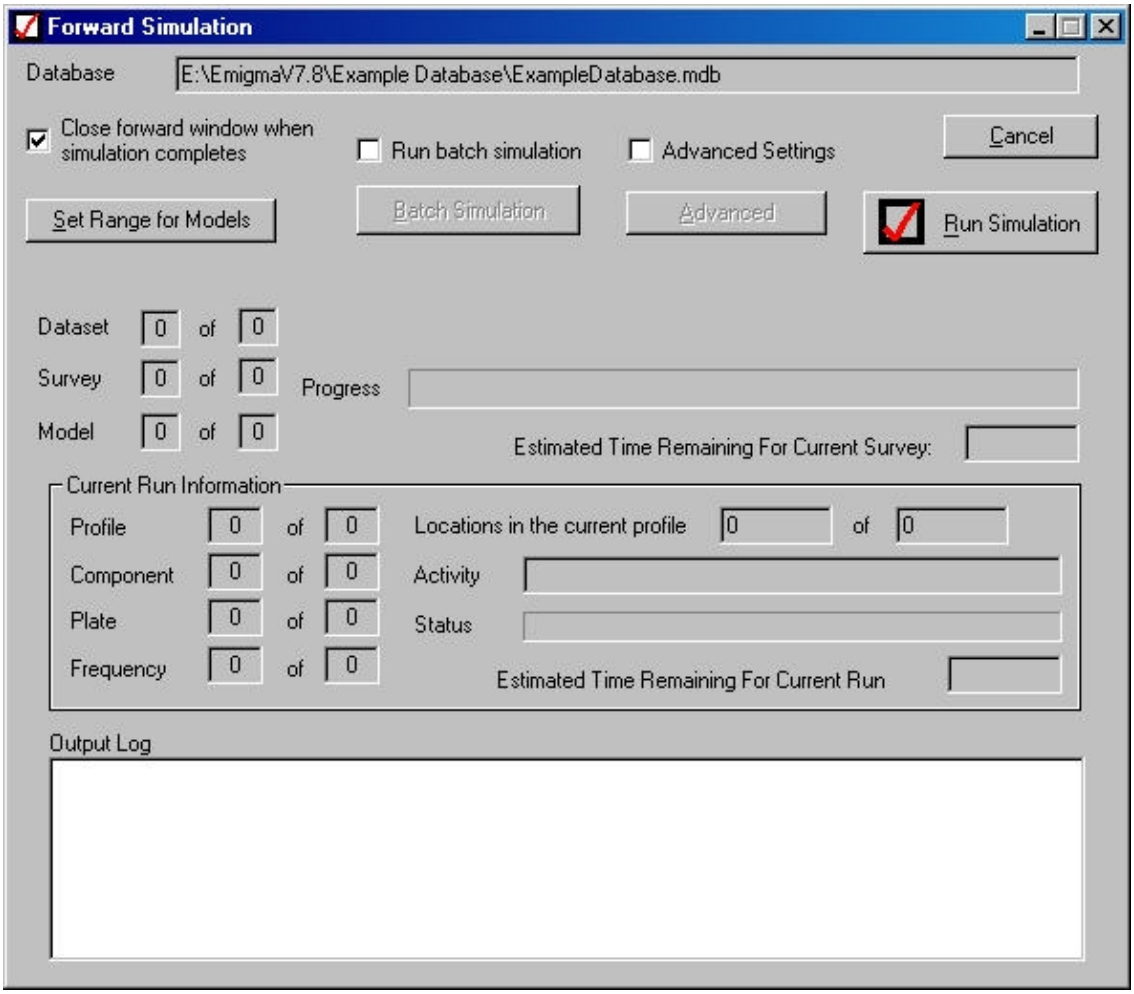
Table of Contents

Forward Simulation Manual

Forward Simulation

Select the data set to subject to simulation in the **Database** dialog and click the **Forward Simulation** button  on the EMIGMA toolbar. A message will ask you whether you want to overwrite the selected data set. Click **Yes** to overwrite it and **No** to create a new data set.

The **Forward Simulation** dialog will appear, with the upper field showing the path to your database:



You can run simulation as is or specify one of the four available modes (see **Related Topics** below). During simulation, which may take a long time, the **Forward Simulation** dialog will keep you updated on the progress, providing information on the status and estimated simulation time of a current survey on the whole and each current run in particular.

Note. *Y*ou can minimize the **Forward Simulation** dialog to do some other things maximizing it from time to time to check on the progress. The **Cancel** button stops the operation and closes the **Forward Simulation** dialog upon completion of a current run.

Related Topics

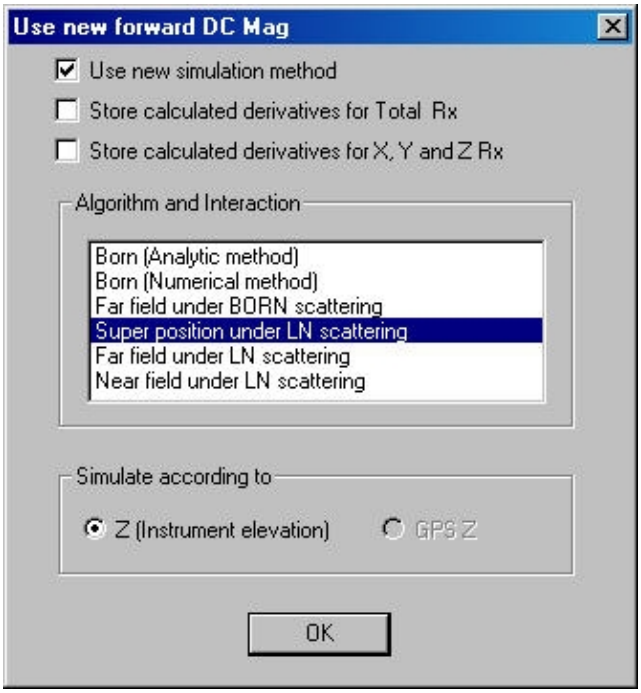
Forward Simulation As Is

Model Suite Generation - Setting Ranges for Models

Forward Simulation As Is

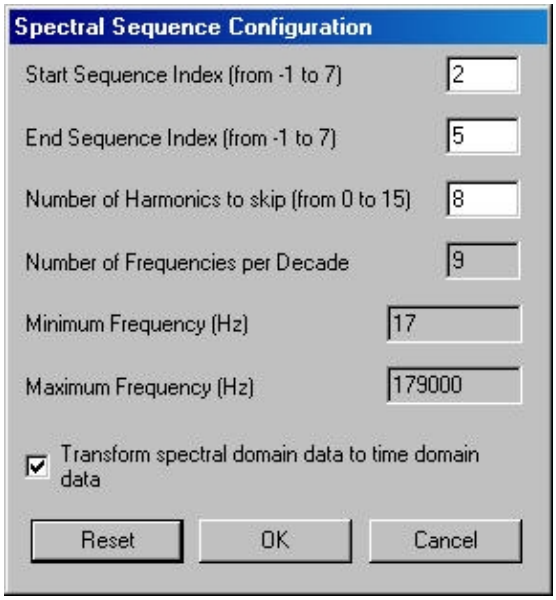
In the **Forward Simulation** dialog, click **Run Simulation** to perform simulation as is.

1. With FEM data, simulation will start right away.
2. In the case of magnetic, gravity or resistivity system, the **Use New Forward** window will open offering you to select the scattering algorithm and the type of interaction to be used in simulation:



- Check the **Use new simulation method** box in the upper part of the dialog to enable the list of methods below and select a method to apply. This option is available only for magnetic or resistivity data
- Check **Store calculated derivatives** to save the derivatives in the same data set. This option becomes active in case you are using a new simulation method.
- If the data set contains a GPS Z channel, these values can be used for the simulation by choosing the **Simulate according to GPS Z** option.
- Click **OK** to start simulation.

3. In the case of TEM, the **Run Simulation** button will bring up the **Spectral Waveform configuration** dialog:



- Check or change, if necessary, the starting and ending frequency sequence indices and the number of harmonics to skip in the respective boxes. The values in the **Number of Frequencies per Decade**, **Minimum Frequency** and **Maximum Frequency** boxes will change accordingly.
- Click the **Reset** button to retrieve the original values.
- Check the **Transform Spectral Domain Data to Time Domain Data** box to run this operation automatically.

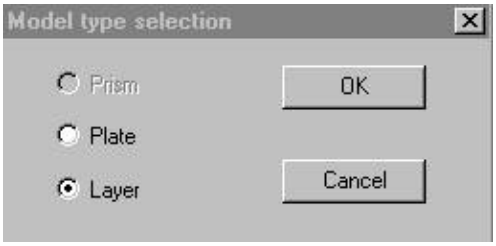
This option is useful when you have modified an already available time-domain model and want to subject it to another forward simulation. In this case, you will not need to run the standalone transform to convert your spectral data into time-domain; this option will provide this operation automatically

- Click **OK** to apply changes and close the **Spectral Waveform Configuration** dialog. Click **Cancel** to cancel the changes, if you have made any, and close the dialog.

Model Suite Generation

Click **Set Range for Models** if you want to subdivide your model into parts differing from each other in a certain property, such as thickness, resistivity or susceptibility. The objective of this operation is to create a number of models to compare your measured data sets with and to select the one that complies best with your survey results

The **Model Type Selection** dialog appears offering you to select between **Plate**, **Prism** and **Layer.** As of now, only the **Plate** and **Layer** options are accessible, with the **Layer** option selected by default:



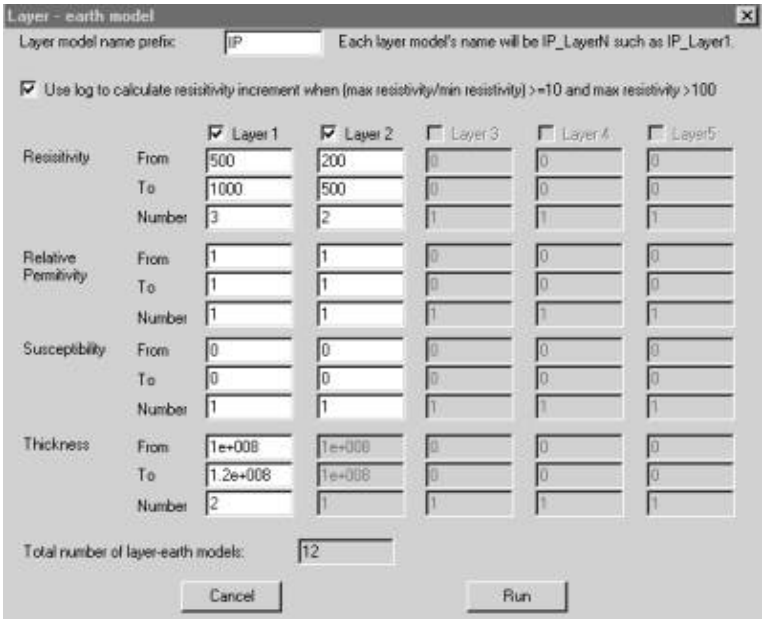
See

Specify Ranges of a Layer-Earth Model

Specify Ranges of a Plate Model

Specify ranges of a layer-earth model

- In the **Model Type Selection** dialog, leave the **Layer** option on and click **OK**. The **Layer - earth model** dialog will open:



- In the **Layer Model Name Prefix** field, type in the name of your model so that the resulting data sets (models) could have the same name prefix followed by the underscore and a subsequent number (e.g. IP_Layer1, IP_Layer2 and so on)
- If max/min resistivity is equal to or more than 10 and maximum resistivity is more than 100, leave the box below the model name prefix checked to use logarithmic function and calculate resistivity increment
- Check the layer you want to subdivide into models . The column below will become active
- Change any property range in the **From** and **To** boxes across **Resistivity**, **Susceptibility**, **Relative Permittivity** and **Thickness** and specify the number of models you want to simulate in the respective boxes

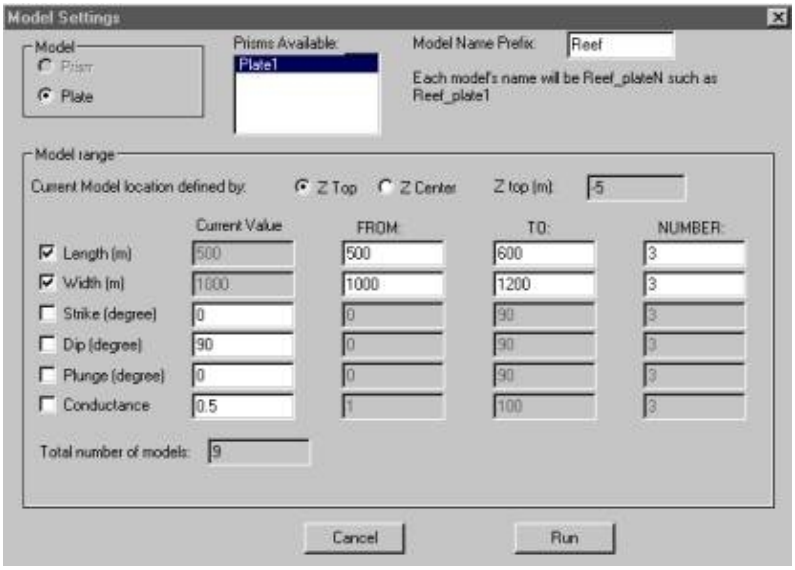
You can select several properties or several layers at a time. In this case, the number of models to be simulated will be a multiplication product of all **Numbers**. For example, if, in the Layer 1 column, you have 3 in the **Number** field of one property and 2 in the **Number** field of another, the total number of the Layer 1 models will constitute 6. If you add 2 in the **Number** field of any property in the Layer 2 column, the number of models to be simulated will total 12. You can view the total number of layer-earth models in the respective box in the bottom part of the dialog

- All parameters specified, click **Run** to start simulation

Specify the ranges of a plate model

- Select the **Plate** option in the **Model Type Selection** dialog and click **OK**

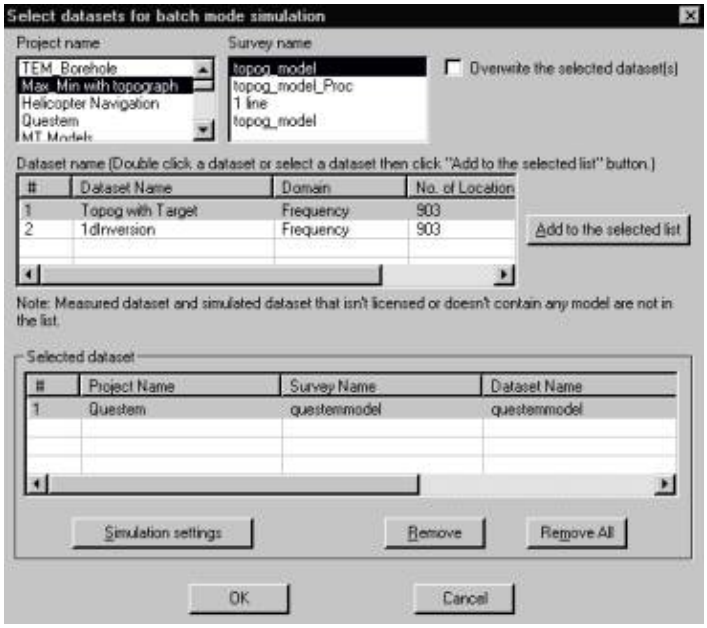
The **Model Settings** dialog will open:



- Select the model to specify parameters for in the **Prisms Available** field
- Type in the model name prefix in the respective box on the right. The resulting data sets (models) will have the same name followed by the underscore and a subsequent number as shown in the example (Reef_plate1)
- In the **Model Range** section, select between the center point and top center modes of the plate position in space
- Check the property you want to specify the ranges for. The **From**, **To** and **Number** fields will become active
- In these fields, specify the ranges and the number of models and click **Run**

Batch Simulation

In the **Forward Simulation** dialog, check the **Run Batch Simulation** box. The **Select Data Set for Batch Mode Simulation** dialog will open:



In the **Project Name** and **Survey Name** lists, you will see selected the names of the current project and survey.

In the **Dataset Name** table below:

- Select the data set you want to subject to simulation and click **Add to the Selected List** button

OR

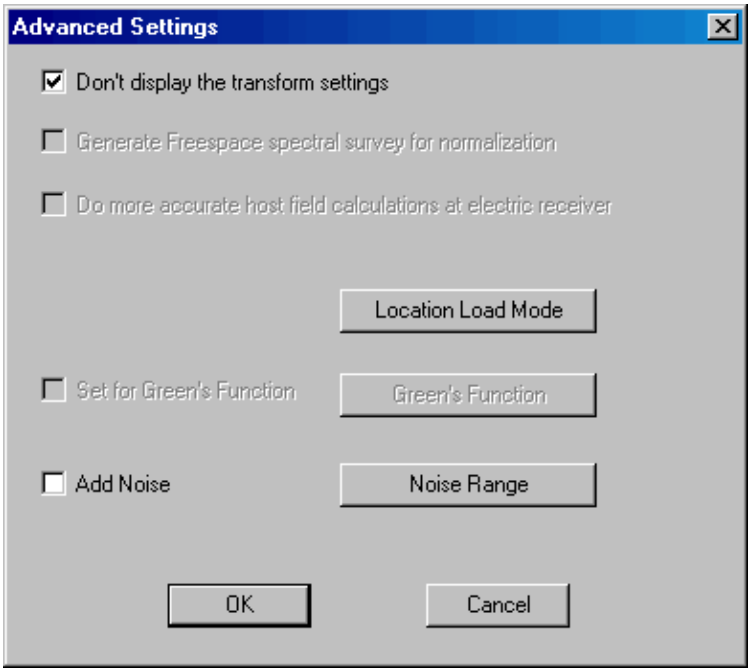
- Double-click on the required data set

The data set will appear in the **Selected Dataset** table

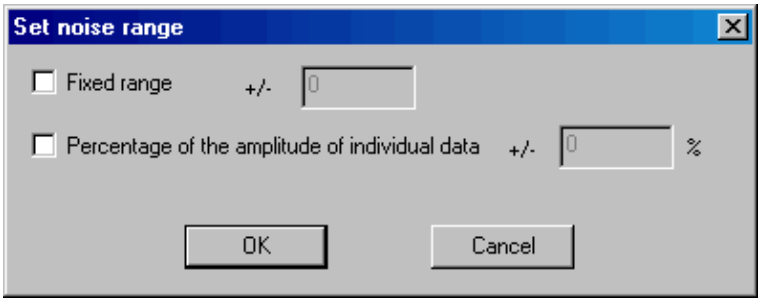
- To remove a data set from this table, select it and click **Remove**; to remove all data sets from this table, click **Remove All**
- To overwrite the selected data set by the one obtained through forward simulation, check the respective button in the top right-hand corner of the dialog
- If your data are time domain, gravity, magnetic or resistivity, the **Simulation Settings** button becomes active. In the case of TEM it opens the **Spectral Waveform Configuration** dialog in which you can specify the frequency sequence indices, number of harmonics to skip, spectral-to-time-domain data transform. In the case of gravity, magnetics, resistivity, it opens the **Use New Forward** dialog in which you can specify the method of simulation. For details, see **Forward Simulation As Is**
- Click **OK** in the **Select Data Set for Batch Mode Simulation** dialog to return to the **Forward Simulation** dialog and start simulation.

Advanced Settings

In the **Forward Simulation** dialog, click the **Advanced Settings** checkbox to access the **Advanced Settings** window:



- If you are simulating a time domain survey, the **Don't display the transform settings** option will be available. Check the box if you do not want to specify settings for the transform tool. The settings window will not appear when simulation begins.
- It is possible to specify whether a survey will be generated to perform normalization when using spectral surveys. Check the box labelled **Generate Freespace spectral survey for normalization** if normalization is desired.
- The option labelled **Do more accurate host field calculations at electric receiver** will be available for surveys with electric receivers. Check this option to increase the accuracy of the calculations. This option is off by default since it will increase the amount of time for the simulation and is usually not needed.
- Select the Location Load Mode option by clicking its button.
- Access the Green's Function settings by checking the box labelled **Set for Green's Function**.
- To add noise to your data, click the box labelled **Add Noise** and the **Set Noise Range** window appears:



- The noise introduced to the data can be either a **Fixed range** or a **Percentage of the amplitude of individual data**. Check one or both of the available options and enter the value in the box labelled **+/-**. Selecting both options adds the two noise values together.

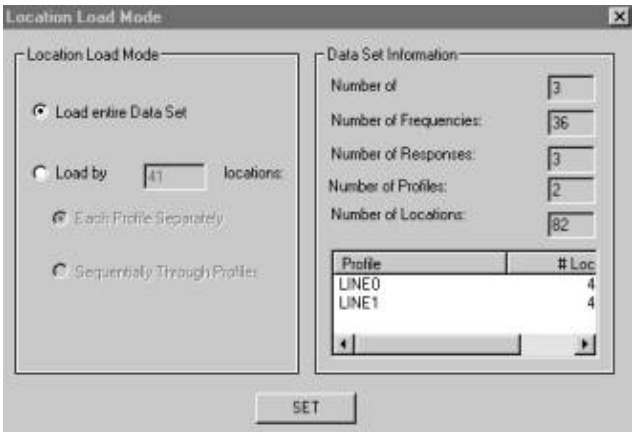
Click **OK** to save your settings and return to the **Forward Simulation** dialog.

Click **Run Simulation** to launch forward simulation.

Location Load Mode

In the **Forward Simulation** window, click the **Advanced Settings** checkbox. Next, click the **Location Load Mode** button in the **Advanced Settings** window to specify the order for the algorithm to pick up locations during simulation. This option is not needed if you are applying a new simulation method to your magnetic or resistivity data as well as in the case of gravity systems.

The **Location Load Mode** dialog opens:



In the right-hand section, you can see the general information on your data set, the list of profiles and the number of locations per profile.

In the left-hand section:

- Select **Load Entire Data Set** if your data set contains not too many locations and you want to subject them to simulation all at once
- Select the **Load by** option if your data set is too big. If it contains more than 4000 locations, this option will be selected for you automatically. Specify the number of locations to be loaded at a time and select between the two ways for the algorithm to go through these locations. If, for example, the number you specified is 20 and your profile contains 41 locations, the **Each Profile Separately** mode will pick up the first 20, then the second 20 and then the last 1 location, whereas the **Sequentially Through Profiles** mode will process sets of exactly 20 locations no matter which profile they belong to.

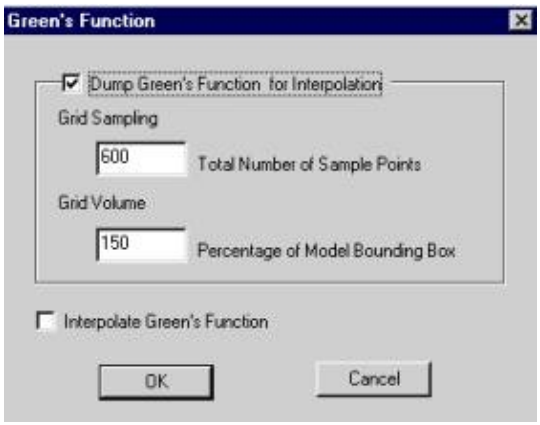
Click **SET** to save your settings, click OK in the **Advanced Settings** window and return to the **Forward Simulation** dialog.

Click **Run Simulation** to launch forward simulation.

Green's function

Green's function is an integral part of the forward simulation process. The respective calculation is run automatically as a routine program based on certain default settings. However, in some cases, it may be useful to view the result of this calculation, especially if your models represent slight variations of each other differing, for example, in the dip of structures or their electrical properties. The **Set for Green's Function** option will be accessible only if your model is 3D; in layered earth models, i.e. in the absence of scatterers, it will be inapplicable.

- Check the **Advanced Settings** box in the **Forward Simulation** window. Next, check the **Set for Green's Function** box in the **Advanced Settings** window. This will bring up the **Green's Function** dialog:



- Check the **Interpolate Green's Function** box and click **OK** to launch simulation using the settings offered by default and displayed in the **Dump Green's Function for Interpolation** section

Or, you can replace the default settings with your own:

- Check the **Dump Green's Function for Interpolation** box. The respective section will become active
- Specify the values in the **Grid Sampling** and **Grid Volume** fields

Grid Sampling allows up to 1000 points within the **Grid Volume**, a model region, which may be any multiple of the bounding box around the model, e.g. 150%. The larger the model region, the more points will be required to maintain a fixed accuracy and the more complex (and time-consuming) the resulting interpolation will be

- Check the **Interpolate Green's Function** box and **OK** to start simulation using the specified settings.