

Table of Contents

Getting Started.....	1
First and Subsequent Use.....	1
First Use.....	1
Subsequent Use.....	2
Database Dialog: EMIGMA's Starting Point.....	4
Creating Projects From Scratch.....	5
System Configuration.....	6
Specify a gravity system.....	7
Current Dipole.....	11
Specify Separation.....	16
IP/Res Wizard.....	17
Specifying Waveform.....	18
Specify the spectral mode.....	19
Specifying Profile Information.....	20
Rename a profile.....	21
Import Profiles.....	25
Specifying Output.....	32
Saving System Configuration.....	36
Model Configuration.....	36
Specifying the Number and Properties of Layers.....	36
Replace a layer.....	38
Specifying the Number and Properties of Targets.....	40
Insert a target.....	42
Import a target.....	43
Import topography.....	44
Replace a target.....	48
Saving model configuration.....	49
Managing Project Information.....	51
Projects.....	51
Create a New Project.....	51
Change the name of a project.....	52
Delete a project.....	52
Surveys.....	52
Add a new survey.....	53
Delete a survey.....	53
Rename a survey.....	53
Write survey comments.....	53
View or change survey comments.....	53
Create a backup survey.....	54
Data Sets.....	54
Rename a data set.....	55
Rename a model.....	55
Delete a data set.....	55
Edit or add a model.....	55
View or edit survey configuration.....	55
View data set information.....	56
View, edit and export grid information.....	56
Remove non-data points from your grid.....	58
Export a grid into a Profile Data Set.....	58
Export a grid into the Geosoft format.....	59

Table of Contents

Managing Project Information

Export a grid into the XYZ format.....	60
Calculate the Difference of Two Grids.....	61

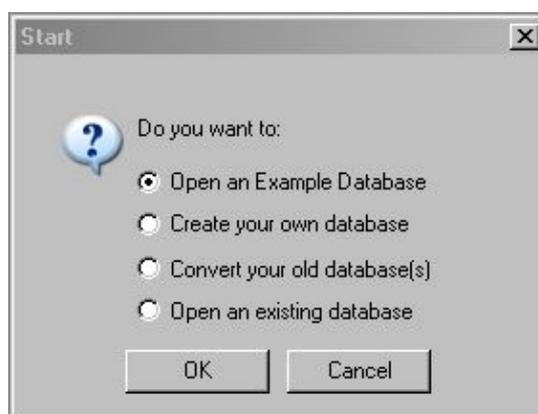
Getting Started

First and Subsequent Use

Start **EMIGMA V8.1** from its group in the Start menu. In the warning message to appear click **OK**. This message will in no way prevent you from executing the program.

First Use

If you are using EMIGMA for the first time, the **Start** dialog will appear offering you four options **Open an Example Database**, **Create your own Database**, **Convert your old database(s)** and **Open an Existing Database**:



To open an example database:

- Select **Open an Example Database**. The **Database** dialog appears containing a number of demo projects.

To create a new database:

- Select **Create your own Database**. The **Save New Database** dialog opens.
- Enter the new database name. It is a standard recommendation to create a new folder to store your file in.
- Click **Create**. The blank **Database** dialog appears, bearing the name assigned by you.

Note. If you select an already existing database, it will be overwritten to result in a blank database.

To convert old databases:

This option is designed for the cases when you use the latest EMIGMA 8.1 version to work with the databases created in earlier versions.

- Select **Convert your old databases**. The **Start Dialog** appears, with your newly converted databases displayed in the lower **Open an Existing Database** field.

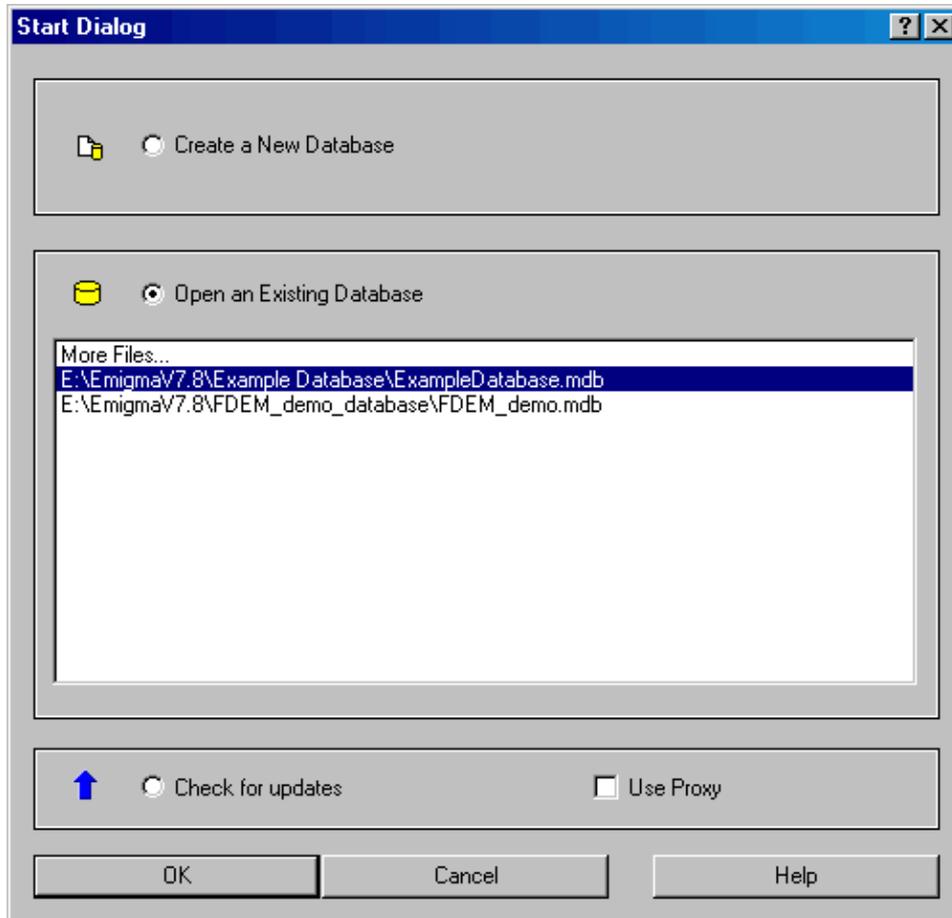
Note. The conversion is carried out automatically provided the old databases are stored in the computer you are using.

To open an existing database:

- Select **Open an existing database**. The **Open Database File** dialog appears.
- Select desired database and click **Open**

Subsequent Use

In subsequent times, you will be welcomed by the **Start** dialog:



To open an existing database:

Since you are a regular user of EMIGMA now, the **Open an Existing Database** option will be selected by default. In the field below you will see the list of the databases you created in previous times, with the one on top being highlighted. 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 required database and click **Open**

To create a new database:

The **Start Dialog** allows you to create a new database in the same manner as when you started EMIGMA for the first time.

- 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

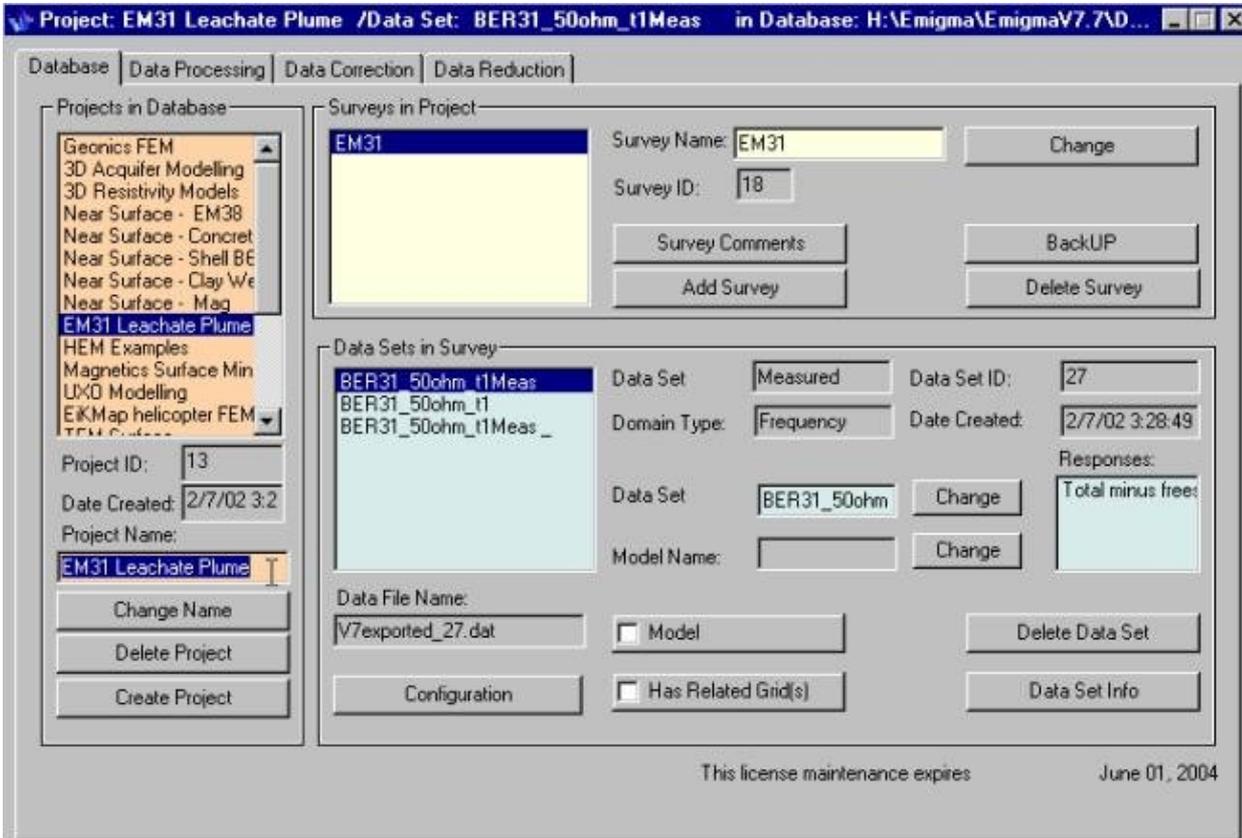
Note. If you select an already existing .mdb file, it will be overwritten to produce a blank database

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 level 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 type, 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:

[Projects](#)

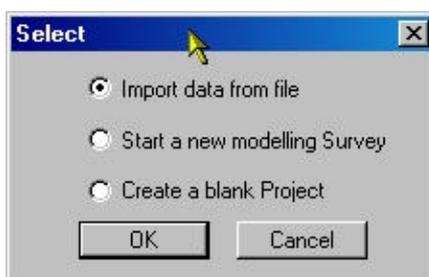
[Surveys](#)

[Data Sets](#)

[Creating Projects from Scratch](#)

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 **Property Pages** dialog to open contains four tabs allowing you to specify your system geometry, waveform, profile information and output.

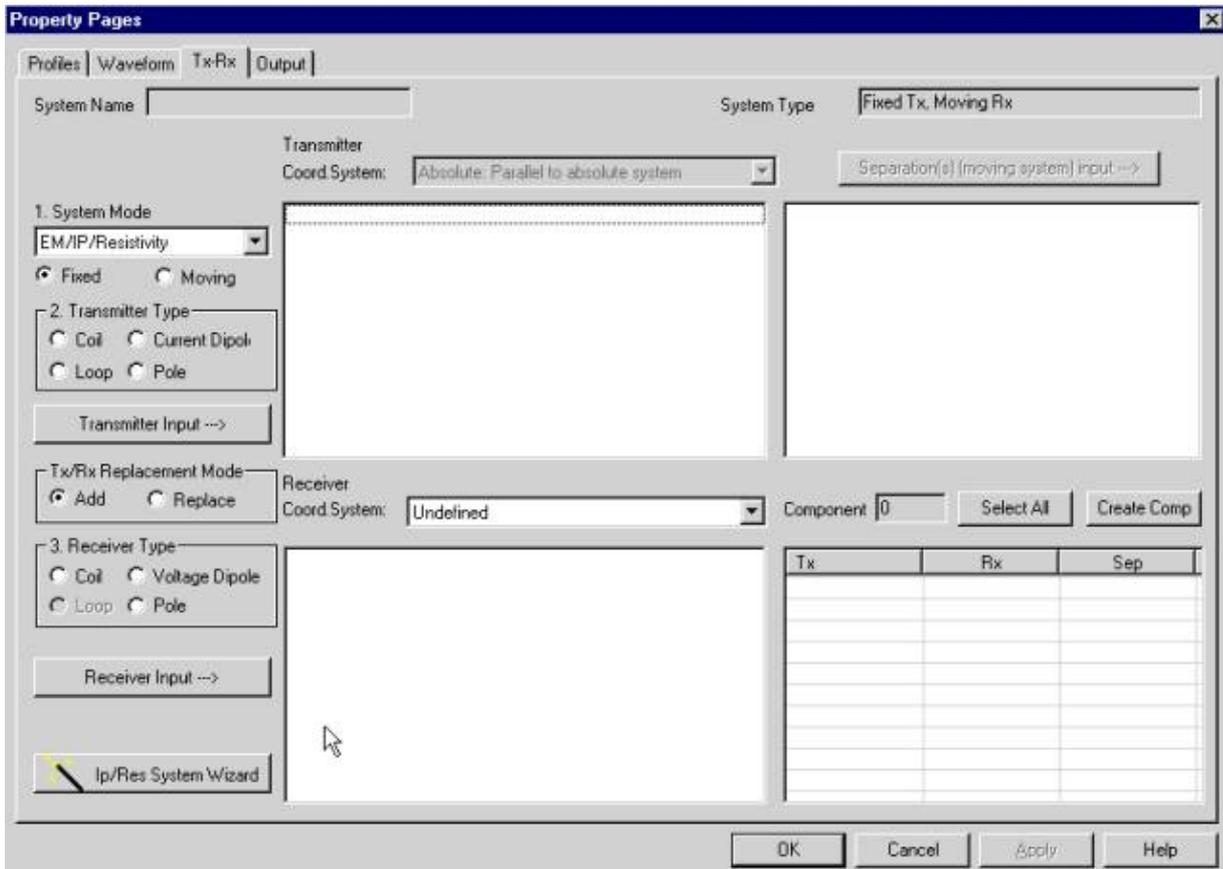
See

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	
Saving System Configuration	

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 an EM/IP/Resistivity System](#)

[Define System Components for Modeling](#)

[Specify the Coordinate 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

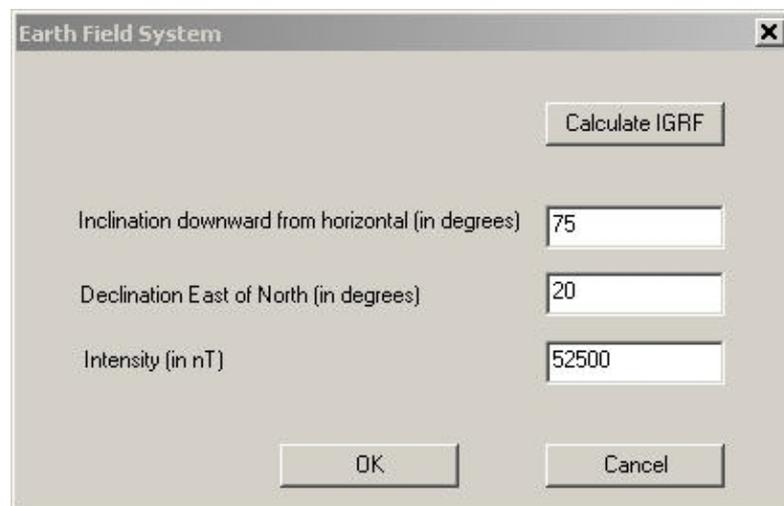


- Specify the number and orientation of the dipoles and click **OK**. The rest of the **Tx-Rx** tab will be filled out automatically
- To add a dipole, click the **Receiver Input** button to reopen the **Gravity Field System** dialog and check dipoles as needed

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 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

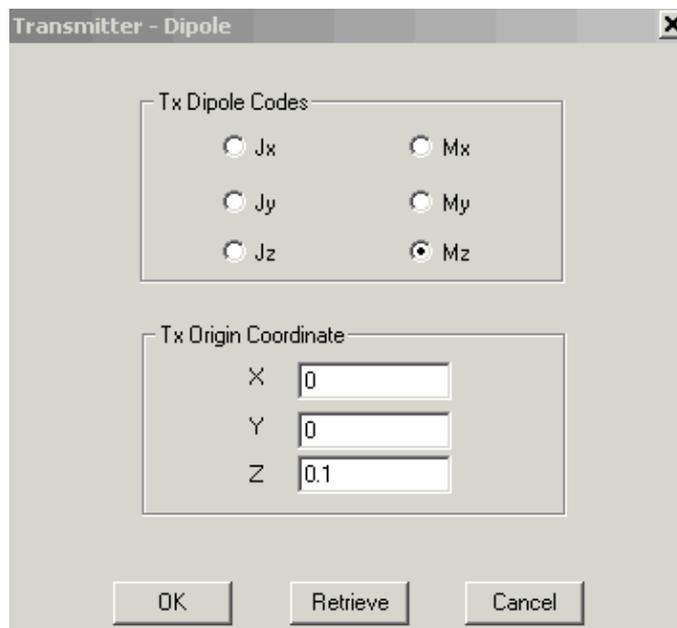
Specify Receiver Type

Specify Separation

IP/Resistivity Wizard

Specify Transmitter type: Coil Transmitter

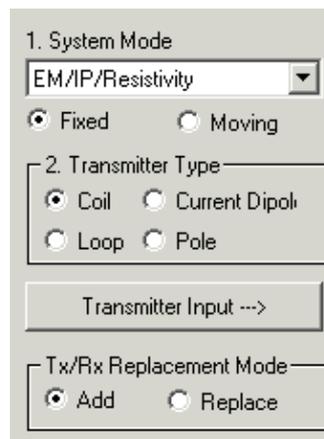
Select **Coil** in the **Transmitter Type** section of the **Tx-Rx** tab. The **Transmitter Dipole** dialog will appear:



- 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.

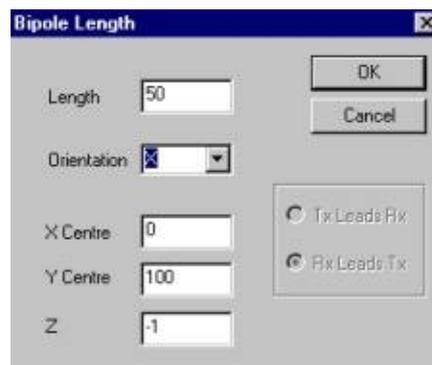
- Click **OK** to close the dialog and view your transmitter information on the **Tx-Rx** tab
- To add a new transmitter, make sure that the **Add** button is on in the **Tx/Rx Replacement Mode** section of the **Tx-Rx** tab, click the **Transmitter Input** button to reopen the **Transmitter Dipole** 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.

Current Dipole

If you select **Current Dipole**, the **Transmitter Input** button will bring up the **Bipole Length** dialog:



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

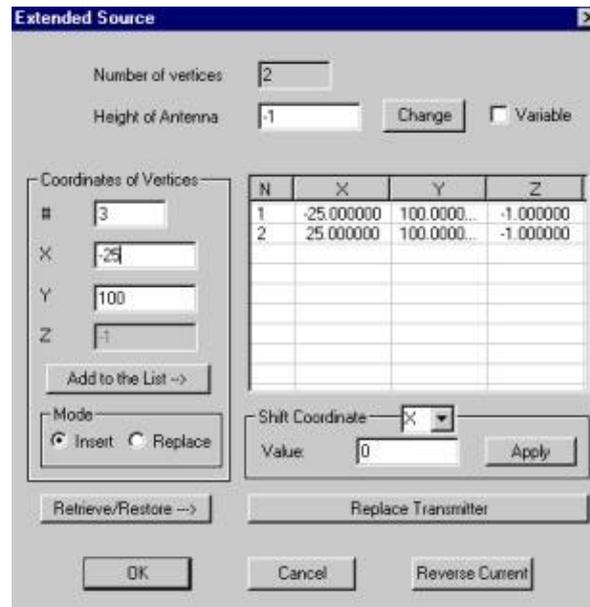
The list contains 4 items: X, Y, Z following a blank item provided for those cases when bipole orientation is to be specified each separate time

- 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:



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 input a bipole from another survey within the same project:

- Click **Replace Transmitter** to open the **Browse For** dialog
- Select the required survey and bipole 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** 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**

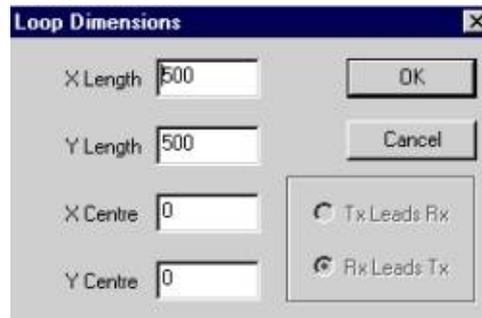
This will bring up the **Extended Source** dialog

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

Specify Transmitter Type: Loop

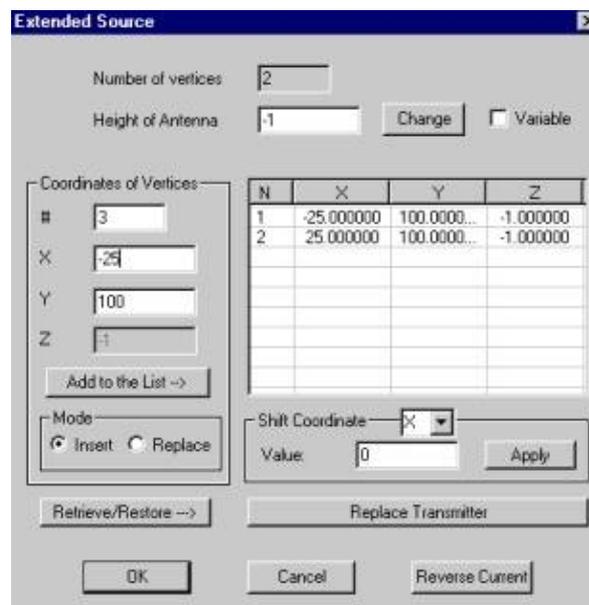
Select **Loop** in the **Transmitter Type** section of the **Tx-Rx** tab and click the **Transmitter Input** button.

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

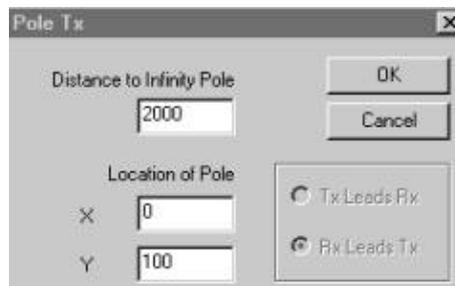
To Add or Replace Loops

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:

Extended Source

Number of vertices: 2

Height of Antenna: -0.1 [Change] Variable

Coordinates of Vertices

3

X -2000

Y 100

Z 0

Add to the List -->

N	X	Y	Z
1	0.000000	100.0000...	0.000000
2	2000.000...	100.0000...	0.000000

Mode: Insert Replace

Shift Coordinate: X [dropdown] Value: 0 [Apply]

Retrieve/Restore ---> Replace Transmitter

OK Cancel Reverse Current

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**

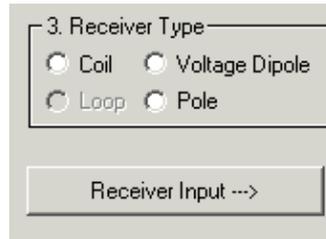
To change the current to reverse

- 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**:



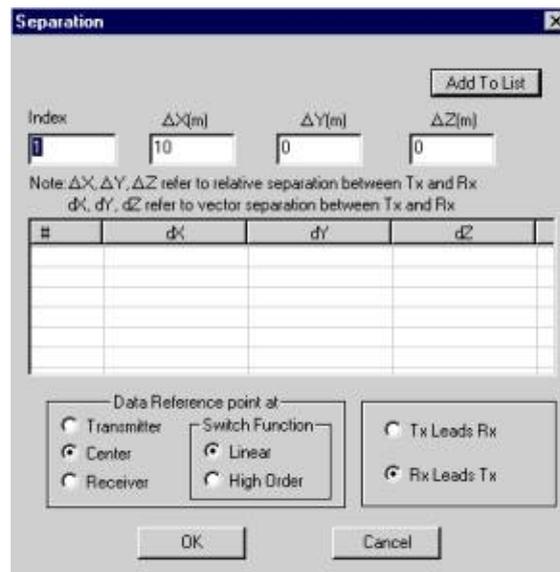
The dialog to appear depends on the receiver type you selected.

- 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**

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**:

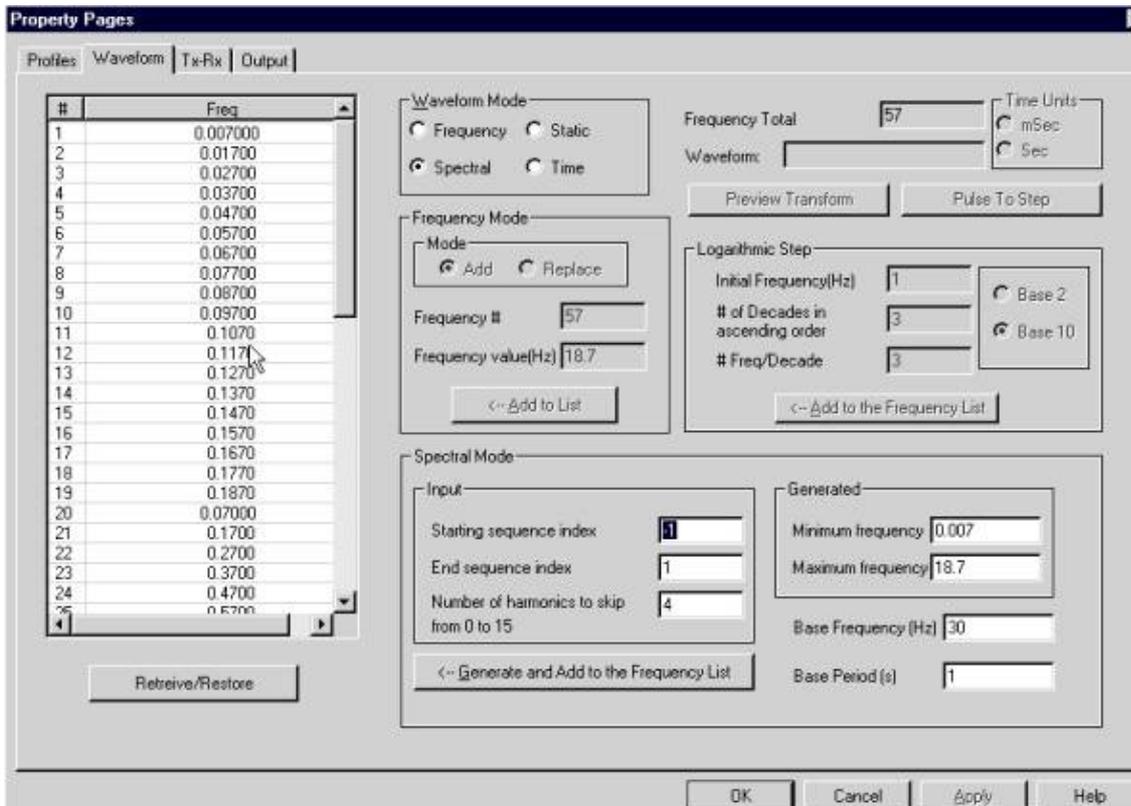
Parallel to Absolute system no matter which coordinate system is chosen for the receiver.

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.

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**

In essence, 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, to run the forward simulation and only then to transform your simulated spectral data to time-domain.

Related Topics

[Specify Frequency-Domain Mode](#)

[Specify Spectral 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 **Waveform Mode** 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** "MsoFooter" style="text-align: justify; text-indent: 0; mso-list: l29 level2 lfo38; tab-stops: list 1.05in; margin-left: in; margin-right: 0in; margin-top: 0in; margin-bottom: 6.0pt">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.

- Select **Spectral** in the **Waveform Mode** section. The **Generate Spectral Suite** dialog opens
- Specify the base frequency in this dialog and click **OK**.

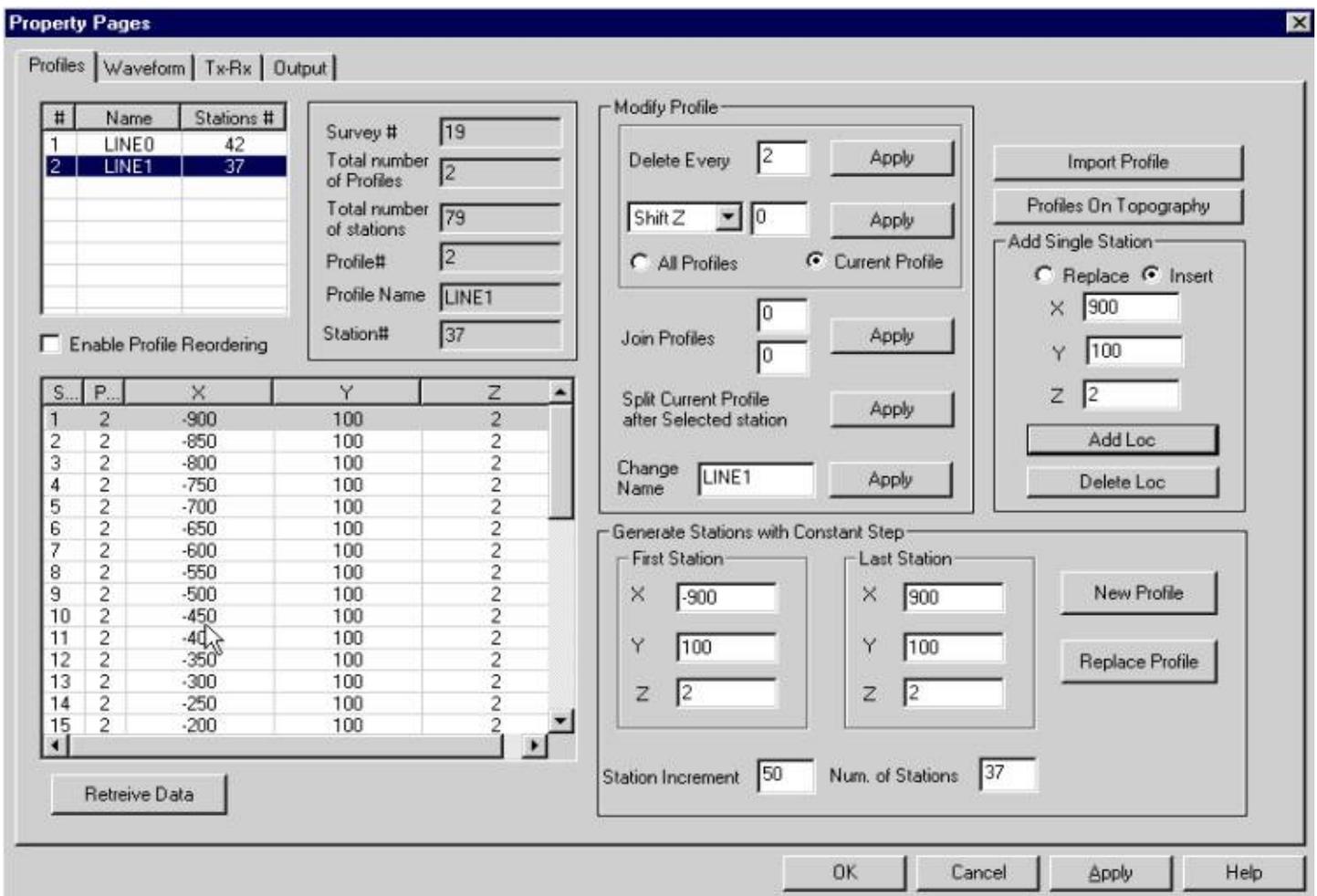
The frequencies will be added automatically to the list on the left. The **Input** section of the **Waveform** tab will display the starting and ending spectral sequence indices dependently of the base frequency you indicated. For example, if your base frequency is less than 20, you will automatically 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.

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](#)

[Reorder Profiles](#)

[Rename a Profile](#)

[Replace a Profile](#)

[Import a Profile](#)

[Import Topography](#)

[Modify a Profile](#)

Add a new profile

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

The screenshot shows a dialog box titled "Generate Stations with Constant Step". It is divided into two main sections: "First Station" and "Last Station". Each section contains three input fields for X, Y, and Z coordinates. The "First Station" fields contain 0, 1, and 1 respectively. The "Last Station" fields contain 0, 79, and 1 respectively. Below these sections are two more input fields: "Station Increment" with the value 1, and "Num. of Stations" with the value 79. To the right of the coordinate fields are two buttons: "New Profile" and "Replace Profile".

- 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

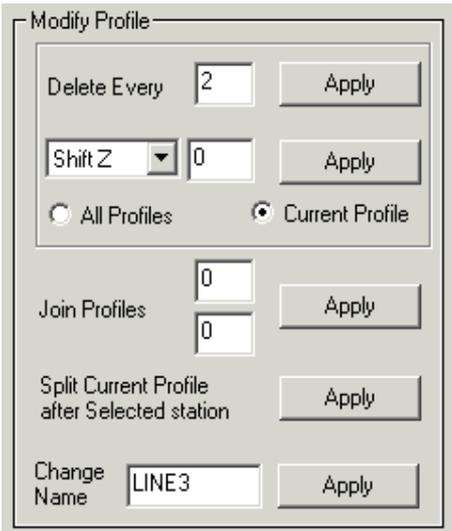
Select a profile to delete in the *List of Profiles* and press **Delete**

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

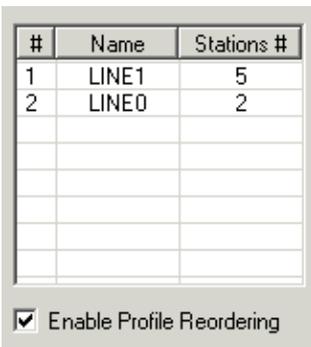
Its name will be displayed in the **Change Name** box of the **Modify Profile** section of the tab:



- 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

Reorder profiles

- Check the **Enable Profile Reordering** box under the list of profiles in the upper left-hand corner of the **Profiles** tab:



- Click and drag a profile to the position you need

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](#).

Modify Profiles

From the **Profiles** tab, you can modify your profiles in very many ways - split and merge profiles, reduce the number of stations, add, delete or replace a single station, shift profiles along axes. You can do all of these in the **Modify Profile** section of the tab:

To split a profile

- Select a profile to split from the list of profiles in the upper left-hand corner of the **Profiles** tab and click on the station you want to split at in the table of stations below
- In the **Modify Profile** section, click **Apply** to the right of the **Split Current Profile at Selected Station** command

The list of profiles will now have two profiles instead of the former one

- If necessary, rename these two profiles as described in [Rename a Profile](#)

To merge profiles

- In the **Modify Profile** section of the **Profiles** tab, type the number of the first profile you want to merge in the upper box to the right of the **Join Profiles** command and the number of the second profile in the lower
- Click **Apply**

The list of profiles will now have one profile instead of the former two

- If necessary, rename this profile as described in [Rename a Profile](#)

To reduce the number of stations

- Select the profile from the list of profiles in the upper left-hand corner of the **Profiles** tab
- In the **Modify Profile** section, set the **Delete Every** box to 2 to delete every 2nd, 3 to delete every 3rd position and so on
- Click **Apply**

In the list of profiles, one of your profiles now has reduced sampling

To shift a profile along the axes

- In the **Modify Profile** section of the **Profiles** tab, select the axis you want to shift a profile along from the **Shift** dropdown list
- Type the value you want to shift your profile by in the box to the right
- Choose the **All Profiles** option to shift data of all profiles and **Current Profile** to shift data only of the profile selected
- Click **Apply**

The coordinates in the bottom left-hand corner table of stations will change accordingly.

To add a single station

- Select the station in the bottom left-hand corner table of stations (**Profiles** tab) *before* which you want to insert a new location
- Leave the **Insert** option on in the **Add Single Station** section
- Specify the X, Y and Z coordinates of the new location
- Click **Add Loc**

To replace a single station

- Select the station you want to replace in the bottom left-hand corner table of stations on the **Profiles** tab
- Turn the **Replace** option on in the **Add Single Station** section
- Specify the X, Y, and Z coordinates of the substitute
- Click **Add Loc**

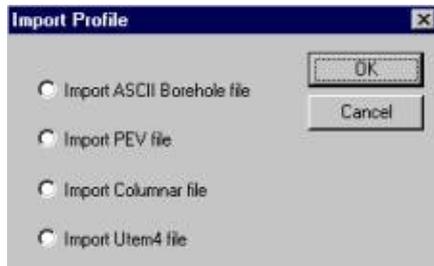
To delete a single station

- Select the station you want to delete in the bottom left-hand corner table of stations on the **Profiles** tab
- Click **Delete Loc** in the **Add Single Station** section.

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 four formats available for import:



Related Topics:

[Import ASCII Borehole Files](#)

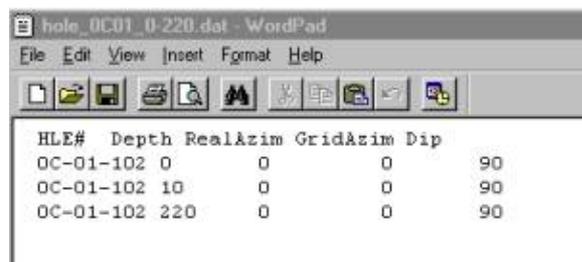
[Import PEV Files](#)

[Import Columnar Files](#)

[Import Utem4 Files](#)

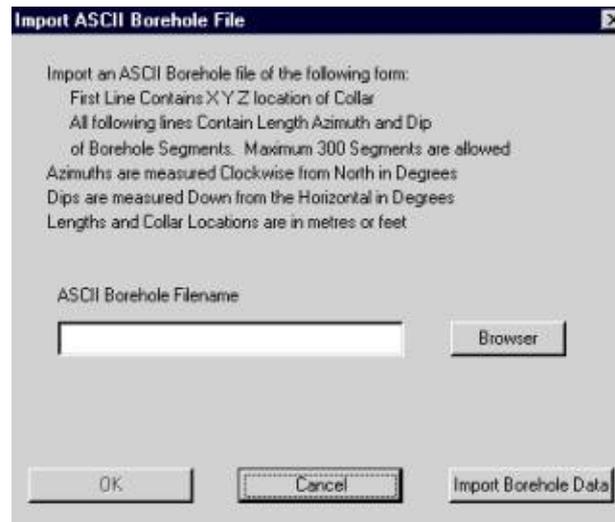
Import ASCII Borehole File

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



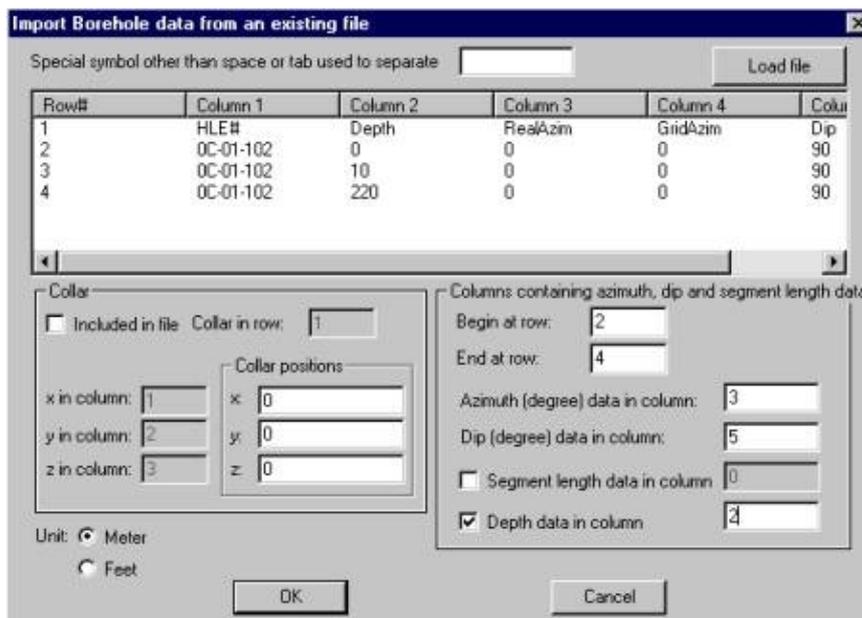
- Select **Import ASCII Borehole File** in the **Import Profile** dialog and click **OK**

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 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:

Segment	Length (m)	Azimuth	Dip	# Locations
Segment 1	10.000000	0.000000	90.000000	2
Segment 2	210.000000	0.000000	90.000000	2

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

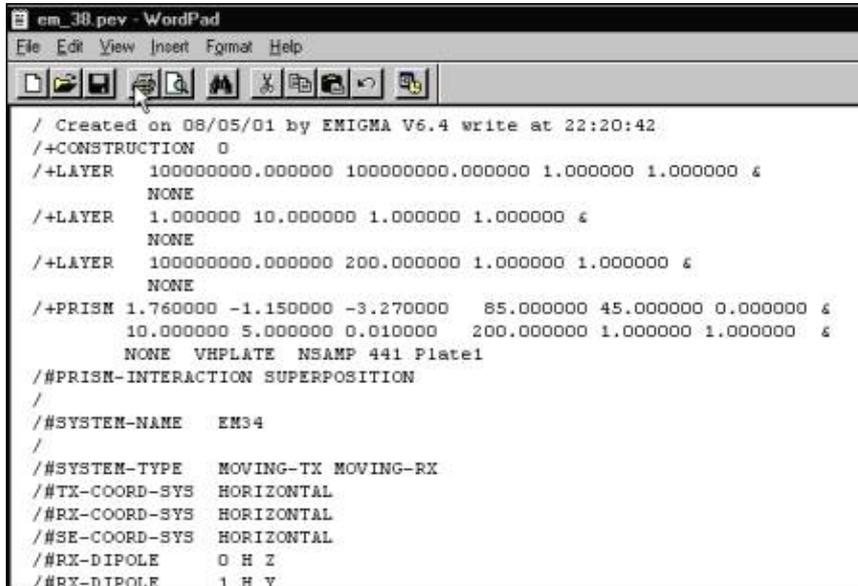
As a result all segments will have the same number of locations and step

- 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:



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

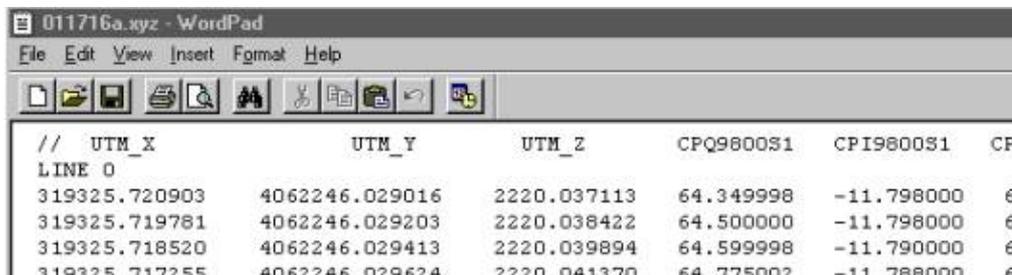
This will bring up the **Import ASCII File with Profile Data** dialog, a standard Windows-style dialog for searching and opening files

- 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.

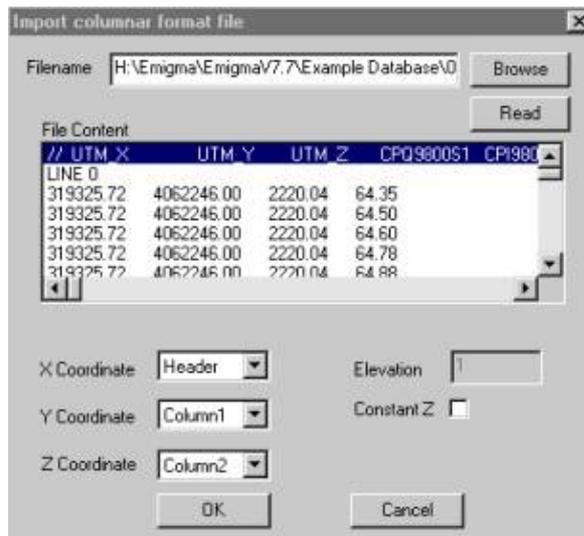
Columnar files

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:



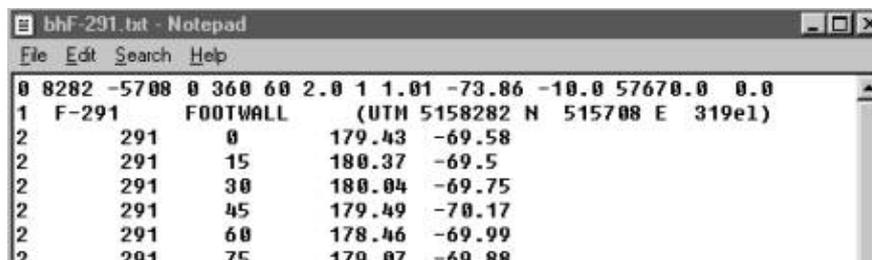
- Click **Browse** to search for a file to import. In a standard Windows-style dialog to open, select a required file and click **Open**

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.

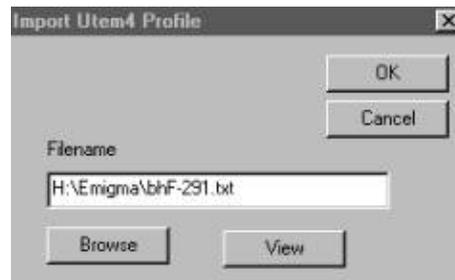
Import UTEM4 format

UTEM4 files contain borehole data in the following format:



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

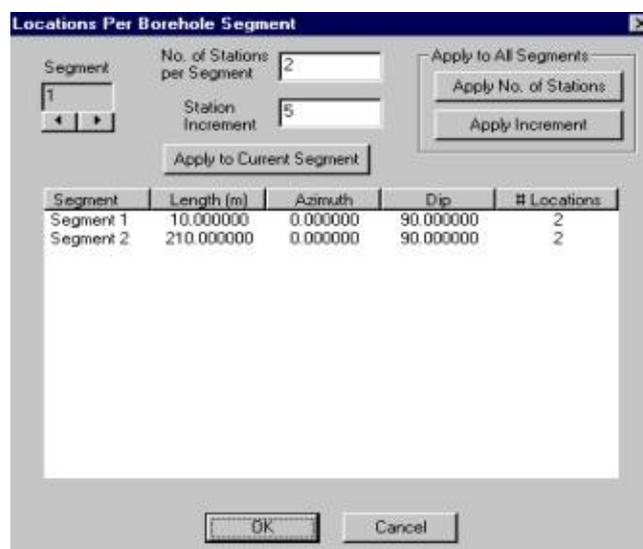
This will bring up the **Import UTEM4 Profile** dialog:



- 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

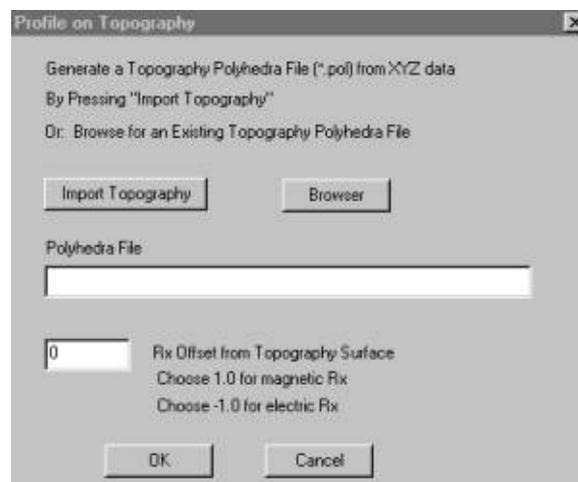
- Click **OK** to complete the import procedure

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 topography

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

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

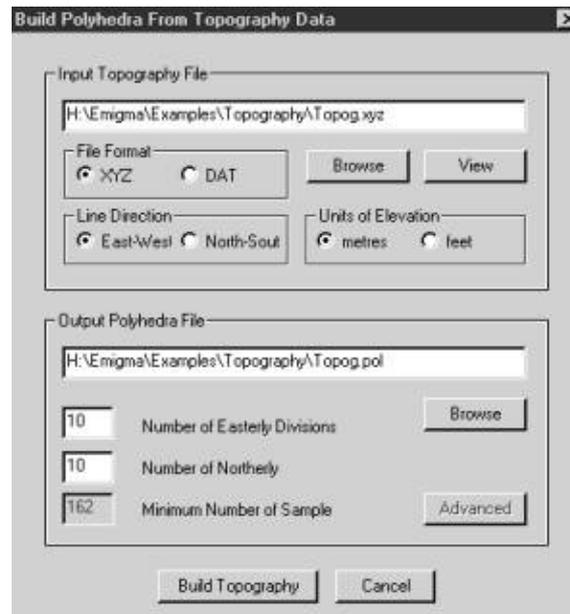


- Click **Browse** to find an existing polyhedra file (.pol)

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 line direction and units of elevation in the respective sections
- 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**

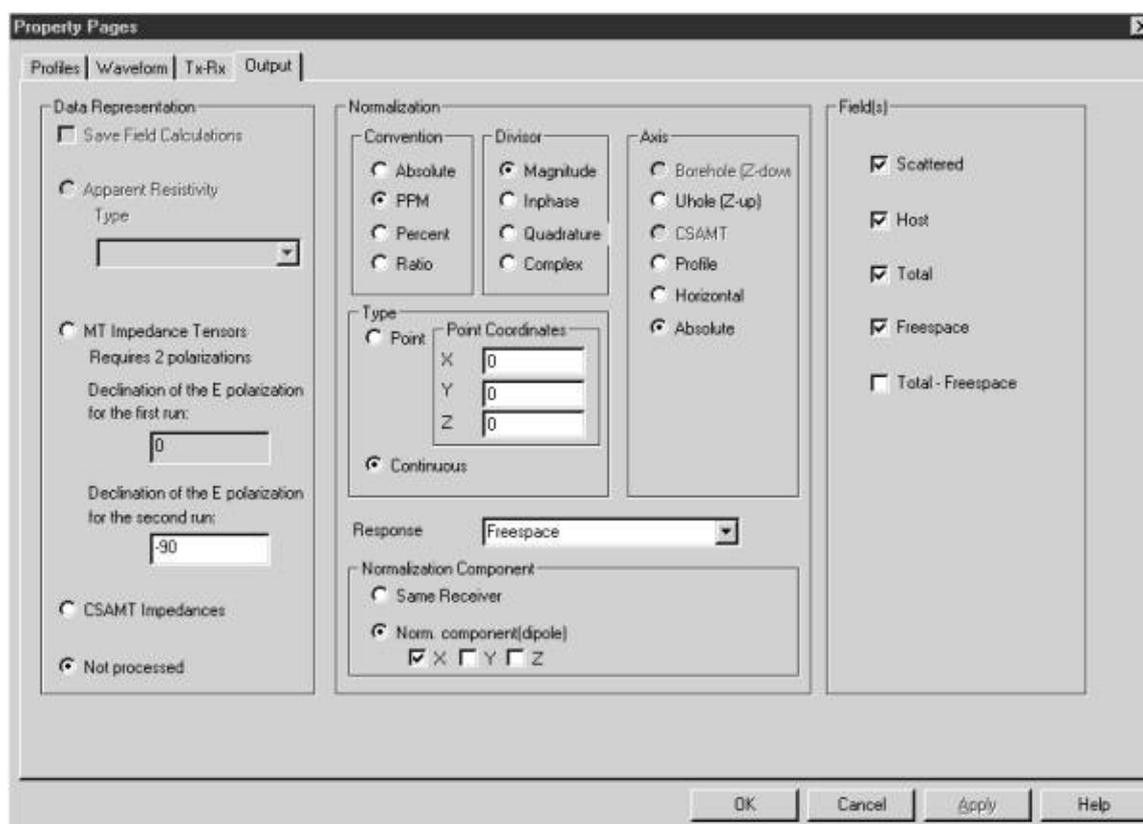
This will bring back the **Profile on Topography** dialog, with the newly created .pol file displayed in the **Polyhedra File** field

- 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 all 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 your data are MT, 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 your data are CSAMT, the **CSAMT Impedances** option will be selected automatically
3. 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 the azimuth direction of the borehole
- 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 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 conductivity structure of the ground. 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 consists of two half spaces with a number of layers sandwiched in between.

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). 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.

Model Configuration

Prisms/Plates/Polyhedra **Layers**

N...	Permeability	Resistivity	Permittivity	Thickness
1	1	1e+010	1	1e+030
2	1	0.25	1	20
3	1	10	1	1e+030

Configuration

Survey Name: LN 12 prisms
Model Name: 12 prisms

Total Number of Layers 3

Depth

Top Depth: -1e+030
Bottom Depth: -1e+008

Cole-Cole Polarization Mode Parameters

C (exponent) parameter: 0
M parameter (chargeability) dimensionless: 0
T (time constant) parameter seconds: 0

Resistivity & Susceptibility Grid Data Files

View
 Extrapolate to Basement
Convert to GPSZ

Edit Mode

Insert Layer
Replace Layer
Delete Layer
Undo Delete
Restore
<-- Import Layers

Layer Parameters

Layer #: 4
Resistivity (Ohm.m): 1e+010
Relative Permeability: 1
Relative Permittivity: 1
Thickness (m): 1e+030
Susceptibility: 0
Density (g/cm³): 0

OK Cancel Apply Help

The Layers tab

To reach the **Layers** tab from the **Database** dialog, click the **Model** button in its bottom part.

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](#)

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

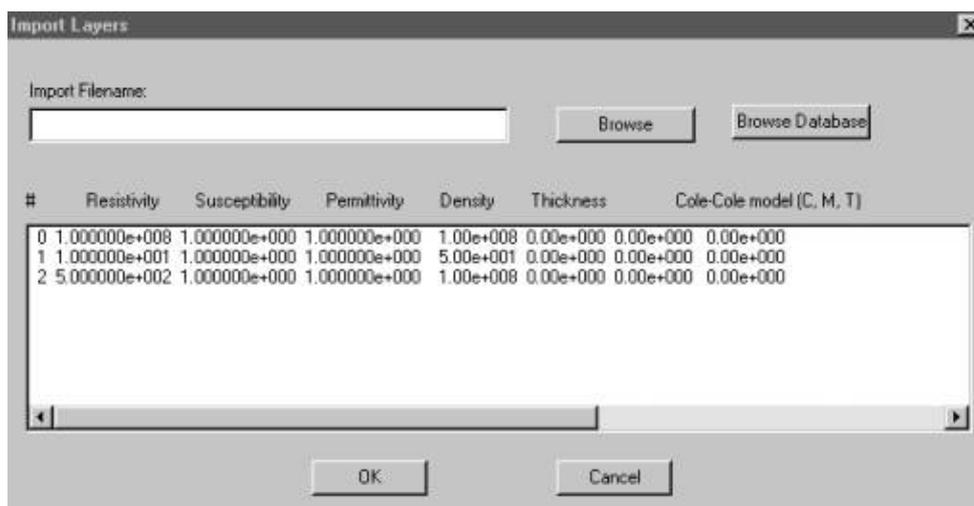
- To restore the layer, click **Undo Delete**.

The layer will reappear in the list

Import a layer

In the **Layers** tab:

- Click **Import Layers** in the **Edit Mode** section. The respective dialog will appear:



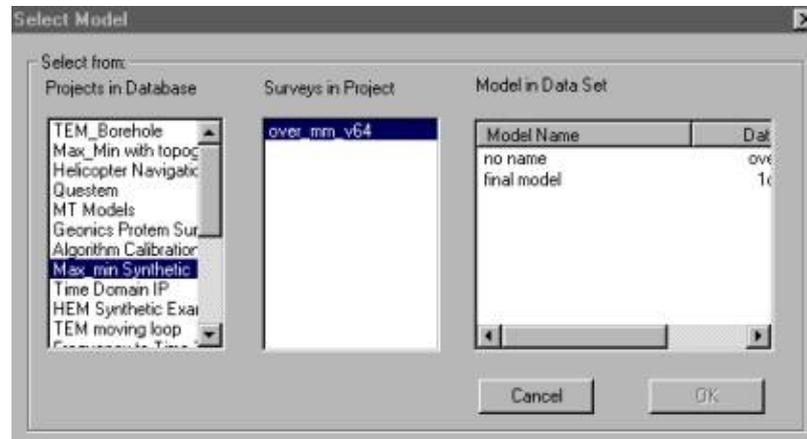
The **Import Layers** dialog

- In this dialog, click **Browse** to open the standard Windows-style **Open** dialog and select a PetRos EiKon file (.pev, .dat, .spt) to import layers from. Click **Open**. This returns you to the **Import Layers** dialog, in which the **Import Filename** box now contains the filename you selected and the field

below shows the layers to import and their properties

OR

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



The **Select Model** dialog

- Select the project, survey and model in the respective lists and click **OK** to return to the **Import Layers** dialog

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

- Click **OK** in the dialog to complete import and return to the **Layers** tab. The list of layers in the upper left-hand corner now contains the imported layers and their properties

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, τ , and a frequency dependence, c , - the three parameters that describe conductivity/resistivity, magnetic permeability and electric permittivity in EMIGMA

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.

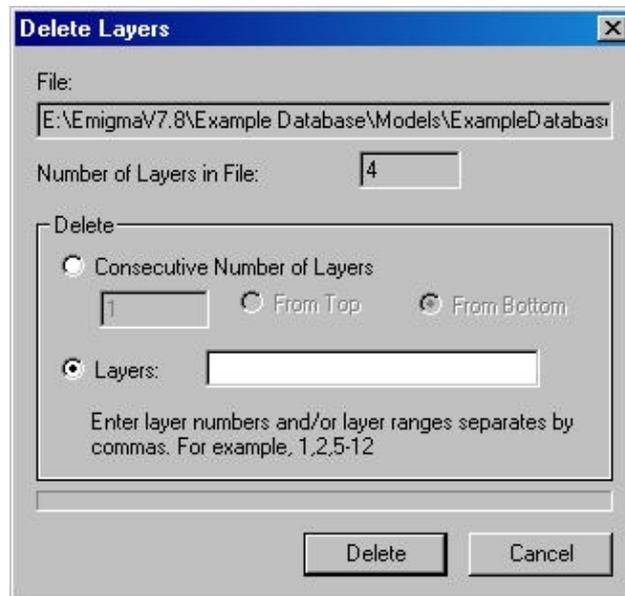
Note. To view these data as a grid, save your model and close the **Model** dialog. In the **Database** dialog, select the PEX file from the **Data Sets in Survey** list and click the **PEX Show** button  on the main toolbar.

- 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

To prevent the resulting pattern from being ragged and disrupted at the bottom, check the **Extrapolate to Basement** box prior to the conversion.

Note. A new GPSZ data set will be added to the **Data Sets in Survey** list of the **Database** dialog. Click the **PEX Show** 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:

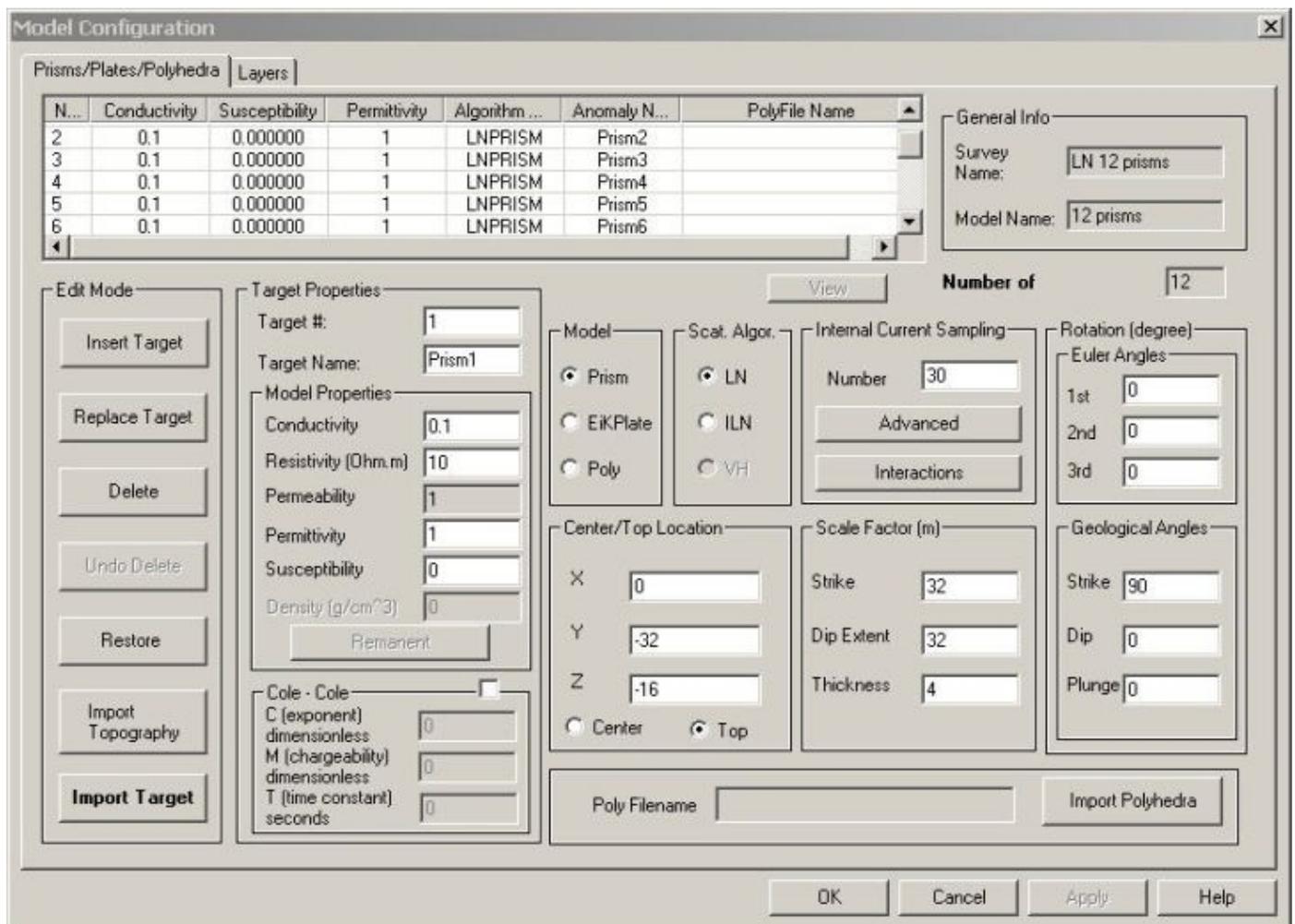


- The number of layers in the PEX file are displayed under the filename. If **Layers** is selected, the layer numbers can be entered as ranges or individually separated by commas. Layers are numbered from ground level to the basement.
- A shortcut option is available if the layers to be deleted are consecutive 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). 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** dialog after having specified the layered-earth model:



The Prisms/Plates/Polyhedra tab

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

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](#)

Insert a target

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

- Select between **Prism**, **Plate** and **Poly** 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**, whereas in the case of a plate, the **VH** option will be on by default

Note. For more information on these algorithms, see *PetRos EiKon's manual - Forward 3-D Electromagnetic and Magnetic Simulation Platform for Comprehensive Geophysical Modeling*, Feb. 2000

- 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 remanent magnetization in magnetic systems, click the **Remanent** button. In the **Remanent Magnetization** dialog to open, specify the inclination, declination and relative magnetization parameters

- 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

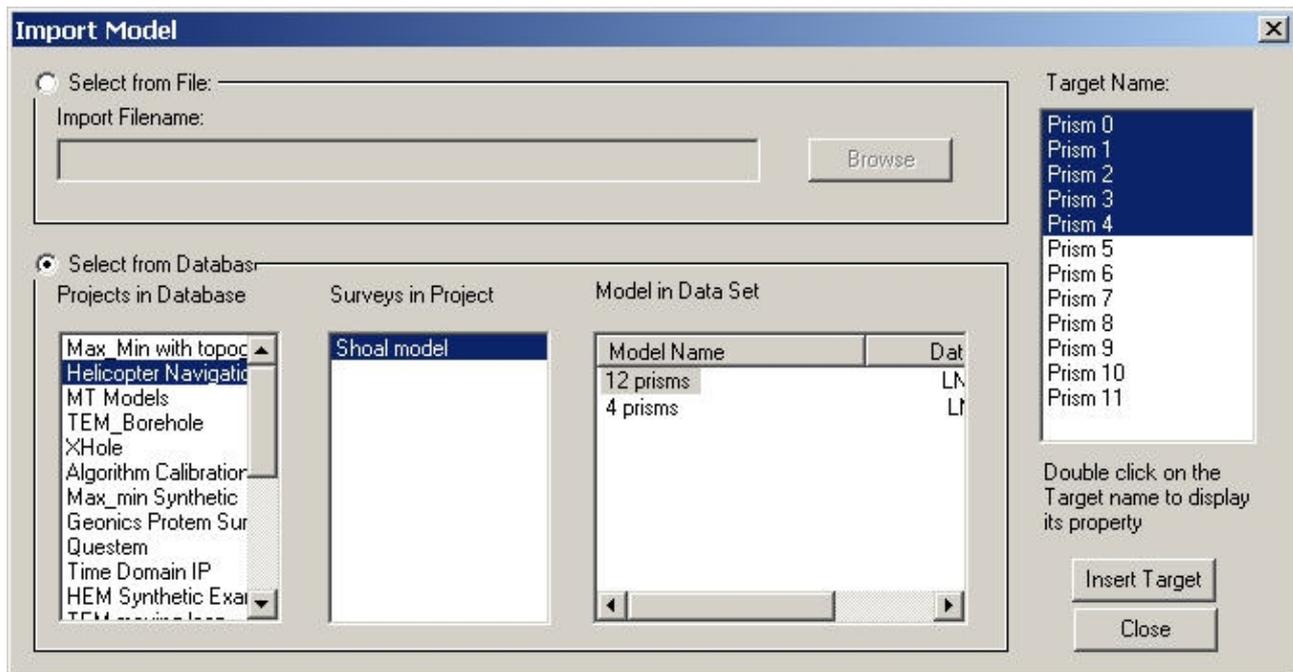
OR

- 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** dialog appears



To import a model from an existing PEV file:

- Click **Browse** in the **Select from File** section to open the standard Windows-style **Open** dialog and select a PetRos EiKon file (.pev, .dat, .spt) containing a required model
- Click **Open**. This will return you to the **Import Model** dialog. The **Import Filename** box will now contain the filename you selected, and you will see the name of the target to import in the respective field on the right
- Click **Insert Target** to complete the import

To import a model from a current database:

- Click **Select from Database** and select the project, survey and model from the respective lists

The name of the model will be displayed in the **Target Name** field on the right

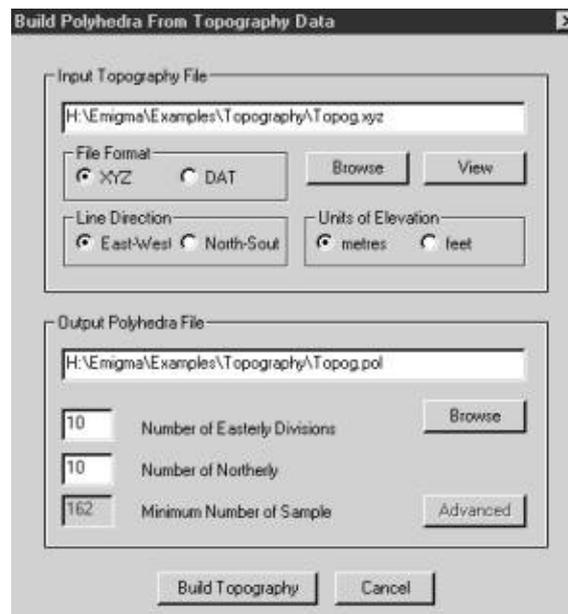
- Click **Insert Target** to complete the import

As you return to the **Prisms/Plates/Polyhedra** tab, you will see the imported model added to the list of targets.

Import topography

For more realistic representation of your model, you synthesize topography and polyhedra. In the **Prisms/Plates/Polyhedra** tab:

- Select **Poly** in the **Model** section
- Specify the physical parameters of your target in the **Target Properties** section
- Click the **Import Topography** button in the **Edit Mode** section. The **Build Polyhedra from Topography Data** dialog appears:



In the upper **Input Topography File** section

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

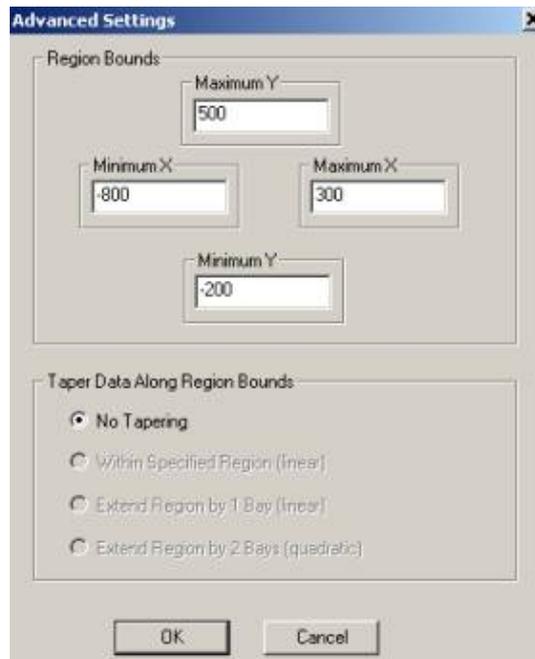
See Also [Topography file format](#)

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 dialog

- 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 below will change accordingly. To properly carry out simulation, this number cannot exceed 250 (for ILN) and 1000 (for LN)
- Click **Advanced** to specify the boundaries of the topography you are about to import in the **Advanced Settings** dialog to appear:

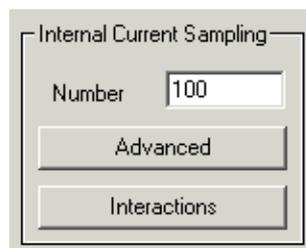


All specified, click **Build Topography** in the bottom part of the **Build Polyhedra from Topography Data**. This will return you to the **Prisms/Plates/Polyhedra** tab. In the **Poly Filename** field, you will see the newly created .pol file

- In the **Edit Mode** section of the tab, click **Insert Target** to add your polyhedron, topography inclusive, to the list of targets

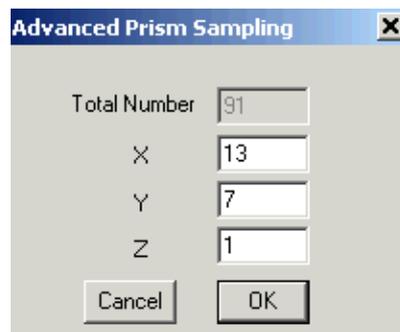
Specify the number of sampling points

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:

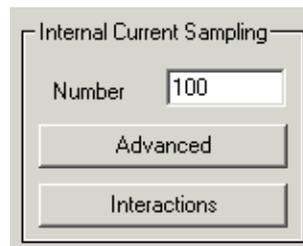


- Type your values in the **X**, **Y** and **Z** boxes. The multiplication product of these values can be viewed in the **Total Number** field on top of the dialog
- Click **OK** to close the dialog and view the result in the **Number** box of the section

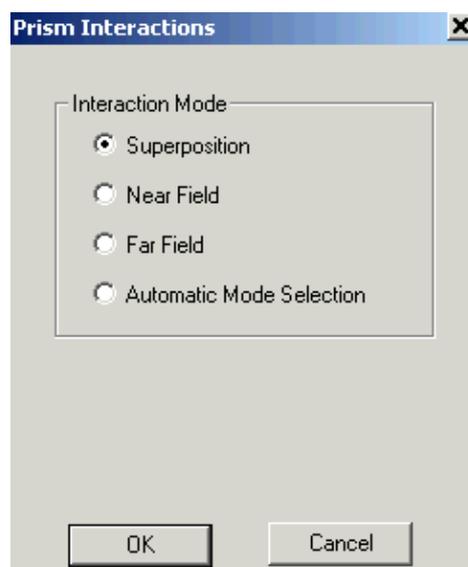
Note. If your model is a plate, the **Internal Current Sampling** section will be disabled and the respective number will be set to 441. This value represents a mesh of 21 by 21 used to calculate the scattered current distribution in the VH scattering algorithm and cannot be changed.

Specify the type of interaction between targets

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



- Click the **Interaction** button. This brings up the **Prism Interactions** dialog
- Select between the four options available in this dialog:



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

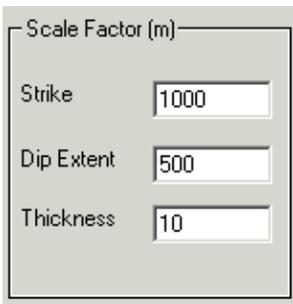
The **Far-Field** option includes the primary effects of multiple conductors not in contact with your target

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

- Click **OK**

Specify the model size and position

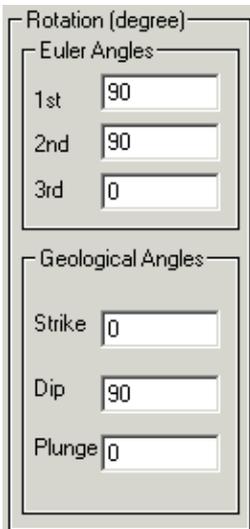
In the **Scale Factor (m)** section of the **Prisms/Plates/Polyhedra** tab, specify the geological dimensions of your target, including its strike, dip and thickness:



Scale Factor (m)

Strike	1000
Dip Extent	500
Thickness	10

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:



Rotation (degree)

Euler Angles

1st	90
2nd	90
3rd	0

Geological Angles

Strike	0
Dip	90
Plunge	0

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**

Saving model configuration

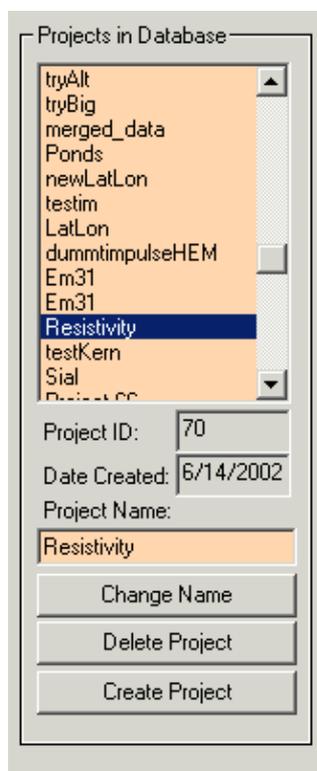
- Click **Apply** in the bottom right-hand corner of the **Model Configuration** dialog (either in the **Prisms/Plates/Polyhedra** or the **Layers** tabs)
- Click **OK** to close the **Model Configuration** dialog and return to the initial **Database** dialog. The box on the **Model** button in the bottom 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](#)

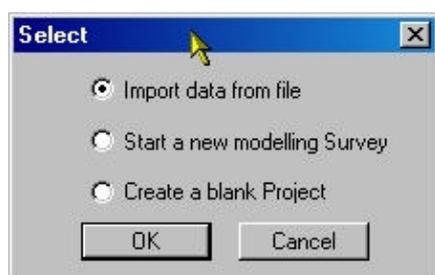
[Change the Name of the Project](#)

[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.

The screenshot shows a dialog box titled "Surveys in Project". On the left, there is a list box containing two items: "Big Survey" (which is highlighted) and "Big Survey_backup". To the right of the list box, there are two input fields: "Survey Name:" with the text "Big Survey" and "Survey ID:" with the text "858". Below these input fields are five buttons arranged in two columns: "Change" (top right), "Survey Comments" (middle left), "Add Survey" (bottom left), "BackUP" (middle right), and "Delete Survey" (bottom right).

Related Topics

[Add/Delete a Survey](#)

[Rename a Survey](#)

[Write, View and Edit Survey Comments](#)

[Create a BackUP 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

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 needs so, 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](#)

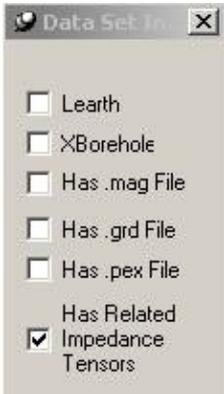
[Specifying Waveform](#)

[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 respective dialog will appear:

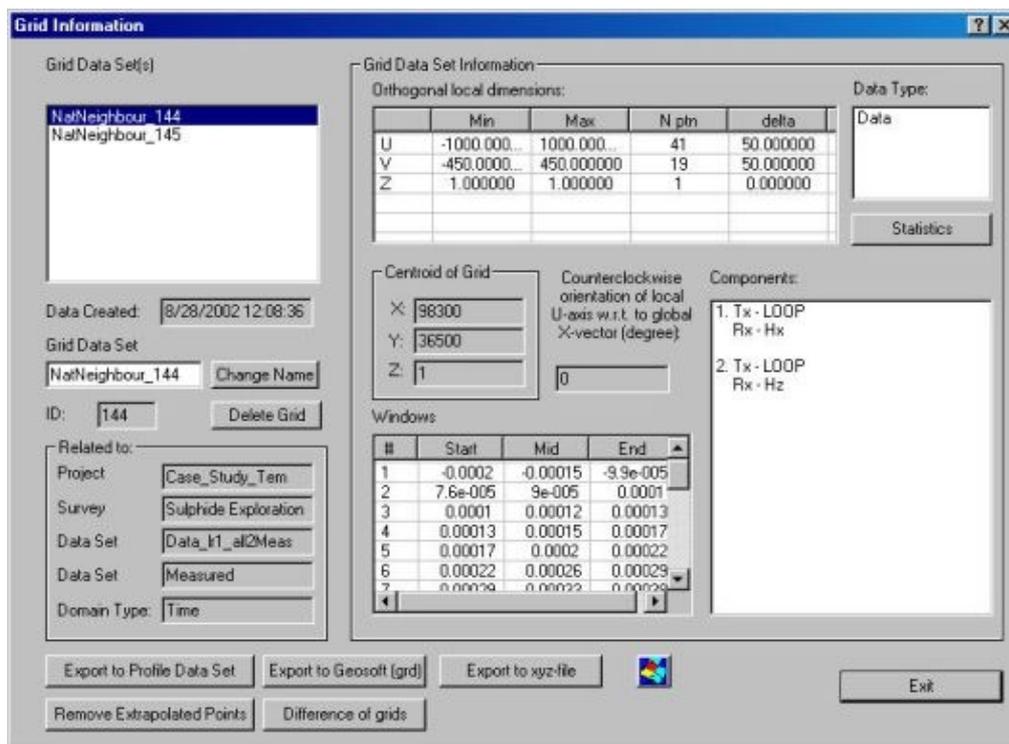


To search through a number of data sets without the window closing, it can be pinned by clicking on the pin icon in the upper left hand corner.

- The checked **Learth** box means that the data set contains a background field model.
- The checked **Xborehole** box means that the data set represents a cross-hole survey.
- The checked **Has .mag File** box means that the data set has an attached MAG file resulting from 3D magnetic inversion within EMIGMA
- The checked **Has .pex File** box means that the data set has an attached PEX file resulting from 1D inversion within EMIGMA
- The 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** 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 through 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.

See

[Remove "No Data" Points from your Grid](#)

[Export a Grid into a Profile Data Set](#)

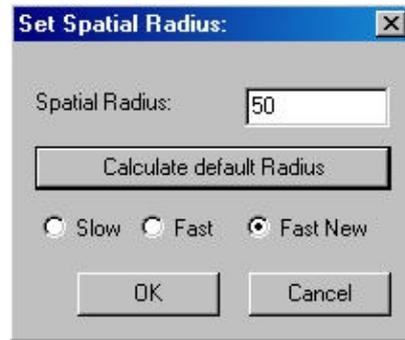
[Export a Grid into the Geosoft Format](#)

[Export a Grid into the XYZ 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

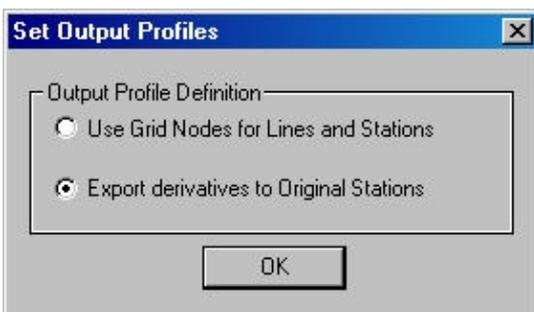
In the present example: a spatial radius of 50 means that if there are no data in the radius of 50 m around a given point – a grid cell center, – this cell will be removed from interpolation

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

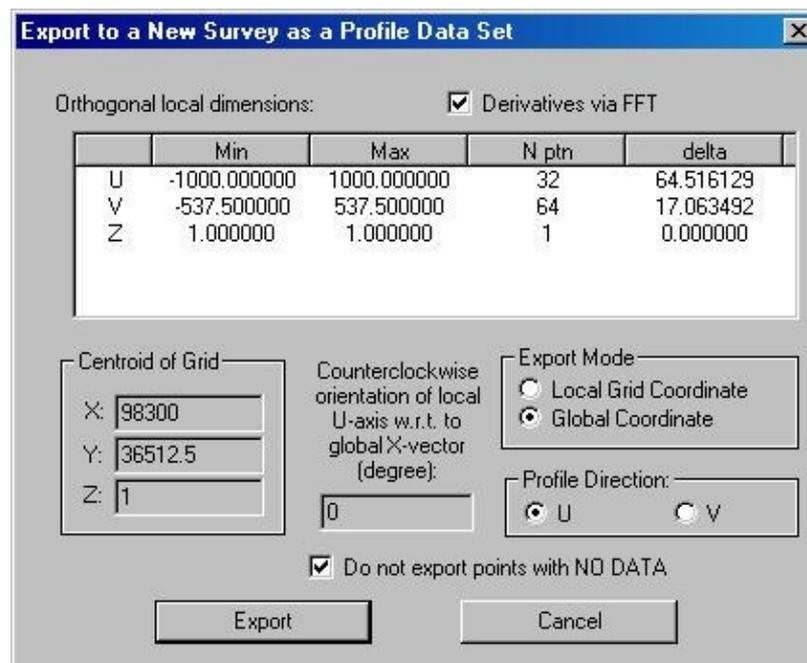
Your new grid will appear in the **Grid Data Set(s)** list of the **Grid Information** dialog.

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:

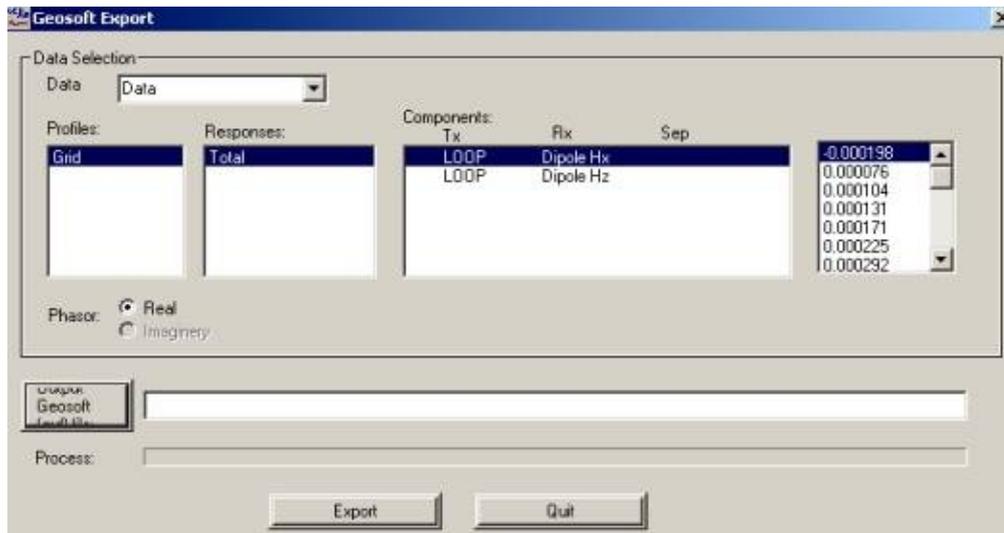
- Select the **Global Coordinate** button
- Select between the two options for profile direction in the respective section.

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 be 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:



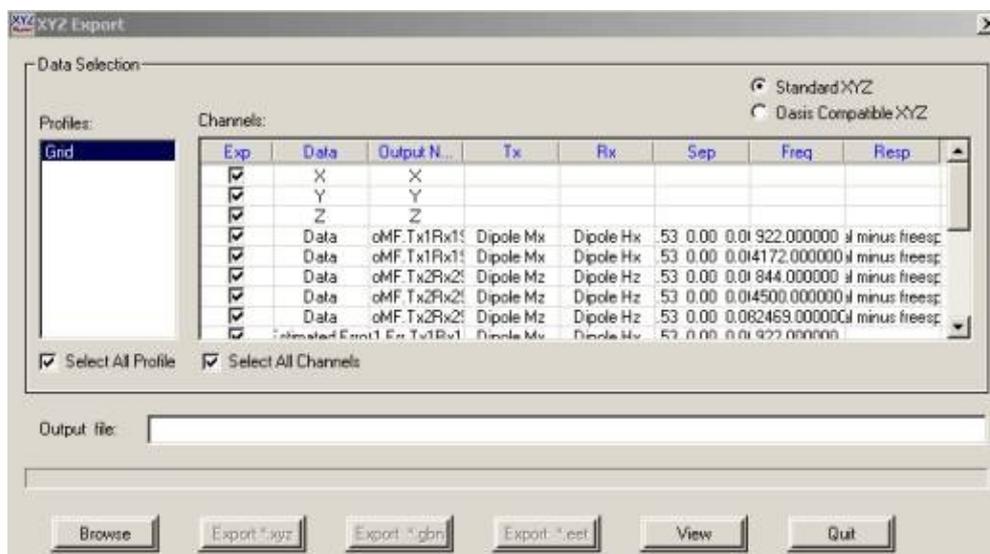
- Select the data type you want to export from the **Data** dropdown list

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 a grid into the XYZ format

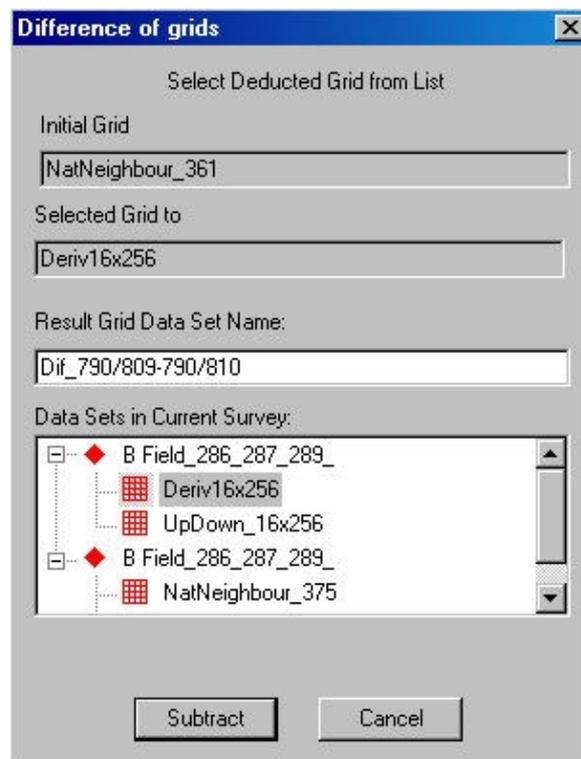
- Click the **Export to xyz-file** button in the bottom of the **Grid Information** dialog. The **XYZ Export** dialog will open offering two different options of export: one to transfer data to the standard XYZ format and the other to the Geosoft's Oasis Compatible XYZ format:



- Select 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
- 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
- To view the output file in the text format, click **View** in the bottom of the dialog
- Click **Export *.xyz** if you export your data into the standard XYZ format, **Export *.gbn** if you export them into the Geosoft's Oasis compatible XYZ format and **Export *.eet (.qct)** if you export your data into the XYZ format to further subject them to quality control in PetRos EiKon's QCTool

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 be modified. Click **Subtract** to create the new grid data set.

