an Existing Map

- Click the **Load** button in the **Map** section of the **GridPresentation** dialog or click the  button on the toolbar. Browse for the required *.map file in the Windows-style Open dialog to appear.

  Also see:

  Saving To An Image File

  Generate a Map

## Save your grid settings

- Click **Save** in the **Settings** section of the **GridPresentation** dialog
- In the **Save Settings** dialog to appear, type in the name for your grid to be saved with or select the existing file from the list and click **Save**. In the latter case, the existing grid will be overwritten with the new one.

## Save your grid to a file

You can save your grid as a *.xyz file

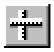
- Select **File/Save Displayed Data in File** from the menu
- Choose a folder and a name for your file then click **Save**

### Load previously saved grid settings

- Click the **Load** button in the **Settings** section in the left panel of the **GridPresentation** window
- In the **Load Settings** window to appear, select the file to open and click **Load**

## Measuring Distance on Grid

To measure the distance between two points on the grid:

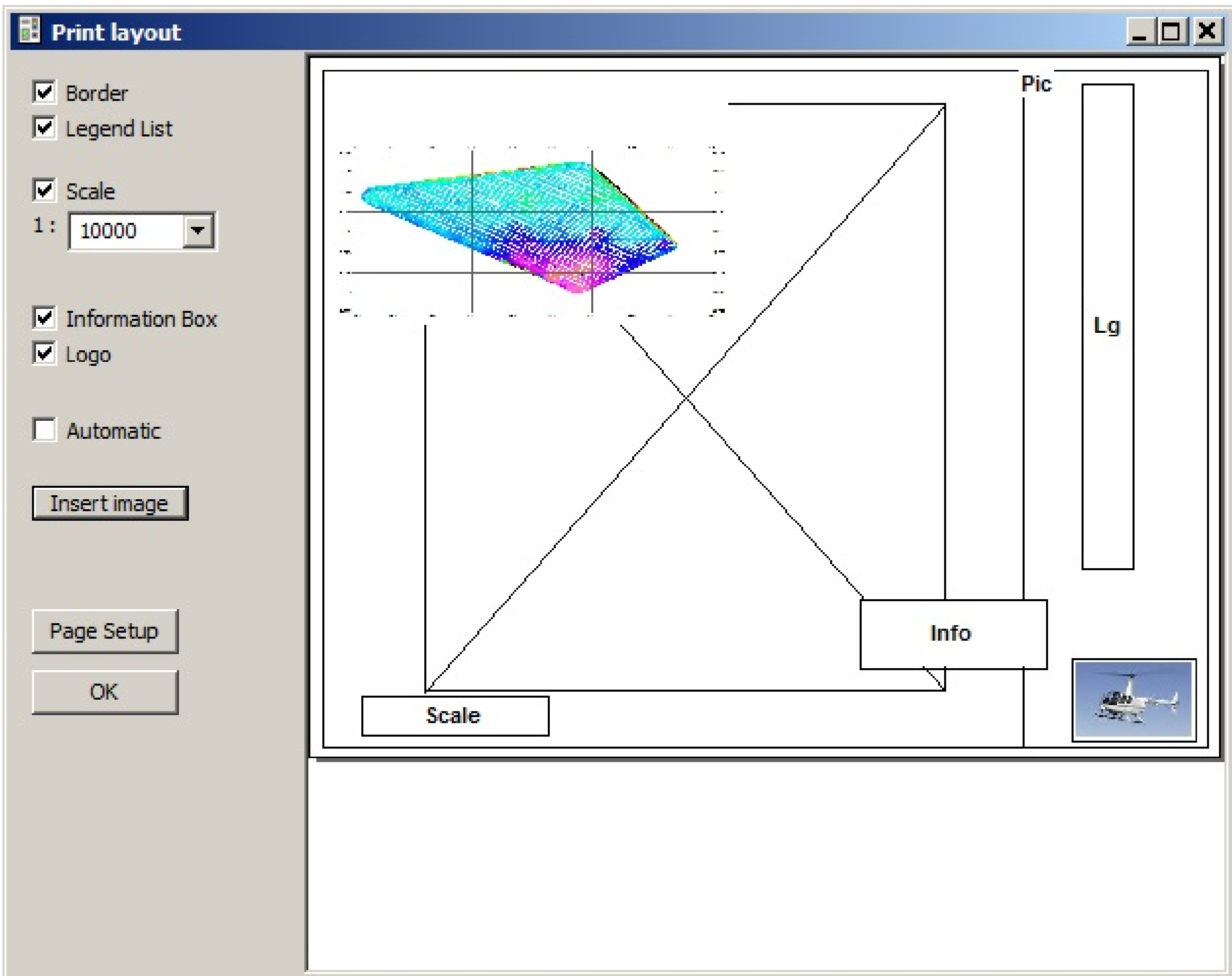
- Click the  button on the toolbar.
- Click on the point on grid where you want to measure from. Don't release the mouse button and drag to the point where you want to measure to. A line is drawn between the two points on the grid display and the distance appears next to the coordinates on the status bar

## Print your grid as is

- Select **File/Print** or click the **Print** button on the GridPresentation toolbar to bring up the standard **Print** dialog and specify print setup

## Specify grid surroundings

- Click the **Print Settings** button  on the **GridPresentation** toolbar. The **Print layout** dialog will open with options listed to the left and a diagram of the page layout to the right:




To print a grid without any surrounding:

- Leave the **Automatic** option checked (it will contain a flag by default)

To print the legend list, scale bar, information box and logo alongside your grid:

- De-select the **Automatic** box.

- Check the **Border** box to draw a border around your grid. The width of the border can be resized with the mouse.
- Check the **Legend List** to have the legend printed to the right of the grid. Double-click on the **Lg** component on the layout diagram to bring up a settings dialog.
- Check the **Scale** box and select a required scale from the dropdown list below. If the scale you want is not listed, type it in manually. Nothe that rescaling in "Print Layout" dialog is reflected also on the screen image.
- Check **Information Box** to have it added to your printed grid. Double-click the box labelled **Info** on the layout diagram to customize the information box.
- Check **Logo** to have a logo on your printout. Double-click the box labelled **Logo** on the display to specify an image file as well as its size.
- Reposition any of the items on the layout diagram by dragging it with the mouse. The grid cannot be repositioned.
- Use the **Page Setup** button to specify page layout and printer settings. Click **OK** to close the dialog and select **File/Print Preview**.

- Click **Insert Image** button to add more raster images to your document. Select image file you want, and then click on the page where you want to place it.
  Images can be moved, resized or deleted. Note that page layout dialog reflects only position of each component, but not its relative size. You can set metric sizes of raster images and info box by double-click on them and entering required numbers.

## Analytic Signal (AS) and Horizontal AS

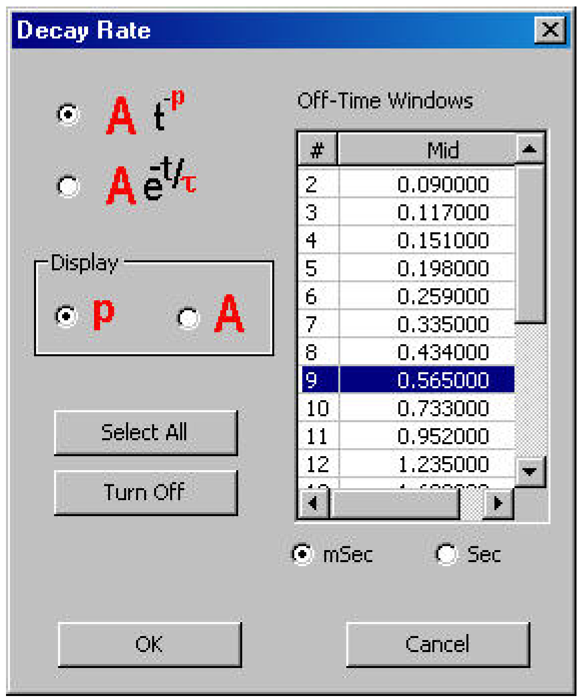
Provided the derivatives have been calculated in your magnetic or gravity survey, you can display the results of processing by the Analytic Signal and Horizontal Analytic Signal techniques

- Under **Processing** in the second toolbar, select **Analytic Signal (AS)** or **Horizontal AS** from the **Processing** drop list to launch the respective processing and view its results

## Decay Rate

For time-domain systems, the decay rate calculation is provided from **Processing**

- Select **Decay Rate** from the **Processing** drop list. The following dialog will open:




- Select the type of decay and the range of decay windows in the respective list on the right to be used in the decay rate calculation

  ***Note**. Only multiple selections are applicable*

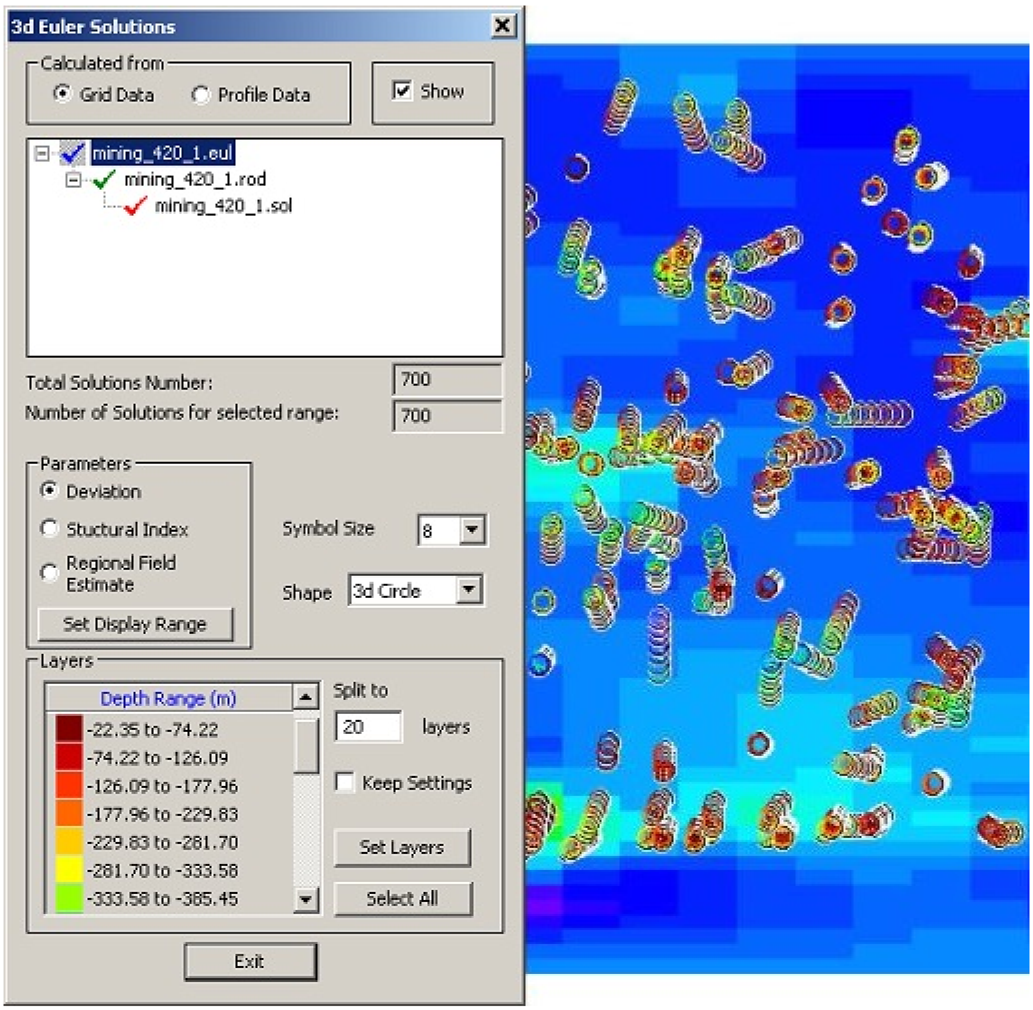
- To select all windows, click the **Select All** button
- To cancel selections and return to initial data, click **Turn Off**
- Select between the two algorithms on the left, then select between the two display options.
- Click **Apply** to view the results.

# Euler Overlays

Provided your data set contains Euler solutions:

- Select **3D Euler** from the **Processing** drop list. The Euler solutions will appear over your grid

  The 3D Euler window opens:



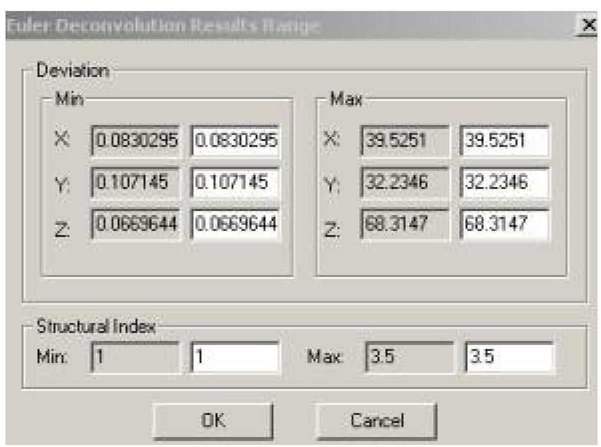

- Leave the **Show** box checked. De-selecting it will switch the Euler overlay off
- In case your data set contains more than one Euler overlay, select the one to view from the list in the upper part of the dialog. The box below

displays information on the total number of solutions.

- Any Rodin and Final solutions available are indicated a plus sign beside the Euler solution file. Click the plus sign to access the post-processed solutions.
- Select **Profile Data** to display a list of Weiner solutions.
- Edit the number of available depths with the box labelled **Split to __ layers**. Edit depth settings in more detail by clicking **Set Layers**.
- Select a particular depth to see its Euler solution or click **Select All** to display all Euler solutions

  The **Number of Solutions for Selected Depth** will change accordingly with your choice of depths

- Select between the **Deviation**, **Structural Index** and **Regional Field Estimate** options in the **Parameters** section and click the **Set Display Range** button to specify the range to be taken into account:




  The gray fields to the left will show you the initial deviation and structural index minimums and maximums. Adjust the minimums and maximums in the active boxes as needed and click **OK** to close the dialog

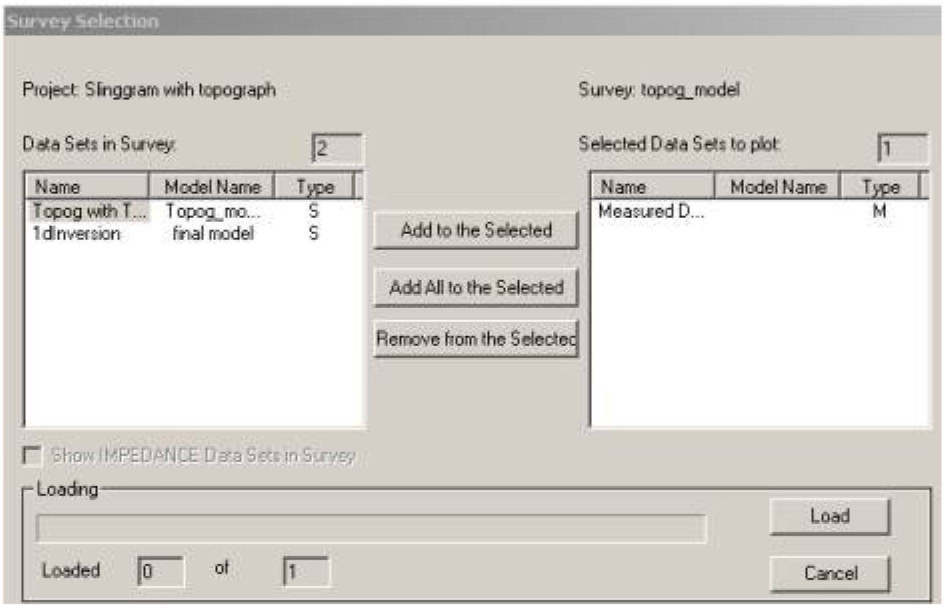
To customize the appearance of your Euler overlay:

- Make selections in the **Symbol Size** and **Shape** sections.

# MultiGrid

The MultiGrid tool is designed for comparison between various components of the same grid. It allows viewing of up to 4 grids at a time and is provided with an easy-to-use interface.

Click on the **MultiGrid** button  of EMIGMA's main toolbar. The **Survey Selection** dialog will appear:




If your survey contains only one data set, the latter will be loaded automatically; if it has two or more data sets:

- To view only the current data set, click **Load**
- To compare your current data set with some other data set in the same survey, select this other data set in the left table, click **Add to the Selected** and **Load**
- To compare your current data set with all the data sets available in your survey, click **Add All to the Selected** and **Load**
- To remove a data set from the **Selected Data Sets to plot** list, select it from the list and click the **Remove from the Selected** button

If your data set contains more than one grid, the most recent one will open. Click in any point of your grid to view its value and x and y coordinates.

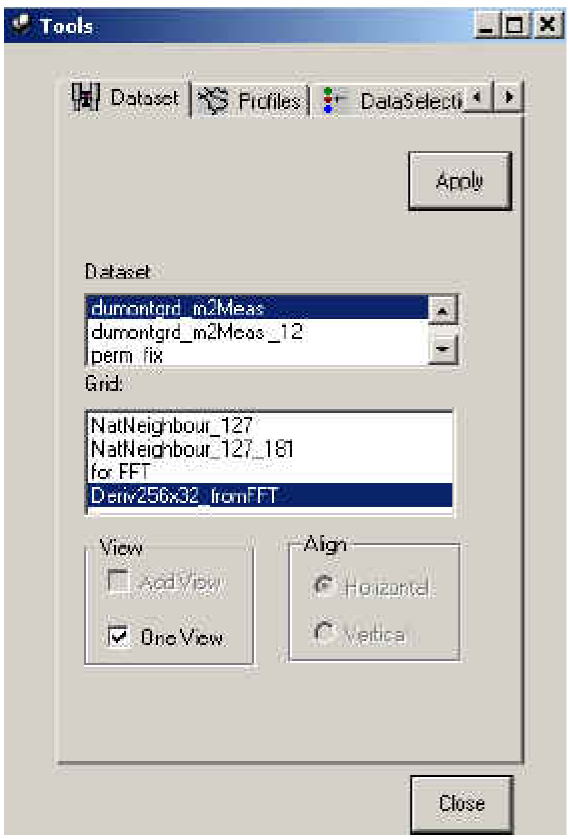
**Related Topics:**

| | |
|---|---|
| Switch between Grids | Display and Customize the Coordinate Grid |
| Display Multiple Grids at a Time | Display and Customize Grid Locations |
| Remove a Grid from the Multiple Grid Display | Load a Map Underlay |
| Compare Various Components of the Same Grid | Add and Customize Titles |
| Display and Customize Profiles | Switch to Proportional Axes |
| Switch between Profile Data and Grid Data | Zoom in and out |
| Specify the Draw Mode | Move a Grid |
| Display and Customize Contours | Hide a Grid |
| Display and Customize the Legend and Scale Rule | Print/Print Preview a Grid |
| Add Annotations to your Grid | |

## Switch between grids

If your data set contains more than one grid, the most recent one will open. To switch to another grid:

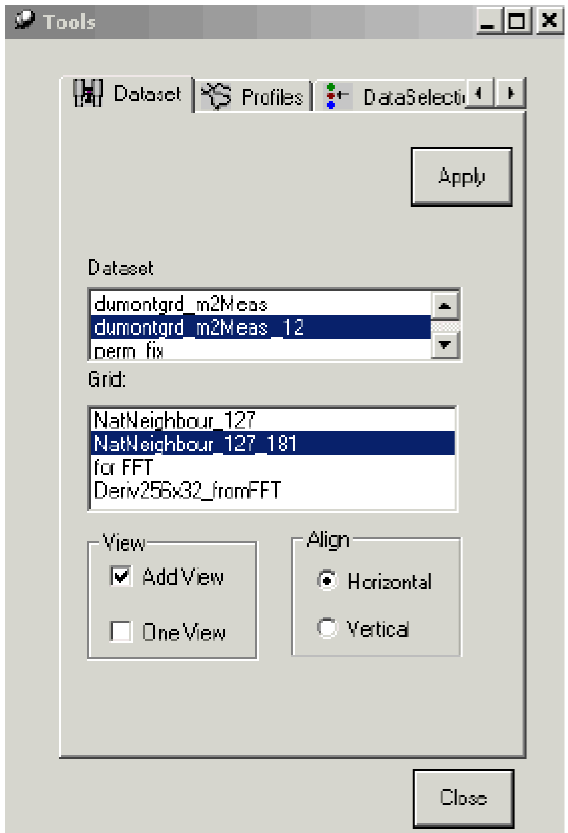
- Click the **Tools** button on the **MultiGrid** toolbar to appear under EMIGMA's main toolbar or double-click anywhere in your grid. The **Tools** dialog will open. **Note**. You can pin the dialog by clicking on the pin in the left corner of its header




- Select a required grid from the **Grid** list on the **Dataset** tab and click **Apply**

## Display multiple (up to 4) grids at a time

Click the **Tools** button  to open the respective dialog. Pin it, if necessary, by clicking on the pin in the left-hand corner of its header:




- On the **Dataset** tab, de-select the **One View** box in the **View** section. This will automatically activate and check the **Add View** box

- Choose between **Horizontal** and **Vertical** in the now active **Align** section to have your grids arranged horizontally or vertically

  ***Note***. To switch from one option to the other in the process of your work, right-click anywhere in your grid and select the required option from the popup menu to appear)

To view grids from the same data set:

- From the **Grid** list of the **Dataset** tab, select the grid you want to view in addition to the one displayed and click **Apply**
- Repeat this procedure, if necessary, to add a third and a forth grid

To view grids from different data sets:

- From the **Dataset** list of the **Dataset** tab, select a required data set from the respective list
- Select the grid you want to view in addition to the one displayed and click **Apply**
- Repeat this procedure, if necessary, to add a third and a forth grid

***Note***. *To switch back to the single-grid presentation, go to the* ***Dataset*** *tab of the* ***Tools*** *dialog and select the* ***One View*** *box; this will bring up the grid that was loaded by default (i.e. the most recent) at the very beginning.*

## Remove a grid from the multiple grid display

- Select a required grid
- Right-click anywhere in the grid and select **Hide Grid** in the popup menu to appear
- To return the grid, see the procedure in **Display Multiple Grids at a Time**

## Compare various components of the same grid

- Click the **Tools** button  to open the respective dialog. Pin it by clicking on the pin in the left-hand corner of its header

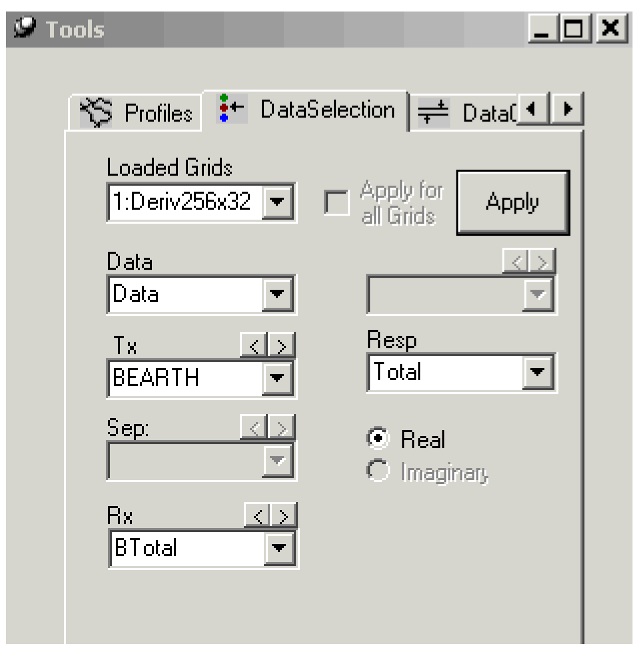
- On the **[Dataset]** tab, de-select the **One View** box in the **View** section. This will automatically activate and check the **Add View** box

- Choose between **Horizontal** and **Vertical** in the now active **Align** section to have your grids arranged horizontally or vertically

- Select the same grid to have its duplicate displayed

- If necessary, repeat this procedure two times more. You will have four similar grids displayed at a time

- Go to the **Data Selection** tab of the **Tools** dialog and select the first grid from the **Loaded Grids** dropdown list (***Note***. You can also select by clicking it on the screen):




- Select a required component you want to display from the dropdown lists below and click **Apply**

- Repeat the same procedure for three other grids

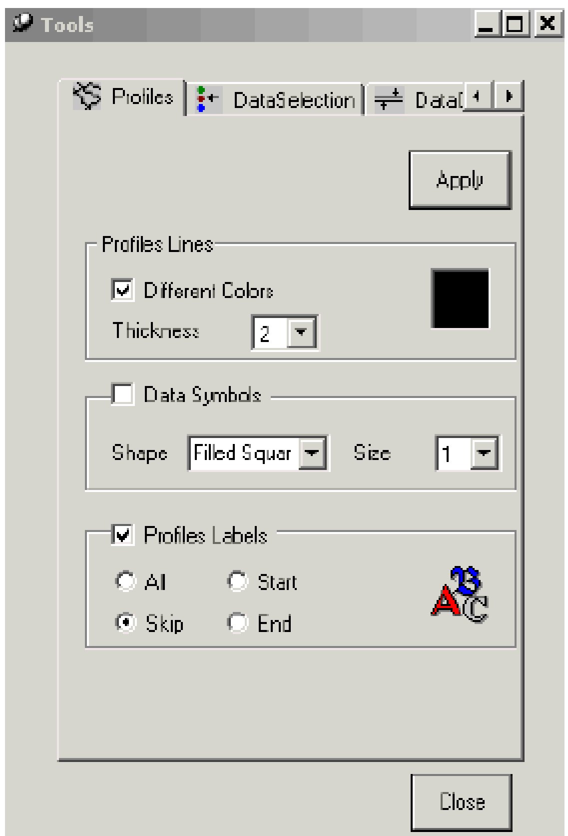
***Note**. To switch back to the single-grid presentation, go to the **Dataset** tab of the **Tools** dialog and select the **One View** box; this will bring up the grid that was loaded by default (i.e. the most recent) at the very beginning.*

## Display and customize profiles

- Click on the grid display you want to overlay with profiles
- Click on the **Show Profiles** button on the **MultiGrid** toolbar

OR

- Right-click anywhere in your grid and choose **Show Profiles** from the popup menu to appear. The profiles will show over the selected grid
- Click on the **Tools** button to open the respective dialog (pin it in the left-hand corner of the header) and go to the **Profiles** tab to customize the appearance of the profiles




- In the **Profile Lines** section, click the color square to specify the color of the profiles using the standard color palette or check the **Different**

**Colors** box to have all the profiles colored differently. Select the thickness of the profiles from the respective dropdown list

- Check the **Data Symbols** box to display the profiles as a set of locations (symbols), each having its own color. Select the shape and size of the symbols to be used from the respective dropdown lists

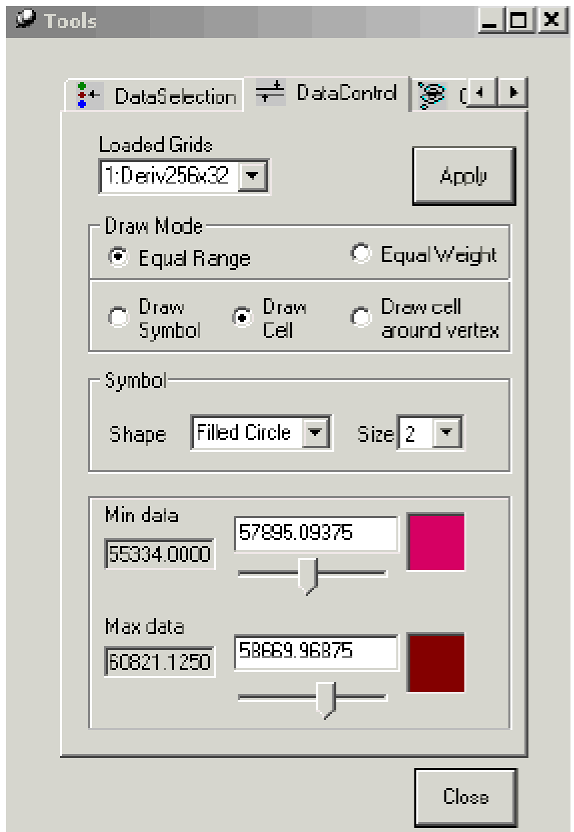
- Check the **Profile Labels** box to display the labels. Select one of the four options below: **Skip** will display labels on every second profile, **Start** – only at the beginning of the profile, **End** – at its end, **All** will display labels at both ends. Click the **ABC** icon to change the font.

## Switch between Profile Data and Grid Data

When you click on a given point in your grid, you can see the x and y coordinates and the data value of this point. When profiles are toggled off, this information is automatically linked to a grid cell (accordingly, the **Point on Grid Data** button  of the **MultiGrid** toolbar is engaged). When profiles are toggled on, the **Point on Profile Data** button  becomes active and can be engaged to display profile-related data

## Specify the draw mode

- Click the **Tools** button  to open the respective dialog. Pin it by clicking on the pin in the left-hand corner of the header and go to the **Data Control** tab




- Select a grid display to specify a draw mode for from the **Loaded Grids** dropdown list or just click on the display of this grid

- Select **Equal Range** to assign different colors to equal ranges independently of the number of points in each range. Select **Equal Weight** to assign different colors to different ranges covering the same number of points

- Select the way for the color to fill the cells (**Draw Symbol**, **Draw Cells**, **Draw Cell Around Vertex**)

**Draw Symbol** displays each grid cell as a set of four data (vertices) and assigns a certain color to each data dependently of its value. **Draw Cell** calculates the average of the data located in the vertices of a grid cell. The cell will be filled with a certain color assigned to the average value. **Draw Cell around Vertex** displays your grid as a set of cells drawn around each grid vertex and filled with a certain color assigned to the data value in the vertex

- If you select **Draw Symbol**, specify the shape and size of the symbols to be used in the **Symbol** section

- Specify the range of data to be displayed (75% of data are displayed by default) using the sliders to the right of the **Min Data** and **Max Data** fields that show the absolute minimums and maximums of your data. Or, type in these values manually. Click **Apply**

- Click on the colored squares to the right to open the standard Windows-style palette and to specify colors for the start and end of the data range

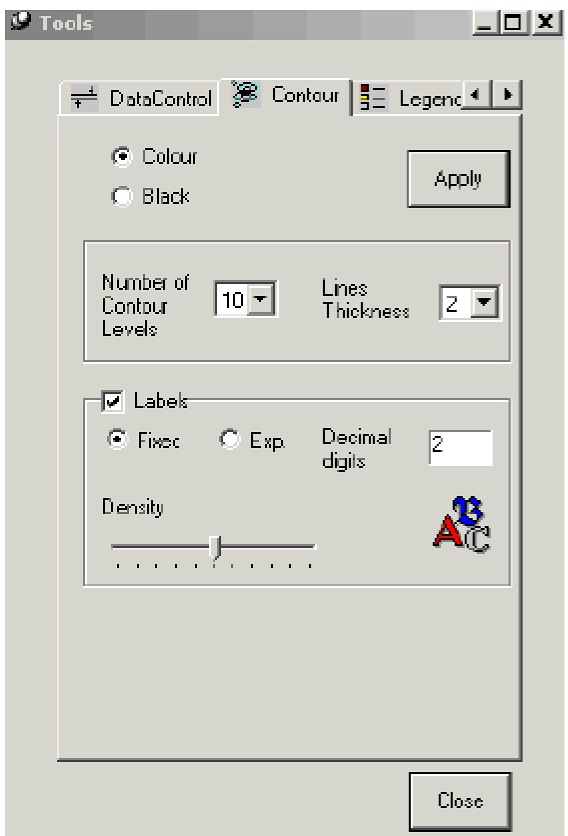
## Display and customize contours

- Click in a required grid display
- Click the **Show Contour Lines** button on the MultiGrid toolbar to toggle the contours on and off

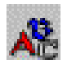
OR

- Right-click anywhere in your grid and choose **Show Contour** from the popup menu to appear

- Open and pin the **Tools** dialog (the **Tools** button on the MultiGrid toolbar) and go to the **Contour** tab




- Select between the **Black** and **Color** options to have your contours black or colored

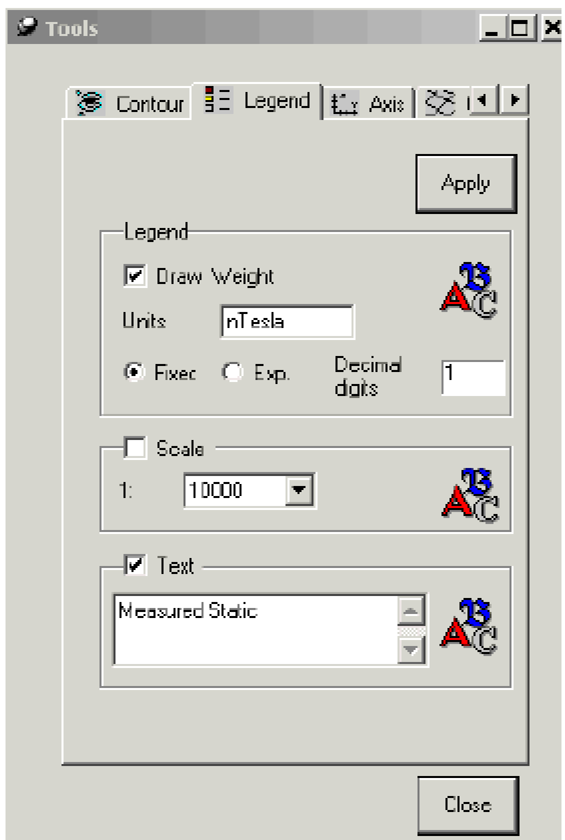
- Select the number of contour levels from the respective dropdown list and the thickness of contour lines to be drawn from the respective list to the right
- Check the **Labels** box to activate the respective section and set the format of the contour labels (fixed or exponential, number of decimal digits). Click on the  symbol to specify the font and style of your label and use the **Density** slider to decrease or increase their number

## Display and customize the legend and scale rule

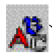
- Click in a grid display
- Click the **Show Legend** button on the **MultiGrid** toolbar

OR

- Right-click anywhere in your grid and choose **Show Legend** from the popup menu to appear. You will see the legend to the left of your grid
- The legend can be repositioned by dragging it with the mouse.
- Open and pin the **Tools** dialog (the **Tools** button on the MultiGrid toolbar) and go to the **Legend** tab




- Select the **Draw Weight** box to display the number of points corresponding to each color in the legend

- Specify the format (fixed or exponential), the number of decimal digits and the font ( ) for the values in the legend
- Check the **Scale** box to authorize scale changes and bring up the scale rule
- Choose a required scale from the respective dropdown list and click **Apply** in the upper right-hand corner of the dialog (or click anywhere in your plot) to view the result. The scale rule will change accordingly

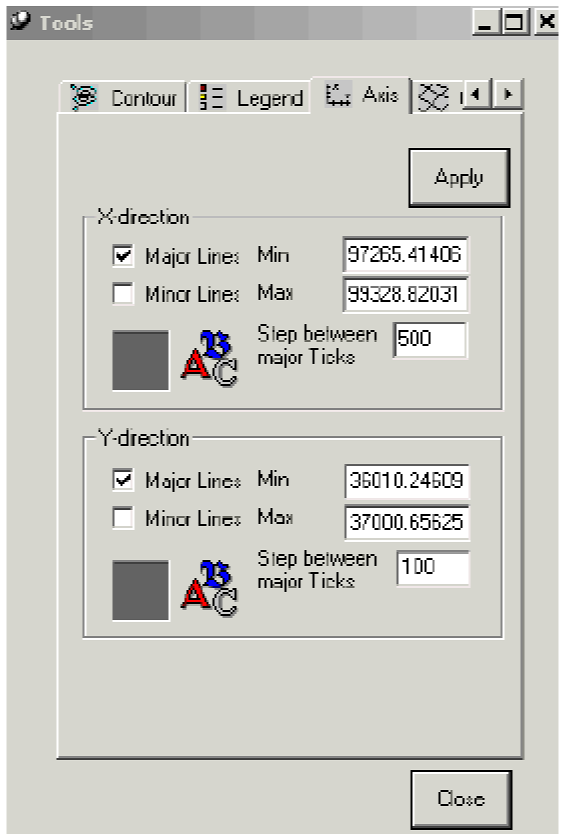
## Add annotations to your grid

- Click in a required grid display
- Open and pin the **Tools** dialog (the **Tools** button  on the MultiGrid toolbar) and go to the **Legend** tab
- Check the **Text** box to add a title or comments

  Type in the field below, press **Ctrl+Enter** to insert a carriage return, click the  button to adjust the font and style of your labels and texts in the standard Windows-style **Font** dialog to appear

## Display and customize the coordinate grid

- Click in a required grid display
- Open and pin the **Tools** dialog (the **Tools** button  on the MultiGrid toolbar) and go to the **Axis** tab:




- Check the **Major Lines** and **Minor Lines** boxes in the **X-direction** and **Y-direction** sections to show the coordinate grid
- Adjust the minimum and maximum coordinates in the respective boxes to display the grid portion you are primarily interested in
- Change the step between major ticks in the respective boxes to increase or decrease the density of your coordinate grid
- Click the color square in the bottom left-hand corner of the sections to edit the color using the standard Windows-style palette to appear

## Display and customize grid locations

- Click on a required grid display
- Click the **Show Grid Locations** button on the **MultiGrid** toolbar to toggle grid locations on and off

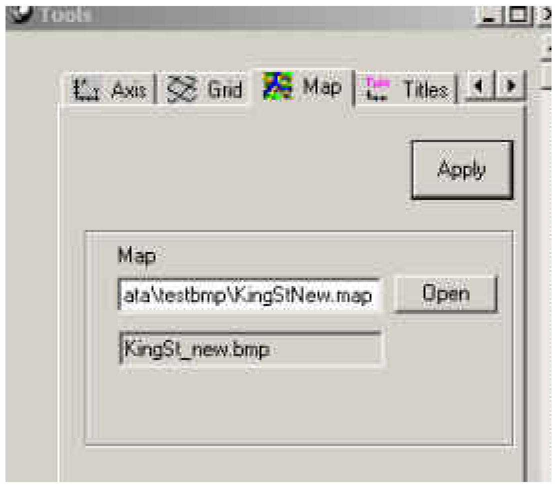
OR

- Right-click anywhere in your grid and choose **Show Grid Locations** from the popup menu to appear
- Open and pin the **Tools** dialog (the **Tools** button on the MultiGrid toolbar) and go to the **Grid** tab:


- Select the grid line thickness from the respective dropdown list and click on the colored square on the right to open the standard color palette and to specify the color
- Change the density of your grid as needed both in the **X-direction** and **Y-direction** boxes in the **Skip Lines** section

## Load a map underlay

- Click on a required grid display
- Open and pin the **Tools** dialog (the **Tools** button on the MultiGrid toolbar) and go to the **Map** tab:




- Click the **Open** button to open the **Select a .map File** dialog, a standard Windows-style Open dialog. Select and load a required map

  *This map is to be calibrated and saved as a text *.map file containing information on externals, internals, etc., and a bitmap (raster) file.*

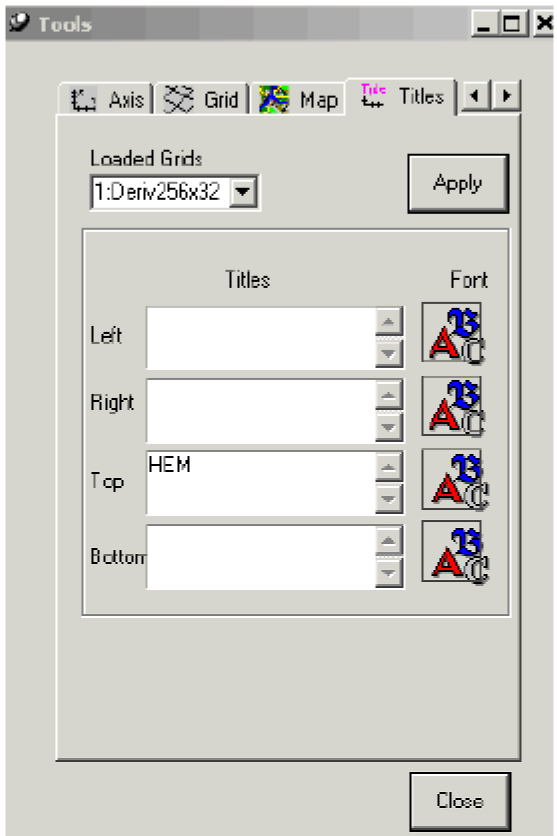
- To toggle the map underlay on and off, use the **Show map** button on the **MultiGrid** toolbar

## Add and customize titles

- Right-click anywhere in a required grid and choose **Show Titles** from the popup menu to appear

  OR

- Open and pin the **Tools** dialog (the **Tools** button on the MultiGrid toolbar) and go to the **Titles** tab:

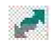



- Select the grid you want to add a title to from the **Loaded Grid** dropdown list or just click on the grid display to select it
- Type in your title in the **Left**, **Right**, **Top** or **Bottom** fields wherever you want it to appear (or create several titles)
- Click on the **Font** button to the right of each field to specify the format of your title

If your title is too big to fit in your display, diminish the grid area using the **Zoom Out** button  on the **MultiGrid** toolbar

- Click **Apply**

## Switch to proportional axes

- Click on a required grid display
- Engage the **Show Proportionally** button  on the **MultiGrid** toolbar. Your grid axes will become of the same scale
- Disengage this button to return to the initial view

*__Note__. If you have multiple grids displayed, all of them will be changed respectively*

## Zoom in and out

- Click on a required grid display
- To zoom in on the whole grid, click the **Zoom In** button  on the **MultiGrid** toolbar. Repeat it as many times as you need
- To zoom out, click on the **Zoom Out** button  as many times as needed
- To zoom in on a grid fragment, engage the **Zoom Selected** button . Click and drag to select a grid fragment to be magnified
- To return to the initial scale, click the **Home View** button 

***Note.*** *If you have multiple grids displayed, all of them will be zoomed in or out respectively, even if you apply this operation only to one of them*

## Move a grid

- Click the **Moving** button  on the **MultiGrid** toolbar
- Click and drag your grid in a desired direction
- To return your grid to its initial position, click the **Home View** button 

## Hide a grid

- Click on a required grid display
- Click the **Show Grid Data** button  on the **MultiGrid** toolbar to toggle it off

OR

- Right-click anywhere in your grid and de-select **Show Grid Data** in the popup menu to appear
- To toggle the grid on again, use the same button on the toolbar or the same command from the popup menu

## Print/print preview a grid

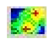
- Select **Print/File Print** and specify printing settings in the standard Windows-style dialog to appear

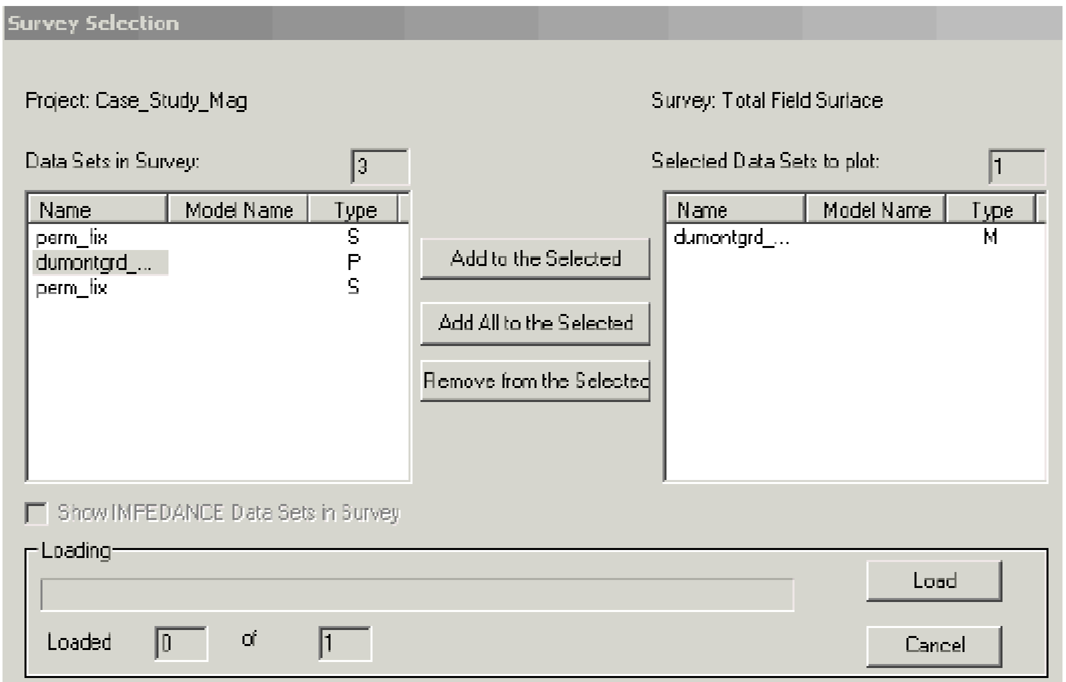
OR

- Select **Print/Print Preview**. Check the appearance of your grid/grids in the print preview window to open and click the **Print** button. Specify printing settings in the standard Windows-style dialog to appear

# Contouring

## Loading Data Set(s)

Click the **EM Contour** button  on the main toolbar of EMIGMA. The **Survey Selection** dialog will open:




If your survey contains only one data set, the latter will be loaded automatically; if it has two or more data sets:

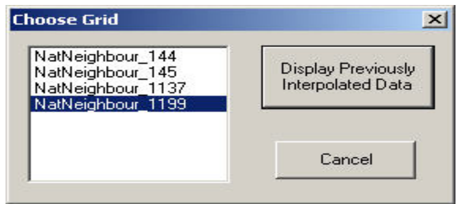
- To view only the current data set, click **Load**
- To compare your current data set with some other data set in the same survey, select this other data set in the left table, click **Add to the Selected** and **Load**
- To compare your current data set with all the data sets available in your survey, click **Add All to the Selected** and **Load**

**Related Topics**

Previously Interpolated Data Sets

## Previously Interpolated Data Sets

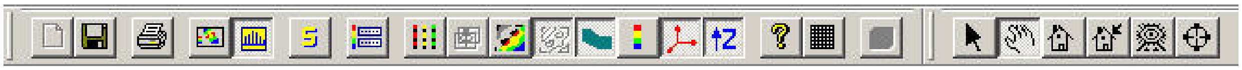
The contour tool will only display data that has already been interpolated. If your data has been interpolated earlier, the **Choose Grid** dialog will open offering you to select the grid to display:




- Select the grid from the list in the left-hand part of the dialog and click **Display Previously Interpolated Data**
- If you wish to switch to a different grid after your selection, you can access the **Choose Grid** dialog by selecting Tools/Gridding/Load Stored Data.

# Working with 3D Surface Contour Displays

Your contour displayed, the dbContour toolbar will appear below the main EMIGMA toolbar:



To the right of your contour display, you will see the **Data Selection** dialog that opens automatically at the same time with contour generation. To close it, disengage the **Select Data**  button on the dbContour toolbar.

**Related Topics**

Adjust Data Selections

Select the Type of your Contour Display

Switch between 2D and 3D Displays

Move and Rotate your Contour Display

Zoom in and out on your Contour Display

Change the Colour of your Contours/Contour Background

Display Profiles

Display Transmitters

Adjust Data Range

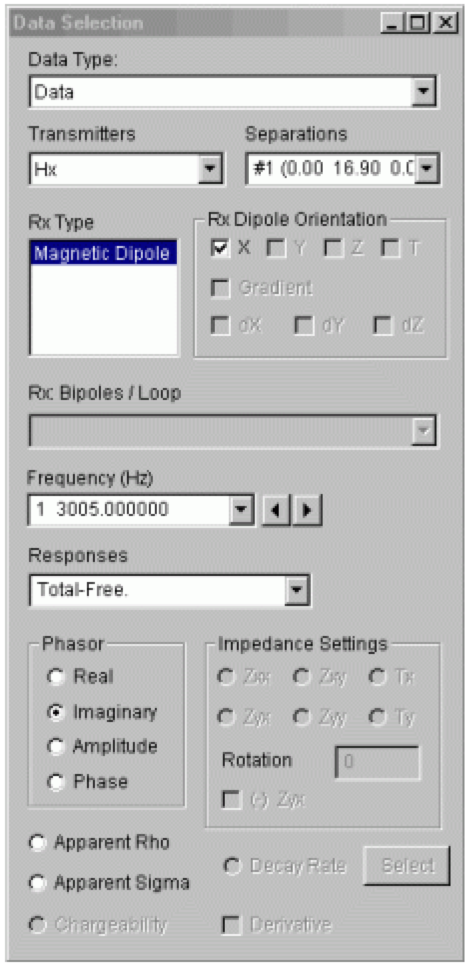
Display the Legend

Adjust Axes

Add a Title/Caption

## Adjust Data Selections

You can check and adjust your selections in the **Data Selection** window to appear simultaneously with your contour display:




Active fields show default selections. E.g., if there is a choice of channels, the channel selected by default will be the first one

Adjust the selections as needed, viewing simultaneously the changes in your contour display:

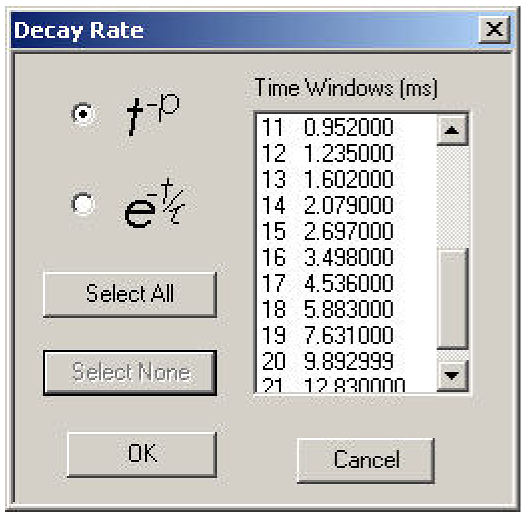
- To switch to a different transmitter, separation, receiver or response, select it from the respective dropdown list or use the scroll buttons on the right side.

- To switch to a different channel (frequency, time), select it from the respective dropdown list or use the scroll buttons on the right side.

- Depending on your system, make required selections in the **Dipole Orientation**, **Gradient**, **Phasor** or **Impedance Settings** sections

- Select **Apparent Rho**, **Apparent Sigma** or **Chargeability** to switch to the respective response

Select the **Decay Rate** option or its **Select** button to display a contour of decay rates (for time-domain systems only). In the interface to open:



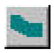

- Select between the two algorithms on the left
- Select the range of decay windows on the right to be used in the decay rate calculation. To select all windows, click the **Select All** button. To cancel selections, click **Select None**

  ***Note***. *Only multiple selections are applicable.*

- Click **OK** to close the window and view the **Decay Rate** contour display.

## Switch between 2D and 3D display

There are two modes of contour display: **Plan** (2D) and **Examiner** (3D). To switch to 2D, select **View/Plan** from the menu. To switch to 3D, select **View/Examiner**.

Clicking the  button on the toolbar will also switch between the two display modes.

## Move and Rotate your Contour Display

To move your contour in the **Plan** view:

- Switch to the hand manipulator 
- Ctrl- or Shift-click the contour display and, without releasing the button, drag it right/left and up/down.

To move your contour in the **Examiner** view:

- Switch to the hand manipulator. This will allow you to examine your contour display from various sides.
- Click the **Set Home** button  on the dbContour toolbar to define the "home" position of your display, which is the position you can always switch back to from any other position
- Click on the plot and, without releasing the button, rotate it in any direction. You can also rotate it relative to the X- or Y-axis by using the dial bars in the bottom left-hand corner of the window
- Press the Shift or Ctrl keys to move the plot up/down and right/left

To make the contour rotate by itself:

- Click on the contour display and, holding the button down, move your mouse in the desired direction of rotation
- Release the button while moving the mouse. The contour will continue rotating
- To stop it, click anywhere in the screen.

## Zoom in and out on your contour display

To zoom in or out:

- Click anywhere in your contour display and, without releasing the button, move your mouse down to zoom in and up to zoom out. If viewing in 3D examiner mode, it is also necessary to hold the SHIFT and CTRL keys while moving the mouse.

***Note***. *You can also use the* ***Zoom*** *dial bar in the bottom right-hand corner of the window*

To zoom in on a certain area of your contour display:

- Click the **Seek** button on the EM Contour toolbar. The message will prompt you to press ‘s’ on the keyboard to seek

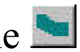
- Click **OK** and press ‘s’ on the keyboard. The hand will change into the crosshairs cursor

- Click the crosshairs cursor over the area you want to examine more closely

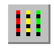
  The display will move nearer, with the area you clicked shifted to the center of your view.

***Notes***. *To return your display to the home position, click the* ***Home*** *button*  *on the EM Contour toolbar. To bring everything into your field of view, click the* ***View All*** *button* .

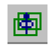
## Change the Color of your Contour and Contour Background

- To switch between the color and black-and-white contour display, click the **Grayed/Color View** button  on the dbContour toolbar.
- To remove the contour shading, select Examiner view using the  button, select Grayed view using the  button and then remove the shading using the  button on the toolbar.
- To change the color of the background, select **View/Edit Background Color.** This will bring up the standard color palette. Set the color and click **OK**.

## Display profiles

Use the **Toggle Profile** button  on the dbContour toolbar to switch the profile display on and off over your contour display.

## Display Transmitters

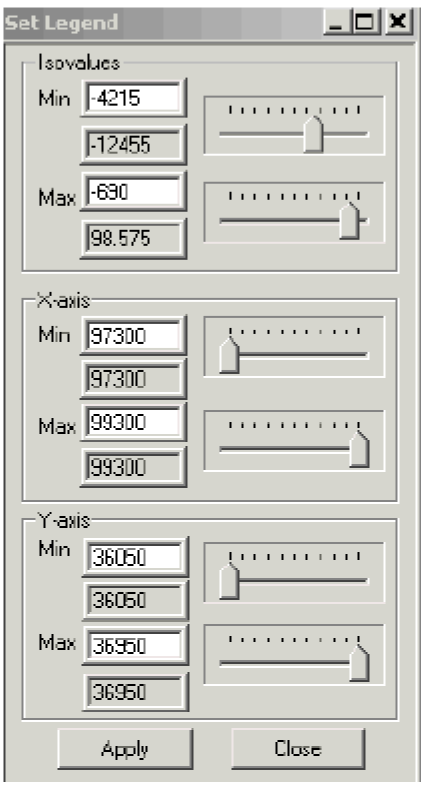
Use the **Toggle Transmitter** button  on the toolbar to switch the transmitter display on and off over your contour display.

## Adjust Data Range

To adjust the range of data to be displayed:

- Click the **Adjust Legend** button  on the dbContour toolbar.

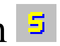
  The **Set Legend** interface appears:

  In the **Isovalues** section of this dialog, you can see the maximum and minimum values of your data to be used as cutoffs. By default, this range covers 97.5% of data

- If required, change the maximum and minimum isovalues. You can do it by typing your values in the active **Min** and **Max** boxes or by using the slider on the right. The disabled boxes below show respectively absolute minimums and maximums of your data

- Click **Apply**. The contour display will change accordingly

- To change back to initial settings, click the **Reset Scaling** button  on the dbContour toolbar.

## Display the legend

- Click the **Toggle Legend** button on the dbContour toolbar to display the legend in the frame on the left of your plot. The range of contour values and colors is generated automatically.

  If you have made changes in the **Isovalues** section, this will show in your legend

- To move the legend to a different place on the screen, click and drag it with your hand manipulator

- To switch the legend off, click the **Toggle Legend** button again.

## Adjust axes

The axes are on by default. To turn them off, click the **Toggle Axis** button  on the dbContour toolbar. Click it again to bring the axes back.

See also

Change the Length of Axes

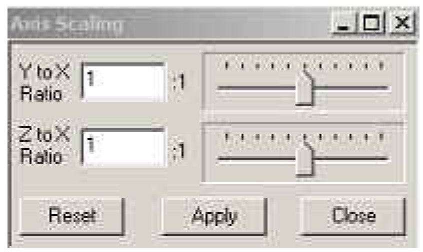
Change Axis Labels

Change Coordinate Settings

## Change the length of axes

- This is enabled through the Z icon on the dbContour toolbar

- Maximize the **Axis Scaling** dialog that appears in the bottom left-hand corner of the screen simultaneously with contour generation



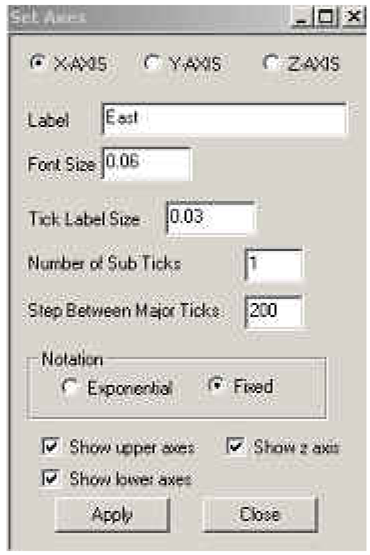

- Type in the required values of Y to X and Z to X ratios (or use the slider to the right) and click **Apply**

- To change back to the initial value, click **Reset**

***Note***. *The* ***Z-Scaling*** *button*  *on the dbContour toolbar switches the* ***Axis Scaling*** *(minimized) dialog on and off*

## Change axis labels

- Select **Plot->Plot Settings/Axes** to bring up the **Set Axes** dialog




- Type new labels for your plot and edit their font size, determine the step between major ticks and the size of their labels, and specify the number of subticks in the respective boxes

- Click **Apply** to view the changes

- Click **Close** to close the dialog

## Change coordinate settings

- Click the **Adjust Legend** button  on the dbContour toolbar.

  The respective window appears (see Adjust Data Range)

- Edit your settings in the X-axis and Y-axis areas of the dialog to see only a section of the contour.

- Click **Apply** to check your changes without closing the window

- Click **Close** to close the window when finished

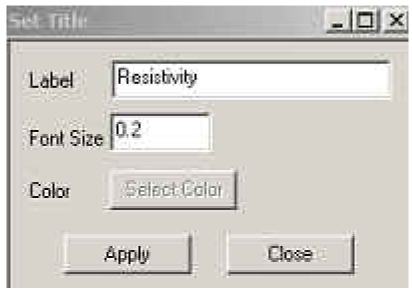
***Note***. *To view the coordinates of any point of your contour display, hold your arrow manipulator over this point. Read the coordinates in the bottom left corner of the window right under the dial bars.*

## Add a title/caption to your contour display

To add a title/caption to your contour plot:

- Select on the main menu **Plot/Plot Settings/Title**

  The **Set Title** dialog appears:



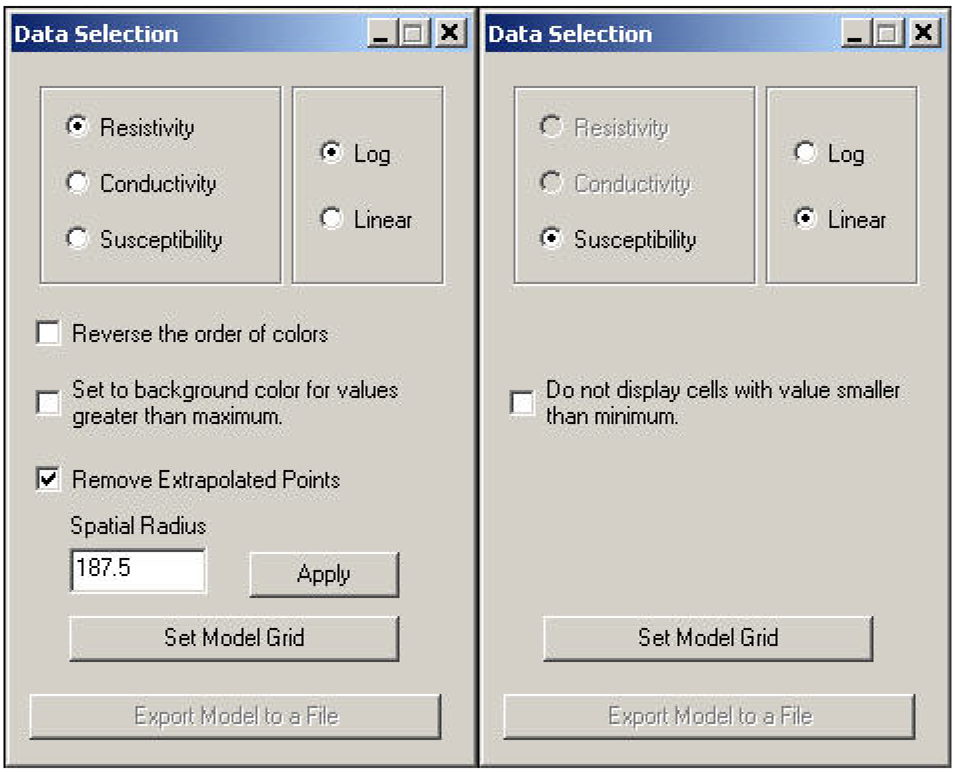

- Type your title in the **Label** field and specify the font size in the box below

- Click **Apply** to view the result

- Click **Close** to close the dialog.

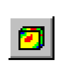
# Viewing Inversion Results

You may load either 3D inversion results or multiple profiles of stacked 1D inversions. On loading a data set with inversion results, it will be possible to choose from displaying the data as Resistivity, Conductivity or Susceptibility(Density for gravity data). Also, the legend can be toggled between a log and linear scale:




Select the checkbox labelled **Set to background color for values smaller than minimum** and the background colour will be assigned to those values that are less than the legend's minimum value or **Do not display cells with value smaller than minimum** for magnetic or gravity data.

The display can be further analyzed using the **Section Cutting** tool available by selecting Tools/Section Cutting from the menu.

Switch to gridded data by deselecting the  button on the toolbar.

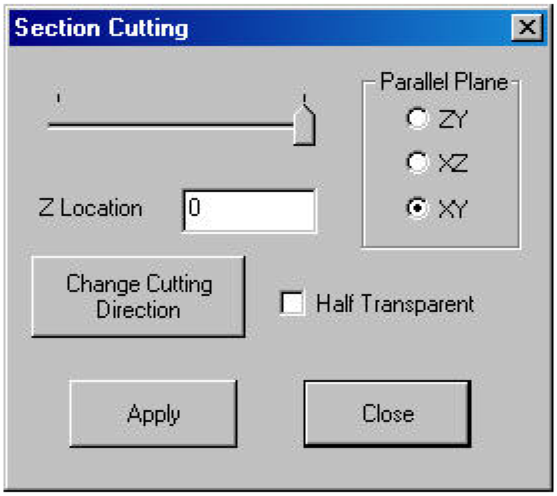
You may process and export your inversions through the toolkit(**Processing and Filters**) on the main **EMIGMA** menu through the **Inversion Model Processing** selection

For 3D volume 1D inversions, select **Export Depth Slices/Cross Section**

For 3D inversion volumes, there are a wide variety of processing and export capabilities for your 3D inversion whether Magnetic, Gravity, Resistivity, CSEM, MT or CSAMT

## Using the Section Cutting Tool

The **Section Cutting** tool is available by selecting Tools/Section Cutting from the menu:




To view only a section of the inversion result:

- Select the axis that the desired cutting plane is parallel to in the **Parallel Plane** section.
- In the **Location** box, enter the location where you want your inversion cut.
- Click **Apply** and the inversion volume beyond the cutting plane you have chosen will disappear.

To dynamically change the location of the cutting plane:

- Move the slider located at the top of the dialog box and the size of the volume will change as you move the slider.

To view the section on the other side of the cutting plane:

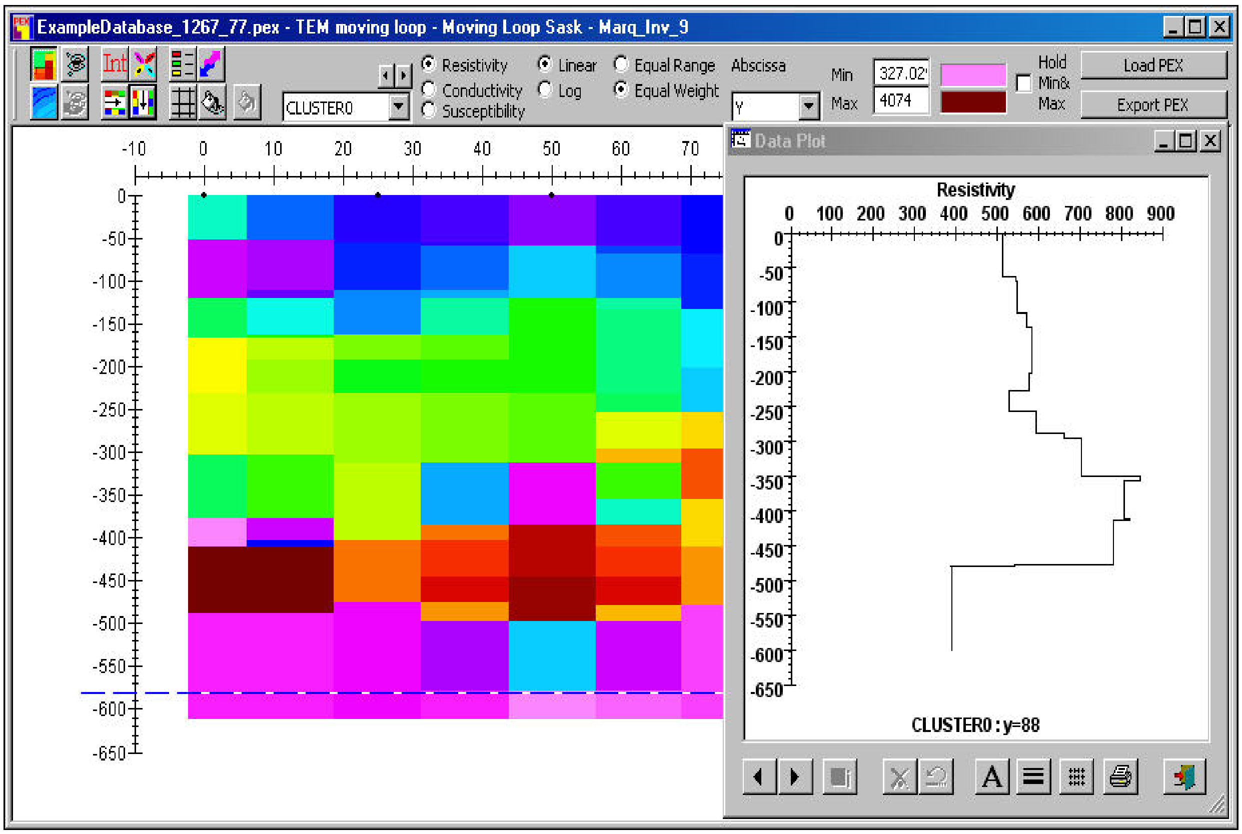
- Click **Change Cutting Direction** and then either click **Apply** or move the cutting slider at the top of the dialog.

To make the inversion partially transparent:

- Click the **Half Transparent** checkbox

# CDI Viewer

The CDI Viewer (PEX Show) tool displays your inverted EM, Resistivity, MT and CSAMT data two-dimensionally. If you performed 1D inversion within EMIGMA, the **Data Sets in Survey** list on the **Database** page of the main dialog will contain a data set linked to a *.pex file. Select it and click the now active **CDI Viewer** button  on EMIGMA's main toolbar to open the respective application.




By default, the CDI (Conductivity Data Image) to appear will be the resistivity pattern for your first line. The separate **Data** dialog to simultaneously open on the right will contain the resistivity versus depth response for the first location of this line.

*Note*. If the **Data** dialog closes (and it does if you make any changes to your grid display), double-click anywhere in the grid to bring it back. The response it will show on reopening will match your current location.

**Related Topics**

| | |
|---|---|
| Go through all Locations of a Line | Customize the Range of Data to be Displayed |
| View Cell Information | Perform Interpolation |
| Better View the CDI Pattern | Display and Customize the Legend |
| Switch between Lines | Customize the Axes |
| Switch between Resistivity, Conductivity and Susceptibility | Draw and Customize Contours |
| Switch between Linear and Logarithmic Scales | Print your CDI |
| Specify the Draw Mode | |
| Switch between the X and Y Axes | |

## Go through all locations of a line

- Use the **Previous** and **Next** buttons in the bottom of the **Data** dialog appearing over your CDI in the main **CDI Viewer** window).

  The plot in the window will switch to the previous or next location, respectively. The vertical red bar will simultaneously move across the CDI indicating the exact location of the plotted data

## View cell information

- Click and hold down in a required cell of the CDI.

  The location, depth and data will be displayed, while the **Data** dialog will automatically switch to the response obtained in the corresponding location.

## Better view the CDI pattern

- Click the  button in the CDI Viewer toolbar. Your grid will be divided into equal cells, each containing one depth data

## Switch between lines

- Select a required line from the dropdown list  in the upper left-hand corner of the **CDI Viewer** window or toggle through all the lines using the scroll buttons above this list.

## Switch between resistivity, conductivity and susceptibility

- Select the desired option on the CDI Viewer toolbar

## Switch between linear and logarithmic scales

- Select between **Linear** and **Log** on the CDI Viewer toolbar.

## Specify the draw mode

On the CDI Viewer toolbar:

- Select **Equal Range** to assign different colors to different ranges which are equal independently of the number of data falling within these ranges
- Select **Equal Weight** to assign colors to different ranges which are unequal but covering the same number of data

## Switch between the X and Y axes

- Select between the **X** and **Y** buttons on the CDI Viewer toolbar. They both will be active unless the direction of survey lines is known exactly.

## Customize the range of data to be displayed

On the CDI Viewer toolbar:

- Type in new minimum and maximum data in the respective boxes to set the new extreme data values. You can select "Hold Min & Max" option to keep displayed data extremes while you navigate through profiles
- Click on the color squares to the right to bring up the standard color palette to specify the range of used color

# Perform interpolation

In the CDI Viewer toolbar:

- Click on the **Interpolate** button

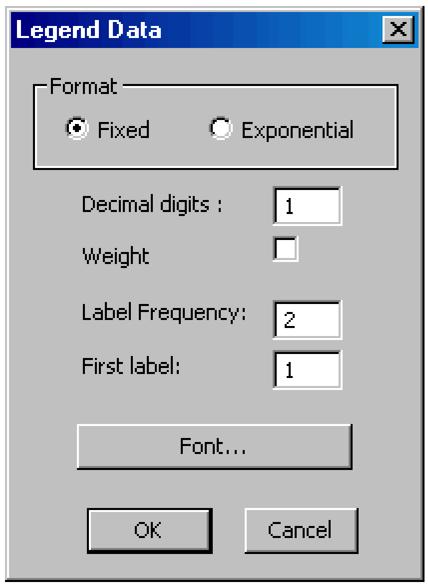
OR

- Use the **Single Row Interpolation (Depths)** and **Single Column Interpolation (Locations)** buttons to specify the kind of linear interpolation. The first interpolates between different depths at the same location, the second – between different locations at the same depth
- To cancel interpolation results and return to the initial view, click the **Reset** button .

# Display and customize the legend

- Click on the **Show Legend** button on the CDI Viewer toolbar or select **View/Show Legend** from the menu. The legend will appear to the right of the CDI. It can be repositioned by dragging it with the mouse.

- To customize your legend, open the **Legend Data** dialog by double-clicking anywhere in the legend area or select **Settings/Legend** from the menu:



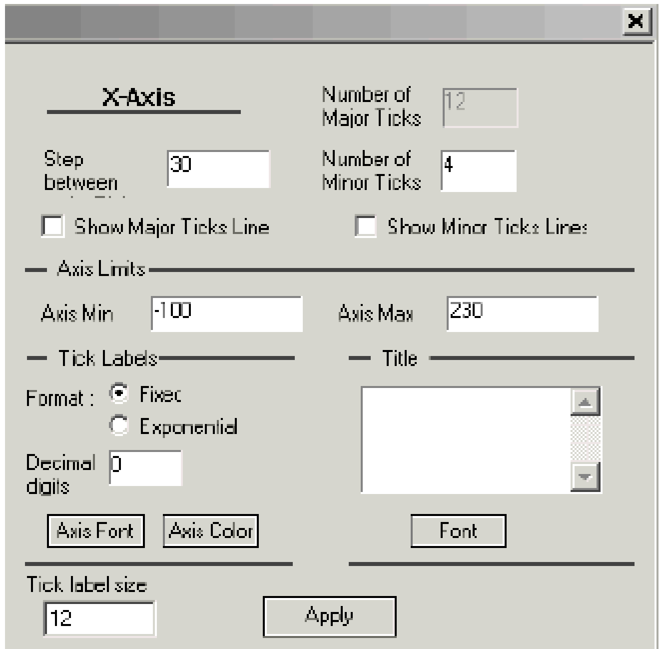

On this interface:

- Select between **Fixed** and **Exponential**
- Specify the number of decimal digits in the respective box
- Check the **Weight** box to display the number of data covered by each color range
- Specify the amount of value labels on the legend in the **Label Frequency** box.
- Specify how many labels to skip before displaying the first label in the **First label** box.

- Click the **Font** button to change the font, size and style of the legend labels in the standard Font dialog to open
- Click **OK**

## Customize the CDI axes

- Double-click in the region of the axis you want to adjust. The respective dialog will open:

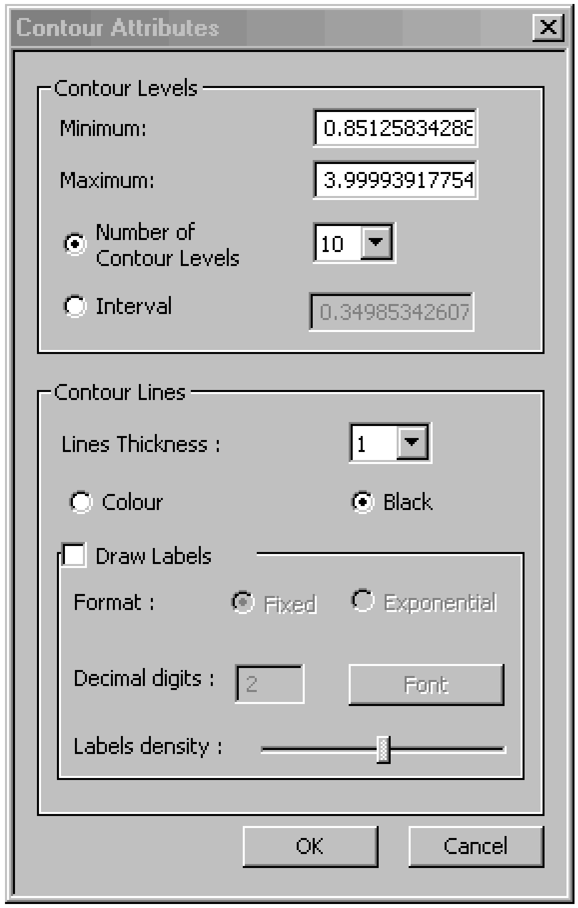



- Edit the step between major ticks in the respective box. The number of major ticks to be displayed will change accordingly
- Check the **Show Major Tick Lines** box and the **Show Minor Tick Lines** box to display the coordinate grid
- Edit the **Axis Min** and **Axis Max** values as desired in the **Axis Limits** section
- Specify the format (fixed or exponential, number of decimal digits), font, size and color of the tick labels in the respective boxes
- Type in the title of your X or Y axis in the **Title** field; use the **Font** button to specify the format of the title
- Click **Apply**

*Note*. For the proportional scaling of the axes, use the **Proportional View** button on the CDI Viewer toolbar.

## Draw and customize contours

- On the CDI Viewer toolbar, click the **Show Contour** button . Select **Settings/Contour Attributes** and the **Contour Attributes** dialog will open:




- The limits of the data values can defined in **Minimum** and **Maximum**.
- Specify the number of contour levels in the respective dropdown box
  OR
  enter the interval between contour lines in the **Interval** box.

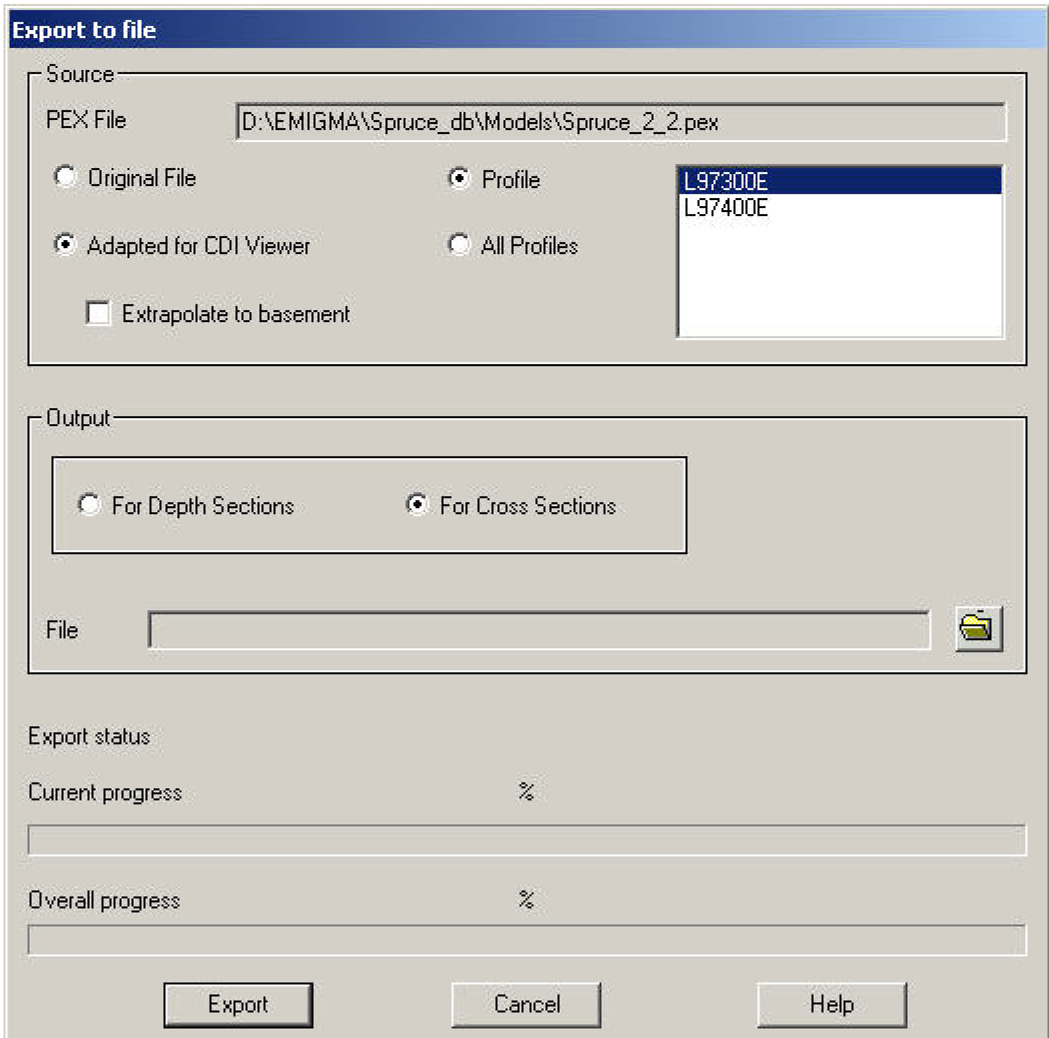
- Select between the black and color options to have your contours in black or different colors

- Choose the contour line thickness from the respective dropdown list

- Select the **Draw Labels** box to add contour labels
- Specify the label format (fixed or exponential, the number of decimal digits, the density of labels and their font)
- Click **OK** to close the dialog and view the results

# Export PEXShow sections to XYZ or QCT

Click the **Export PEX** button on the CDI Viewer toolbar. The following window appears:




**Original File** vs **Adapted for CDI Viewer**
The inversions results are stored in a .pex file in the /models subdirectory. The "original file" selection simply exports this file to a new file. The "adapted" selection reorders the inverse model data for viewing either as a pseudosection or a depth slices for other software applications.

**Profile** vs **All Profiles**
Output a selected profile or all profiles

Two choices of format : ASCII xyz format or output as .qct for opening in QCTool

**Output**
**For Depth sections**: outputs one spreadsheet per depth and per profile for .qct files . For xyz files, one header per depth for each profile
**For Cross sections**: outputs one spreadsheet per profile for .qct and one header per profile for xyz format

Click the  button to select a name for the output file.

Click **Export** to create the file.

## Load another PEX file

- Click the **Load PEX** button on the CDI Viewer toolbar.
- In the window to appear, select the file to open and click **Load**.

## Print your CDI

The **File** menu offers the standard Windows-style **Print**, **Print Preview** and **Print Setup** interface.

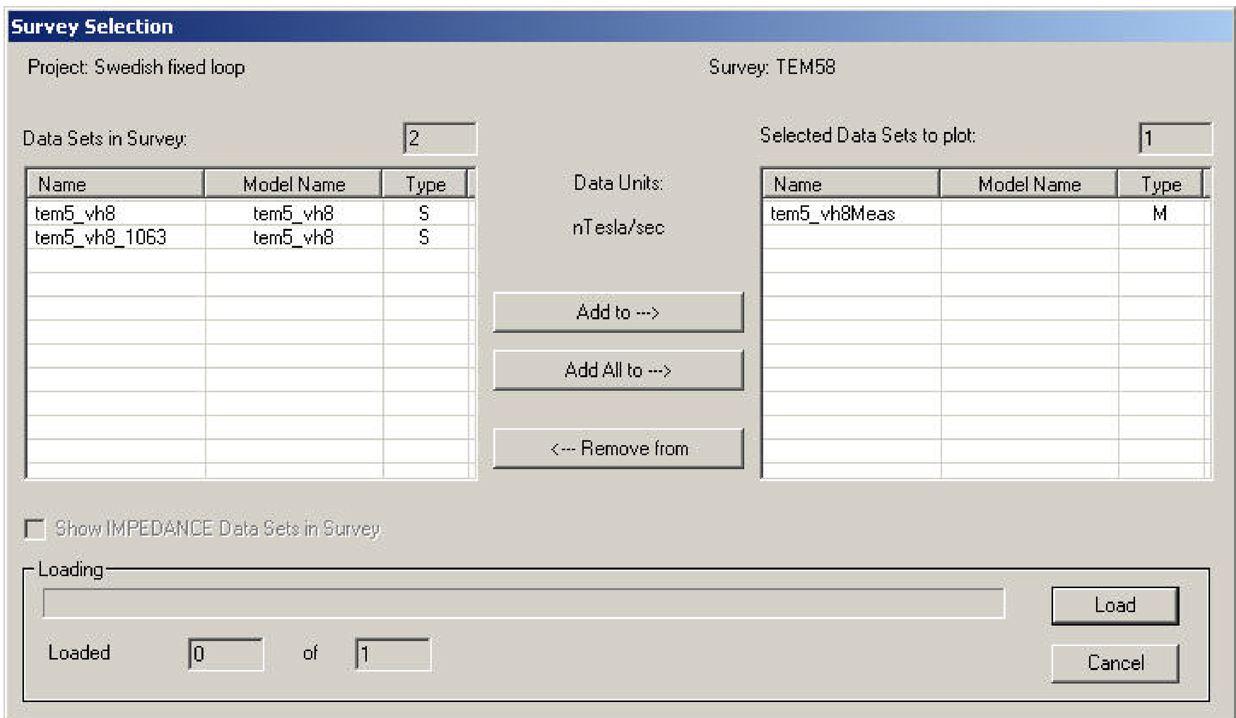
# Visualizer

## Starting the Visualizer

To use the visualizer select a data set you would like to view from the [Database]() page. Click the 

button on the main EMIGMA toolbar.

- The **Survey Selection** dialog will open offering you to choose data sets to be compared with your current data set:




- Select a data set from the list on the left and click **Add to**. In case you want to compare all the available data sets, click **Add All to**
- To remove a data set from the list on the right, select it and click **Remove from**.
- Click **Load**

# Visualizer. Toolbar.

| Click | To |
|---|---|
| | Add or remove data sets to be analyzed in Visualizer. Launches Survey Selection dialog. |
| | Save model to the database |
| | Print the active view |
| | Toggle transparent views of layers on and off |
| | Toggle anomalies on and off |
| | Toggle data on and off |
| | Insert a new prism or plate into the scene |
| | Import model object from a survey in the current database |
| | Duplicate a selected prism or plate |
| | Split the selected layer in half. The two new layers can be given different attributes. More easily done in Model Configuration |
| | Toggle axis on and off |
| | Step back one frequency or time channel |
| | Step forward one frequency or time channel |
| | Rescale data so maximum value appears at a reasonable height above the profiles |
| | Show a vector field created from Source Distribution application. Launches Background Fields window. |
| | Saves the current model to a new data set if the survey contains only measured data. Otherwise, save to a new dataset or overwrite the present dataset |
| | Select/Pick Button |

Selects object manipulation or pick mode (and deselects camera or viewer mode). The cursor shape will change to an arrow. In this mode, the user is manipulating objects in the scene graph. You can select and edit objects in the viewer by clicking and dragging with the mouse.

View Button

Selects camera or viewer mode (and deselects object manipulation or pick mode). The cursor shape will change to a hand icon. In this mode, the user is moving the camera in 3D space. You can change your viewpoint by dragging the mouse in the graphic area.

Home Button

Returns the camera to its home position (initial position if not reset).

Set Home Button

Resets the home position to the current camera position.

View All Button

Brings the entire scene graph into view.

Seek Button

Allows the user to select a new center of rotation for the camera. When clicked on (and in viewer mode) the cursor changes to a crosshair. The next left mouse buttonpress causes whatever is underneath the cursor to be selected as the new center of rotation. Once the button is released, the camera animates to its new position.

Camera Alignment Buttons

Select the axis of alignment (X, Y, or Z) of the camera. Only displayed for Plan Views.

## Visualizer. Background or Source Fields.

These are calculated in **EMIGMA** by utilizing the **Source Distribution** calculations *(Lyr)*

Here, you may compute either the background fields and/or the fields of the source over a volume grid

A new dataset is produced in the survey and when you load this dataset, the survey is shown.

To show the source/background fields select the Lyr button on the Visualizer toolbar

The following window will appear:

**Response, Phasor, Tx, Frequency/Window** Select the desired data.

**Vector Comp**
Choose the components of the vector to be displayed.

**H Color, E Color**
Set the colors for the magnetic and electric fields.

**Line Thickness**
Set the thickness of the vector lines.

**Scale**
The slider controls the length of the vector lines
Select **Normalize Vector** to set all the vector lines to an equal length.

## Visualizer. Menu commands.

The following options are available from the menus:

| | |
|---|---|
| **File** | Retrieve, save, print, and export files |
| **View** | Change the display of parts of the view |
| **Data** | Change data display parameters |
| **Model** | Add or edit models |
| **Manips** | Change manipulation options |
| **Lights** | Add or edit lights |
| **Options** | Set viewing parameters |
| **Window** | Manage multiple windows |
| **Help** | Help options |

## Visualizer. File menu commands.

The following options are available from the menus:

| | |
|---|---|
| **Open** | Add or remove data sets to be analyzed in Visualizer. Launches Survey Selection dialog |
| **Close** | Closes an opened document. |
| **Save** | Saves a model with a prompt to create a new data set. |
| **Save Image** | Saves to a 3d pdf, a cgm file or bmp file type. |
| **Print** | Prints the screen. |
| **Print Preview** | Displays the document on the screen as it would appear printed. |
| **Print Setup** | Selects a printer and printer connection. |
| **Exit** | Exits Visualizer application. |

## Visualizer. View menu commands.

The View menu offers the following commands:

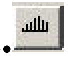
| | |
|---|---|
| **Toolbar** | Shows or hides the toolbar. |
| **Status Bar** | Shows or hides the status bar. |

**Viewer types:**

| | |
|---|---|
| **Examiner** | Set the viewer type to the examiner viewer. |
| **Plan** | Set the viewer type to the plan viewer. |

## Visualizer. Data menu commands.

| | |
|---|---|
| **Show** | Show the data |
| **Plot Style** | Select whether you want a vector, curve or surface data representation |
| **Plot Settings** | Edit data plotting parameters and the appearance of the axis |
| **Filters** | Apply a reduction to the data. Useful when data values are large but have little variation. This function will be performed automatically for static DC magnetic data. |
| **Model Label** | Choose whether you want the data set to be identified by the Model Name or Survey Name on the **Select Data** dialog. |

## Visualizer. Select Data. 

**Data Type** Select which kind of data to be displayed, when there is more than one kind of data in a data set.

**Responses**
List of available responses.

**Phasor**
Select the desired phasor to be displayed

**Transmitters**
List of available transmitters. Use arrows to step through transmitters quickly.

**Receiver**
List of available receivers.

**Separations**
List of available separations.

**Frequency/Time Window**
List of available frequencies/time windows. Use arrows to step through frequencies/time windows.

**Rx Comp & Projection**
Choose any combination of the x, y and z components. If more than one component is selected, the data displayed will be the square root of the sum of squares of the selected components. Choose more than one component by right clicking on the component. A left click will deselect any other components.

**Derivative**

Check off to show inline derivative of data.

**MT Settings**
Select the desired impedance tensor. Any change to the MT rotation angle will be applied when an impedance tensor is selected.

**Plot Data**
Select which data set will be displayed. Both or just one can be chosen. If data set has not been chosen before, a default data set will be displayed. The third data set will display the difference between the first two data sets. This means to see residual data, data must first be selected for data set one and two.

**Change Settings**
Select which data set will be modified

**Gradient**
Show the derivative of the total data with respect to the selected direction. More than one derivative can be selected and length of the resulting vectors is calculated and displayed.

**Scale**
Modify the maximum amplitude of the data. The amplitude can be modified further using Max Amp under the menu item Plot Settings.

**Const**
Data will not be rescaled when stepping through frequencies or time channels.

**Auto**
Data will always be rescaled so its maximum value will be at the same height.

**Show Surface**
Display the data as a surface for the selected data set specified under Change Settings.

## Visualizer. Plot Style.

**Vector Line** Display data as a vector. Direction of the vector will depend upon projection defined in Plot Settings or Select Data dialog.

**Line Interpolation**
Display data as a line interpolated along the data points.

**Surface Interpolation**
Display data as a surface interpolated across the data points.

**Data Set**
See the plot style settings for the selected data set.

**Line Thickness**
Change the thickness of the vector or curve

**Order**
Use linear or cubic interpolation

**Colour**
Change the colour of the vector, curve or surface.

**Visible**
Toggle on or off

# Visualizer. Plot Settings.

## Data Settings.

**Max Amp** Data is scaled according to this value. Enter a larger value to increase the z variation in the data.

**Constant**
Data will not be rescaled when stepping through frequencies or time channels.

**Automatic**
Default. Data will always be rescaled so its maximum value will be at the same height.

**Logarithmic**
Log to base 10 of data is displayed.

**Linear**
Default.

**Amplitude**
Show the amplitude of the data in the selected direction: X,Y or Z

**Change Sign**
Multiply all the data by minus one

**Vector**
Direction of the data is determined by three components of the data: X,Y and Z

**Flip Quadrature Sign For Simulated Data**
Multiply data by minus if it is simulated quadrature data

**PetRos EiKon(-Y,X) ==> Crone(X,Y)**
Convert to and from PetRos EiKon and Crone axis conventions.

## Axis Settings

**X,Y,Z Axis**
Select the axis to be modified, make changes and press the apply button to update the display of the axis.

**Label**
Text to be displayed next to the selected axis.

**Font Size**
Size of the axis label.

**Tick Label Size**
Default is half the axis label font size.

**Number of Sub Ticks**
This is the number of lines that appear between each tick label. Create a finer or coarser grid on the axis.

**Step between major ticks**
The value difference between two consecutive tick labels.

**Depth of Z Axis**
Useful for setting the location of the grid at the top and bottom of the axis.

**Format**
Choose whether or not to display tick labels in exponential form or not with power of ten displayed at the end of the axis.

## Visualizer. Filters (Reduce Data).

**Data Set** Parameters on this dialog will be shown for the selected data set.

**Reduction Value**
Default value is equal to the average of all the data being displayed in current data set. A negative value will add to the data.

**Max Data, Min Data, Average Value**
Values after reduction

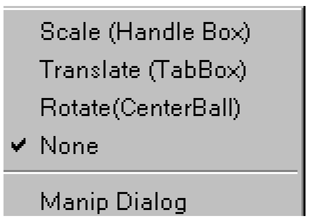
**Reduced by**
Amount the data has been reduced by.

## Visualizer. Model menu commands.

| | |
|---|---|
| **Transmitter** | Show/hide transmitter drawing and change its line thickness and colour. |
| **Profile** | Edit profiles. Toggle Off/On<br>Make the profiles visible or invisible |
| **Layer** | Edit selected layer |
| **Polyhedron** | Edit selected polyhedron |
| **Prism** | Edit selected prism |
| **Sphere** | Edit selected sphere properties |
| **3D Inversion model** | Modify display of data in mag/grv/res file |
| **Euler Deconv** | Display a solution created with the 3D Euler Deconvolution tool. |

## Visualizer. Manips menu commands.

The Manips menu from the mouse right-click upon selecting an object, offers the following commands, which enable you to set the type of manipulator and toggle the display of the manipulation dialog. These commands are available when a model is selected using the arrow tool. Manipulators cannot be used on the Euler solution prisms.




Manipulator types:

**Tab Box**

| | |
|---|---|
| *Corner Tabs* | Click and drag to scale about opposite corner |
| *Edge Tabs* | Click and drag to scale about opposite edge |
| *Faces* | Click and drag to translate freely in the plane of the face<br><Shift> click and drag to translate along one axis<br>(determined by next user gesture) |

**Handle Box**

| | |
|---|---|
| *Corner Cubes* | Click and drag to scale uniformly about center |
| *Center Cubes (on sides of box)* | Click and drag to scale about center |
| *Faces* | Click and drag to translate freely in the plane of the face<br><Shift> click and drag to translate along one axis<br>(determined by next user gesture) |

**Centerball**

| | |
|---|---|
| *Stripes* | Click and drag to rotate around that axis |
| *<Shift> Surface* | Click and drag to position user axis (user axis stripe appears) <Shift> click and drag pole to intersection of two normal stripes to remove user axis |
| *User Axis Stripe* | Click and drag to rotate around user axis |

Toggle manipulation dialog:

**Dialog option (Manips menu)**

Use this option to toggle display of the manipulation dialog.

Dialog shows centre and dimensions of body when using handle box and tab box manipulators. Dialog shows axis and angle of rotation when using centerball manipulator.

## Visualizer. Lights menu commands.

**To enhance visualization, several types of lights with controls are offered**

**Create Dir Light** Add a directional light source that illuminates along rays parallel to a given 3-dimensional vector.

**Create Spot Light**
Add a spotlight style light source. A spotlight is placed at a fixed location in 3-space and illuminates in a cone along a particular direction. The intensity of the illumination drops off exponentially as a ray of light diverges from this direction toward the edges of the cone.

**Create Point Light**
Add a point light source at a fixed 3D location. A point source illuminates equally in all directions; that is, it is omni-directional.

**Edit Ambient Lighting**
Ambient lighting is the amount of extra light impinging on each surface point. Launches color editor.

**Turn all ON**
Turn all lights on.

**Show all Icons**
Make all the light manipulators visible.

**Hide all Icons**
Make all the light manipulators invisible.

## Visualizer. Options menu commands.

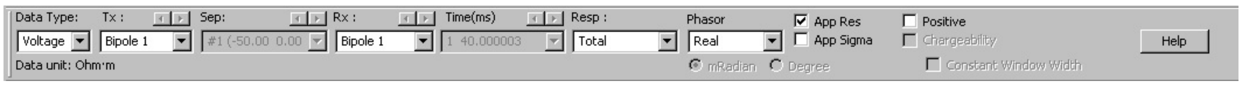
The Options menu offers the following commands, which enable you to configure the viewing parameters of the graphic viewer:

| | |
|---|---|
| **Draw Style** | Sets the drawing style of the viewer (solid, wireframe, or both).<br>*Solid* - Objects are always solid.<br>*Wireframe* - Objects are always wireframe.<br>*Solid and Wireframe* - Objects are solid if stationary and wireframe if moving. |
| **Colour Ranges** | Sets the ranges of the types of data which are used to set the colours of the model constructions in the graphic viewer.<br><br>Choose whether new colour range is going to be applied to layers or prisms<br><br>*Field* - Parameter used to determine minimum and maximum values of range.<br>*Range* - Minimum and maximum values of range assigned to the colours. |
| **Vertical Exaggeration** | Sets the vertical scale of the viewer.<br>Use this command to activate the vertical scale dialog box. |
| **Transparencies** | Set the transparency type:<br>*Screen Door* - Uses stipple patterns for screen door transparency<br>*Blended* - Uses alpha blending<br>*Delay Blended* - Uses blending, rendering all transparent objects after opaque ones<br>*Sorted Blended* - Same as Delayed Blended, but sorts transparent objects by distances from camera |
| **Edit Background Colour** | Change the background colour of the scene.<br>Launches colour editor. |

| | |
|---|---|
| **Axis** | Toggle simple axis on and off or toggle grid axis on and off. |
| **Reference Plane** | Display a horizontal plane at a specified depth |

# Pseudo Sections

## Upper Toolbar.




Data Type: This could be Data, Apparent Resistivity, Apparent, Depth, Voltage, etc

Tx: If more than one transmitter, select your desired transmitter

Sep: Separation. If pseudosection of frequencies or time windows, then select your separation

Rx: Select the desired receiver

Time: If Data is selected and section is by separation, then Time window or Frequency selection is provided

Resp: Response. In some cases, there is more than one type of response available

Phasor: In some cases, there may be Real and Imag and/or Amplitude

App Res: For some data, an apparent resistivity formula is available, select this box and the data will be displayed in apparent resistivity. App Sigma is just the inverted apparent conductivity

Positive: In some cases, the calculated apparent resistivity can be negative because the voltages are negative. Setting Positive keeps everything absolute value.

Chargeability: In the case of IP data, then chargeability can be sectioned

## Lower Toolbar.

| Click | To |
|---|---|
| | Insert additional Rows. This will linearly interpolation between 2 datapoints and insert rows halfway between 2 depths, frequencies or time windows. |
| | Insert additional Columns. Equivalent to inserting Rows but for Columns |
| | Show data at locations as colored points |
| | Toggle Contour. Only available after interpolation has been performed. |
| | Natural Neighbour or Minimal Curvature Interpolation |
| | Open last interpolation settings. Allows modification and then interpolation |
| | Fill in blanks if existing in the grid by linear interpolation. |
| | Reset Initial Data Settings (remove all interpolations and inserted rows and columns). |
| | Change the aspect ratio of the display. The default setting is to fit the screen. |
| | Change the size of the data symbols when they are being used. |
| | Toggle Legend. |
| | Modify the contour line attributes. |
| | Toggle filled contour. |
| | Select Frequency (Time channel) or Separation |
| | Display data as linear or logarithmic |
| | Set Equal Range or Equal Weight data color distribution mode. Equal Weight is not available |

when contour is displayed.

Set profile location display as X or Y coordinates on the plot display.

Change profile lines

Enter the minimum and maximum data values that the colour range applies to. Change data color range by clicking on colored Min/Max Rectangles. Select the **Hold Min Max** option to stop the minimum and maximum values from updating when a new profile or a different range of time windows, frequencies or separations is selected.

Save settings or load settings

Export currently displayed section to a file. More details

**Save** and **Load** - allows one to save and restore current settings

Different data channels can be selected from the upper toolbar

When the mouse is clicked, current coordinates and data are displayed near the clicked point.

**Related Items**

Axis Settings

## Axis Settings

Double-Click the mouse on the axis you would like to edit and the **Format Axis** window appears:

You can:

- change Number of Steps between Major Ticks
- show/hide Major/Minor Ticks Lines (Major Line – solid, Minor – dashed)
- change Axis Limits
- enter an axis title in the **Title** section
- alter which parameter is used to map depths in **Z mapping**
- change axis tick labels to Fixed/Exponential
- change axis label fonts
- change the tick color by clicking the **Color** button in the **Tick Labels** section

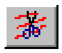
## Exporting a Section

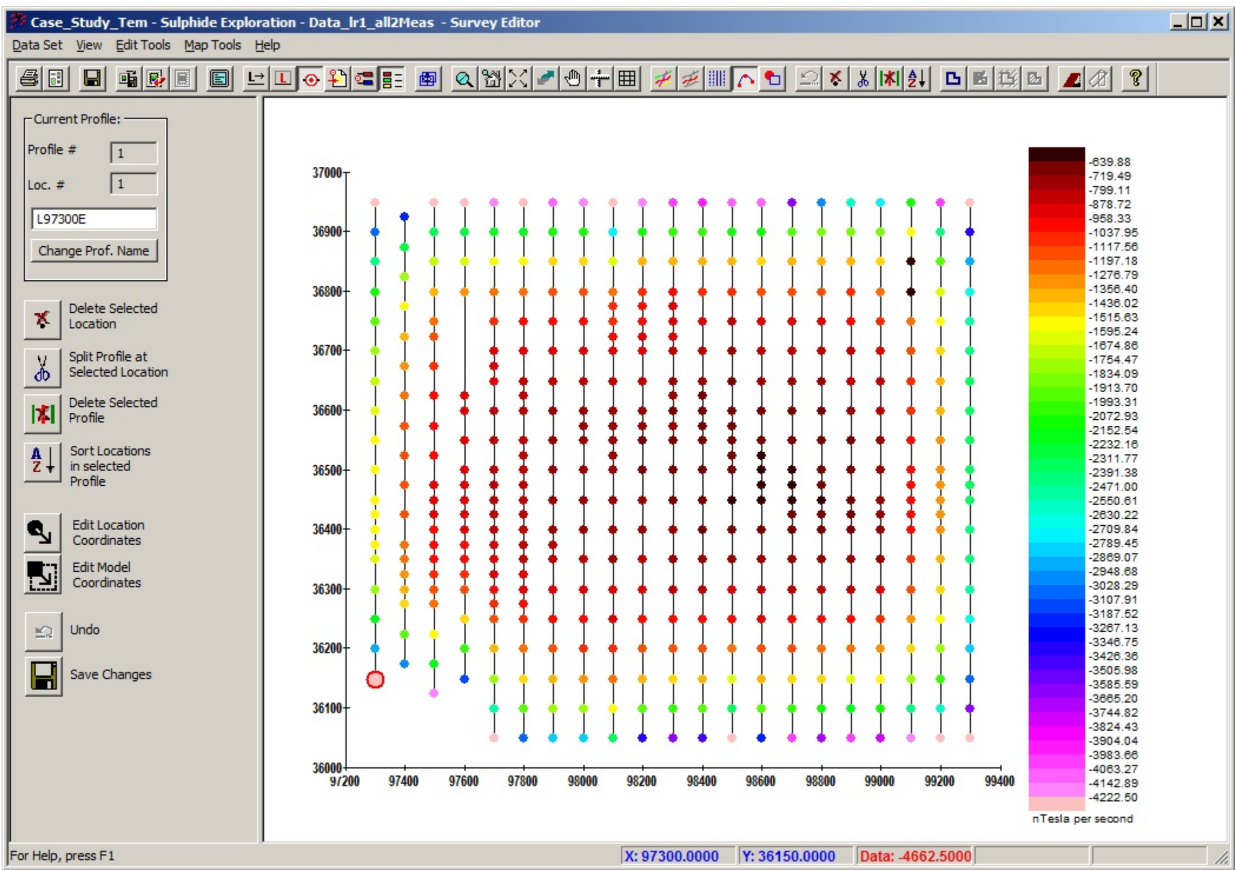
Clicking the button will bring up the **Output data** window:

- Click **Browse** to select a file to save your data
- The data that is currently displayed will be saved to the **Output File** in an xyz format. Only the data for the currently selected channel will be saved. To save the data for a different channel, make your selection on the toolbar and click **Export Section** again.
- In the **Line Direction** section, choose **Horizontal** to save horizontal lines in the output file at different depths or choose **Vertical** to save vertical lines at different positions.
- Click **Export** to save your file.

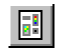
# Survey Editor

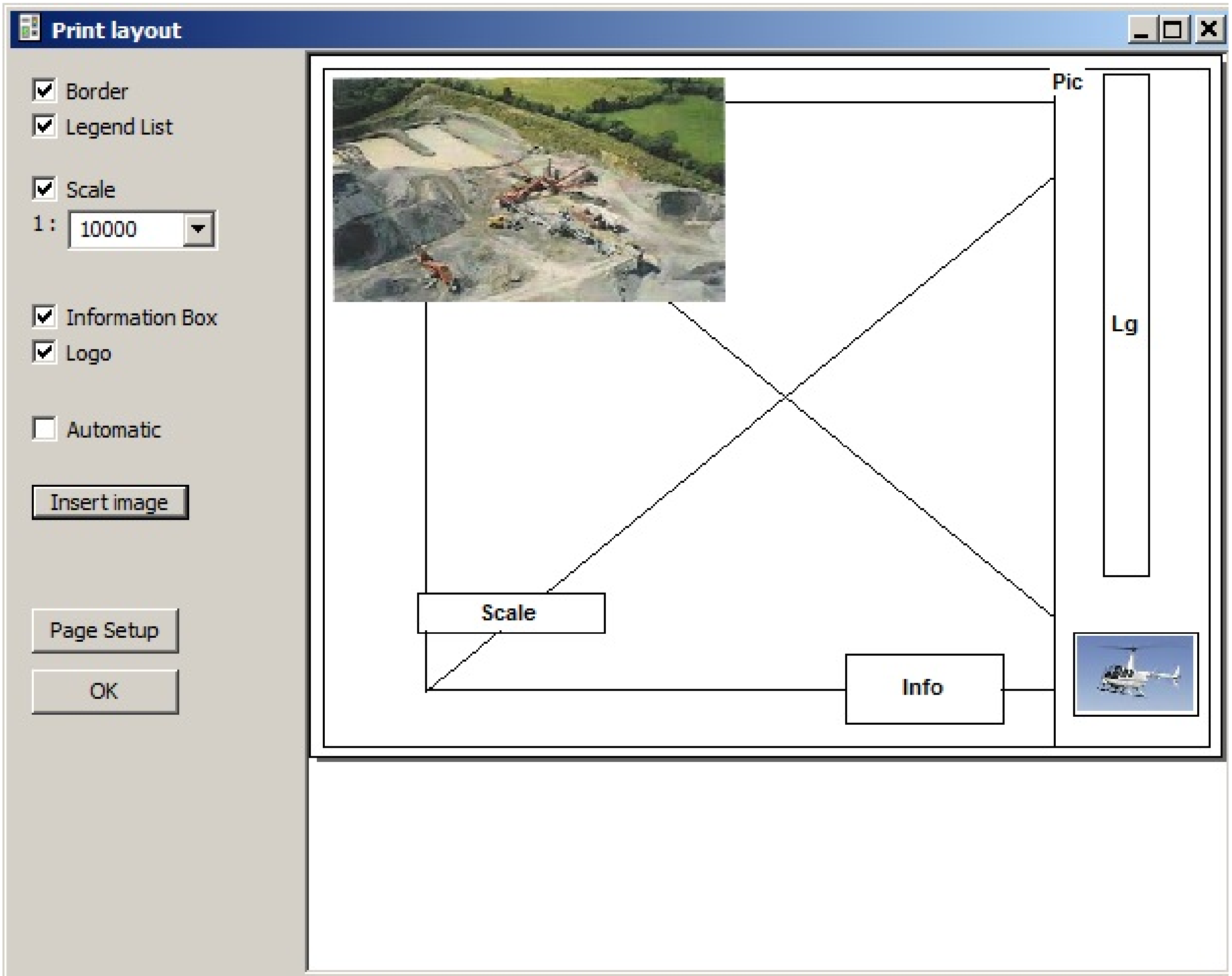
## Starting the Survey Editor

To open the Survey Editor, select the survey you wish to work with from the Database dialog and click the  button on the main toolbar:



# Printing

To set up the layout of your hard copy, click the  button on the toolbar:




- Check off **Border** to reserve a section of the page for the Legend(Lg) and the Information Box(Info).
- Check off **Legend List** to display a legend for the data on the print out.
- Check off **Scale** to choose a proportional size for the survey to appear on the page
- Check off **Information Box** to display a box in the bottom right hand corner which contains the survey name and date.
- Check off **Automatic** to print the survey as displayed in the Survey Editor window.

- The information box, legend and scale can be repositioned by dragging the desired item with the mouse.

**Creating Map Underlays**

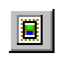
Survey Editor allows you to load an already available map or generate a map to be used as an underlay. Calibrated files must have a *.map extension.

See

Saving To An Image File

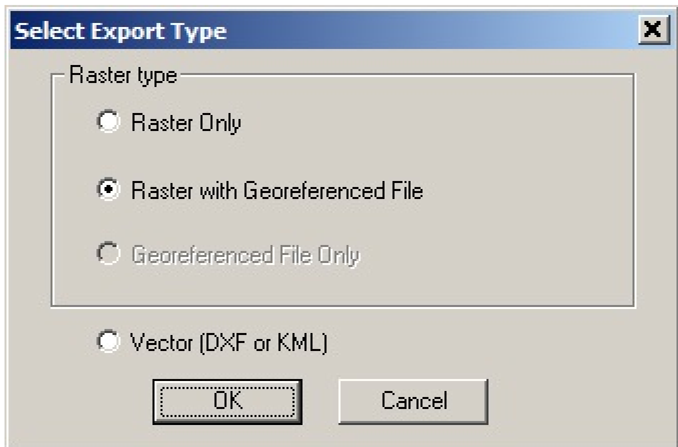
Generate a Map

Load an Existing Map

To toggle the map underlay on and off, use the  button on the toolbar.

## Save to an Image File

To save the image displayed to a file, click the  button on the toolbar. The **Select Export Type** window appears :



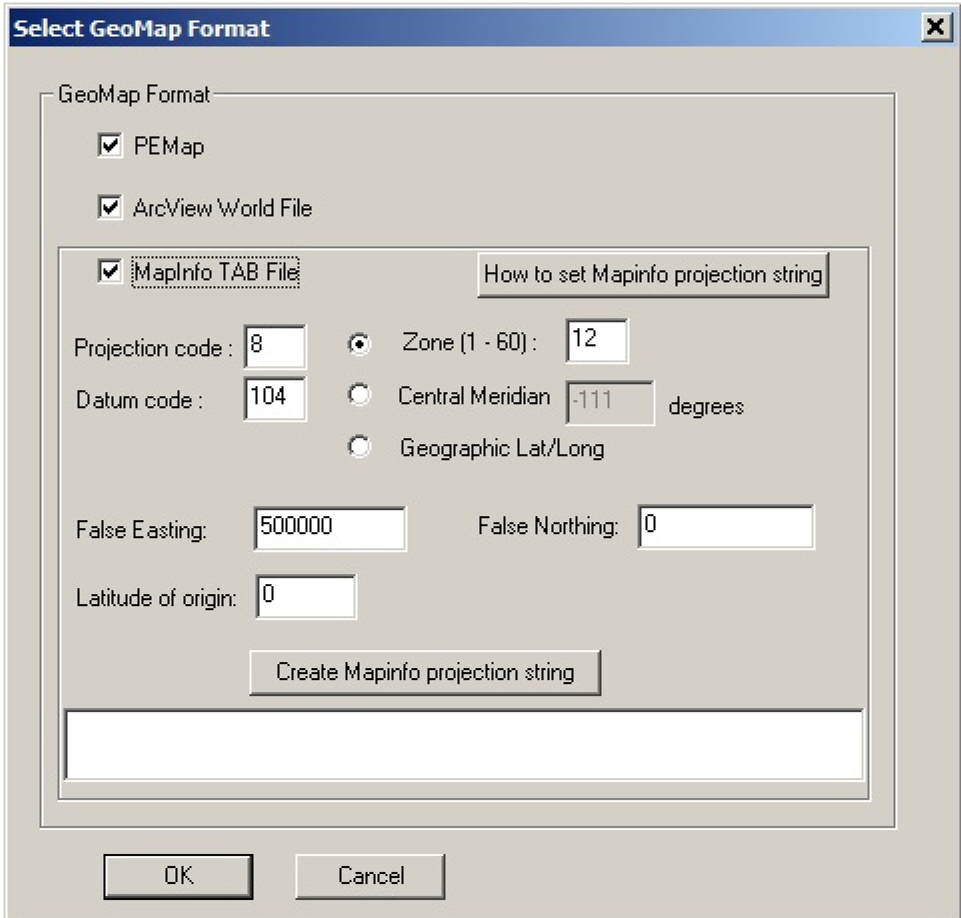

- **Raster with Georeferenced File** will create both an image and a map file. The available map file formats are for EMIGMA/QCTool, ArcView and MapInfo. If you do not desire a map file select **Raster Only**.

Click **OK** and enter a name for the image file and select the file type before clicking **Save**. Available output file types are jpeg, bmp, gif, png, tiff, tga, pcx, wmf and emf. For jpeg and tiff formats, you will be asked for some compression settings. The default values work very well. The **Bitmap Export** window then appears:


To modify the resolution:

- Enter new values into the **Width** or **Height** boxes to modify the size of the image. Check the respective box to maintain aspect ratio.
- Enter new values into the **Horizontal** and **Vertical** boxes to modify the resolution. One value will be updated to be the same as the other if **Maintain aspect ratio** is selected.
- Select **Original** from the drop down box if it is displaying **Custom** to return to the original values.
- Click **OK**

If you had chosen to create map files, the **Select GeoMap Format** window will appear:




One or all of the options can be chosen. Select **PEMap** for use in EMIGMA and QCTool. Select **ArcView World File** for use in ArcView. Select **MapInfo TAB file** for use in MapInfo. Click **OK** and a window will appear with information on the saved files.

See PEGeoMap if you wish to customize your map file.

**NB:**
Note that when you save a plot which is not drawn as "proportional" (i.e., width and height ratio does not reflect real topographic sizes), saved raster also will not be proportional. However, obtained georeferenced image will be geographically correct. When read into mapping software (PEGeoMap, ArcMap, GlobalMapper, etc.), it will be positioned accordingly to its coordinates.

**Export profiles to KML:**

- Survey lines can be exported to KML and then viewed over landscape in Google Earth. This provides very precise and convenient view of survey geographic location. To perform this operation, original data must be in UTM or lat/long coordinates.
  In "Select export type" dialog, choose "Vector", and KML format. Then, follow selecting coordinate system parameters for correct positioning.

  Note that if you make 2 kml exports of the same data: raster overlay from QCGrid or Survey Editor and vector survey lines, and then open them both in GoogleEarth, they appear to be slightly differently positioned against each other, especially when they cover comparatively large area (>10 km). This is due to the fact that raster overlay has precisely positioned only its 4 corners, while in vector survey lines each point is positioned in its exact georaphic place.

### Load an Existing Map

- Click the  button on the toolbar.

  The available map file formats for export are:
  EMIGMA/QCTool .map
  ArcView World files
  MapInfo .tab
  AutoCAD .dxf
  GoogleEarth .kml

  Last 2 formats are vector, and can contain only survey lines with no additional data.

  Browse for the required file in the Windows-style Open dialog to appear.
  If KML export was selected, datum/zone selection dialogs follow to perform conversion from geographic lat/long coordinates to UTM.

  Also see:

  Saving To An Image File

## Switch to proportional axes

- Engage the **Show Proportionally** button on the toolbar. Your grid axes will become of the same scale
- Disengage this button to return to the initial view

## Zoom in on a selected area

- To zoom in on a grid fragment, engage the **Zoom Selected** button . Click and drag to select a grid fragment to be magnified
- To return to the initial scale, click the **Home View** button 

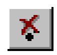
## Editing Profiles

On the left panel:

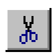
To change the text of a profile label:

- Select any location on the profile whose name you want to change. Enter the new label in the **Current Profile** box and click the **Change Prof. Name** button.

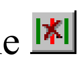
To delete a location:

- Select a location and Click on the  button

To split a profile into two separate profiles:

- Select the location situated before where you would like the split to occur and click the  button.

To delete a profile:

- Select any location in the profile you want to delete and click the  button.

To sort the locations in a profile:

- Select any location in the profile you would like to sort and click the  button. Select the parameter to sort by on the window that appears and click **Sort**.

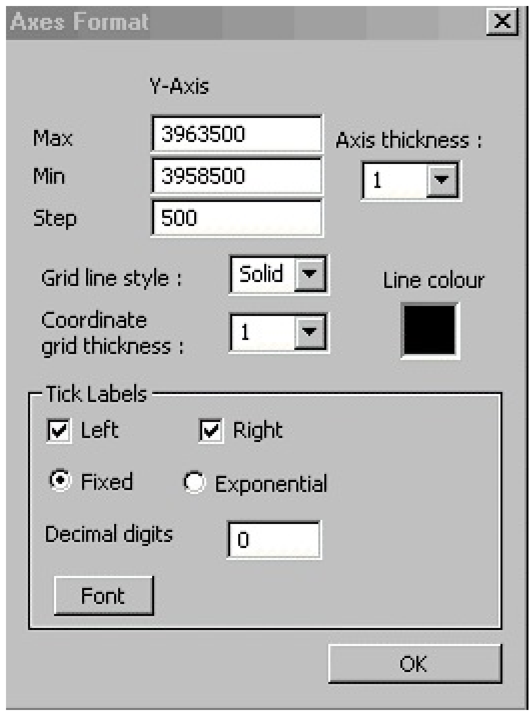
To undo any of the above changes:

- Click the  button.

## Adjust axes

In the image field of the Survey Editor:

- Double-click in the region of the X axis to bring up the **X-Axis** dialog or in the region of the Y axis to display a similar **Y-Axis** dialog:



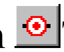

- Toggle the display of an axis on either side of the display using, in the case of the Y-Axis, the **Left** and **Right** checkboxes.
- Edit the **Step** between major ticks in the respective box.
- Edit the **Min** and **Max** values as desired in the **Axis Limits** section.
- Specify the fixed or scientific notation, the number of decimal places and the font of the tick labels in the respective section.
- Type in the title of your X or Y axis in the respective field; use the **Font** button to specify the format of the title.
- Click **Apply**.

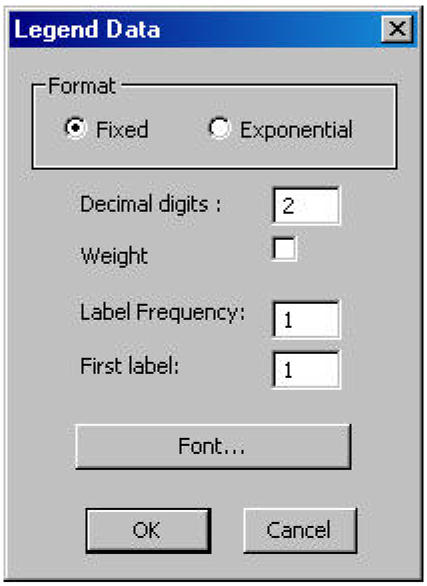
***Note***. To provide the proportional scaling of the axes, use the **Show Proportionally** button .

## Toggle the coordinate grid on and off

- Use the **Show Coordinate Grid** button on the toolbar.

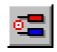
## Display the legend

- To display the legend, use the **Show Legend** button.  on the toolbar. It is enabled by first clicking the **Show Data** button The legend can be moved to a new location by dragging it with the mouse.
- Click once on the legend and a resizing box will appear to allow the legend to be resized.
- Double click on the legend and the **Legend Data** dialog will appear:

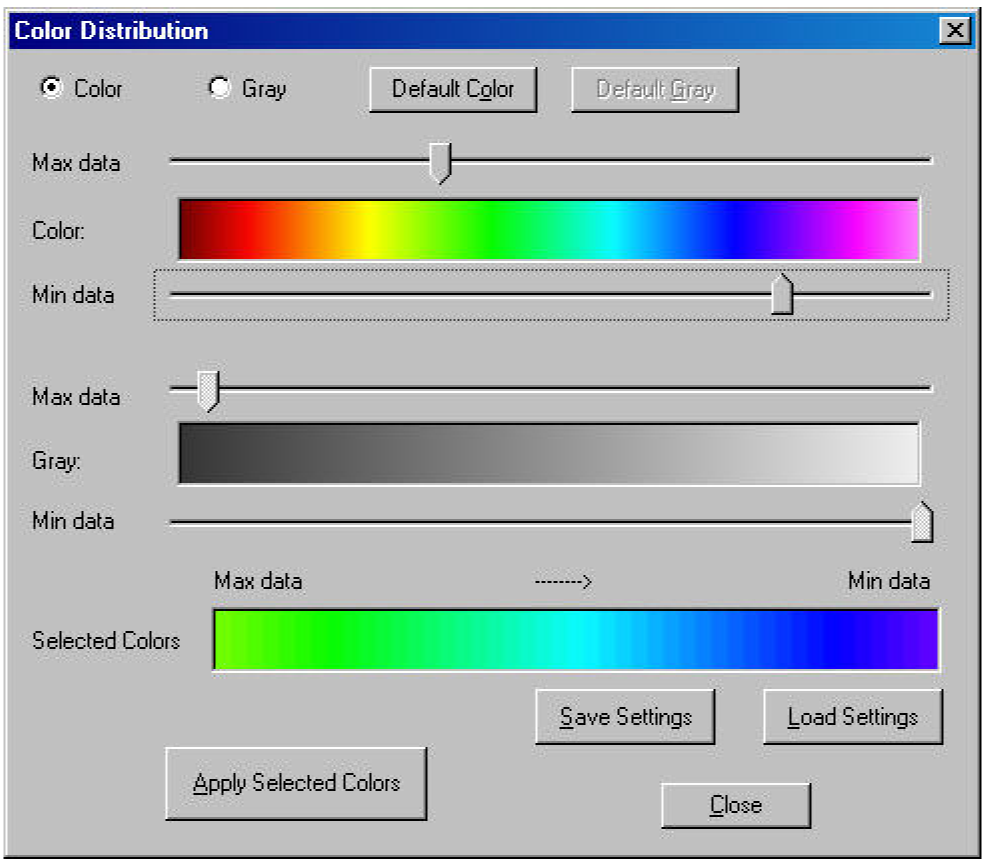



- The format of the legend data values can be set here.
- Check **Weight** to display how many data values fall into a certain interval
- Control the amount of intervals with **Label Frequency**.
- Leave a number of intervals without a label using the **First label** value.

## Adjust the range of data to be displayed

On the toolbar click the  button

- Insert the minimum and maximum values manually in the active **Min Data** and **Max Data** boxes or use the respective sliders. The absolute minimum and maximum values are displayed in the disabled boxes on the left
- Click on the boxes displaying the colors of the minimum and maximum data values to bring up the color distribution window:




- Choose between color and gray. You can now choose new colors for the minimum and maximum data values by moving the appropriate sliders to a specific color. The range of colors you have selected will be displayed in the color spectrum labelled **Selected Colors**

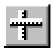
- To return to the original color range. Click the Default Color or Default Gray button. The appropriate one will be enabled.
- The color range that has been selected can be saved by clicking the **Save Settings** button so that it can be loaded later with **Load Settings**.
- Click **Apply Selected Colors** to see the data with the new color range.

## Switch to the full-screen view of your survey

Click the **Full Screen View** button  on the toolbar. You can also do it from the **View/Full Screen** menu

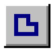
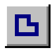
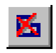
## Measuring Distance

To measure the distance between two points:

- Click the  button on the toolbar.
- Click on the point where you want to measure from. Don't release the mouse button and drag to the point where you want to measure to. A line is drawn between the two points on the display and the distance appears next to the coordinates and data value on the status bar in green.

## Selecting a Group of Locations

To select an irregularly shaped group of locations:

- Click on the  button
- On the survey image, click on the vertices of the polygon which contains the locations you would like to select. To complete the polygon, click the  again so the first and last vertices will be joined to close the polygon. Click the  button if you would like to remove the polygon.

To delete locations found outside the polygon:

- Click the  button

To delete locations found inside the polygon:

- Click the  button

## Drawing models

Models listed in "Prisms/Plates/Polyhedra page" (see Additional Tutorials/Emigma Modelling of TEM Data) can be visualized by projecting over "Survey editor" plot :

- Click button  "draw models" on toolbar.
- Click button  "draw models outline" on toolbar.

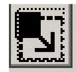
- In "View\Model display setting", change colours and other features for models drawing.

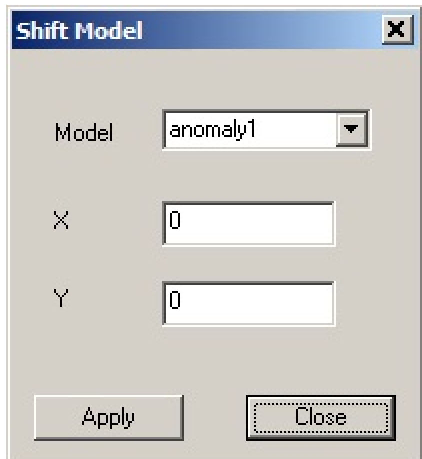
- Symbol ├ near model name reflects strike/dip identifying model's orientation in space: short line shows dip - direction of the slope (down), longer - strike (horizontal line along the slope side).

Model projections can be exported to KML together with survey lines (see Survey Editor\Creating Map Underlays\Saving to an image file: Export profiles to KML) and viewed over the landscape in Google Earth.

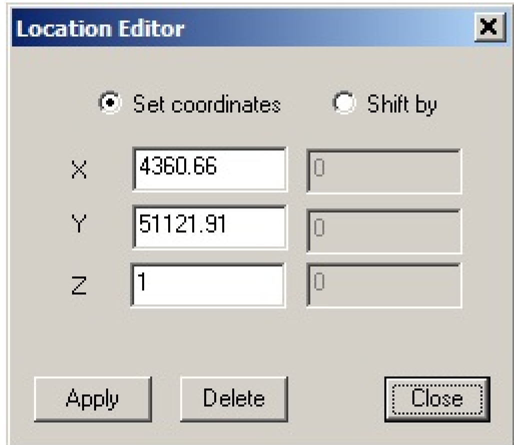
## Editing locations

Positions of profile locations, transmitter nodes, and models can be changed in Profile viewer.

- To change model postion, click button  "Edit model coordinates" on the left side of the splitter. This option is available only if models are drawn. In the dialog

  type values for the shift along X and Y axes. Click "Apply".
- To change postion of profile location or transmitter node, click button  "Edit location coordinates" on the left side of the splitter, or select required location, right click, and select "Edit" in the floating menu. In the dialog



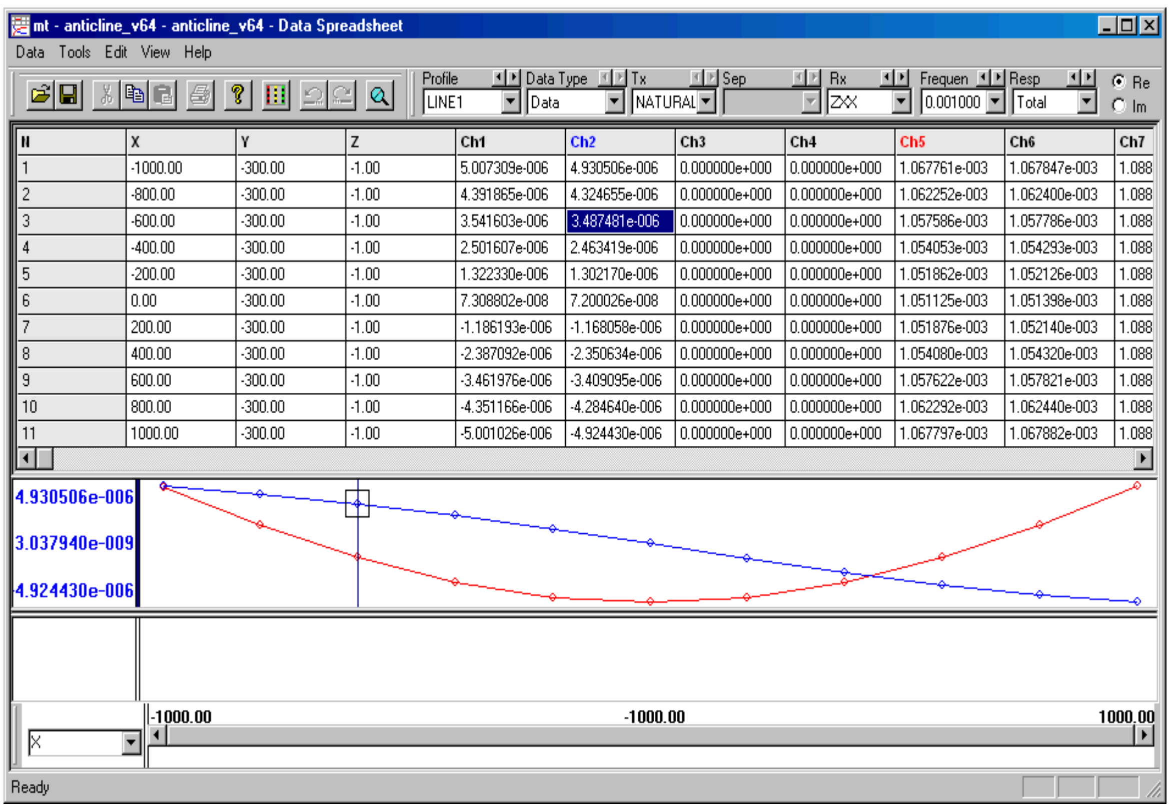

type coordinate values or the shift along X and Y axes. Click "Apply". Also, in the same dialog currently selected node can be deleted.

# Data Spreadsheet

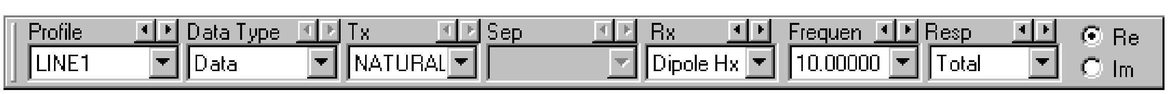
## Using the Data Spreadsheet

To start the Data Spreadsheet:

- Select the data set you wish to work with from the Database dialog.
- Click the  button on the main toolbar and the Data Spreadsheet window appears with the data set displayed in spreadsheet format in the top half of the screen.:




To use the channel toolbar:




- Select the type of data on the channel toolbar and the spreadsheet cursor will move to the appropriate column which the data selection specifies.

- Click on a channel's column and the channel toolbar will be updated with information about the selected channel.

To display impedance data:

- Save any changes that have been made using the  button.
- Select **Data/Impedance Data** from the menu and the impedance data set will be loaded. Click on any channel column to see the information about the data on the channel toolbar.
- A checkmark beside the **Impedance Data** menu item will indicate that current data set is impedance data.

To display channel statistics:

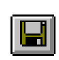
- Right click on the column you want information on and select **Statistics**.

To edit data:

- Click on any spreadsheet cell and enter a new value.

To edit data using a plot window:

- Right click in the column of the channel which has data you want to edit. Select **Plot Channel**. The plot appears in the bottom half of the screen.
- Right click on the plot and select **Edit Data**.
- Click on a point and it will be highlighted by a cursor.
- Drag the point to the desired position. The value will be updated in the spreadsheet.
- Edit a point in a different channel by either clicking on the channel's column or double clicking on the point on the plot.
- Call up the right mouse menu. Select **Undo Modify** to restore the original data value. Select **Save Data** to keep the new value.
- Any data modification can be undone by clicking the 
- An undone edit can be redone by clicking the  button.

A new data set with the modified data can be created by using the  button.

## Spreadsheet Functions

To copy and paste a data value:

- Select the cell you would like to copy, click the  on the toolbar then select the cell you like to copy to and click the  button.

To keep columns from scrolling:

- Right click on the column you would like to remain stationary and select **Freeze Columns**. Any columns before and including the one you selected will turn gray in colour and will not move when scrolling.

To perform a mathematical operation on the data

- Select the range of data you would like to work on either on the spreadsheet or plot.
- Right click on the selection on the spreadsheet and select the operation you would like to perform. Available operations are **Multiply**, **Shift Data** which require a value to multiply by and add to respectively. Also available are **Reverse Sign** and **Set NODATA**.
- Undo an operation using the  button.

To change the data format:

- Right click on the column you would like to work on and select **Column Format**. The window to appear will allow you to specify the number of decimal places and whether the data should be in fixed or scientific notation.
- Your changes can be applied globally by activating the checkbox labelled **Apply to all data channel columns**.

## Plotting Channels

To plot a channel:

- Right click on the column of the channel you would like to plot and select **Plot Channel**
- The plot will appear in the bottom half of the screen. The colour of the column heading will match the colour of the plot and y axis tick labels.
- Repeat this procedure to plot other channels in the same panel.

To plot a channel in a different panel

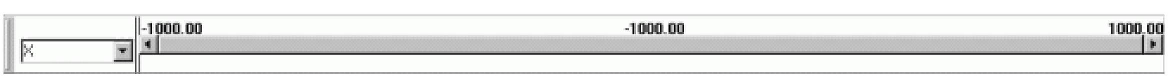
- Click in the panel you would like the plot to be in. The y axis will become dark blue to indicate the panel has been selected.
- Select Plot Channel for the channel you would like to plot.

To change the appearance of a plot

- Click on the column of the channel you would like to modify.
- Right click on the plot and select **Channel Settings**.
- You will be able to change the symbol type, symbol size, line colour, line style and line width.
- Modify the plot of a different channel by selecting a new channel in the drop down box. The channels are numbered according to the order they were plotted.

To modify the x axis:

- Select the channel to plot on the x axis using the drop down box in the bottom left hand corner.




To zoom in on a section of the plot:

- Click the  button to zoom in on a section of the plot. Drag the mouse on the plot to create the zoom window. Move the window to the section to zoom in on and click on that point.
- Use the scrollbar to move left and right through the plot.

- Right click on the plot and select **Rescale All** to return to the original zoom level.

To display the locations which have non-data values:

- Right click on the plot and select **Panel Settings**.
- Deselect the checkbox labelled **Do not show locations with NO DATA.**

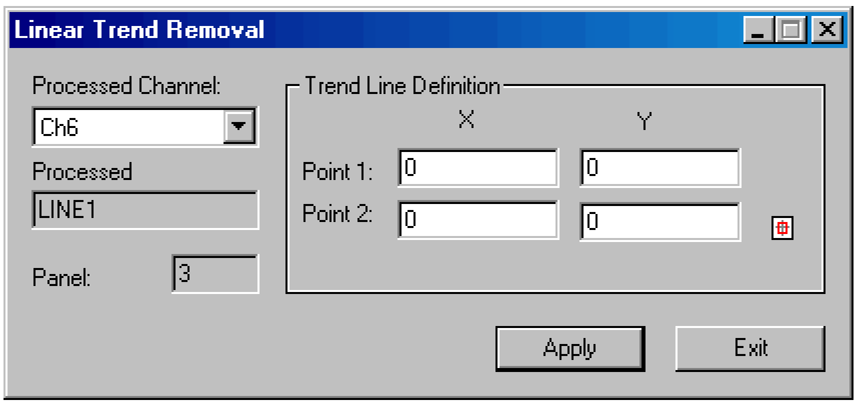
To change the scaling preferences:

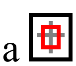
- Right click on the plot and select **Panel Settings**.
- Select **Scale each channel separately** to have all the plots in the panel use the full panel height. The y axis will display data values for the currently selected channel.
- Select **Same scale for all channels** to have the y axis apply to all the channels plotted in the panel.

# Data Leveling

To remove a linear trend:

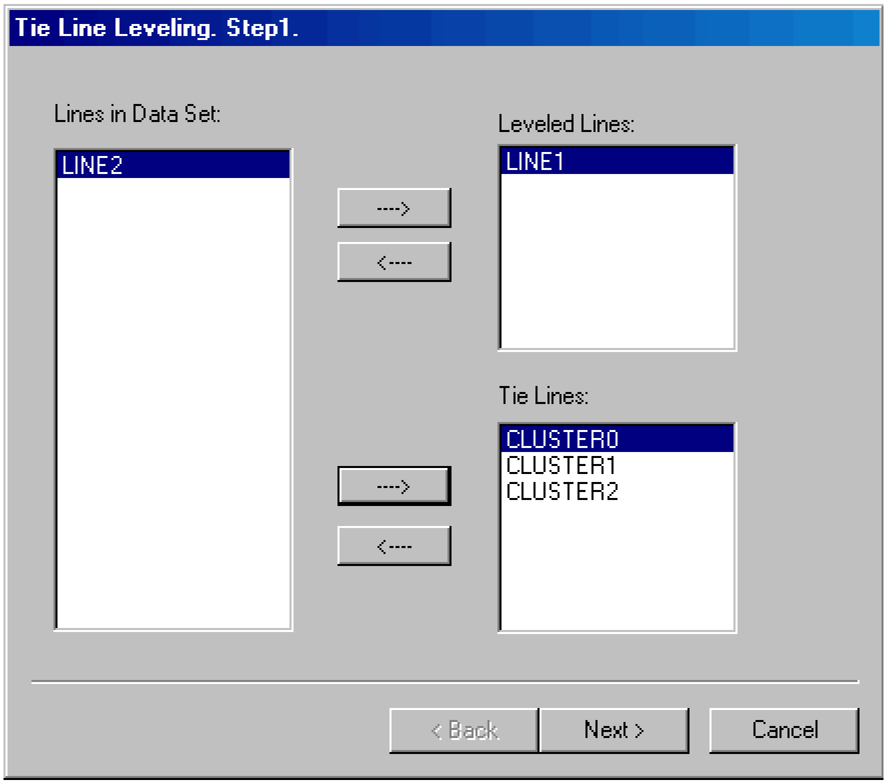
- Click on a plot of the channel you would like to work with.
- Select **Tools/Data Leveling/Trend Removal** from the menu and the **Trend Removal** window appears:




- The index of the panel displaying the channel is displayed as well as the name of the profile the channel belongs to.
- If there are multiple channels plotted in the current panel, the channel to work on can be selected using the **Processed Channel** drop down box.
- Click on one of the coordinates labelled **Point 1**. There will be a  symbol to the right indicating that these values will be updated.
- Click on the position of the panel where the starting point of the trend line will be.
- Click on one the coordinates labelled **Point 2** and then click on the panel where the end point of trend line will be.
- Click Apply and the values of the trend line will be subtracted from those in the selected data channel.

To perform tie line leveling:

- Select **Tools/Data Leveling/Tie Leveling** from the menu.
- The **Tie Line Leveling** window is launched:




- Specify which lines will be leveled and which lines are tie lines by loading the **Tie Lines** and **Leveled Lines** boxes using the 

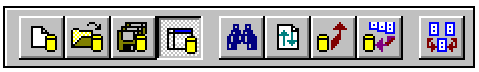
buttons. Any lines in the selection boxes can be removed by using the 

.
- Click the **Next** button.
- Select the tie lines to be used in the tie leveling by loading the **Selected Tie Lines** box.
- Click **Process** and the intersecting points will leveled by the tie line data values.

# Database Tools

## Database Toolbar

This toolbar contains buttons that deal with general database functions:



 Creates a new database. Click this button and the **Save New Database** window appears. Enter the name for the new database, select the folder to save the database and click **Create**. An empty Database window will appear bearing the name you assigned to it.

 Loads a saved database using the Start Dialog.

 Saves the state of the workspace that will be loaded the next time EMIGMA is started. The information saved in the workspace includes the currently open database and plots that are displayed in the 2d plotter. The workspace is also saved when EMIGMA is closed.

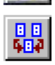
 Toggles the visibility of the main database window.

 Searches for a data set in the currently open database.

 Refreshes the database window.

 Launches the export interface.

 Launches the data import interface.

 Merges two data sets into a new data set.

# Specifying Export Type

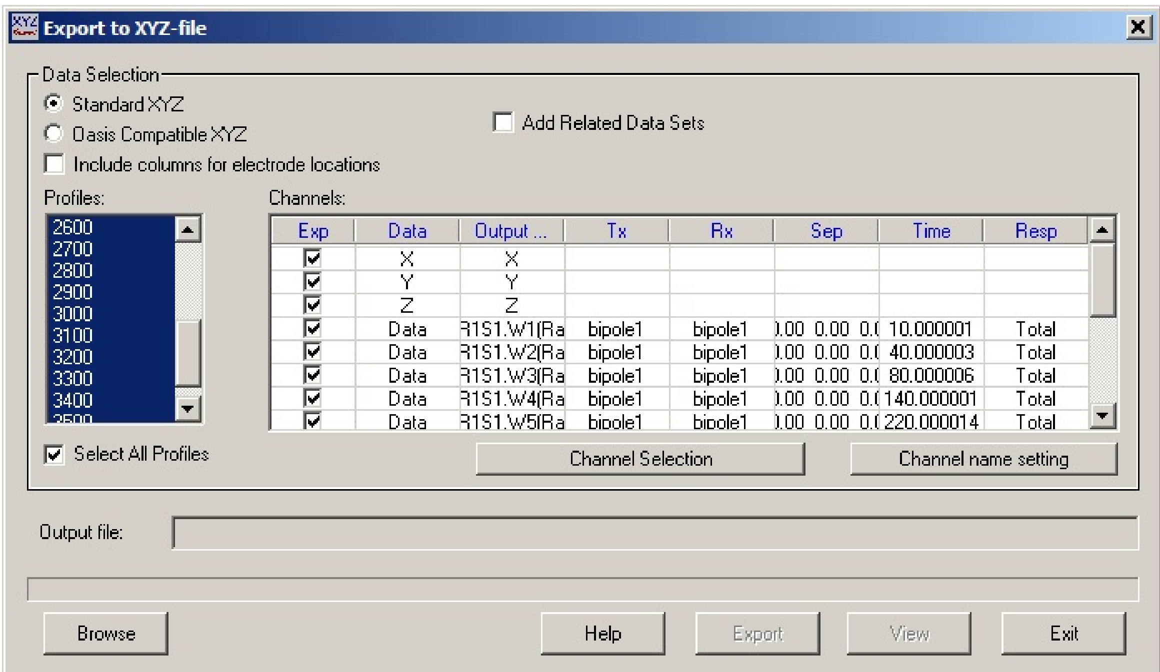
Export functions can be accessed by clicking the  button on the main EMIGMA toolbar.

You are presented with a number of export file types:

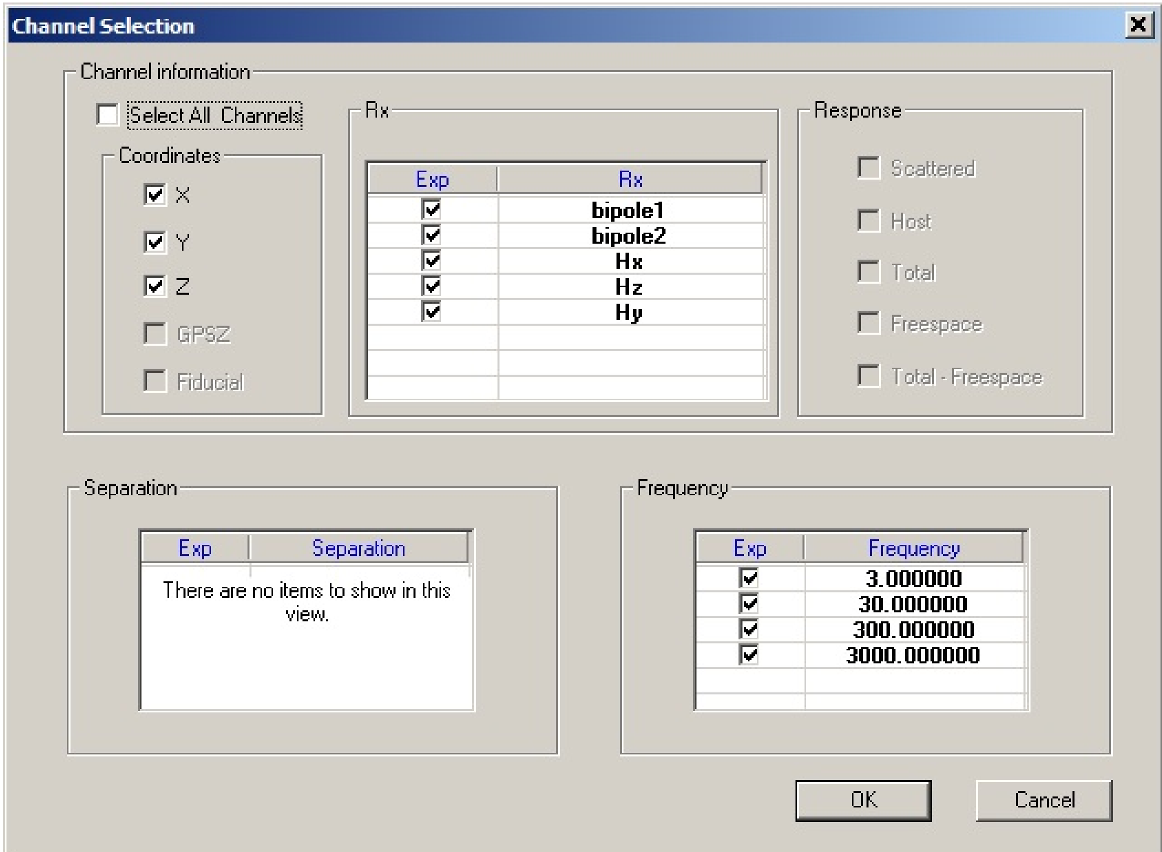
- .XYZ - ASCII format
- .GBN - GEOSOFT format
- .PEV - GeoTutor format
- .QCT - QCTool format
- .MDB - EMIGMA database
- Compressed EMIGMA database
- .KML - Survey lines to GoogleEarth
- .DXF - Survey lines to AutoCAD

# Exporting from Database to a File

After you selected required output format in the export type dialog, the **Export** window will open.
It looks similar for XYZ, GBN and QCT exports.

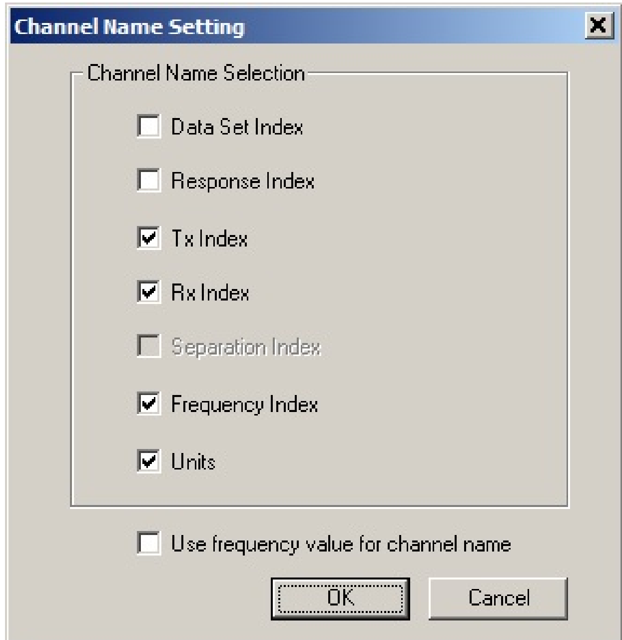



- If XYZ export selected, there would be a choice of **Standard XYZ** or Geosoft Compatible **Oasis XYZ** format.
- Select the profile and channel to export from the respective lists. You can choose to select all available profiles and channels by checking the respective boxes below the lists
- If there are other data sets in the selected survey, you may activate the checkbox labelled **Add Related Data Sets**. A window will appear allowing you to select other data sets from your survey. When they have been loaded, the current data set will be displayed in a control beside the checkbox. This control can be used to switch to a different data set.
- If data to be exported is IP, there will be available checkbox **Include columns for electrode locations**. If check it, each location will also have correspondent transmitter and receiver coordinates.
- If you want to manually define what is to be exported, click button **Channel selection.** In the following dialog




you can select channels and also filter out of export some frequences/separations.

- To set a rule for creation output columns names, click button **Channel name setting.** In the dialog



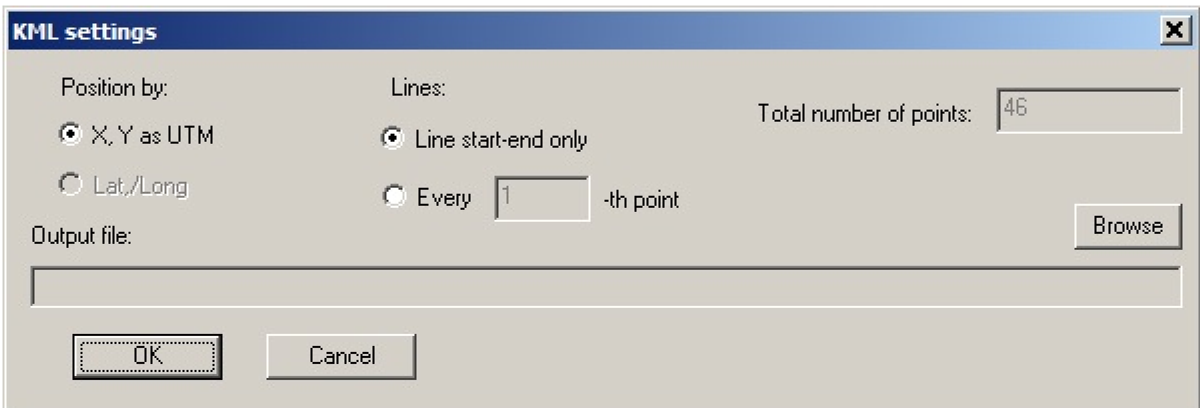

you can choose additional descriptors to be used in output column names.

- Click **Browse** to bring up a standard Windows-style **Open** dialog, specify the directory and folder to save the output file into and click **Open**. The output filename will appear in the **Output File** box
- Click **Export** to create the output file.
- To view the output file in the text format, click **View** in the bottom of the dialog

# Vector Export

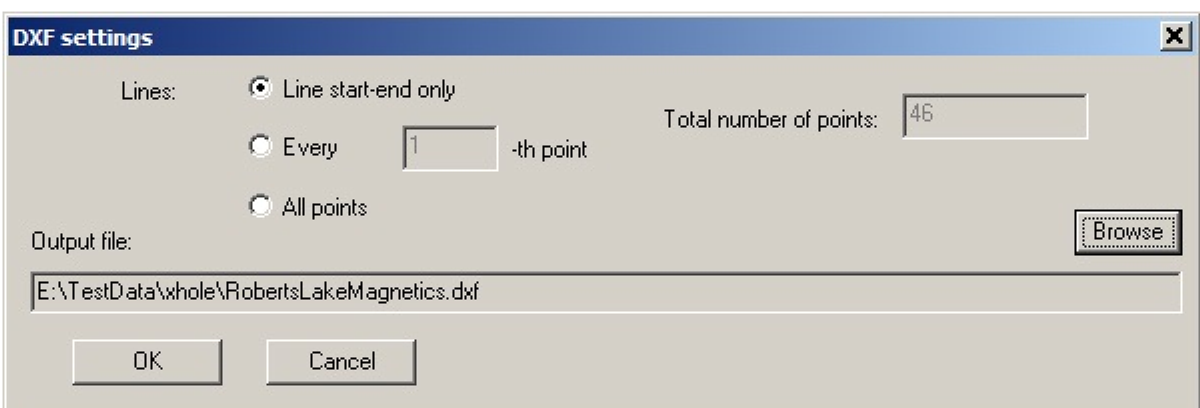
There is a possibility to export survey lines to KML and thus see its geographic position in GoogleEarth, or to DXF and use in AutoCAD drawing.

If you selected **KML**, dialog



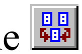

will appear.
Then, follow selecting coordinate system parameters for correct positioning.

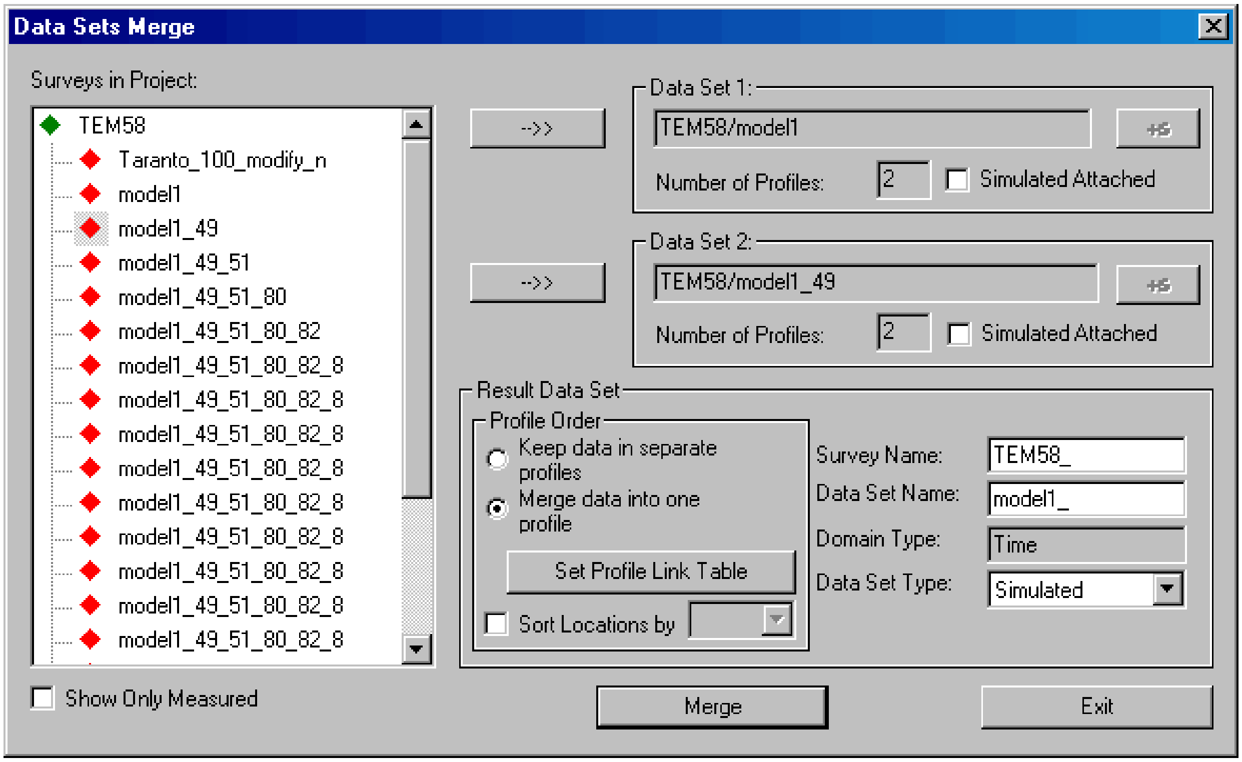
For **DXF**, it will look like

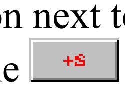



You can choose how many points are exported: only profile start-end, or every n-th point if there number is very large.

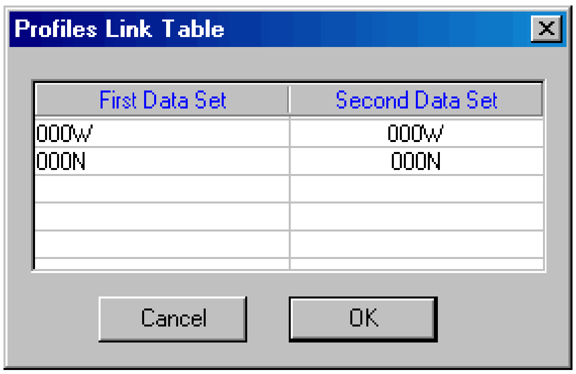
# Merging Data Sets

EMIGMA's merge utility can be accessed by clicking the  button on the main EMIGMA toolbar. The following window appears:




- All the surveys in the selected project appears in the **Surveys in Project** list. They are indicated by a green diamond. The data sets in each survey are indicated by a red diamond. By default, only the measured data sets are displayed. Deselect the checkbox labelled **Show Only Measured** to display other data sets as well.
- Select data sets for the boxes labelled **Data Set 1** and **Data Set 2**. Enter a selected data set by clicking the **-->>** button next to the Data Set box.
- If the selected data set has measured data, the  button will be enabled. Click this button to associate a simulated data set with the selected measured one. The simulated data must be in the same survey as the measured data. Select a simulated data set from the list that appears and click OK. A check mark beside the label **Simulated Attached** indicates that there is an associated simulated data set.

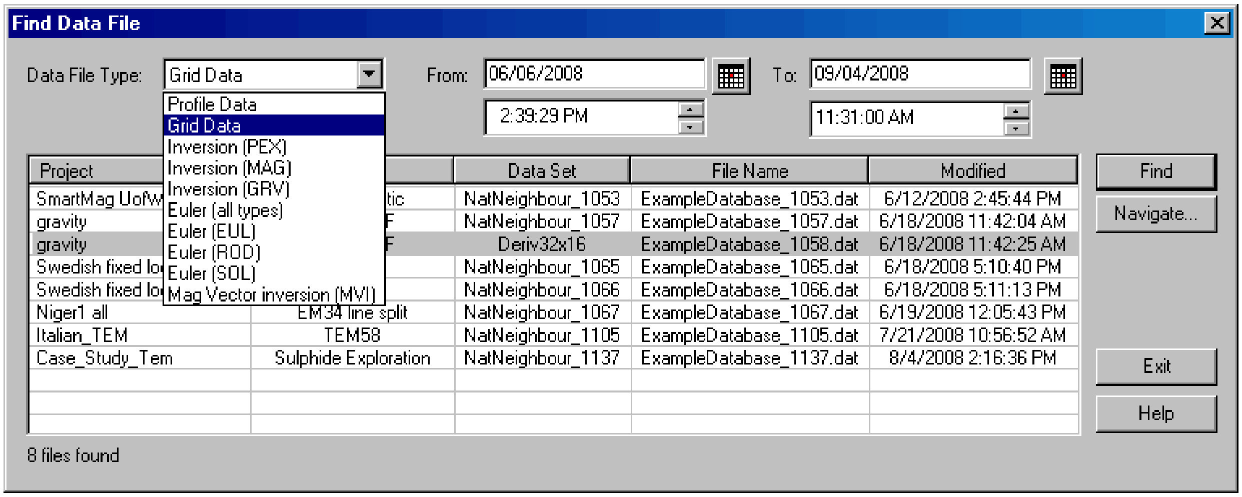
- You may decide to keep the data from the two data sets in separate profiles or to merge the data into one profile in the **Profile Order** section. If the data is going to be merged into one profile, you can choose an index to sort the locations and the **Set Profile Link Table** button will be enabled. Pressing this button will launch the following window:




- This window specifies the profiles which will be merged together. If the current setting needs modifying, select a profile in the **First Data Set** column. The profile to associate with it can be selected from the **Second Data Set** column. Repeat this for each profile pair to be modified and click **OK**.
- Click **Merge** to create a new survey with the merged data set(s).

# Searching for a Data Set

EMIGMA's search utility can be accessed by clicking the  button on the main EMIGMA toolbar. The following window appears:




- Specify what type of data you are searching for in box labelled **Data File Type**. Data file types available are profile data, grid data, three types of inversion files and data generated by the Euler deconvolution tool.
- Specify the time span you would like to search through by entering a date in the boxes labelled **From** and **To**. You may also enter times using the respective boxes. Push the  button to select dates using a calendar.
- Push the **Find** button and the search results will appear on the screen.
- Select a data set and push the **Navigate** button to see this data set in the main [database window](database window).

# Transferring Data Between EMIGMA Databases

To import data sets to the current database from a different one:

- Select **PETROS EIKON database** under the **Other Sources** tab of the **Import** window.

To export data sets from the current database to a different one:

- Click the  button and select **.MDB EMIGMA database** from the list of export formats on the window that appears.

Below is the interface used to transfer a data set between databases:

- The path of your current database will be in either the **To** or **From** field of this window depending on whether data sets are being imported or exported from the current database. All the projects available in your current database will be displayed in the corresponding Database list below the two fields.
- Click the  button next to the field that is empty in the upper part of the window. Browse for the database with which you would like to exchange a project/survey/data set. After you have specified the database, all the projects available in it will appear in the corresponding Database list on the lower part of the window.

  ***Note***. *Projects have a blue diamond next to them, surveys have a green diamond next to them, and data sets have a red diamond.*

To transfer a project:

- Select the checkbox next to the desired project from the **Database FROM** list.

- Click **Copy to Database TO**. The project appears in the **Database TO** list on the right, including all the surveys and data sets it originally contained.

To import a data set into a new project:

- Expand a required project and survey in the **Database FROM** list. Select a data set to import.

- Click **Copy to Database TO** . The project appears in the **Database TO** list. It contains only the survey and data set that you selected in the **Database FROM** list.

To remove any project/survey/data set from the **Database TO** list:

- Select a project/survey/data set from the list and click **Delete from Database TO**.

# Exporting to a compressed EMIGMA Database

Click the  button and select **Transfer to EMIGMA DB and compress** from the list of export formats on the window that appears. Select a name and location for your zip file and you will be presented with the interface used to export data sets to a zip file:

- Click the  button next to the **To** field to change the folder or name for the zip file.
- The path of your current database will be in the **From** field of this window. All the projects available in your current database will be displayed in the **Database FROM** list.

  ***Note***. *Projects have a blue diamond next to them, surveys have a green diamond next to them, and data sets have a red diamond.*

To transfer a project:

- Select the checkbox next to the desired project from the **Database FROM** list. All the data sets for the project will be selected by default. You may deselect any data sets you do not want in the new database.

- Click **Copy to Zip File List**. The project appears in the **Zip File List** box on the right, including all the surveys and data sets that were selected.

To remove any project/survey/data set from the **Zip File List**:

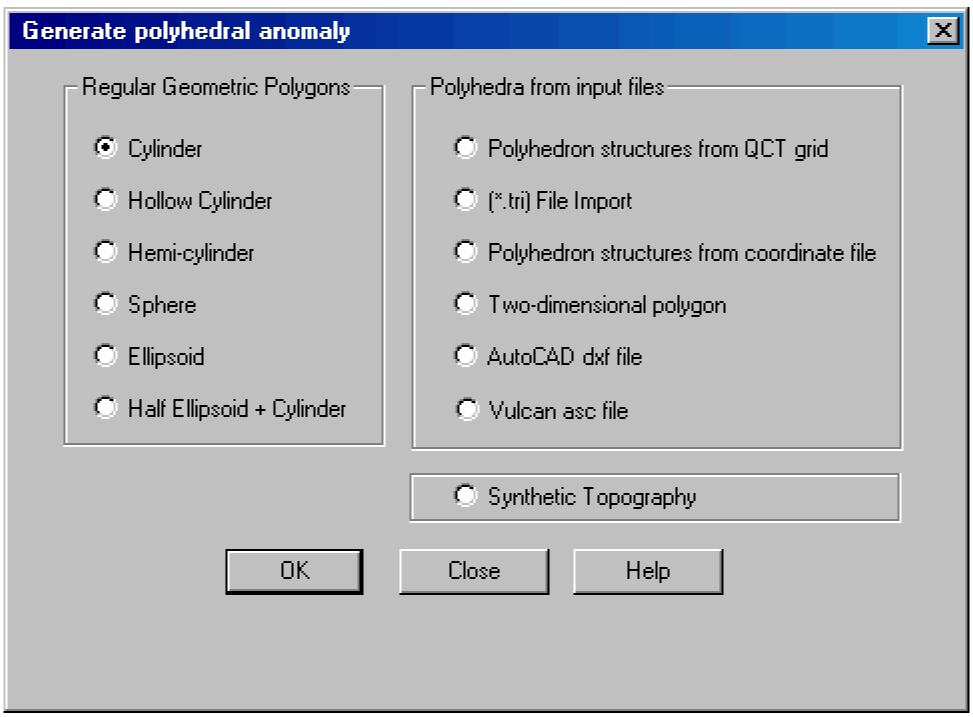
- Select it and click **Delete from Zip File List**.

Once you are satisfied with the data sets displayed in the Zip File List. Click **Create Zip File** to save a compressed database containing the data sets you have selected.

# Poly Generator 

This tool creates both simple and complicated geometric shapes that can be used in EMIGMA. It is started by clicking the  button on the EMIGMA toolbar. You may also select **Poly Generator** from the EMIGMA group on the Windows Start menu to run it as a standalone.

Choose the shape you would like to generate and click **OK** on the **Generate polyhedral anomaly** interface:

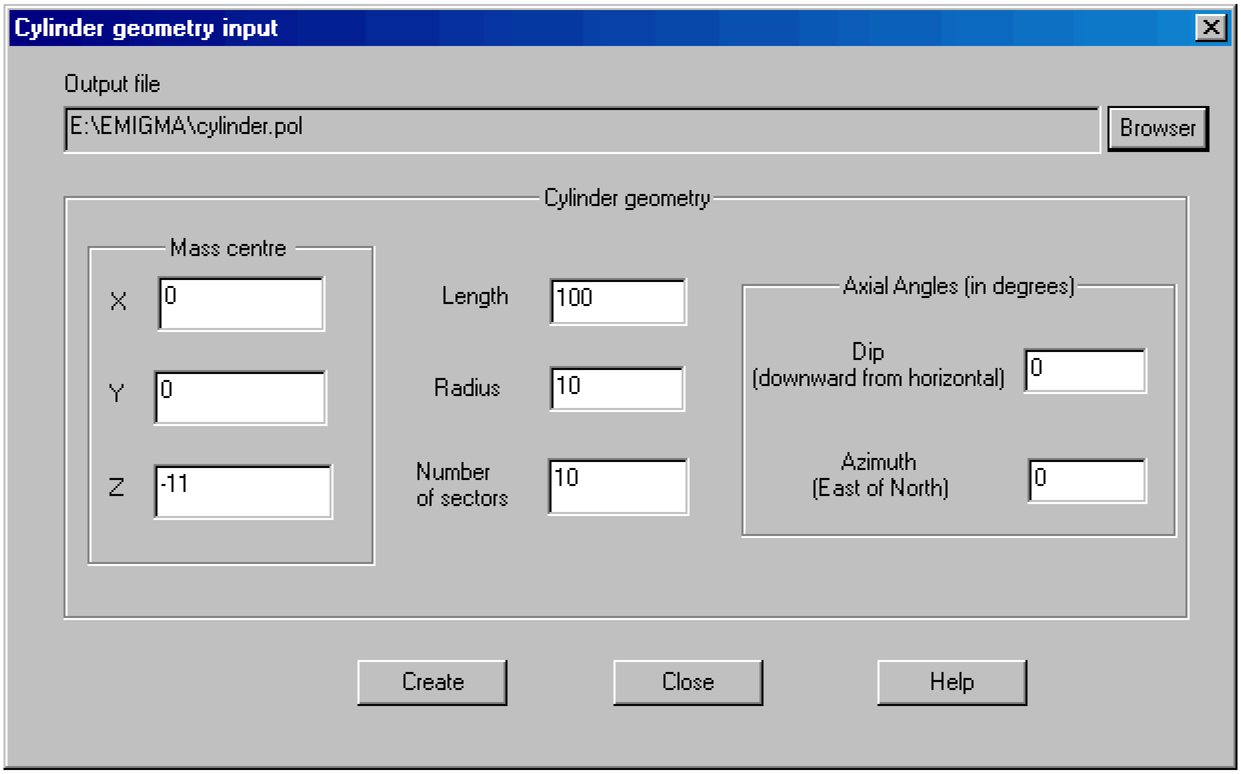



Available options are:

- Cylinder
- Hollow cylinder
- Hemi-cylinder
- Sphere
- Two-dimensional polygon
- Ellipsoid
- Half Ellipsoid + Cylinder

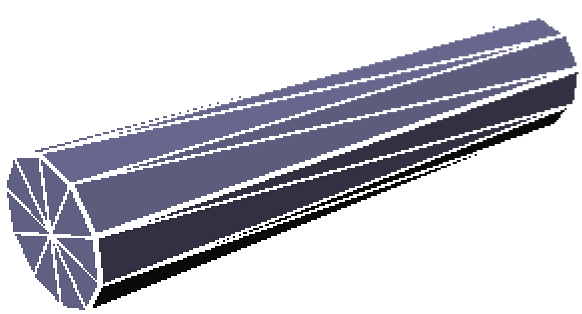
Polyhedron structures from QCT grid
(*.tri) File Import
Polyhedron structures from coordinate file
AutoCAD dxf file
Vulcan asc file
Synthetic Topography
Topography (actual)

# Cylinder Geometry Input




- Specify the **Centre point** of the cylinder.
- Specify the **Length** and **Radius** of the cylinder.
- Specify the cylinder's orientation in the **Axial Angles** section
- Approximate the curvature of the cylinder in the **Number of sectors** section. A higher value will give the cylinder a better curve.

The following cylinder will result from the entries displayed in the above interface:



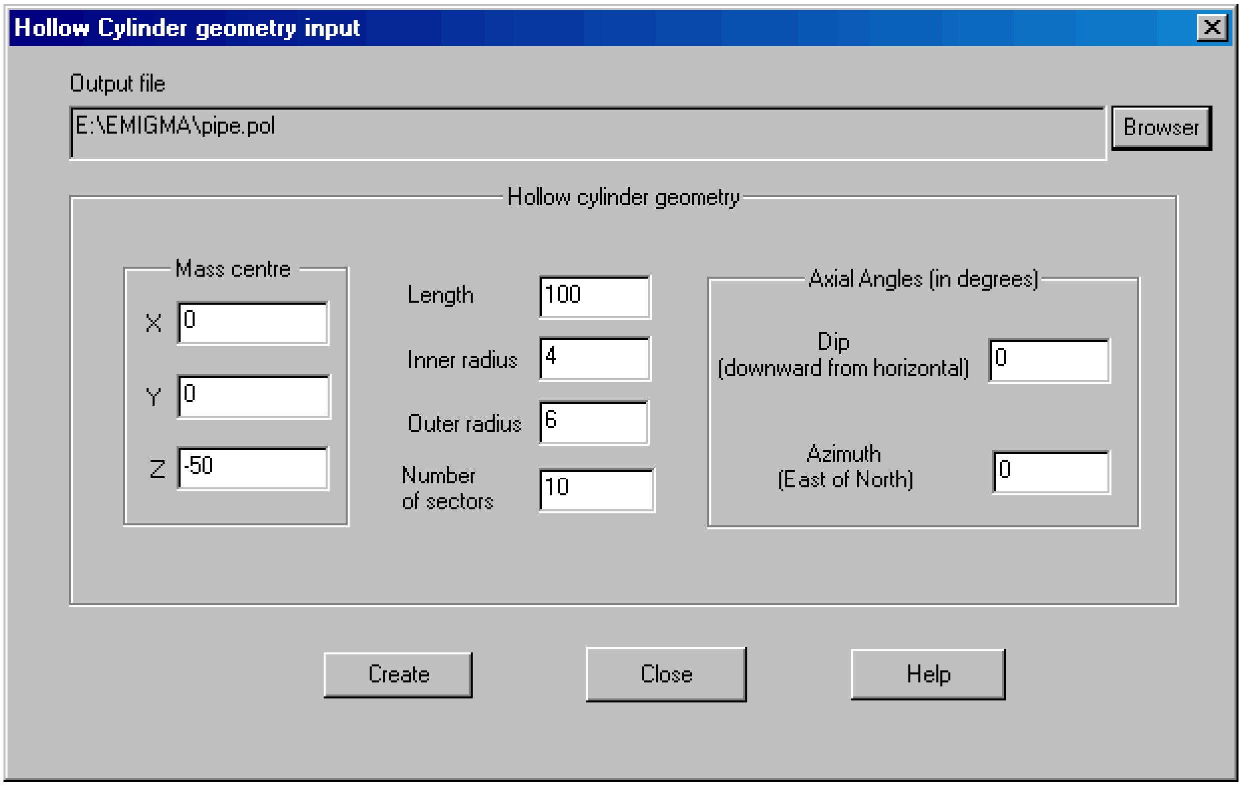
When using the standalone Poly Generator:

- Click **Browser** to specify the *.pol filename and its location.
- Click **Create** to save the *.pol file.
- See Insert a target for details on importing polyhedra files into EMIGMA.

When using Poly Generator from EMIGMA:

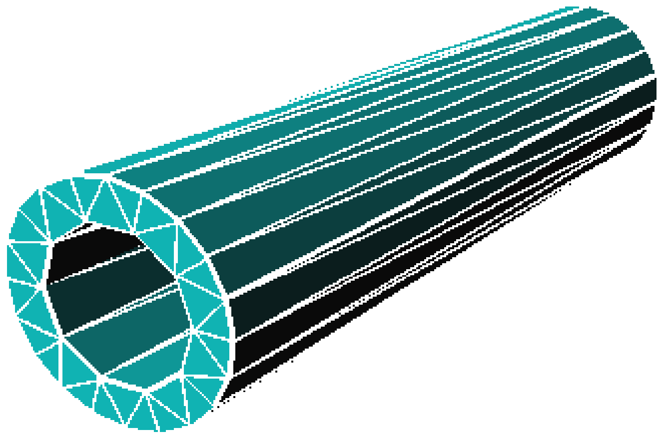
- Click **Create** and the structure will be saved to a new data set in the current survey of the database.

# Hollow Cylinder Geometry Input




- Specify the **Centre point** of the cylinder.
- Specify the **Length** and **Radius** of the cylinder.
- Specify the cylinder's orientation in the **Axial Angles** section
- Approximate the curvature of the cylinder in the **Number of sectors** section. A higher value will give the cylinder a better curve.

The following cylinder will result from the entries displayed in the above interface:



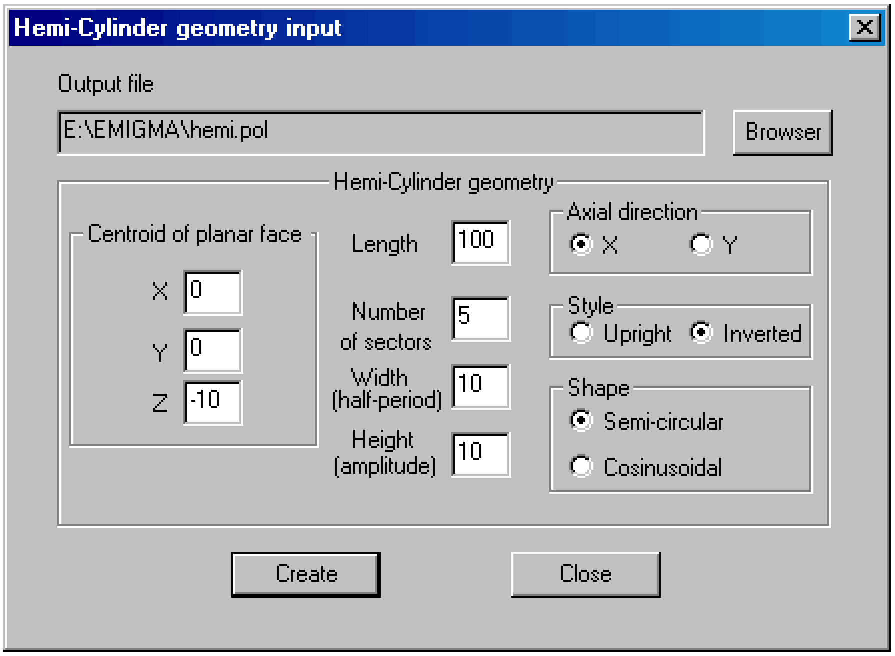
When using the standalone Poly Generator:

- Click **Browser** to specify the *.pol filename and its location.
- Click **Create** to save the *.pol file.
- See [Insert a target] for details on importing polyhedra files into EMIGMA.

When using Poly Generator from EMIGMA:

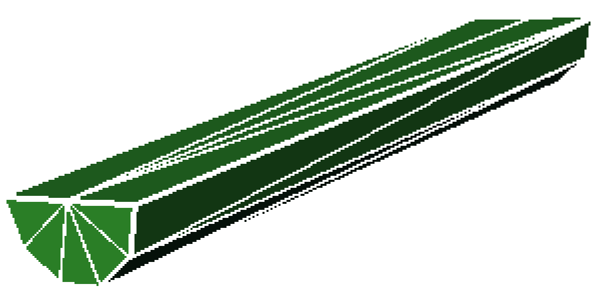
- Click **Create** and the structure will be saved to a new data set in the current survey of the database.

# Hemi-Cylinder Geometry Input




- Specify the **Centroid of the planar face**.
- Specify the **Length**, **Width** and **Height** of the hemi-cylinder.
- Specify the cylinder's orientation in the **Axial direction** section
- Approximate the curvature of the hemi-cylinder in the **Number of sectors** section. A higher value will result in a better curve.
- Specify which side will be curved in the **Style** section.
- Specify the type of curve in the **Shape** section. If **Cosinusoidal** is selected, **Width** will apply to the half-period and **Height** will apply to the amplitude.

The following hemi-cylinder will result from the entries displayed in the above interface:



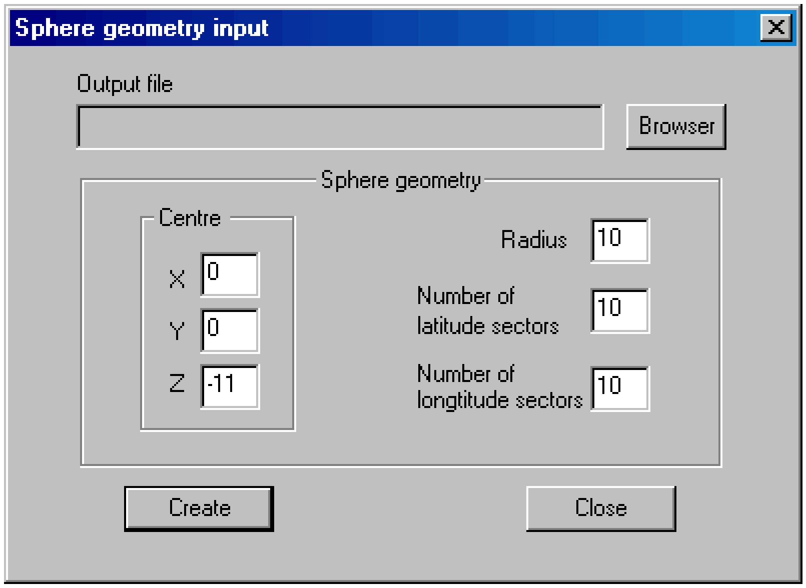
When using the standalone Poly Generator:

- Click **Browser** to specify the *.pol filename and its location.
- Click **Create** to save the *.pol file.
- See Insert a target for details on importing polyhedra files into EMIGMA.

When using Poly Generator from EMIGMA:

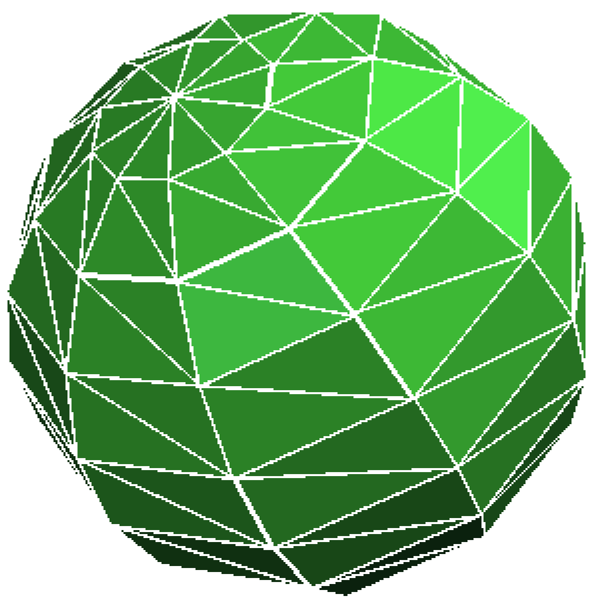
- Click **Create** and the structure will be saved to a new data set in the current survey of the database.

## Sphere Geometry Input




- Specify the **Centre** position and **Radius** of the sphere.
- Approximate the curvature of the sphere in the **Number of latitude sectors** and **Number of longitude sectors** sections. A higher value for these parameters will result in a better curve.

The following sphere will result from the entries displayed in the above interface:



When using the standalone Poly Generator:

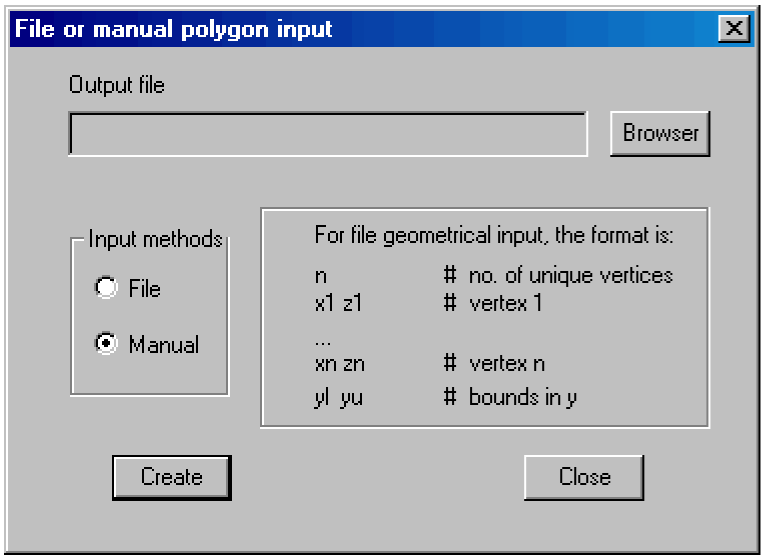
- Click **Browser** to specify the *.pol filename and its location.
- Click **Create** to save the *.pol file.
- See Insert a target for details on importing polyhedra files into EMIGMA.

When using Poly Generator from EMIGMA:

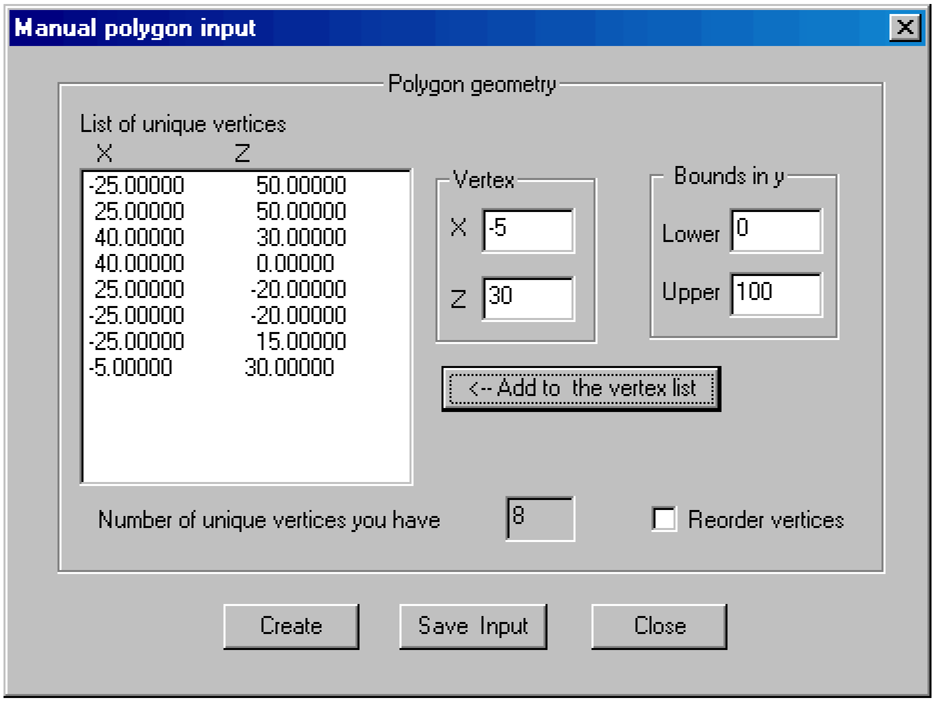
- Click **Create** and the structure will be saved to a new data set in the current survey of the database.

## 2D Polygon Geometry Input

- The polygon vertices can be specified in a **File** or entered manually. Choose a method for entering the vertices and click **Create**.

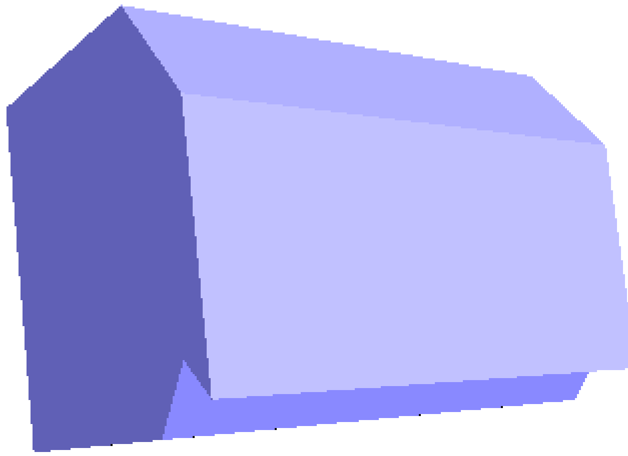



Choose **File** and you will receive a prompt for a *.in file which needs to follow the format described on the above interface. Choose **Manual** and the following window is displayed:




- Specify the x and z coordinates of a vertex and click **Add to the vertex list**.
- Repeat the above step for all the vertices of the polygon.
- Specify the thickness of the polygon and y position in the **Bounds in y** section.
- Click **Reorder vertices** to have the polygon creation algorithm attempt to detect any vertices in the list which appear to be out of order and make corrections.
- Click **Save Input** to create a *.in file that can be used on the previous interface to create a polygon.

The following polygon will result from the entries displayed in the above interface:



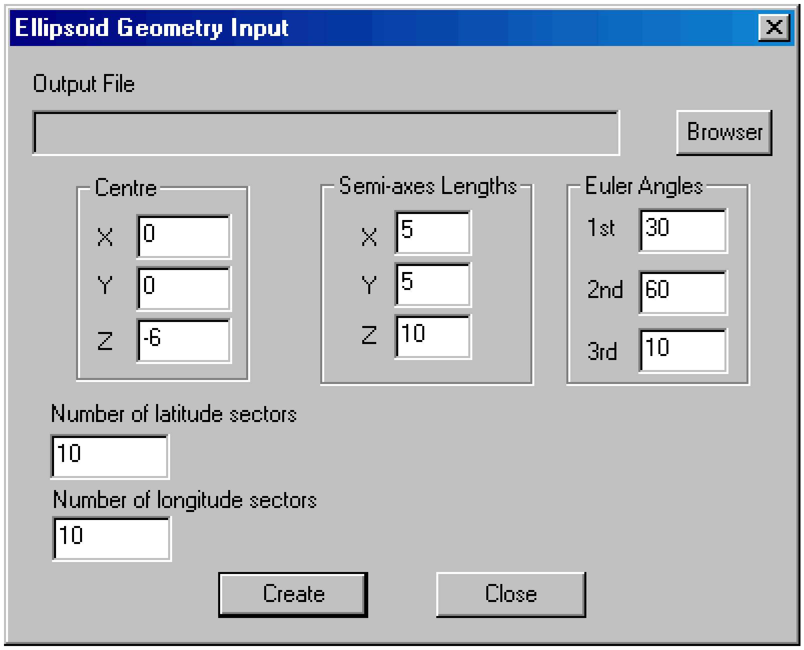
When using the standalone Poly Generator:

- Click **Browser** on the first page to specify the *.pol filename and its location.
- Click **Create** to save the *.pol file.
- See Insert a target for details on importing polyhedra files into EMIGMA.

When using Poly Generator from EMIGMA:

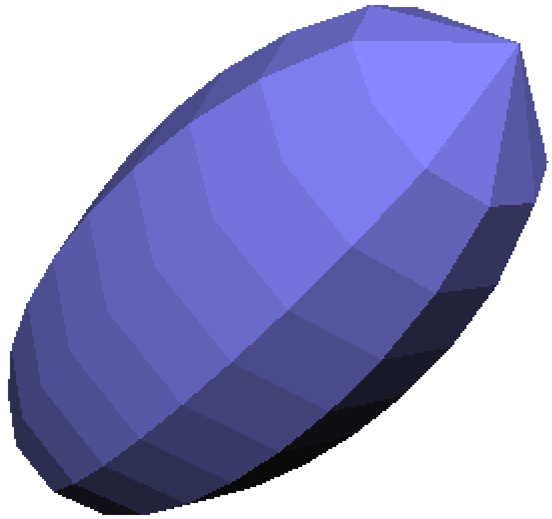
- Click **Create** and the structure will be saved to a new data set in the current survey of the database.

## Ellipsoid Geometry Input




- Specify the **Centre** position of the ellipsoid.
- Specify the ellipsoid's orientation in the **Euler Angles** section.
- Specify the dimensions of the ellipsoid in the **Semi-axes Lengths** section.
- Approximate the curvature of the ellipsoid in the **Number of latitude sectors** and **Number of longitude sectors** sections. A higher value for these parameters will result in a better curve.

The following ellipsoid will result from the entries displayed in the above interface:



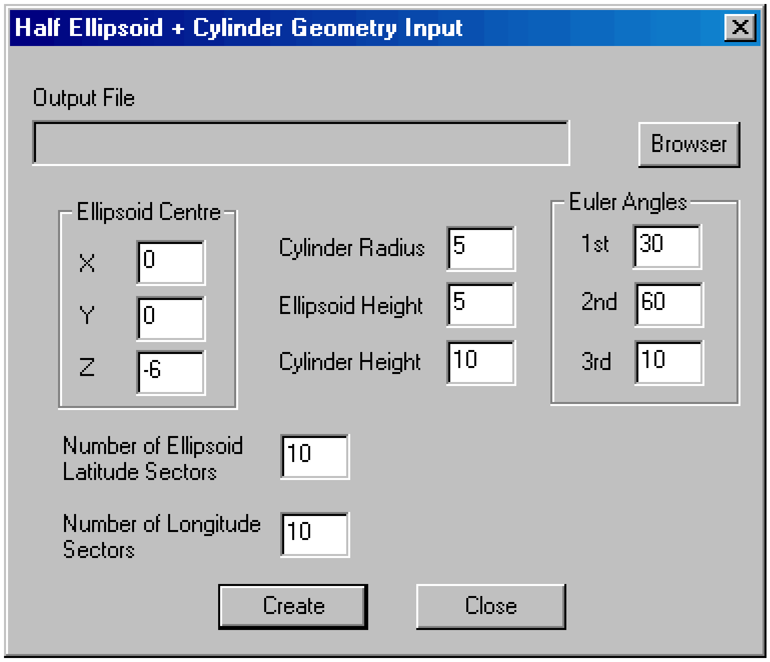
When using the standalone Poly Generator:

- Click **Browser** to specify the *.pol filename and its location.
- Click **Create** to save the *.pol file.
- See Insert a target for details on importing polyhedra files into EMIGMA.

When using Poly Generator from EMIGMA:

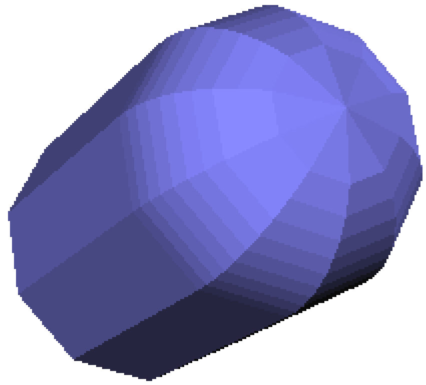
- Click **Create** and the structure will be saved to a new data set in the current survey of the database.

## Half Ellipsoid + Cylinder Geometry Input




- Specify the **Centre** position of the ellipsoid part of the shape.
- Specify the orientation in the **Euler Angles** section.
- Specify the dimensions of the ellipsoid in the **Semi-axes Lengths** section.
- Approximate the curvature of the ellipsoid in the **Number of latitude sectors** and **Number of longitude sectors** sections. A higher value for these parameters will result in a better curve.

The following figure will result from the entries displayed in the above interface:



When using the standalone Poly Generator:

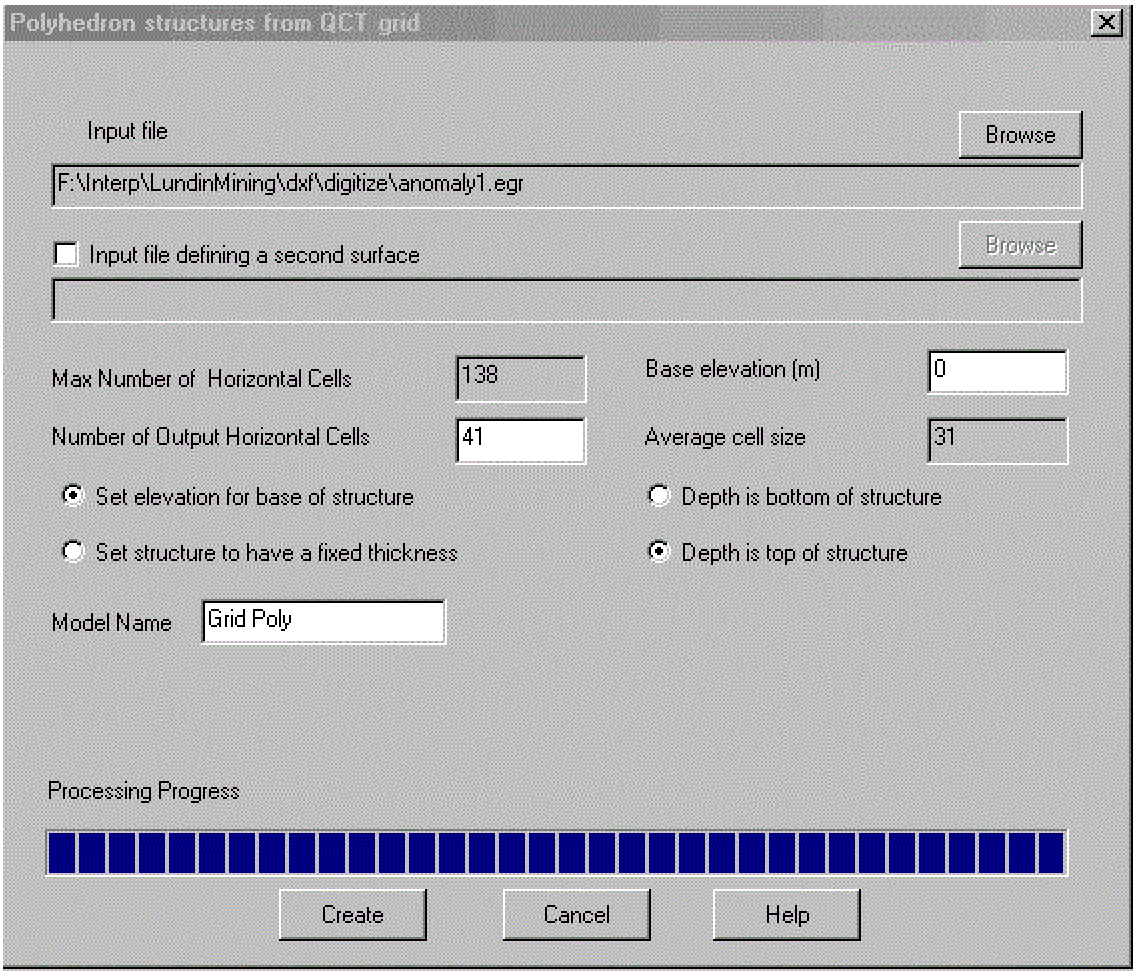
- Click **Browser** to specify the *.pol filename and its location.
- Click **Create** to save the *.pol file.
- See Insert a target for details on importing polyhedra files into EMIGMA.

When using Poly Generator from EMIGMA:

- Click **Create** and the structure will be saved to a new data set in the current survey of the database.

## Polyhedron structures from QCT grid

Your data for this polyhedron is first imported to QCTool. This data is to be based upon one or two complex surfaces (x,y,z). The surfaces are then interpolated and saved to a QCTool Grid file (.egr) for use in this tool.



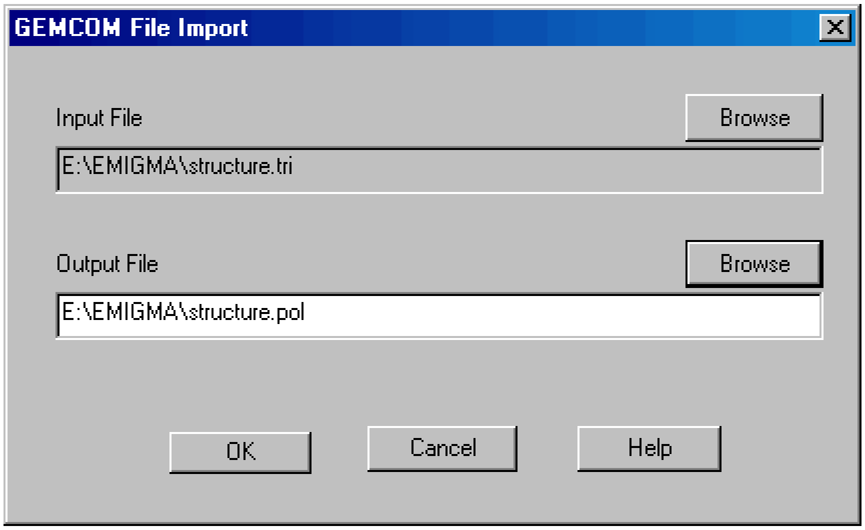

- Click the first **Browse** button to specify the **Input File** from QCTool. The file will have a .egr extension.
- If utilizing one surface, select whether the elevation represents top or bottom of the surface.
- Define whether the structure has a constant thickness (Set structure to have a fixed thickness) or either a fixed elevation for the top or bottom of the structure depending on whether the elevation in the grid is the bottom or the top of the structure

- Specify the **Number of Output Horizontal Cells** to obtain a smaller poly file but a less detailed structure. The total number of cells defined in the QCTool grid file is displayed as **Max Number of Horizontal Cells**. If utilizing two surfaces, the first file is the top surface and the second file the bottom surface of the structure
- Click **Create** and the structure will be saved to a new data set in the current survey of the database.

When using the standalone Poly Generator:

- Click the second **Browse** button to specify the *.pol **Output File** and its location.
- Click **Create** to save the *.pol file.
- See Insert a target for details on importing polyhedra files into EMIGMA.

# *.tri File Import




It is possible to create polyhedra from the *.tri file format.

- Click the first **Browse** button to specify the *.tri **Input File**.

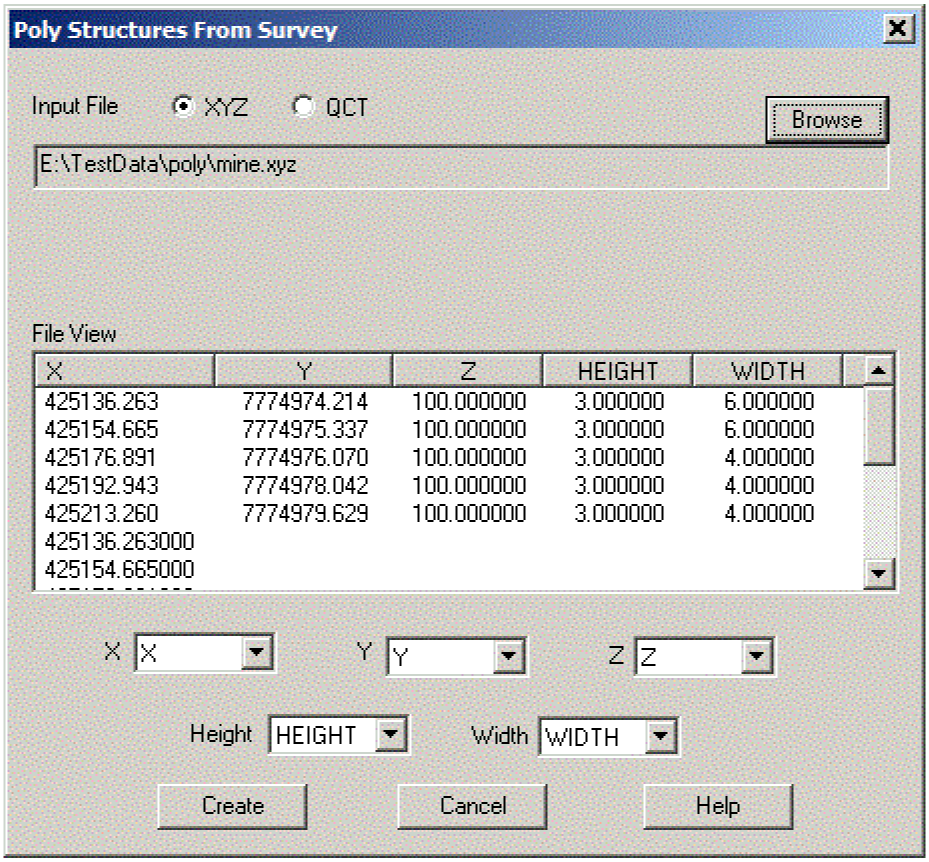
When using the standalone Poly Generator:

- Click the second **Browse** button to specify the *.pol **Output File** and its location.
- Click **OK** to save the *.pol file.
- See Insert a target for details on importing polyhedra files into EMIGMA.

When using Poly Generator from EMIGMA:

- Click **OK** and the structure will be saved to a new data set in the current survey of the database.

# Polyhedron structures from coordinate file




Use this tool to create the polyhedron based on a line, e.g. a mine void dimension survey.

- Choose **xyz** or **qct** format for the input file.
  Click the first **Browse** button to specify the **Input File**. The file must have a .xyz or .qct extension respectively.

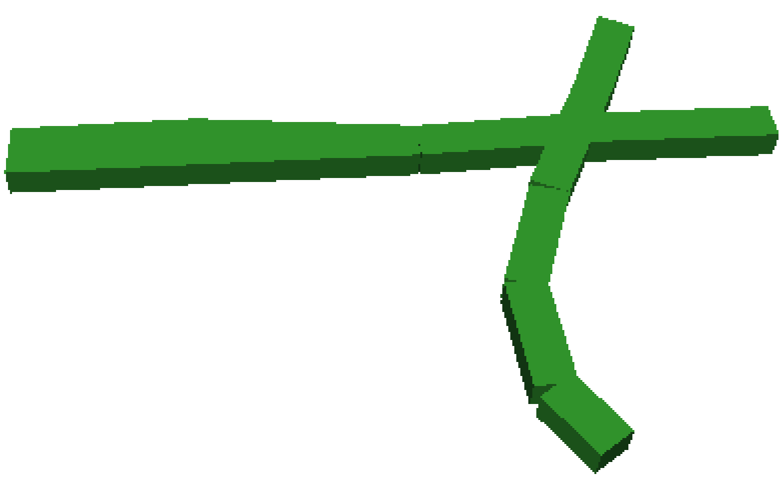
  Normally, the xyz file has 5 columns. In order, they are x, y, z, height and width. QCT file might have them in no specific order. Program tries to find them by their names. If columns for X, Y, Z, Height, Width are not correctly determined automatically, select them in the comboboxes.

  In **.xyz**, a "LINE" label is required immediately before the listing of the column data. Multiple lines can be defined in the same file by starting each line with a unique "LINE" label. Comments can be added at the beginning of the file before the "LINE" label.

An example file:

| x | y | z | height | width |
|---|---|---|---|---|
| LINE 1 | | | | |
| 425136.263 | 7774974.214 | 100.000000 | 3.000000 | 6.000000 |
| 425154.665 | 7774975.337 | 100.000000 | 3.000000 | 6.000000 |
| 425176.891 | 7774976.070 | 100.000000 | 3.000000 | 4.000000 |
| 425192.943 | 7774978.042 | 100.000000 | 3.000000 | 4.000000 |
| 425213.260 | 7774979.629 | 100.000000 | 3.000000 | 4.000000 |
| LINE 2 | | | | |
| 425199.359 | 7774994.402 | 100.000000 | 3.000000 | 4.000000 |
| 425195.478 | 7774984.233 | 100.000000 | 3.000000 | 4.000000 |
| 425189.368 | 7774970.076 | 100.000000 | 3.000000 | 4.000000 |
| 425185.921 | 7774958.594 | 100.000000 | 3.000000 | 4.000000 |
| 425187.480 | 7774947.618 | 100.000000 | 3.000000 | 4.000000 |
| 425191.867 | 7774942.397 | 100.000000 | 3.000000 | 4.000000 |

This example file generates the following figure:



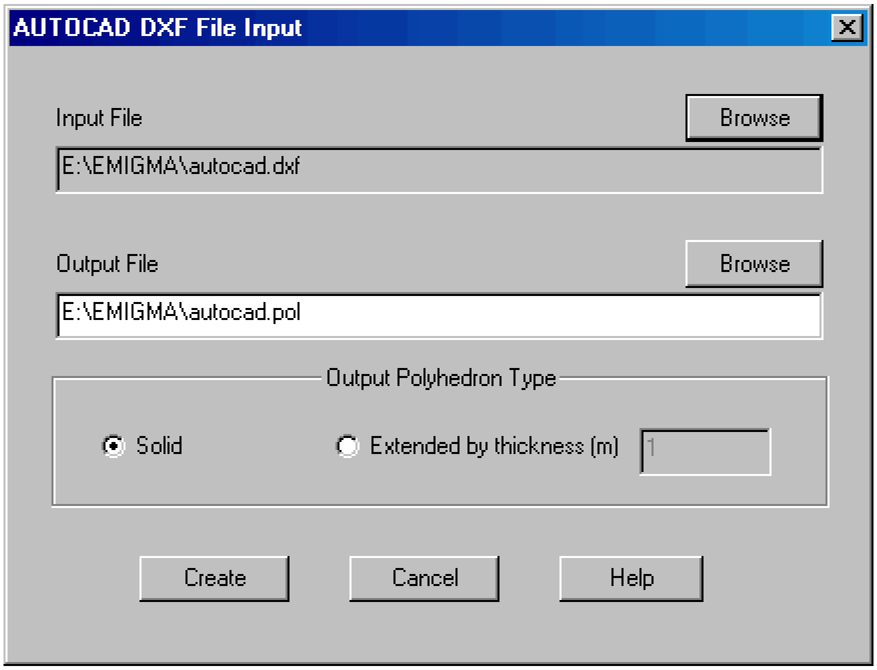
When using the standalone Poly Generator:

- Click the second **Browse** button to specify the *.pol **Output File** and its location.
- Click **Create** to save the *.pol file.
- See Insert a target for details on importing polyhedra files into EMIGMA.

When using Poly Generator from EMIGMA:

- Click **Create** and the structure will be saved to a new data set in the current survey of the database.

# AutoCAD dxf file




It is possible to create polyhedra from AutoCAD's dxf file format.

- Click the first **Browse** button to specify the dxf **Input File**.
- Select **Extended by thickness** to modify the model in the dxf file by adding a thickness to the model surface.

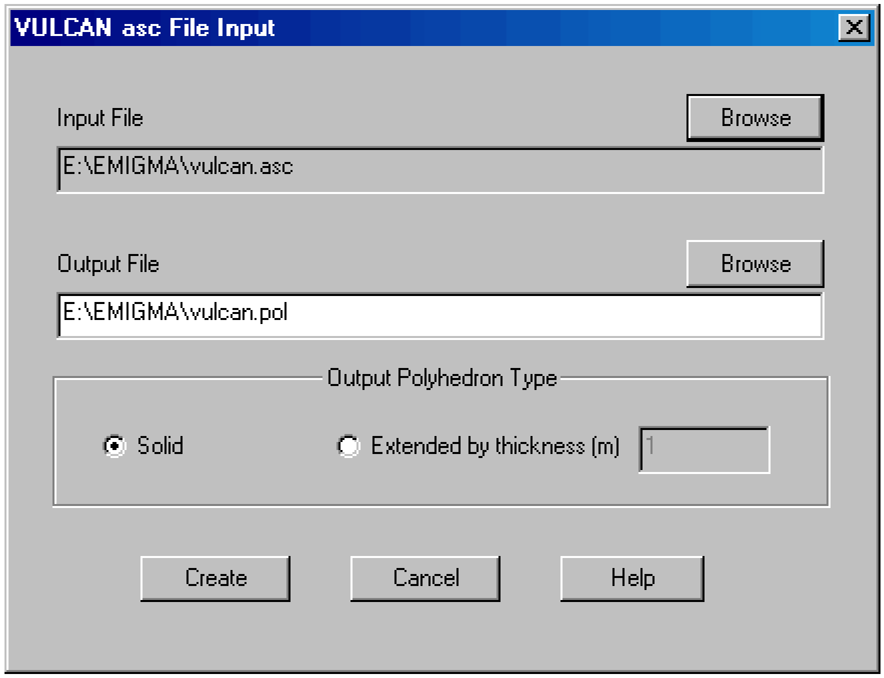
When using the standalone Poly Generator:

- Click the second **Browse** button to specify the *.pol **Output File** and its location.
- Click **OK** to save the *.pol file.
- See Insert a target for details on importing polyhedra files into EMIGMA.

When using Poly Generator from EMIGMA:

- Click **OK** and the structure will be saved to a new data set in the current survey of the database.

## Vulcan asc file




It is possible to create polyhedra from Vulcan's asc file format.

- Click the first **Browse** button to specify the asc **Input File**.
- Select **Extended by thickness** to modify the model in the asc file by adding a thickness to the model surface.

When using the standalone Poly Generator:

- Click the second **Browse** button to specify the *.pol **Output File** and its location.
- Click **OK** to save the *.pol file.
- See Insert a target for details on importing polyhedra files into EMIGMA.

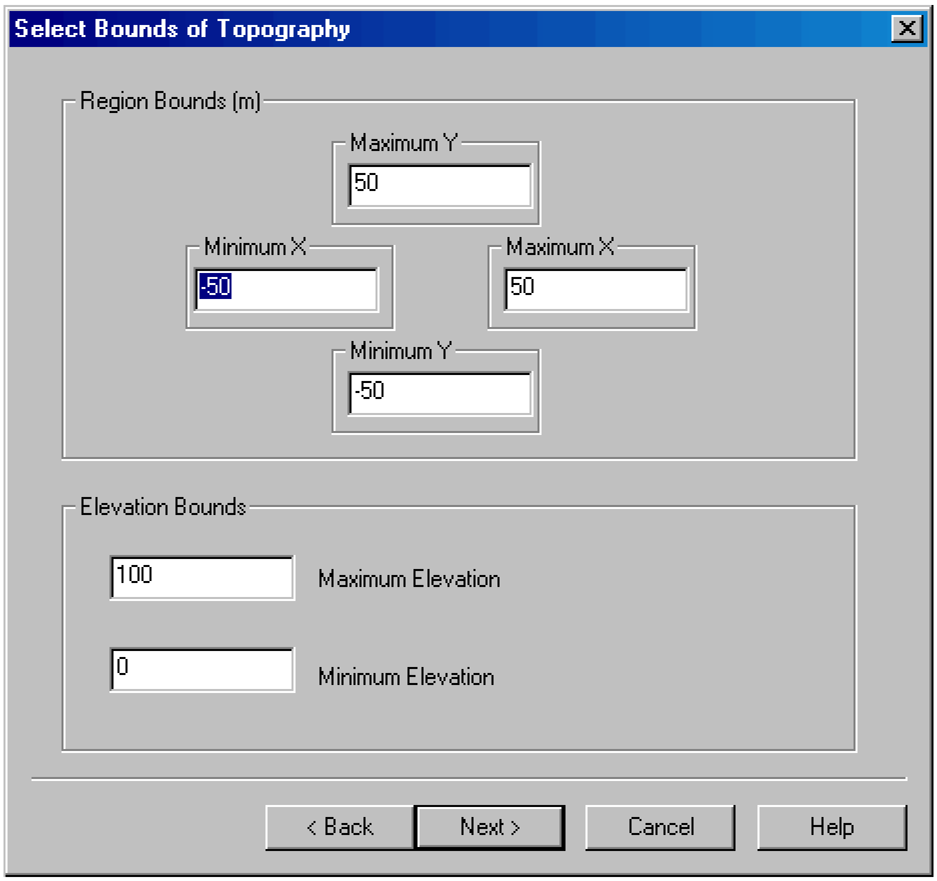
When using Poly Generator from EMIGMA:

- Click **OK** and the structure will be saved to a new data set in the current survey of the database.

# Synthetic Topography

## Selecting Topography Bounds

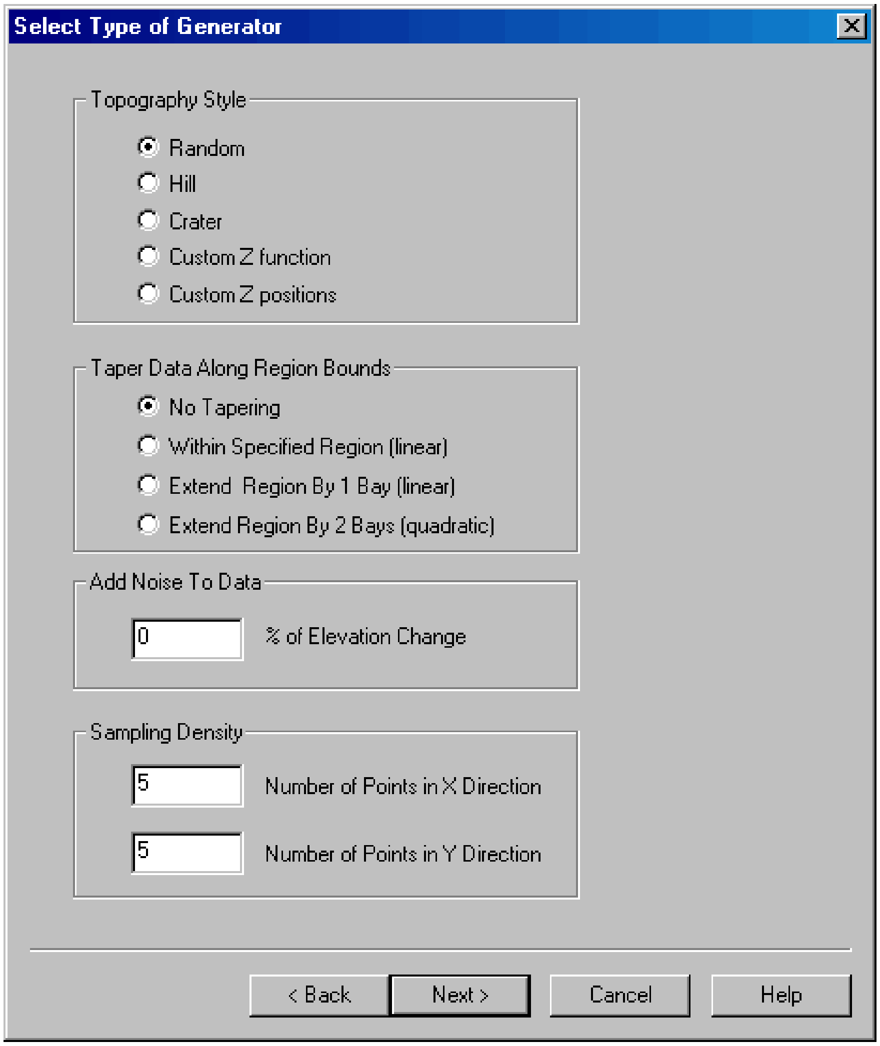
To generate a topographic surface, select **Synthetic Topography** from the initial Poly Generator window and click **OK**. This brings you to the **Select Bounds of Topography** interface:




- In the **Region Bounds** section, specify dimensions of the topographic region to be generated by entering the minimum and maximum values for both x and y in metres.
- In the **Elevation Bounds** section, specify range of possible values for the elevation by entering the minimum and maximum elevation values.
- Click **Next**.

# Selecting Topography Type

The **Select Type of Generator** interface is displayed:

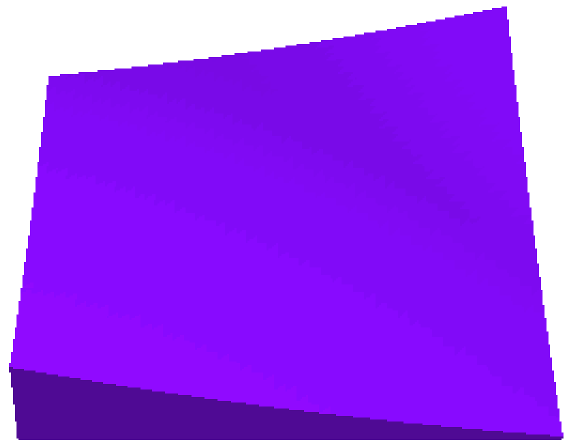



- In the **Topography Style** section, choose the general shape your region. Available options are:

Random

Hill

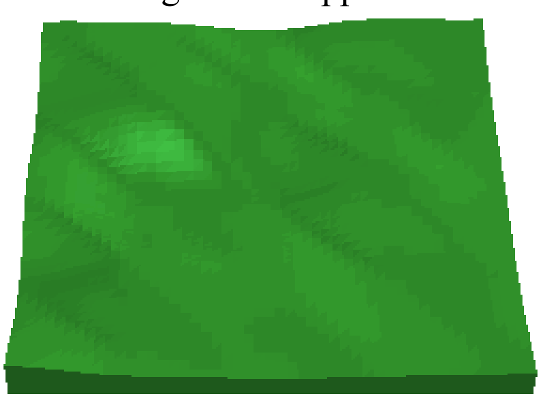
Crater



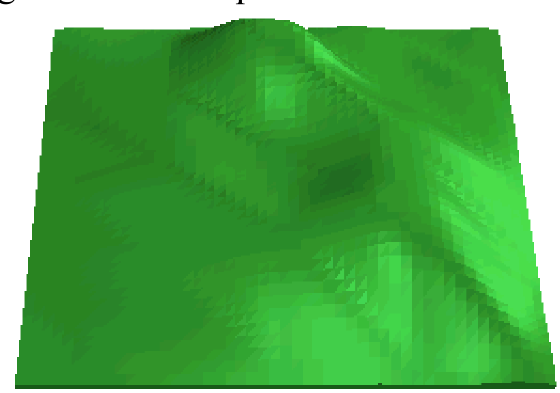
Custom Z function (e.g. $z = 2x^2 + 10xy + 2y^2$)

Another option: Custom Z positions - you must enter each value for the grid positions.

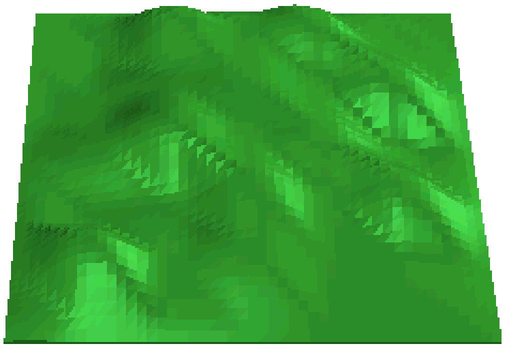
- In the **Taper Data Along Region Bounds** section, specify how the generated region will appear at the edges. Available options are:



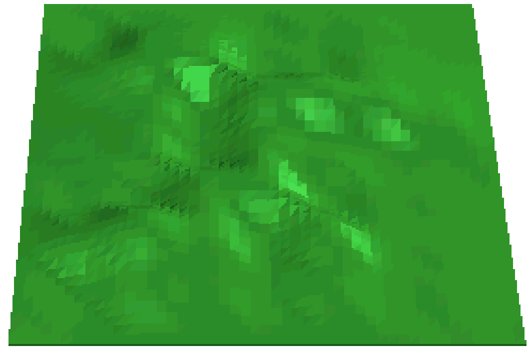
No tapering



Within Specified Region



Extend Region By 1 Bay

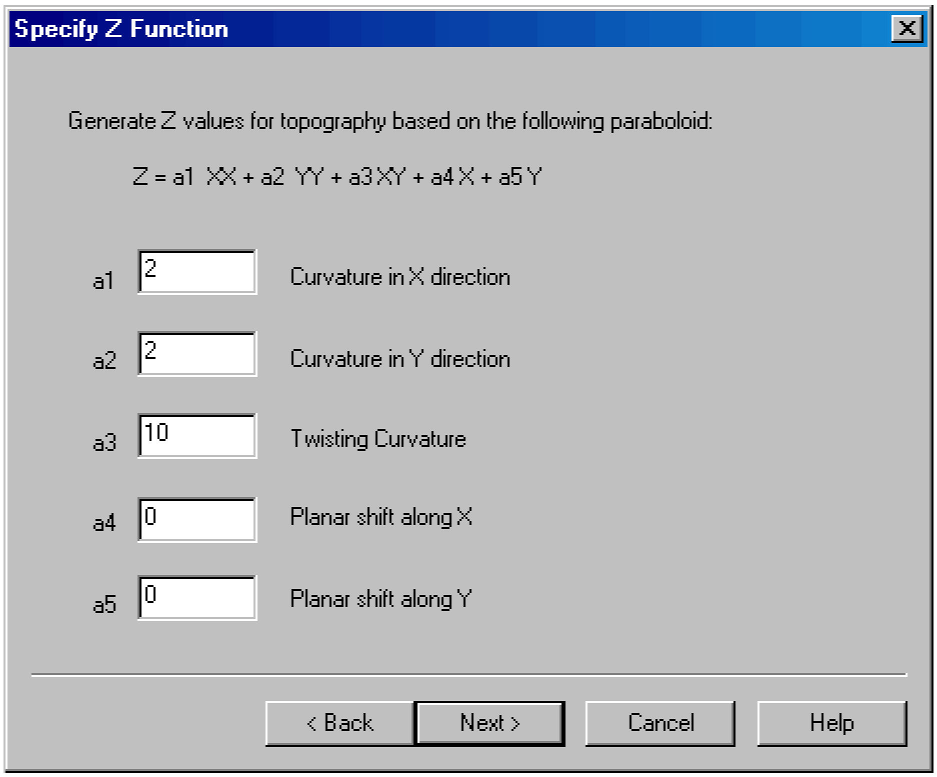


Extend Region By 2 Bays

- In the **Add Noise To Data** section, specify the level of randomness for the generated surface.
- In the **Sampling Density** section, specify the granularity of the grid used to generate the surface in both x and y directions.
- Click **Next**.

# Selecting a Custom Z Function

Selecting the **Custom Z function** option will display the **Specify Z Function** interface:

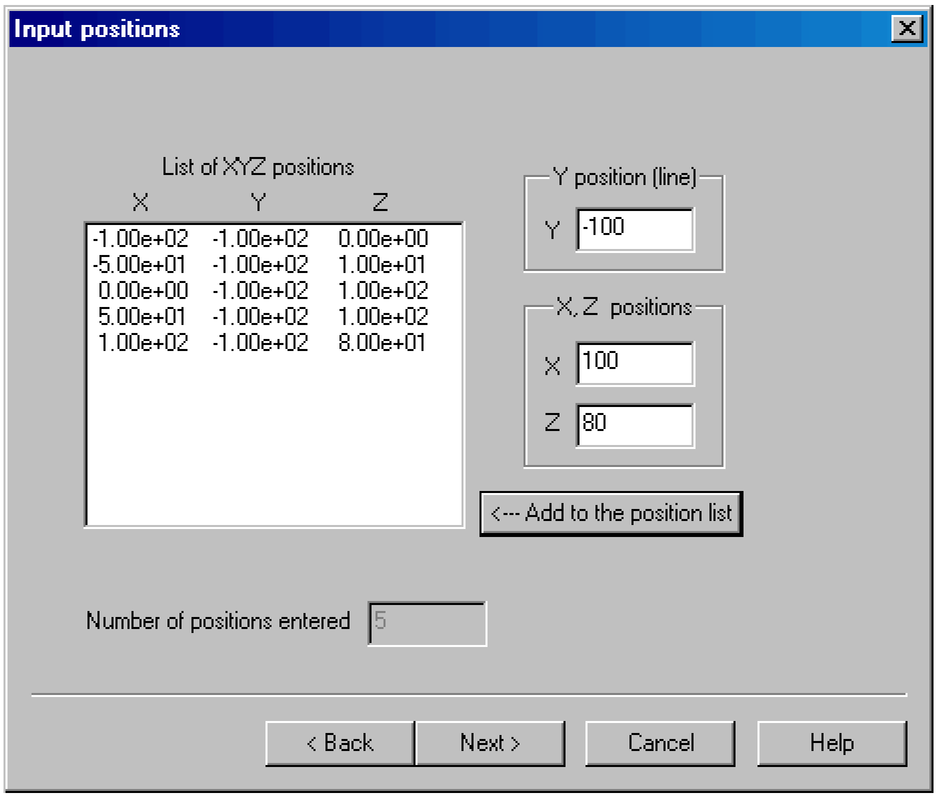



Specify the parameters for the function and click **Next**

The settings displayed in the above interface will create a paraboloid with the function: $z = 2x^2 + 10xy + 2y^2$

# Selecting Custom Z Positions

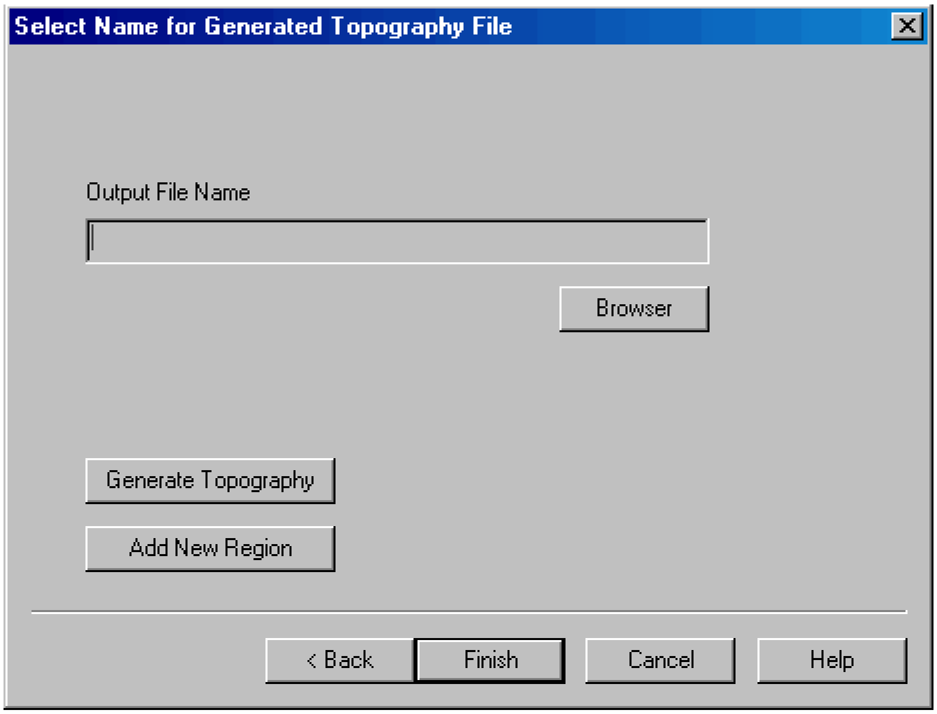
Selecting the **Custom Z positions** option will display the **Input Positions** interface:




- Enter values for **X**, **Y** and **Z**.
- Click **Add to the position list**
- Repeat the above steps until all the positions of the surface grid have been entered. The number of positions for grid was specified in the **Sampling Density** section of the previous page.
- The number of positions entered is displayed below the position list.
- Click **Next**

## Generating Topography

The **Select Name for Generated Topography File** interface is displayed:




When using the standalone Poly Generator:

- Click **Generate Topography** to create the *.xyz topography file. You will be prompted for the name of the file and a save location.
- Click **Finish**.
- See Topography File Format for details on how to import this file into EMIGMA.

When using Poly Generator from EMIGMA:

- Click **Generate Topography** and the topography will be saved to a new data set in the current survey of the database.

# Additional Tutorials

- EMIGMA Quick Reference
- Imaging and Inversion of One Line of EM31/34 data
- FDEM Manual - opening databases, importing data, working with data
- Surface Protem Survey and Model
- Importing GEM data
- Using PEImport to model Geonics Field Data
- 3D Gravity Inversion in EMIGMA
- A Moving Loop Setup
- Sengpiel Depth Sections
- TEM Inversion
- EMIGMA Modelling of TEM Data
- 3D Magnetic Inversion
- 1D FEM Inversion
- Building a Model
- Airborne FEM Tutorial
- Modelling in EMIGMA

# Technical Papers

1. **A Comparison of Airborne and Ground EM Data at a Calibration Site near the Grand Canyon**Nov 2011. Laura Davis, Ruizhong Jia and Ross Groom. 10th CIGEW Workshop, Nanchang, China. paper(PDF) or presentation (PDF)
2. **1D-Time Domain Inversion Incorporating Various Data Strategies with a Trust-Region Method**
   Nov 2011. Ruizhong Jia, L.J. Davis and Ross Groom. 10th China International Geo-Electromagnetic Workshop, Nanchang, China. paper(PDF)
3. **Preliminary Modeling of the Magnetic Effects of Steel Casings**
   May 2011. L.J. Davis and R.W. Groom. presentation(PDF)
4. **Deep, Accurate Structural Interpretations through Time Domain EM Techniques**
   May 2010. L.J. Davis and R.W. Groom. GeoCanada 2010 Conference, Calgary, Canada. poster(PDF)
5. **Calibration of Airborne TEM with Ground TEM. Two Case Studies in Arizona**
   April 2010. L.J. Davis. Workshop on Airborne EM, 2010 SAGEEP Annual Symposium, Keystone, Colorado. presentation(PDF)
6. **A Comparison of airborne and ground Electromagnetic data near the Grand Canyon**
   Feb 2010. L.J. Davis and R.W. Groom. Internal Report. paper(PDF)
7. **Inference of lithologic distributions in an alluvial aquifer using airborne transient electromagnetic surveys**
   2009. Jesse E. Dickinson, D.R. Pool, R.W. Groom, and L.J. Davis. Accepted with minor revisions, Geophysics. paper(PDF)
8. **Some issues on 1d-TEM inversion utilizing various multiple data strategies**
   Oct 2009. Ruizhong Jia, L.J. Davis, and R.W. Groom. 79th SEG Conference, Houston, Texas. paper(PDF)
9. **A comparison of airborne and ground electromagnetic data near the Grand Canyon**

Oct 2009. L.J. Davis and R.W. Groom. 79th SEG Conference, Houston, Texas. paper (PDF) or presentation (PDF)

10. **New Approaches to Topographic Gravity Corrections**
April 2009. Ruizhong Jia, Laura Davis and R.W. Groom. 2009 SAGEEP Annual Symposium, Fort Worth, Texas. paper(PDF)
11. **Airborne - EM Hydrocarbon Mapping In Mozambique**
Feb 2009. Andreas Pfaffling, Ståle Monstad, Ross W. Groom and Jonathan Rudd. 21st ASEG-PESA Conference, Sydney, Australia. paper(PDF)
12. **Traditional EM & onshore hydrocarbon exploration**
Oct 2008. Andreas Pfaffling and Ross W. Groom. EAGE Non-Seismic Workshop, Manama, Bahrain. presentation(PDF)
13. **Processing gradients of magnetic data utilizing an equivalent source technique**
2007. Ruizhong Jia and R.W. Groom. 77th SEG Conference, San Antonio, Texas. presentation (PDF)
14. **Airborne EM Data Comparisons in a Sedimentary Basin**
Sept 2007. Laura Davis and Ross Groom. Exploration 2007, Toronto, Canada. presentation(PDF)
15. **Enhancing Model Reliability from TEM Data Utilizing Various Multiple Data Strategies**
2007. Ruizhong Jia and R.W. Groom. 2007 SAGEEP Annual Symposium, Denver, Colorado. presentation (PDF)
16. **Enhancing Model Reliability from TEM Data Utilizing Various Multiple Data Strategies**
2007. Ruizhong Jia and R.W. Groom. KEGS Symposium, Toronto, Canada. presentation (PDF)
17. **Vertical Spatial Sensitivity and Exploration Depth of Low-Induction-Number Electromagnetic-Induction Instruments**
2006. James B. Callegary, Ty P.A. Ferré and R.W. Groom. Vadose Zone Journal 6:158-167. paper (PDF)
18. **Aerial Transient Electromagnetic Surveys of Alluvial Aquifers in Rural Watersheds of Arizona, United States**
2006. Poole, Callegary and Groom. 2006 AGU Fall Meeting, San Francisco, California. poster (PDF)
19. **Processing gradients of magnetic data utilizing an equivalent technique. Internal paper.**

2006. Ruizhong Jia and R.W. Groom. paper (PDF)

20. **Magnetic Case Study: Raglan Mine**
2006. Laura Davis. Internal Report. paper (PDF)
21. **Geophysical case study of the Iso and New Insco deposits, Québec, Canada: Part II, modeling and interpretation**
2006. Li Zhen Cheng. Reviewed Draft. Exploration and Mining Geology, January 2006, vol. 15, no. 1-2, p. 65-74. paper (PDF)
22. **Comparison of Theoretical and Physical Model Studies of the Responses of Moving Source and Fixed Loop Electromagnetic Exploration Systems**
2005. K. Duckworth. Pure and Applied Geophysics, December 2005, vol. 162, no. 12, p. 2505-2521. paper (PDF)
23. **Inversion of Ground Gravity and Airborne Gradient Data**
Nov 2005. Ruizhong Jia, Ross Groom and Bob Lo. 75th SEG Conference, Houston, Texas. presentation (PDF)
24. **On some issues regarding 3D-gravity inversion**
2005. Ruizhong Jia and R.W. Groom. SEG. 75 Annual Meeting, Houston, TX. paper (PDF)
25. **3-D sensitivity distribution of low-induction-number frequency-domain electromagnetic instruments**
2005. James B. Callegary, Ty P.A. Ferré and R.W. Groom. 2005 AGU Fall Meeting, San Francisco, California. poster (PDF)
26. **On Time-Domain Transient Electromagnetic Soundings, Extended Abstract**
2005. Ruizhong Jia and Ross Groom. 2005 SAGEEP Annual Symposium, Atlanta, Georgia. presentation (PDF)
27. **Final Report On Improved Aeromagnetic Compensation: A New Aircraft Compensation System for Magnetic Terrains**
2004. John Jia, Bob Lo and Ross Groom. Unpublished Report. paper (PDF)
28. **The Use of GPS Sensors and Numerical Improvements in Aeromagnetic Compensation. Extended Abstract**
2004. Ruizhong Jia, R.W. Groom and Bob Lo. SEG. 74th SEG Conference, Denver, USA. paper (PDF)
29. **Magnetic Compensation of magnetic noise related to aircraft's maneuvers in airborne survey. Extended Abstract**
2004. R.W. Groom, Ruizhong Jia and Bob Lo. 2004 SAGEEP Annual

Symposium, Colorado Springs, Colorado. paper (PDF) or presentation (PDF)

30. **Exploring for Groundwater with Electromagnetic Methods**
    2004. Ross Groom. Internal Report. presentation (PDF)
31. **On Inversion of Gradient Magnetic Data for Detection of Multiple Buried Metallic Objects. Extended Abstract**
    2004. Ruizhong Jia and R.W. Groom. 2004 SAGEEP Annual Symposium, San Antonio, Texas. paper (PDF)
32. **Inversion of Magnetic and Gradient Magnetic Data For Detection and Discrimination of Metallic Objects**
    2003. Ross Groom. EAGE, 9th European Meeting, Prague, Czech Republic. presentation (PDF)
33. **Investigations into Inversion of Magnetic and Gradient Magnetic Data for Detection and Discrimination of Metallic Objects. Extended Abstract**
    2003. R.W. Groom, Ruizhong Jia and C. Alvarez. 2003 SAGEEP Annual Symposium, San Antonio, Texas. paper (PDF)
34. **UXO Applications for Geophysics**
    2002. PetRos EiKon. Internal Report. presentation (PDF)
35. **3D EM Modelling - Application of the Localized Non-Linear Approximator to Near Surface Applications. Extended Abstract**
    2002. R. Groom and C. Alvarez. 2002 SAGEEP Annual Symposium, Las Vegas, Nevada. paper (PDF)
36. **Some Pitfalls in Gradient Magnetic Processing or Why Rotate Gradients**
    2002. R.W. Groom and Bob Lo. KEGS Symposium, Toronto, Canada. presentation (PDF)
37. **Improved Depth Distributions by Inversion of Magnetic Surveys Collected at Different Survey Heights**
    2002. Bob Lo and R.W. Groom. KEGS Symposium, Toronto, Canada. presentation (PDF)
38. **An Inverse Magnetic Problem**
    2002. Ruizhong Jia. Internal Report. paper (PDF)
39. **3D Modelling of Near Surface Problems**
    2002. PetRos EiKon. Internal Report. presentation (PDF)
40. **Development in a normal mode helical electrical antennae crosshole instrumentation and integrated interpretation system.**

**Extended Abstract**
2001. R.W. Groom and C. Candy. 2001 SAGEEP Symposium, Denver, Colorado. paper (PDF) or presentation (PDF)

41. **X-hole Tomography. A New Frontier in Equipment and Software**
2001. R.W. Groom and C. Candy. KEGS Symposium, Toronto, Canada. presentation (PDF)
42. **Mag 3D Inversion**
2000. PetRos EiKon. Internal Report. presentation (PDF)
43. **An Interpretation Study of Pulse EM Reconnaissance Data from the Raglan Belt of Northern Canada, Extended Abstracts**
1999. Groom, R.W., Murray, I.R., Alvarez, C. and North, J. 3D Symposium, Salt Lake City, Utah. paper (PDF)
44. **Modelling of complex electromagnetic targets using advanced non-linear approximator techniques, Extended Abstracts**
1999. Murray, I. R., Alvarez, C. and Groom, R.W. 69th SEG Conference, Houston, Texas. paper (PDF)
45. **On Extending the Localized Non-Linear Approximation to Inductive Modes Extended Abstracts**
1997. Murray, I.R. F004, 59th EAGE Conference, Geneva, Switzerland. paper (PDF)
46. **A case study on the application of the EMIGMA modelling package to Crone data over the SOQUEM Lac Volant Region, Sept-Iles, Quebec, Extended Abstracts**
1996. Parker, D.H, Boivin, M., Murray, I. R and Groom, R.W. 67th SEG Conference, Dallas, Texas. paper (PDF)
47. **A case study of the application of the EMIGMA modelling package in interpretation of UTEM data over the Cominco Cerattepe deposit in Turkey, Extended Abstracts**
1996. Groom, R.W., Hyde, C.H. and Lajoie, J. 66th SEG Conference, Denver, Colorado. paper (PDF)
48. **Beyond the Born and Rytov Approximations: A Nonlinear Approach to Electromagnetic Scattering**
1993. Habashy T. M, Groom, R.W. and Spies. B.R. J. Geophys Res., 98, no. B2, p. 1759-1775. paper (PDF)

# License Registration/Deregistration

These instructions only apply to serial number EMIGMA licenses.

**To Register an EMIGMA license** Registration can be done by selecting the **Register EMIGMA** option from the EMIGMA folder on the Windows Start menu. Then, enter your serial number. You must have administrative privileges to do this task.

**To Deregister an EMIGMA license**
Should you wish to move a registration from one computer to another, simply select the **Deregister EMIGMA** option from the EMIGMA folder on the Windows Start menu. Then you may install and register on another computer.

## Topography file format

There are two places to import your topography in EMIGMA using an XYZ format.
1. **Model Configuration**. The user may create a polyhedral model of the topography.
2. Profile tab of Data set **Configuration GUI**. The user may adjust the height of the receiver locations to correspond to the topography.

In both cases, the functions are the same and the required file format is also the same.

The XYZ file should contain elevations along a series of lines, which may be oriented in any direction. This file may be exported from EMIGMA or QCTool

**Format specifics**

1. The first column should be the x co-ordinate.

2. The second column is the y co-ordinate.

3. The third column is the digital elevation value.

4. There must be a label at the beginning of each new line in your DEM XYZ file such as "Line 1".

### Save layer model

The current layer model can be saved to a text file with columns for layer number, susceptibility, resistivity, density, thickness, permittivity, and the three cole-cole parameters (c, m, and tau).

In the **Layers** tab:

- Click **Save Model** in the **Edit Mode** section to open the standard Windows-style **Save** dialog. The default folder will be the Models folder of the current database. The default filename is a combination of the project name and data set name.

**Join layers**

In the **Layers** tab:

- 
- Select two or more adjacent layers in the layer list box then click the **Join Layers** button directly beneath the list box. The selected layers will be replaced by a new layer with an average of the selected layer properties and the sum of the layer thicknesses

**Split a layer**

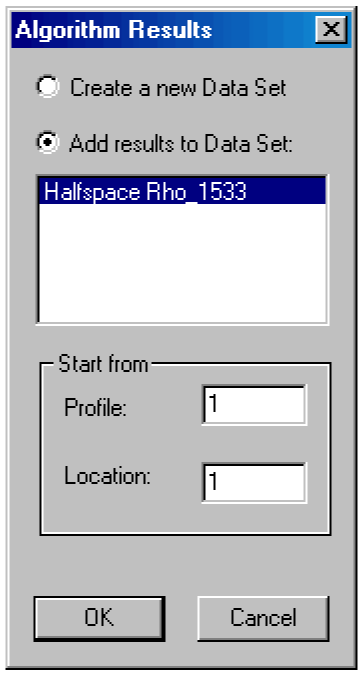
In the **Layers** tab:

- Select a single layer in the layer list box, click **Split Layer** immediately below the list box and the selected layer will be replaced with two layers that have the same properties of the selected layer and half the thickness.

## Calculate both apparent resistivity and depth (CDI)

In the Select Algorithm dialog, choose **CDI (apparent resistivity + apparent depth)** and click **Select**.

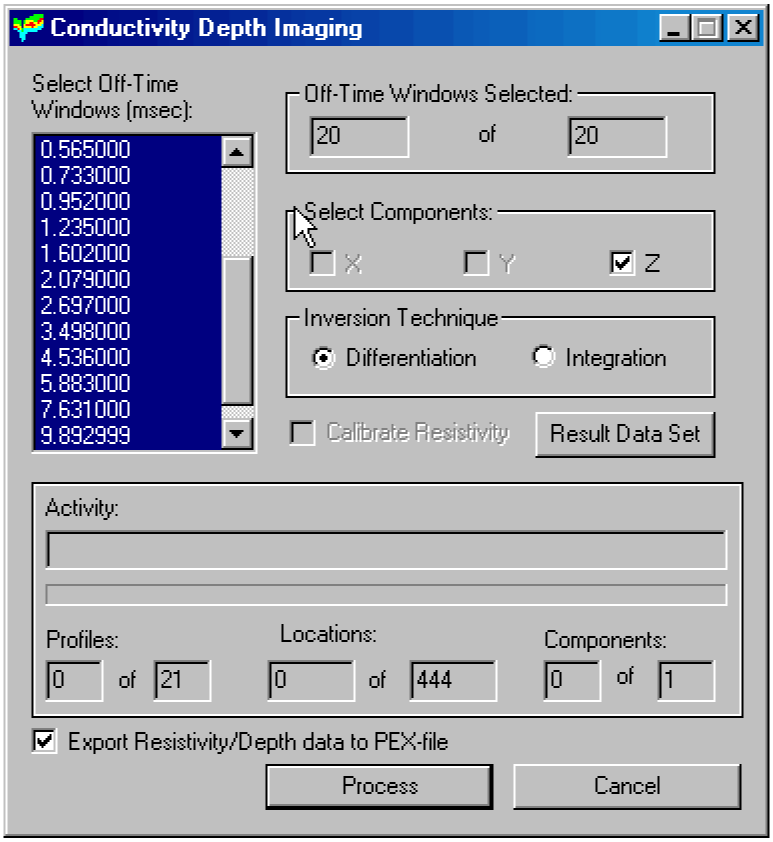
Surveys may have data sets that have been generated by the CDI tool. If you selected a data set belonging to one of these surveys, you will be presented with the **Algorithm Results** window:




You are given the opportunity with this interface to add to the previous results in a data set.

- Select **Create a new Data Set** to continue without using any previous results.
- Select **Add results to Data Set** to continue where previous processing was interrupted or to modify the existing results.
- Next, select a data set from the list on this window.
- The **Start from** section allows you to specify from what location the processing should begin. Any existing results after and including the specified location will be overwritten.
- Click **OK**

In the window to appear:




- Specify the components you would like to use by selecting the appropriate checkboxes in the **Select Components** section.
- Specify the time windows to be used in the processing by selecting the desired windows from the **Select Time Windows** list.
- Specify the **Inversion Technique**
- Click the **Result Data Set** button to display the **Algorithm Results** window and change the data set the results will be saved to.
- Select the option labelled **Export Resistivity/Depth data to PEX-file** to save the processing results to a default PEX file automatically once complete. If you want to choose only some of the processing results for a PEX file then deselect this option and once the processing is complete, you will be able to select the data that will be exported to the PEX file.
- Click the **Process** button.
- Next, if you chose not to export a default PEX file, a window will appear prompting you to perform the export. See Exporting to a PEX

file for more details on how to export.

When finished, you will find a new CDI_ data set on the Database tab of EMIGMA's main interface.

There will be a checkmark on the **Model** button for this data set to indicate a default PEX file was automatically saved. The contents of the PEX file can be viewed by clicking the **Model** button. See View Resistivity & Susceptibility Grid Data Files for more details.

To view the results graphically, select this data set and click the **CDI Viewer** button  on EMIGMA's main toolbar (for further details see **CDI Viewer**).

A separate PEX_ data set will be generated if the PEX file was exported using the Export to PEX-file interface. This data set may also be viewed using the CDI Viewer tool.

## Display profiles

In the right-hand panel of the GridPresentation dialog:

- Click the **Show or Hide Profiles** button to toggle profiles on and off. You can also do it from the **GridPresentation** toolbar. When engaged, this button enables the respective section of the dialog

  In the **Show or Hide Profiles** section:

- Select between the **Grid Data** and **Profile Data** options.

  If you leave the **Grid Data** button on and click in a certain point of a profile, the x and y coordinates to appear will refer to the grid cell. With the **Profile Data** button on, the x and y coordinates will refer to the profile location

To specify the same style and color for all of the profiles:

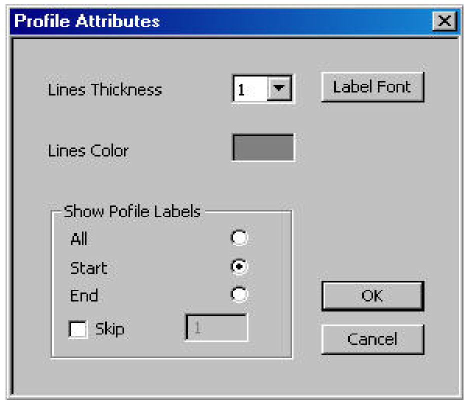
- Click the **Advanced** button and select the line thickness and color in the **Profile Attributes** dialog to appear (see below)

To display your profiles stacked, with different colors assigned to each location dependently of the data value in it:

- Check the **Profile Data Symbols** box and select the size of symbols from the dropdown list to the right. To better view the profiles and data distribution, click the **Hide Grid Mesh** button in the **Draw Mode** section to switch your grid off and leave only profiles on the screen

To display profile labels:

- Check the **Labels** box. Click the **Advanced** button to specify the format, density and location of the labels in the **Profile Attributes** dialog to open:




The **All** option will show all labels; the **Skip** option will show each second label. With the **Start** or the **End** option on, the profile labels will appear respectively at the beginning or the end of the profiles.

Previous/Next

# PEGeoMap

## Using PEGeoMap

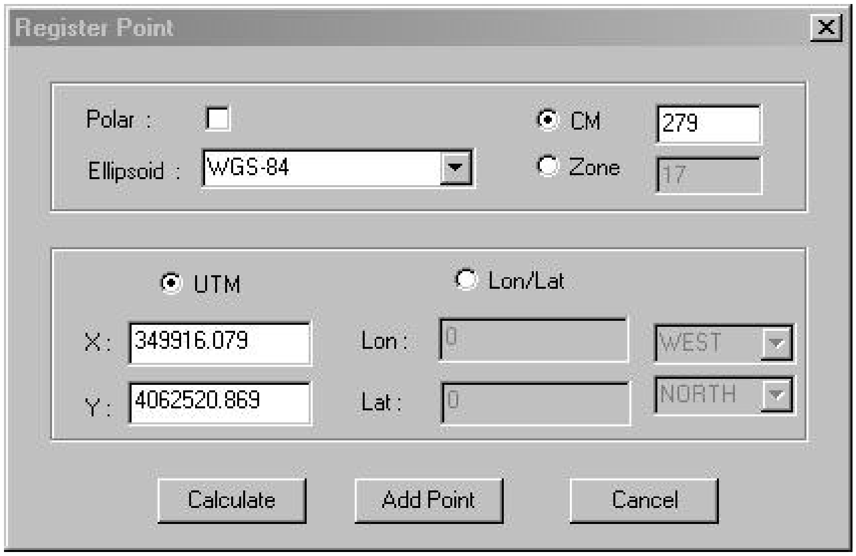
Pegeomap is a tool for working with georeferenced raster images. It can either use existing georeference information compatible with various other mapping software, or create a new information by the "Generate a Map" procedure [see below]. The georeferenced image can be then imported as underlying map into QCTool grid or EMIGMA GridPresentation.

PEGeoMap comes with the QCTool software package or can be downloaded on its own from the QCTool website

**To open georeferenced map:** Select in menu File/Open geomap/ select georeference file: .jgw, jpg, tfw, map, geotill, kml or tiff.
Or, it can be any kind of an image without georeference: bmp, jpg, png, etc.; in this case, use File/Open Raster menu item. Also, you can drag and drop any such file from Windows Explorer into the application window.
If georeferencing was successfully discovered, status bar displays valid X,Y coordinates; otherwise they are 0.

**To Generate a Map (create georeference for raster image):**

- A raster format file (gif, jpg, etc.) needs to be loaded. If a file is not already loaded you will be given the opportunity to load the last used file. If you would like a different one, click **No** and a window will appear offering you to select a file. You can also load a raster file from the toolbar using the  button. You can load a map file using the  button.
- A raster file needs to be calibrated with at least three points.
- Double-click on the first point in the image. The following window appears:




- Check the **Polar** box if your map comes from a polar region
- Select the required ellipsoid datum from the respective dropdown list
- Specify the central meridian(CM) and zone in the respective boxes
- Select between **UTM** and **Lon/Lat**. The appropriate boxes below will become active.

- Type in the coordinates of the point to add and click **Add Point**. Repeat this operation for the rest of the points.
- To convert your UTM into Lat/Lon, select **UTM**, click **Calculate** and the conversion appears in the Lat/Lon area. Select **Lat/Lon** and **Calculate** to convert the other way. Lat/Lon will be recalculated to UTM since raster transformation is based on UTM coordinates.
- Once at least three points have been entered, you may save your map(see below) and the Map Tools will be enabled.

**Managing the registered points:**

- Click the  button. The **Edit Registered Points** window will open. Select the cell you want to edit and type in a new value.
- Click the  button to remove all registered points.

- Use the button to apply a shift to the x and y values of the map. The effect of the shift can be seen by looking at the X and Y values displayed on the status bar located at the bottom of the PEGeoMap window.

**Customizing the appearance of the map:**

- To toggle the display of grid lines on the image display, click the button.
- To toggle between color and grayscale, click the button
- To add a line to your map, click the button and draw the line with the mouse. With the line selected:
    - Click the button to change the line's width.
    - Click the button to change the line's style.
    - Click the button to change the line's colour.
    - Drag the boxes at the ends of the line to modify the start and end positions.
    - Use the [Map Tools] to add a line at specific coordinates.
- To add a symbol to your map, click the button and drag the mouse to specify the desired dimensions. With the symbol selected:
    - Click the button to change the symbol's style.
    - Click the button to change the symbol's colour.
    - Drag one of the resize controls located around the symbol to change its dimensions.
    - Use the [Map Tools] to add a symbol at specific coordinates.
- To add an annotation to your map, click the button. In the **Annotations** window to open, write your text. Use the Font section at the right end of the toolbar to change the appearance of the text. Click on the map at point where you want the annotation to appear and drag the mouse to specify its dimensions.
- To cancel any of the previous operations, click the **Undo** button
- Any items that have been added to the map can be repositioned by dragging with the mouse.

- A popup menu is also available by right clicking on the selected item :
  - **Delete** - Deletes the item.
  - **Edit** - Displays the **Annotation** editing window.
  - **Paste** - Creates a duplicate of the selected item.
  - **Properties** - Display the window to edit the item's properties.

**Viewing the map:**

- To zoom in on your map, click the **Zoom In** button on the toolbar. Then, click on the image to the desired zoom level.
- To zoom out, click the **Zoom Out** button. Then, click on the image until the desired zoom level is reached.
- To zoom to a percentage of the actual size, click the down arrow 75% and select a new zoom value.
- To zoom to 100% of the actual image size, click the button.
- To move a different viewing area, click the button and drag the image with the mouse.
- To measure a distance, click the button and drag the mouse between the two end points of the distance you would like measure. The value of the distance will appear on the right end of the status bar at the bottom of the PEGeoMap window. Three points need to be registered in order to use this feature.

**Saving your work:**

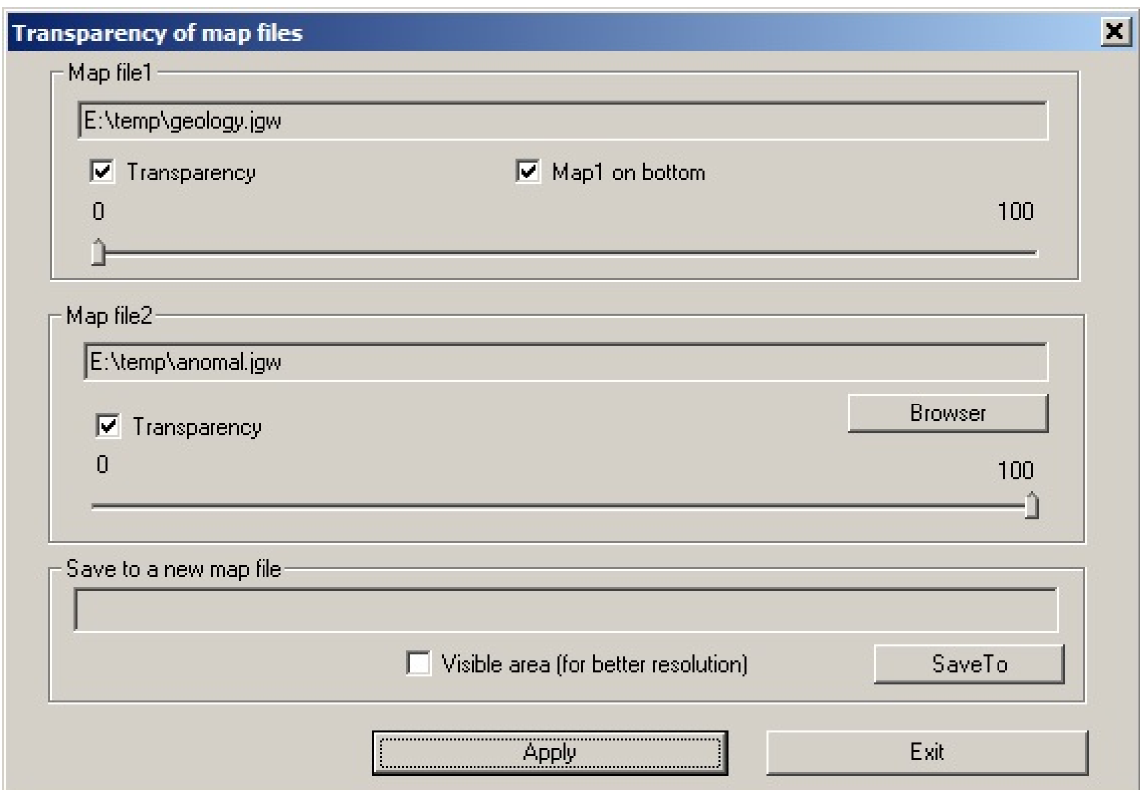
- Save using the button. You may choose to save an image file and/or a *.map file. The *.map file will be available to other EMIGMA tools such as Grid Presentation and Survey Editor which can use the created raster image as an underlay.
- Click the button to quickly save an image file only. Various file types are supported.
- To save a section of the image to a file, click the button. Click and drag the mouse on the image to create a selection box containing the area you would like to save. Then, click the button to save the file.
- To customize the layout of a printed hardcopy, click the button. The Print layout window will appear and then you will be able to edit the

appearance of your printout.

**PEGeomap image layering:**

PEGeoMap can display 2 layers with selected transparency.

- After loading your first map layer, select menu item **"MapTools/Transparency".** "Transparency of map files" dialog appeares.
  Select 2nd layer map by clicking button **"Map file2 - Browser"**.
  Both layers must be georeferenced in the same coordinate system.




Set required transparency so you can see their mutual position.

**For successful saving of the layered image:**

- Both layered images must have same raster type: bmp, jpg, etc. (and correspondingly, bpw, jgw... world files if chosen format is ESRI).

- For the 2nd layer, choose file with a larger white background (i.e. put map on 1st, drawing on 2nd layer).
  NOTE: White background of 2nd layer in saved raster will be transparent (which is not reflected on the screen image).

- The Result image can be characterized by 2 resolutions: digital (resolution of jpg file, which in reality defines jpg compression algorithm), and geographic - number of pixels for the same distance on the ground. Both resolutions and area of saved image are equal to 1st layer's resolutions and area. Generally, select for 1st layer, an image with higher resolution(s). The Result image might have insufficient quality if layers have very different geographic resolution. In this case, try to zoom in to the area of interest, and click the "Visible area" checkbox in "Transparency of map files" dialog before clicking the "SaveTo" button. This way the saved image will be retrieved from the screen with good quality.

## Visualizer. Layer.

**Split Layer** Convert the selected layer into two layers, each half the thickness of the original.

Better exercised in Model Configuration

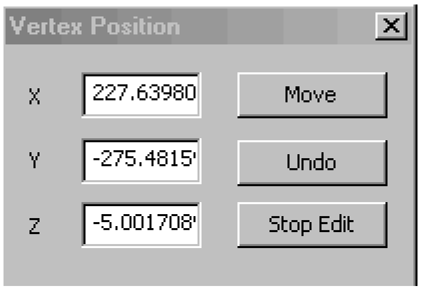
**Properties**
Activate the properties dialog for the selected layer.

## Visualizer. Polyhedron.

**Start Edit** Activate in order to drag a vertex to a new location or add a primitive.

**Stop Edit**
Saves the changes to the polyhedron.

**Vertex Position**
Select a vertex and the following interface opens:

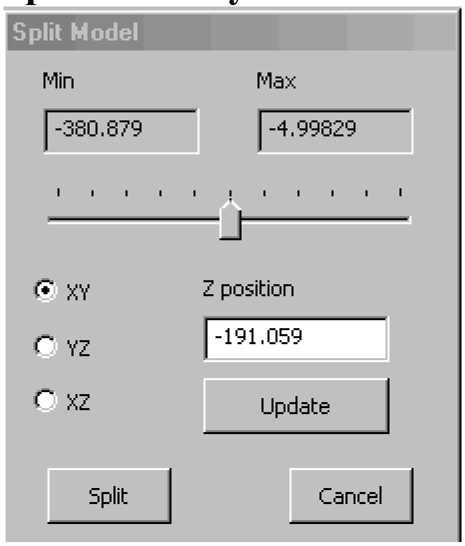



You may drag the vertex to the position desired or insert inthe the interface the new position of the vertex

**Add Primitive**
Select a face of the polyhedron after a start edit command has been issued. A four vertex tetrahedron will be added to the polyhedron. It will share the three vertices of the selected face. The fourth vertex will located at the user specified height above the centroid of the face.

**Properties**
Activate the [properties dialog] for the selected polyhedron.

**Split Anomaly**




Divide the selected polyhedron into two separate polyhedra using a desired plane, horizontal **XY** plane or vertical planes, **YZ** or **XZ**, at a user specified location.

## Visualizer. Prism.

**New Prism/Plate** Add a new body to the survey. Select the desired simulation algorithm.

**Split Prism**
Divide the selected prism into two separate prisms along a plane parallel to one of the axes at a user specified location.

**Copy Prism**
Creates a duplicate of the currently selected prism.

**Convert Prism to Poly**
Change the selected prism into a polyhedron

**Properties**
Activate the properties dialog for the selected prism.

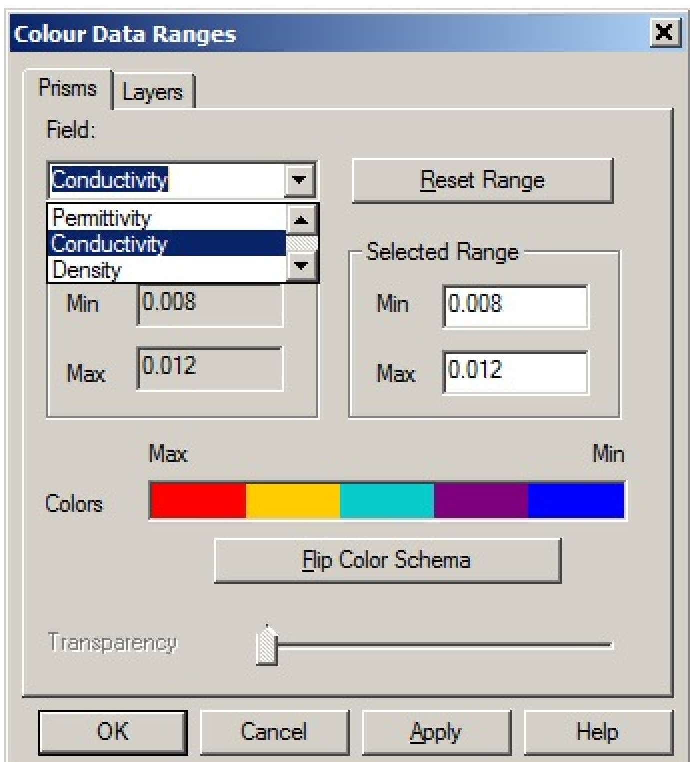
**Undo Last Split**
Remove the effects of the most recent Split Prism command.

## Visualizer. 3D Inversion model.

**Import 3D Inversion model file** Load the prisms from an existing mag file into the current survey

**Section Cutting**
Take cross sections of volume.
**Sensitivity**
Enter a value for susceptibility. Any prism whose absolute value of susceptibility is less than this will not be displayed.

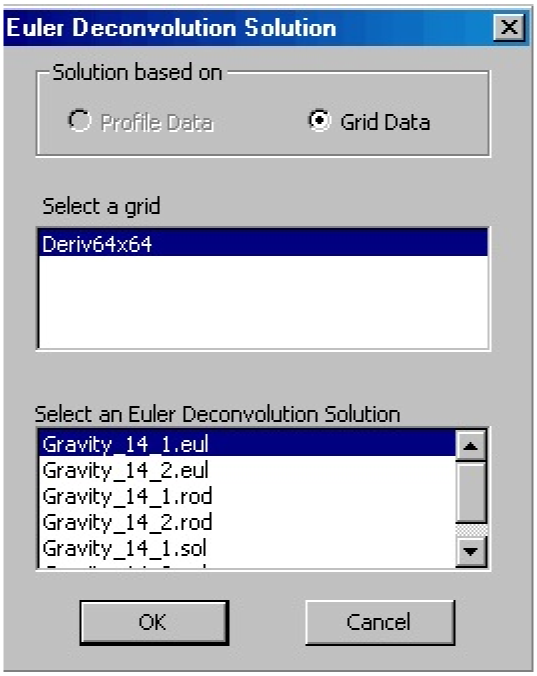
**Colour range**
Change colour range for permittivity, conductivity, density from the dialog




**Toggle On/Off**
Turn prisms defined in mag file on and off

## Visualizer. Euler Deconv.

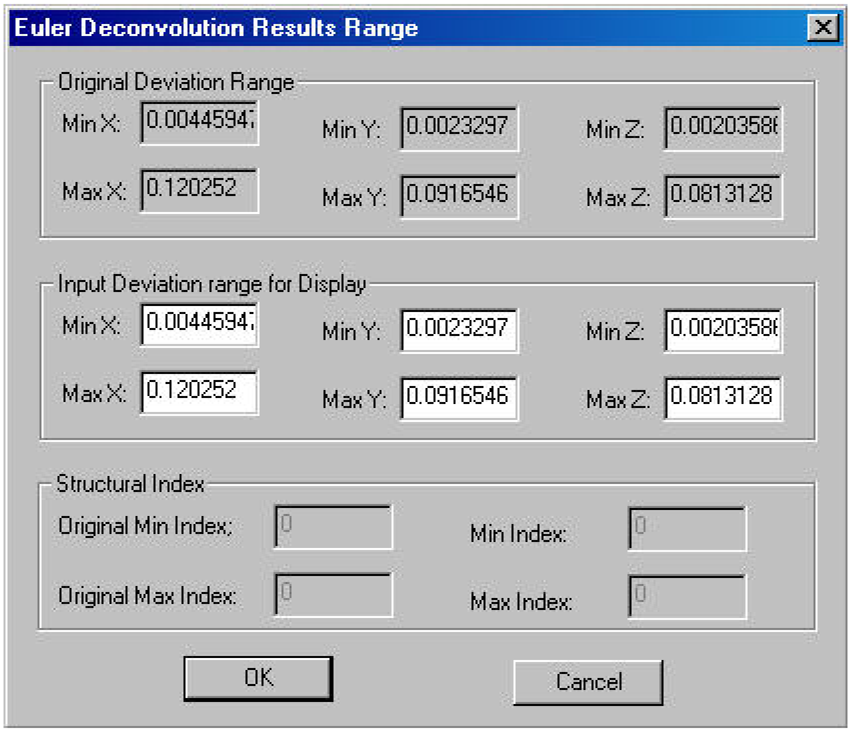
**Toggle On/Off** Toggle the display of the prisms defined in the solution file on and off.




**Select a Solution**
First, select a grid from the current data set. Then, choose an Euler, Rodin or Final solution from those available in the selected grid.

**Display Type**
Choose between Structural Index and Deviation. The colours of the prisms will reflect the data values of display type that was chosen.

**Set Display Range**
The **Euler Deconvolution Results Range** window appears. Depending on which **Display Type** is selected, the minimum and maximum values for the Structural Index or Deviation are displayed. New minimum and maximum values may be chosen to emphasize a certain range of values which will be represented by different prism colours.


Euler Deconvolution Results Range
Original Deviation Range
Min X: 0.0044594
Min Y: 0.0023297
Min Z: 0.0020358
Max X: 0.120252
Max Y: 0.0916546
Max Z: 0.0813128
Input Deviation range for Display
Min X: 0.0044594
Min Y: 0.0023297
Min Z: 0.0020358
Max X: 0.120252
Max Y: 0.0916546
Max Z: 0.0813128
Structural Index
Original Min Index: 0
Min Index: 0
Original Max Index: 0
Max Index: 0
OK
Cancel

## Color Editors.

The color editors are used for editing a number of different properties, including the background color, light colors, and various material colors. They all have this same general appearance. This one, for example, is used for setting a material's diffuse color.

You can drag the marker in the color wheel to select a color. You move the V (value) slider to control the intensity of the color. The color boxes display the currently selected color and the previous color. You can use the arrow buttons to switch back and forth between them. By default, the scene is automatically updated as you modify the color.

**Edit Menu**
The color editor's Copy menu item allows you to copy the current color to the Windows clipboard. The Paste menu item applies that color to the currently selected object. The Paste menu item will also apply the color component from a material that was copied to the clipboard by the material editor.

**Sliders Menu**
The color editor can display sliders that allow you to directly manipulate the components of an RGB or HSV color.

## Visualizer. Layer Properties

**Susceptibility** See EMIGMA full manual for more information.

**Resistivity**
Inverse of conductivity.

**Thickness**
Thickness of selected layer. Top air layer and bottom halfspace layer have thicknesses of 1.0e8.

## Visualizer. Model Properties

**Dimensions and Geological Angles** The primed variables denote the axis system on the body and the unprimed coordinates the absolute system. The body's axis system is oriented such that the x' axis points towards the strike of the body and the y' axis is parallel to its dip extent. For a body with a 90 degree strike angle and 0 degree dip angle, the unprimed axis system is parallel to the primed axis system. The origin of the body's axis system is the point defined as center/top.

The dimension parallel to the x' axis corresponds to the strike extent and that parallel to the y' axis is the dip extent. A positive strike angle rotates the body's x' clockwise about the z' axis looking towards the origin along the positive z' axis, so that with a 0 degree strike for example, the x' axis of the body will be aligned with the absolute y axis. A positive dip angle pushes the body's y' axis downwards rotating around the x' axis, so that a body with a 90 degree dip will have its y' axis parallel to the negative z axis in the absolute system. After the first two rotations, a positive plunge angle will rotate the body clockwise about its z' axis looking towards the origin along the negative z'. The thickness is the dimension of the body which is not the dip extent or strike extent.

**Center/Top**
Click center to display center coordinates of selected body. Click on top to display the coordinates of the uppermost point of the selected body.

**Number of Sample Pts**
VH Thin sheet sample points are fixed at 441. FSPlate has maximum order 9. A prism can have up to 1000 sample points. See Sampling Fields in Prisms for more information.

**Name**
Label associated with selected body.

**Undo**
Undo last change.

**Color**
Change color of body

**Material Properties**
**Conductivity**
See Electrical Conductivity for more information

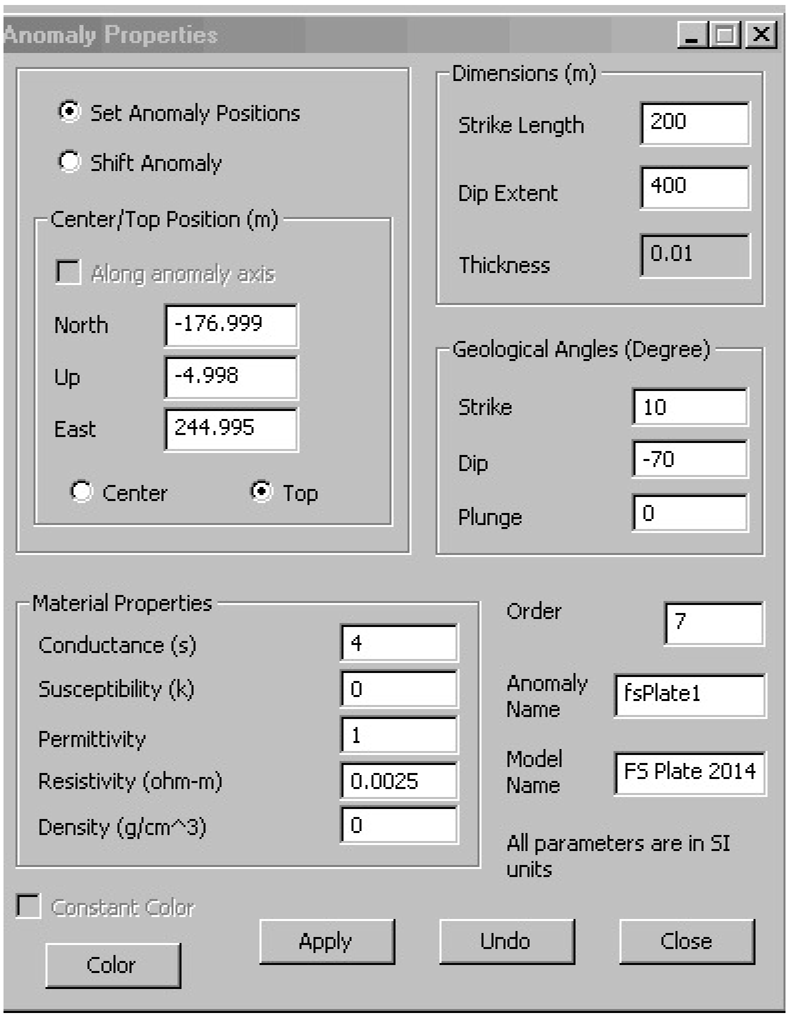
Susceptibility
Equal to permeability minus one. See Relative Magnetic Permeability for more information.

**Permittivity**
See Relative Electric Permittivity for more information

**Resistivity**
Inverse of conductivity.

# Sampling Fields in Prisms.

**When selecting properties for an object in your model, then this interface will appear:**




**There are anumber of different model simulation algorithms but there are 4 different primitives.**

**Rectangular Prisms**

**Thin-Sheet Plates (not available for all survey types)**

**Multi-faced Polyhedra**

For the prisms, 2 algorithms may be selected LN and ILN and the internal sampling is defined via the number of sample points per dimension of the prism. Only the total number is shown in the property page but in Model Conigurationi, one may select each dimension's sampling as well as target interactions.
This primitive is also used with either a Born or LN technique for magnetics and gravity

There are 2 thin-sheet algorithms: FSPlate (freespace plate) when the eigencurrent (eigenvalue) order may be selected on the VHPlate ( 2 potential ) which has a fixed number of samples (441)

The polyhedra may utilize 2 algorithms as in the prisms and the total number of samples is defined. The software determines the best distribution of sample points.

More details are contained in the main EMIGMA manual but a few points are covered here.

THEORETICAL CONSIDERATIONS:

For plates, one has to consider the 2 principal effects of an EM anomaly are induction on the surface of the target or currents which channel through or around the target. The FSPlate computes the response of the background layers and adds to this the inductive response of the target.

VH computes both induction and current channelling. FSPlate is much faster than VHPlate and so one should consider if current channelling could be a factor.

When we consider the *LN* approximation, the spatial variation of the total internal electric fields in any prism must be considered. Internal electric field sampling requires the modeller to consider such effects as skin depth and the proximity of the conductor to the source and to the receiver. When the skin depth of the layer in which a prism sits is small compared with a dimension of the prism, the field will be strongly attenuated inside the conductor. The electric field should thus be sampled densely enough near the edges of the prism to reflect this variation. Similarly, when the prism is located close to either a transmitter or a receiver, sampling of the fields near those points becomes critical if the scattered fields are to be properly calculated.

The second issue to be considered is the sampling of the internal transfer functions, namely, how the incident primary field from the transmitter determines the scattering currents inside the conductor. These transfer functions can be broken into two parts: a part due to charges on the surface of the prism and a part due to currents flowing inside it (induction). In as far as charge effects are concerned, *EMIGMA*'s algorithms compensate well for them. However, the present version only partially compensates for the inductive component of the transfer function.

PRACTICAL CONSIDERATIONS:

When deciding if sampling is adequate, the best course of action is to test convergence. Convergence testing is typically done by recursively running the model and recording the change in the model response with the number of sample points. A correctly sampled model's response will show no change with the number of sample points. Unfortunately, this can only be detected by running models that in retrospect have had too many sample points included in it.

The LN capability to accommodate many points, yet still execute rapidly, allows convergence to be easily checked. This is a feature of numerical modelling that has often been overlooked until now, simply because computing proper convergence checks was impossible.

INTERNAL ELECTRIC FIELD SAMPLING:

Once the model parameters from the control file have been input, sampling points are located on a regular grid inside each prism. So specifying the location of each point individually is not necessary. EMIGMA assumes that a fixed number of points are needed in each prism ( 500, but this however, depends upon the operating system). The number of points used affects the total program memory requirements, and the suggested sampling in the x, y and z directions. This suggested sampling is based on an algorithm which creates the condition where the distances between the sample points in the x, y and z directions are approximately equi-dimensional subject to the condition that the total number of points is approximately the preset limit. Given this suggested sampling, it is then up to the modeller to accept it or to modify it interactively.

## PERFORMANCE HINTS

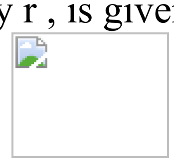
*EMIGMA* differs from conventional electric field integral equation solutions because the computation of the electric fields inside the prisms is so rapid. In many practical problems then, this part of the calculation is insignificant compared with the actual computation of the incident fields. Thus, the part of the computation that was relatively fast in other solutions, the incident field calculation, is now relatively slow. For optimal performance with *EMIGMA*, it is wise to reduce the number of points used to sample the scattered fields. As stated previously, the scattered field must be sampled densely where it exhibits rapid variation, such as near the edges of the conductor if the skin depth inside the conductor is large, or near transmitter and receiver dipoles. In such cases, it is often useful to compose a single prism out of two or more prisms, so that prisms where the field varies rapidly can have dense sampling. Prisms lying where the fields vary less rapidly can be less densely sampled.

# Electrical Conductivity.

Electrical conductivity is defined as the inverse of resistivity and describes a material's ability to transmit conduction current. There are three ways that electric current can propagate through rocks and minerals; dielectric (discussed above), electronic, and electrolytic. The conductivities of common materials may be calculated from the resistivities in Table 1.

ELECTRONIC CONDUCTIVITY:

In metals, electrons are the charge carriers, so the conductivity is said to be electronic. The electrons in a metal are not tightly bound to their atoms, so the conductivity of metals is high. For a metal rod of cross-sectional area A, length L and resistance R, the resistivity r , is given by



where the resistance is the ratio of the electrical potential across the rod to the current through the rod. The conductivity of the rod, s, is simply

$$\sigma = 1/\rho.$$

Electronic conductivity is a relatively rare phenomenon in the earth, and its occurrence is usually confined to metals and certain massive sulphide ores.

ELECTROLYTIC CONDUCTIVITY:

Electrolytic conductivity is common in the ground in comparison to electronic conductivity, and is associated with the presence of water where ions are the charge carriers. Much of the conductivity encountered in geophysics is electrolytic and because of this, is is difficult to characterize conductivities which are associated with particular lithologies since water content is such a dominating factor when determining conductivity.

Electrical resistivity can thus be quite variable depending on mobility, concentration, and the degree of dissociation of the ions. Electrolytic conduction is a slow process because the movement is actually a transfer of material and perhaps even a chemical change, and so polarization phenomena often occur at low frequencies.

The effective resistivity of a porous rock, $\rho a$, was determined empirically by Archie (1942) to be

$$\rho e=a\phi-ms-n\rho\omega$$

where $\phi$ is the porosity, s is the fraction of the pores containing water, $\rho\omega$ is the resistivity of the water, n  2, and a and m are constants (0.5 <= a <= 2.5, 1.3 <= m <= 2.5).

## ANISOTROPY OF RESISTANCE

The earth is neither uniform nor homogenous. Futhermore, it is not anisotropic. Small fractures associated with strain, bedding planes and preferential crystalline alignments due to schistosity and gneissosity all contribute to presenting paths of conduction which are preferential to current flow in one direction over another.

To understand anisotropy, consider a layered material where the layers are stacked vertically. If a current is passed through the stack from top to bottom, the system becomes a circuit of resistors in series. The equivalent resistance of such a circuit is equal to the sum of the individual resistances. To summarize,

Now consider the same stack of layers, but with current being passed through all the layers from side to side. The circuit that would represent this situation is one with the resistors in parallel. The equivalent resistance for this circuit is

Since the equivalent series resistance is not equal to the equivalent parallel resistance, seeing why the resistance in a layered material is anisotropic is easy. The *anisotropy coefficient* is the ratio of the maximum resistivity to the minimum resistivity. For graphitic slate, this coefficient may be as high as 2, but is in the range of 1 to 1.2 for limestone, shale and rhyolite. Anisotropy can be caused by many effects: preferential crystal orientation such as happens with olivine in the mantle, banding of minerals, preferential orientation of water-filled fissures and fractures, and micro-layering of sediments.

In general the anisotropy is best described by a tensor, since the coefficient of anisotropy can vary in three directions. The effect of this is that current is not necessarily parallel to the electric field as the scalar Ohm's Law would suggest.

# Relative Magnetic Permeability.

When a material is placed in a magnetic field, it may acquire a magnetization in the same direction as the field. This *induced magnetization*, or *magnetic polarization*, results from the alignment of the ions or molecules with the magnetic field. The intensity of the induced magnetization, **I**, is related to the magnetizing force, **H**, of the field as

$$\mathbf{I}=k\mathbf{H},$$

where $k$ is the magnetic susceptibility of the magnetic material.

In a vacuum, the relationship between the magnetic field **B**, and the magnetizing force **H**, is

$$\mathbf{B}=\mu o\mathbf{H},$$

where μo is the magnetic permeability of free space. Since air and water have magnetic permeabilities close to μo, the above equation represents the magnetic field of the earth in the absence of magnetic materials. When a magnetic material is introduced, the induced magnetization of the magnetic object adds to the field of the earth and the total field becomes

$$\mathbf{B}=\mu o\mathbf{H} + \mu o\mathbf{I} = \mu o(\mathbf{H}+k\mathbf{H}) = \mu o(1+k)\mathbf{H} = \mu o\ \mu R\mathbf{H}.$$

μR is the dimensionless *relative magnetic permeability* which corrects for the additional magnetic field of the region due to the magnetic material. In general, $\mathbf{B}=\mu\mathbf{H}$, where $\mu=\mu o\ \mu R$. Table 1 lists magnetic permeabilities of some common materials.

Diamagnetic and paramagnetic materials have weak magnetic susceptibility, so their relative permeability is approximately that of a vacuum. Ferromagnetic materials exhibit strong spontaneous magnetization which may exist in the absence of a magnetic field. Antiferromagnetic materials have no external magnetic field, and parasitic antiferromagnetism creates a negligible field. Ferromagnetic materials have strong spontaneous magnetization and a high susceptibility.

# Relative Electric Permitivity.

Poor conductors and insulators conduct current by dielectric conductivity. Dielectric conduction occurs when atomic electrons are slightly displaced from their nuclei in the presence of a varying electric field. This polarization may occur with ions or molecules, and is the means for dielectric conduction. The parameter of interest in dielectric conduction is the dielectric constant, *k*.

Like magnetic fields and magnetic polarization, electric fields can induce electric polarization in materials. The intensity of the electrical polarization, **P**, is related to the electric field, **E**, by

$$\mathbf{P} = \eta \mathbf{E},$$

where $\eta$ is the electric susceptibility of the material.

The electric displacement, **D**, is comparable to the magnetic field, **B**. In a vacuum, **D** is related to the electric field strength by

$$\mathbf{D} = \varepsilon o \mathbf{E},$$

where eo is the electric permittivity of free space. When a dielectric material is present, the electric displacement is altered due to the electric polarization of the material introduced. The overall electric displacement becomes

$$\mathbf{D} = \varepsilon o \mathbf{E} + \mathbf{P} = \varepsilon o \mathbf{E} + \eta \mathbf{E} = \mathbf{E}(\varepsilon o + \eta) = \varepsilon o k \mathbf{E} = \varepsilon \mathbf{E},$$

where the dielectric constant is defined as

$$k = 1 + \eta / \varepsilon o,$$

and the *relative electric permittivity* is defined as

$$\varepsilon = \varepsilon_0 k.$$

Table 1 lists some examples of relative electric permittivity.

Table 1: Typical Values of Electrical and Magnetic constants of Rocks and Minerals

| | Resistivity(Ohm.Meter) | | | | Relative Electric Permittivity | Relative Magnetic Permeability |
|---|---|---|---|---|---|---|
| *Minerals* | *Range* | | | *Average* | | |
| Graphite | 0 | - | 10 | 1000 | - | - |
| Chalcopyrite | 0.00001 | - | 0.3 | 0.004 | - | - |
| Bornite | 0.00003 | - | 0.5 | 0.003 | - | - |
| Pyrite | 0.00003 | - | 1.5 | 0.3 | - | 1.0015 |
| Pyrrhotite | 0 | - | 0.05 | 0.0001 | - | 2.55 |
| Galena | 0.00003 | - | 300 | 0.002 | 18 | - |
| Sphalerite | 1.5 | - | 10000000 | 100 | - | - |
| Bauxite | 200 | - | 6000 | - | - | - |
| Hematite | 0.0035 | - | 10000000 | - | 25 | 1.05 |
| Limonite | 1000 | - | 10000000 | - | - | - |
| Magnetite | 0.00005 | - | 5700 | - | - | 5 |
| Quartz | 4E+10 | - | 2.00E+14 | - | 4.2-5 | - |
| Hornblende | 200 | - | 1000000 | - | - | - |
| Bitum | 0.6 | - | 100000 | - | - | - |
| Coals (various) | 10 | - | 1.00E+11 | - | - | - |
| Anthracite | 0.001 | - | 200000 | - | 5.6-6.3 | - |
| Lignite | 9 | - | 200 | - | - | - |
| Meteoric Waters | 30 | - | 1000 | - | 80 | 1 |
| Surface waters: (ign. rocks) | 0.1 | - | 3000 | - | 80 | 1 |
| (sediments) | 10 | - | 100 | - | 80 | 1 |

| | | | | | | |
|---|---|---|---|---|---|---|
| Soil waters | - | | - | 100 | - | - |
| Sea water | - | | - | 0.2 | - | - |
| Saline water, 3% | - | | - | 0.15 | - | - |
| Saline water, 20% | - | | - | 0.05 | - | - |
| Igneous and Metamorphic Rocks | | | | | | |
| Granite | 300 | - | 1000000 | - | 4.8-18.9 | - |
| Diorite | 10000 | - | 100000 | - | 6 | - |
| Dacite | 2.00E+04 | | | - | 6.8-8.2 | - |
| Andesite | 45000 | - | 170 | - | - | - |
| Diabase porphory | 1000 | - | 170000 | - | - | - |
| Gabbro | 1000 | - | 1000000 | - | 8.5-40 | - |
| Basalt | 10 | - | 13000000 | - | 12 | - |
| Olivine norite | 1000 | - | 60000 | - | - | - |
| Gneiss (various) | 68000 | - | 3000000 | - | 8.5 | - |
| Quartzites (various) | 10 | - | 20000000 | - | - | - |

*From Telford et al.'s Applied Geophysics pp. 451-457*