

## Getting Started

What is Potent?

Main concepts

A simple tutorial

## What is Potent?

*Potent* is a program for modelling the magnetic and gravitational effects of subsurface structures. It provides a highly interactive 3-dimensional environment that is well suited for:

- **Detailed ore-body modelling for mineral exploration.** Potent is used by mining and exploration companies world-wide. You can interpret surface, airborne and down-hole data; separately or simultaneously.
- **Stratigraphic modelling for petroleum exploration.** Potent is an economical, versatile and highly interactive tool for building models of complex layered structures.
- **Education.** Educational establishments around the world use Potent for teaching and research purposes.
- **Environmental and ordnance work.** Potent is used for industrial decontamination studies and to help locate unexploded ordnance.

*Potent* can model multi-component data, such as down-hole  $F_x$ ,  $F_y$ ,  $F_z$  magnetic data, or tensor gravity data.

## Main concepts

The main concepts in *Potent* are:

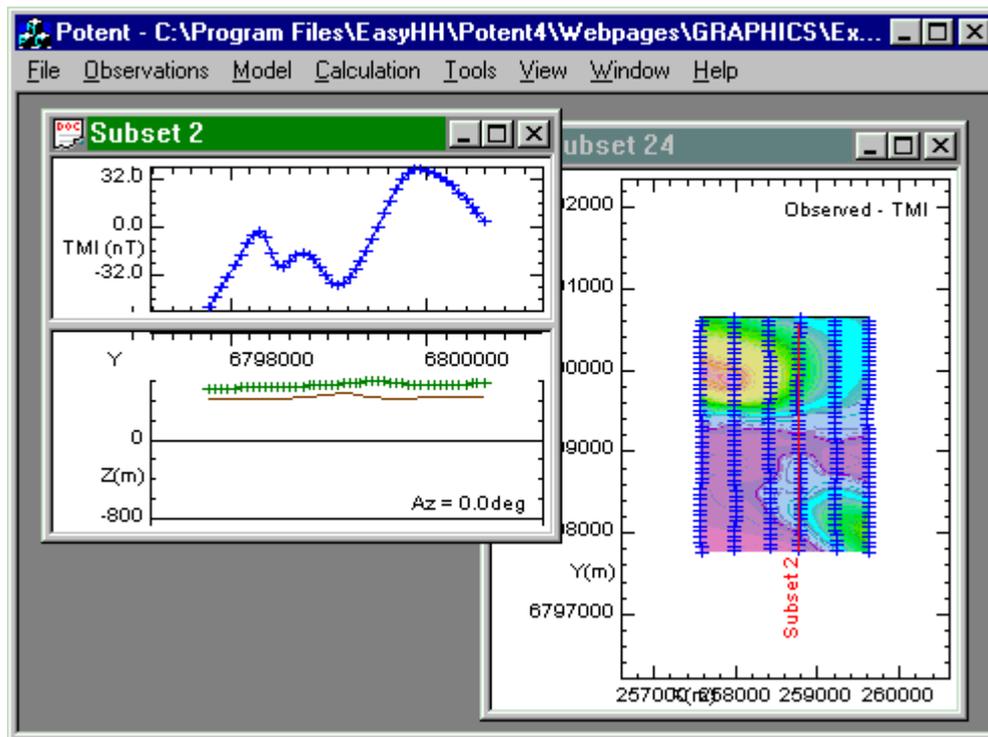
- Observations
- Model
- Calculation
- Visualisation

The primary function of the program is to bring these together in a coherent and intuitive way.

## Observations

These are the “facts”. Someone has measured one or more physical quantities (total magnetic intensity, vertical gravity, etc.) at a set of known locations. The only uncertainty is in any inherent “noise” or, if you are using gridded data, in the interpolation process that was used to generate the grid values from the original “true” values.

This example shows observed TMI data from a small fragment of an airborne survey. The observed values are depicted as an image and as a single N-S profile (labelled “Subset 2” on the plan). The observations are represented as blue crosses on the plan. In the profile window the blue crosses in the top pane represent the observed field values (TMI in this case). The bottom pane is used to depict the subsurface. The green crosses mark the actual position of the observation sensor (the aircraft in this case). The line below the green crosses represents the topography, and has been generated by subtracting the ground clearance (radar altimeter) from the true (GPS) height of the sensor.



There is no inherent limit to the number of sets of observations that you can input to Potent. For example, you could load TMI airborne data, down-hole 3-component magnetic data, and gravity road-traverse data simultaneously.

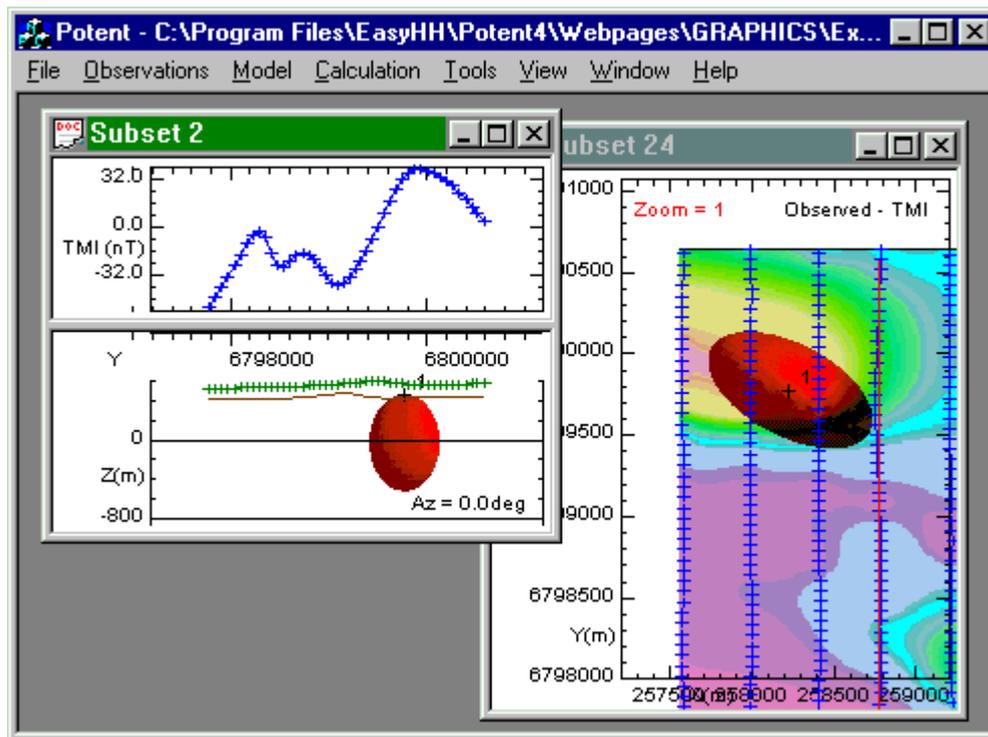
See also: - [Working with Observations](#) in the User's Guide.

Back to [Main Concepts](#)

## Model

A Potent model consists of an assemblage of simple 2-D or 3-D geometrical bodies such as cylinders and ellipsoids. Your main task as an interpreter is to devise a model that is geologically plausible and also is consistent with the observed physical values

This graphic shows the same data as in the previous topic, but with a model consisting of a single ellipsoidal body. This could be a grossly simplified representation of an ore-body. Notice that the right-hand window (the plan) has been zoomed (using the [Tools | Zoom](#) command) to make the body more visible.

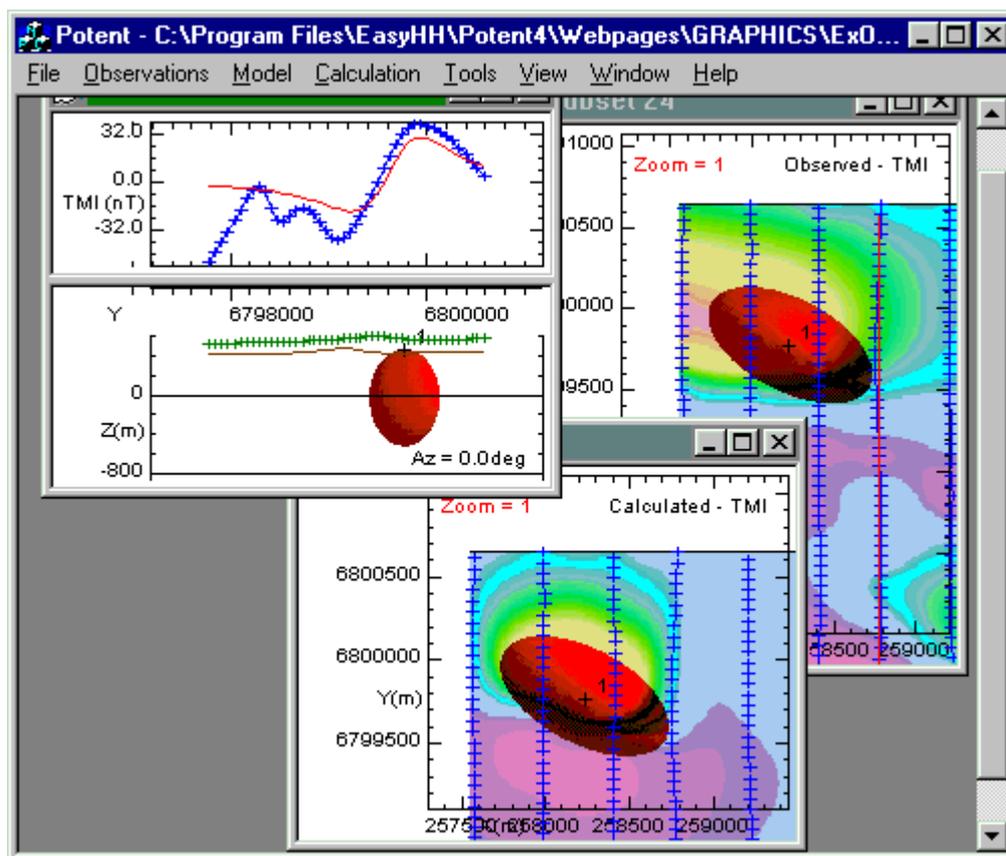


See also - [Working with the model](#) in the User's Guide

Back to [Main Concepts](#)

## Calculation

The model is consistent with the observed physical values if its calculated field matches the observed values to some (subjective) degree of precision. You assess this by calculating the field (TMI in this case) due to the model and comparing it with the observed field.



In this image the red curve in the profile window (top left) is the calculated field due to the model. A third window (bottom centre) has been introduced to show the calculated field as an image. In an actual modelling session you would proceed to adjust the shape, position and physical properties of the model so that its calculated field correlates well with the observed field. Note that *Potent* calculates the field due to the model at the actual observation points. These are the only points at which the observed field is known, and from a modelling point of view it is not necessary to calculate the field elsewhere.

See also - [Calculating the field due to the model](#) in the User's Guide.

[Back to Main Concepts](#)

## Visualisation

You subjectively assess the “match” between the observed and calculated physical values by visualising them in the most appropriate manner. **Visualisation** is an inherent part of the modelling process, as shown in the previous graphics.

See also - [More about visualisation](#)

## A simple tutorial

This tutorial is aimed at familiarising you with the main features of *Potent* as quickly as possible. It takes you through a very simple forward and inversion modelling exercise without giving any in-depth explanations along the way. It is concerned mainly with pointing you at the relevant operations.

Run Potent

Import observations from an XYZ file

Visualise the observed values

Creating a subset

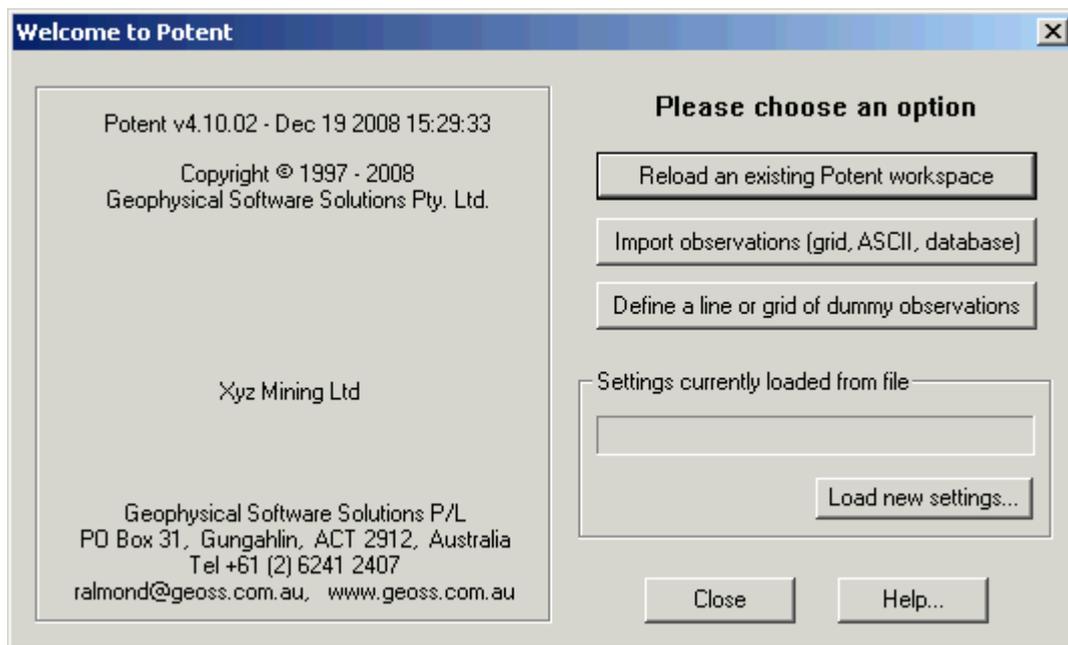
The Model

More about visualisation

Inversion modelling

## Run Potent

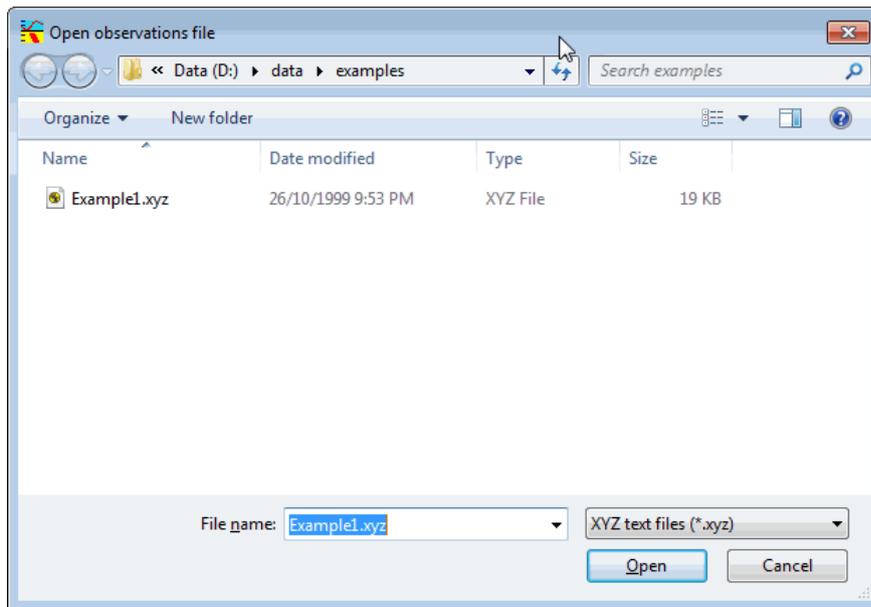
To make sure that you have a “clean” environment, exit *Potent* if it is already running and then run it again. You should see a window similar to this:



Back to [A Simple Tutorial](#)

## Import observations from an XYZ file

We want to import observed data from an XYZ file, so click the button labelled **Import observations...** You will be presented with a standard Windows file-chooser dialog. You need to locate a file named *Example1.xyz* that came with your *Potent* installation disk. By default it will be in a subfolder of the folder into which *Potent* was installed. (We suggest you make a working copy of this folder and its contents somewhere in your Documents area.) This file contains some synthetic magnetic data. Once located the file dialog should look something like this:

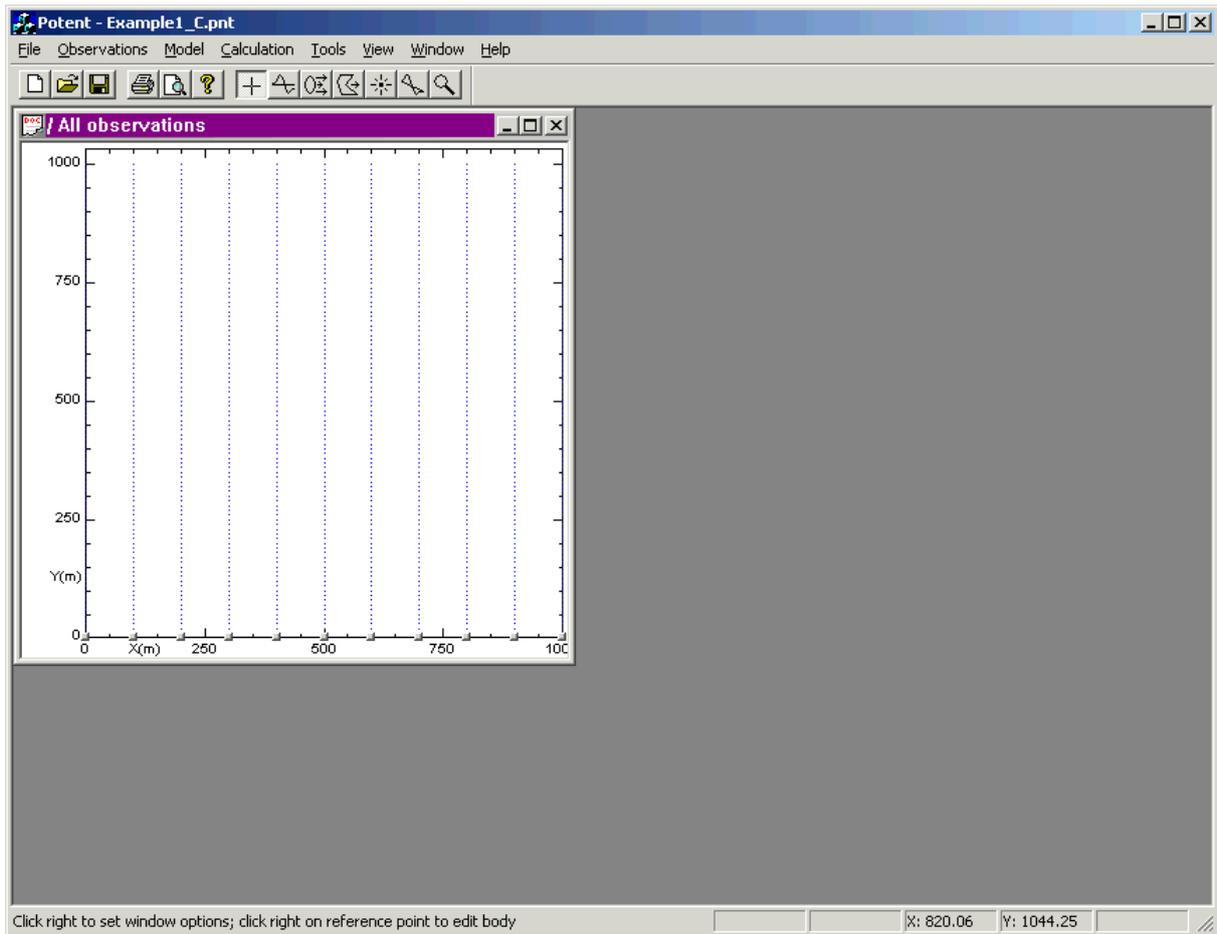


Click **Open** to open the file. *Potent* displays the **Options for reading observations** dialog box (see below) which allows you to specify the format of the data in some detail. The bottom group of controls in the dialog box contains a **Preview** window, which displays the first few lines of the file. Firstly, select **Use smart preview**, then use the mouse to set up the rest of the dialog box as shown below. (Using the mouse animates the procedure, which makes things clearer.) Refer to [Using smart preview](#) in the User's Guide for more information.

You can see that channel 1 contains X (in metres), channel 2 contains Y, channel 3 contains Z (0 in this case) and channel 4 contains the observed TMI field (in nanotesla). These channels are reflected in the dialog box **Column position** settings. (Notice that this is a **Free field** format file and so **Start col** and **End col** are disabled.)

Make sure that **Subtract averages from field values** is cleared, as we know the data is synthetic and contains no background field.





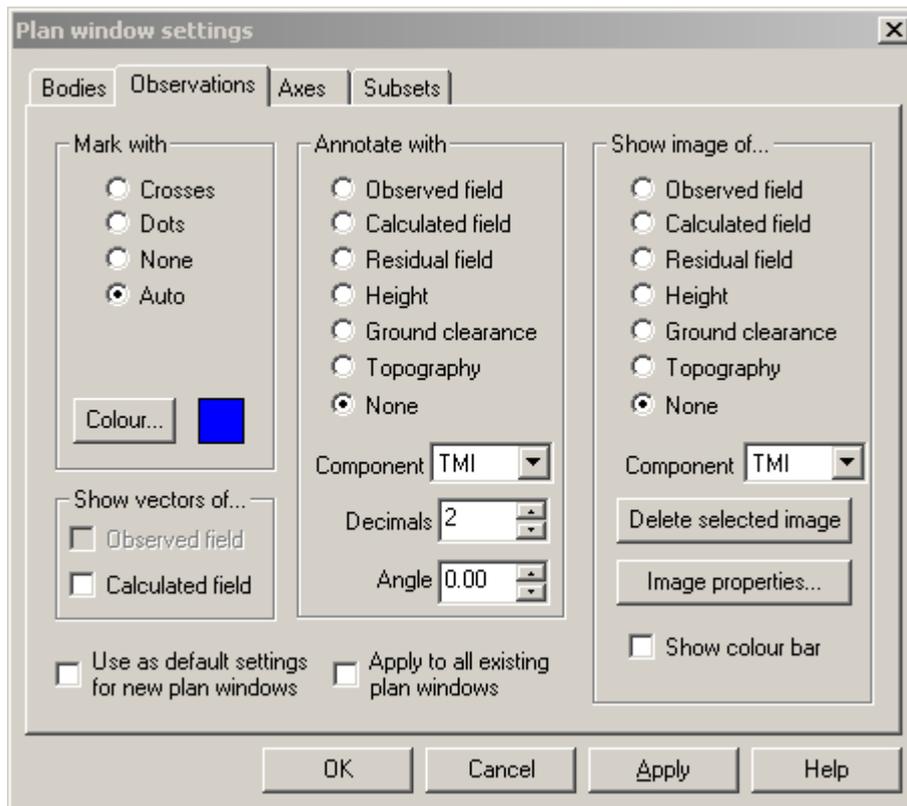
Back to [A Simple Tutorial](#)

## Visualise the observed values

Right click anywhere on the Plan window and click **Options** on the popup menu (equivalent to clicking **Options** in the **Window** menu).



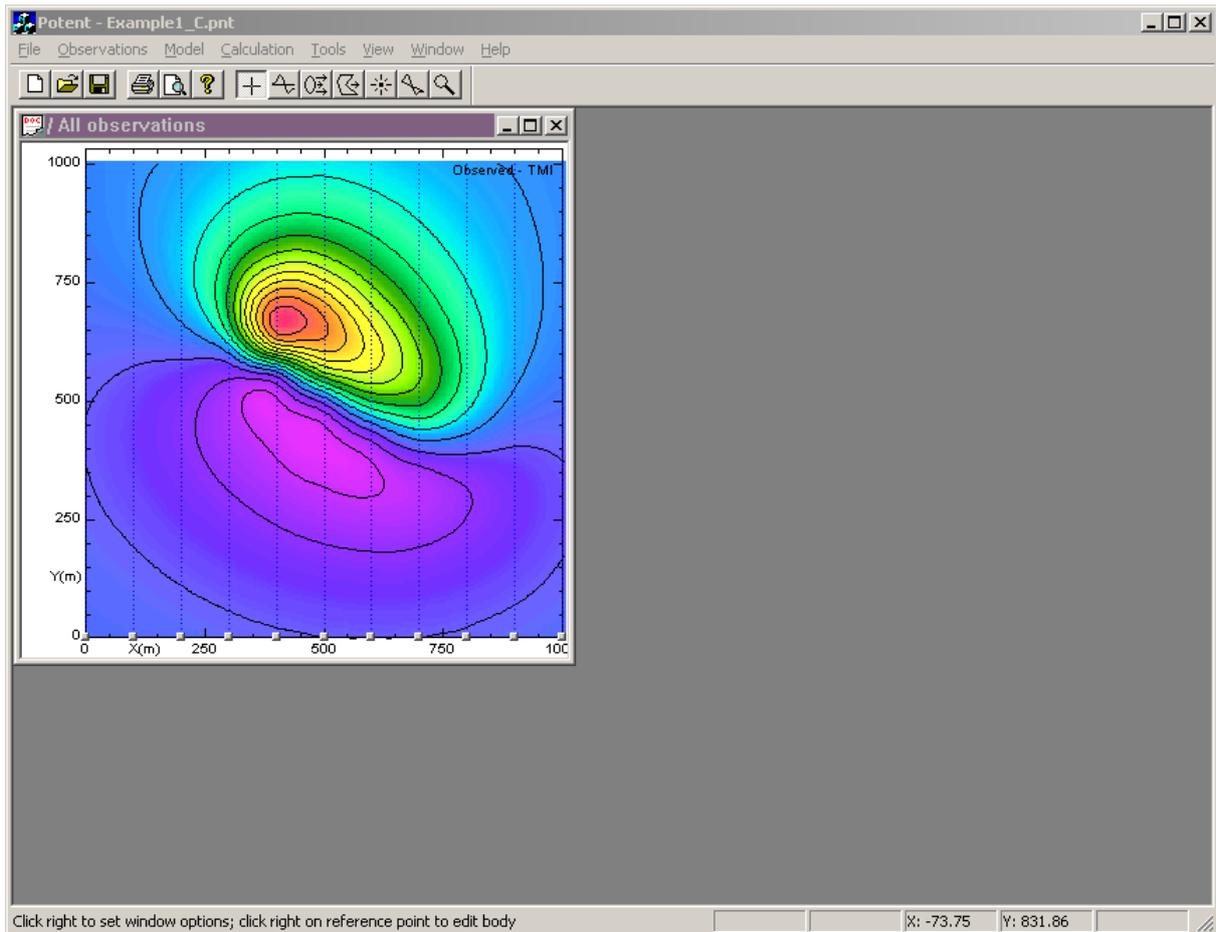
Select the **Observations** tab of the **Plan window settings** dialog box.



In the **Show image of** group, select **Observed field** .



Click **OK**. The Plan window should now look like this:

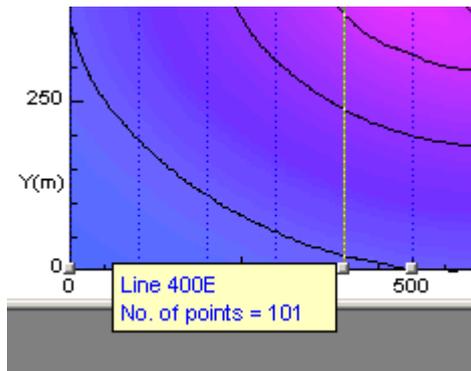


You now have an image of the observed field. As you move the mouse cursor over the image the status bar displays the (X,Y) coordinates and the underlying interpolated field value, F.

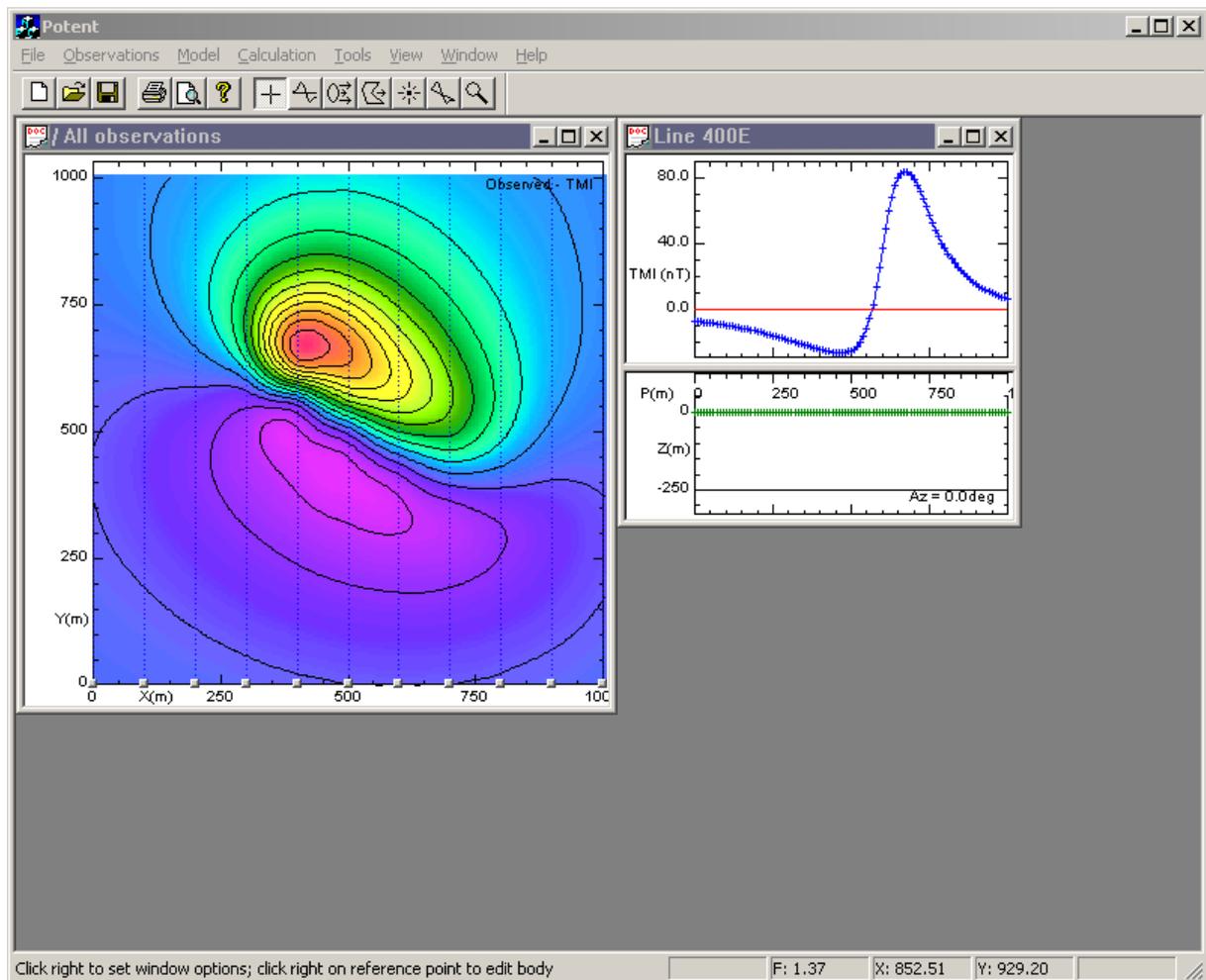


(The wobbly nature of some of the contours in the example makes this a good place to point out that Potent uses a very fast minimum curvature gridding algorithm. The grid, and its associated contours, are present to help you to visualise the interpretation process, and are not intended to compete with the output of specialised gridding applications. In particular, the interpolated grid points are not used in any modelling calculations; these are always done at the observed data locations.)

It is usually desirable to look at the data as one or more profiles. When the data has been imported from an XYZ file the original line structure is already present. These lines are the simplest sections to display in profile form. We shall display the N-S profile for line 400E. If you hover the mouse cursor on the small grey button that marks the end of each line a small popup window is displayed that contains basic information about the line. The path of the line is drawn temporarily on the plan.



Having identified line 400E click the left mouse button on the line marker button. A Profile window is created alongside the Plan window. Resize the profile window so that your configuration looks like this.



This Profile window consists of two panes.

- The top pane displays the original observed field values in profile form, drawn as blue crosses in this case. (The red line is the calculated field. As there currently is no model, these values are all zero.) The horizontal axis P (for “profile”) is the distance in metres from the start of the profile. The vertical axis is the observed TMI in nanotesla. (Note that the interpolated values that form the image on the plan are not used. The only function of the image is to help guide you around the really important entities, the observed values.)

- The bottom pane is the cross-section pane. The vertical axis, Z, is height and so the negative values represent depth. The window essentially contains a cross-section of the sub-surface, and will eventually contain a representation of the model. The green crosses are the observation points, which are all at height 0m. The cross-section pane uses the same horizontal axis as the profile pane.

At this point we could click on further line marker buttons on the Plan to open more N-S profiles. However, we shall look at a more general way of creating a profile.

Back to [A Simple Tutorial](#)

## Creating a subset

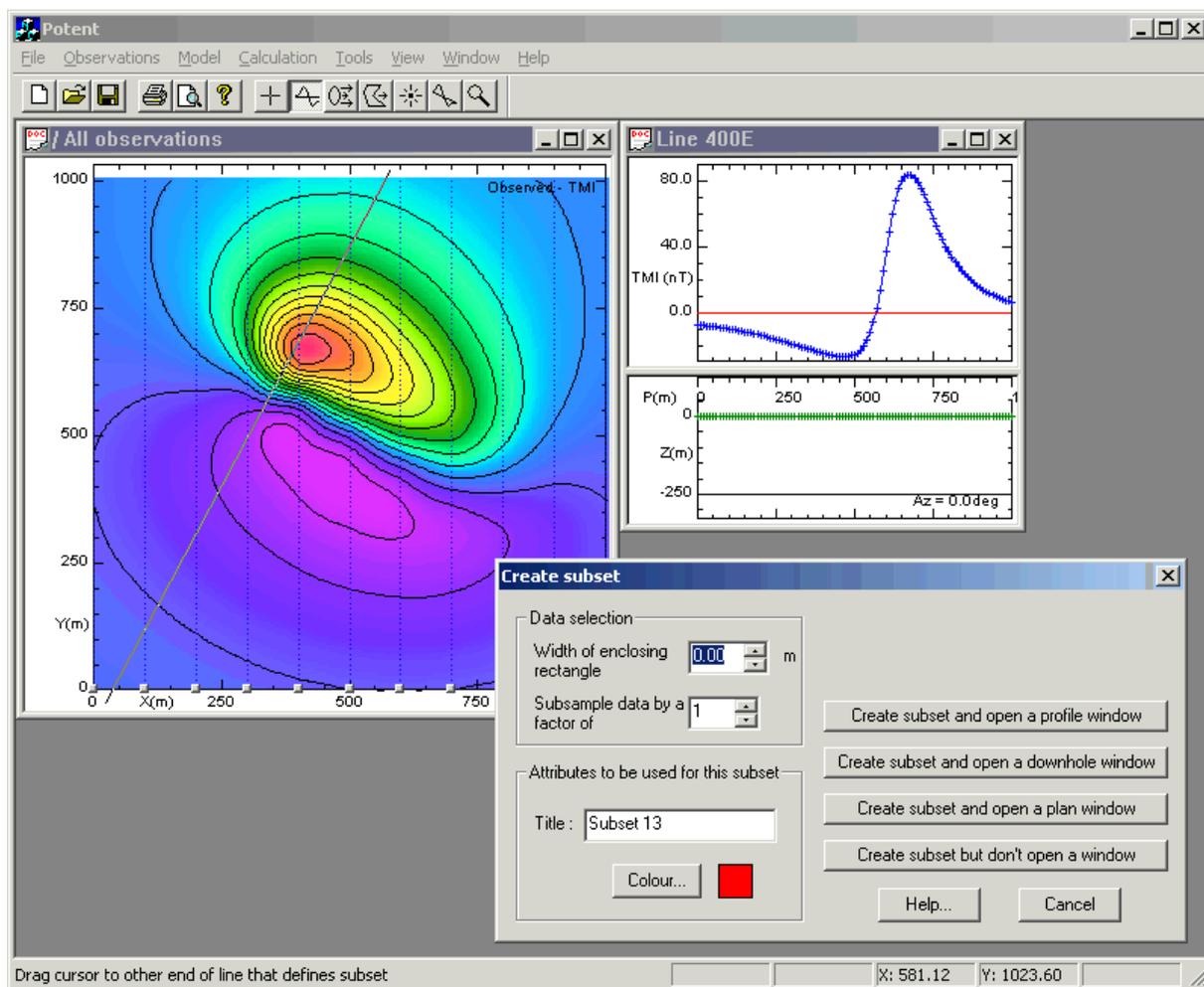
The **subset** is one of the key concepts in *Potent*. In complex or large datasets it is important to be able to home in on features of interest. We can use the current very simple example dataset to demonstrate the subset concept by creating a profile at an oblique angle across the anomaly shown in the Plan.

(For a long, narrow subset the **Subset – box** tool is most appropriate. However, remember that the **Subset – polygon** tool allows you to create a subset of arbitrary polygonal shape.)

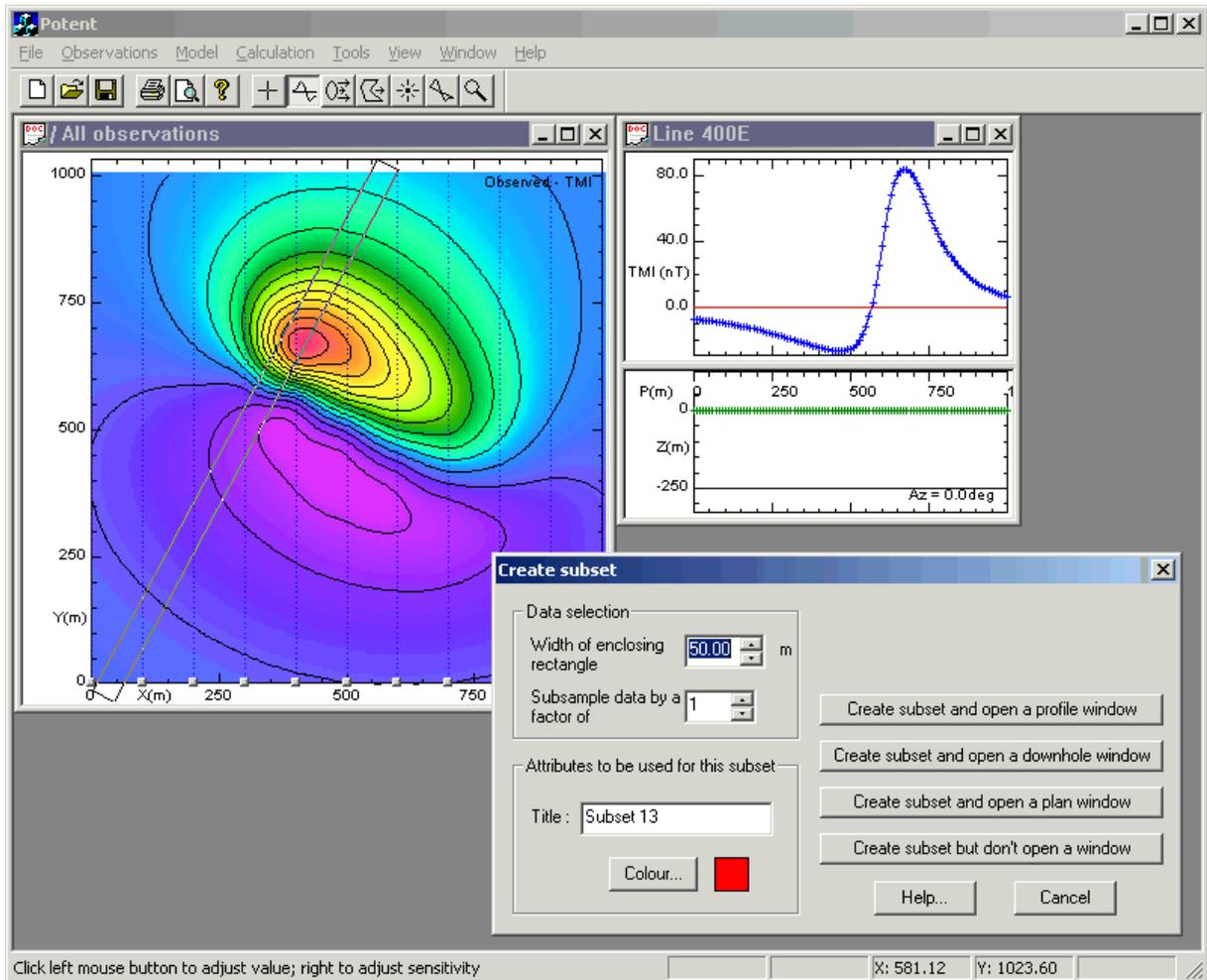
From *Potent's* **Tools** toolbar (or by dropping down the **Tools** menu) select the **Subset - box** tool. The mouse cursor changes to the shape indicated on the toolbar button.



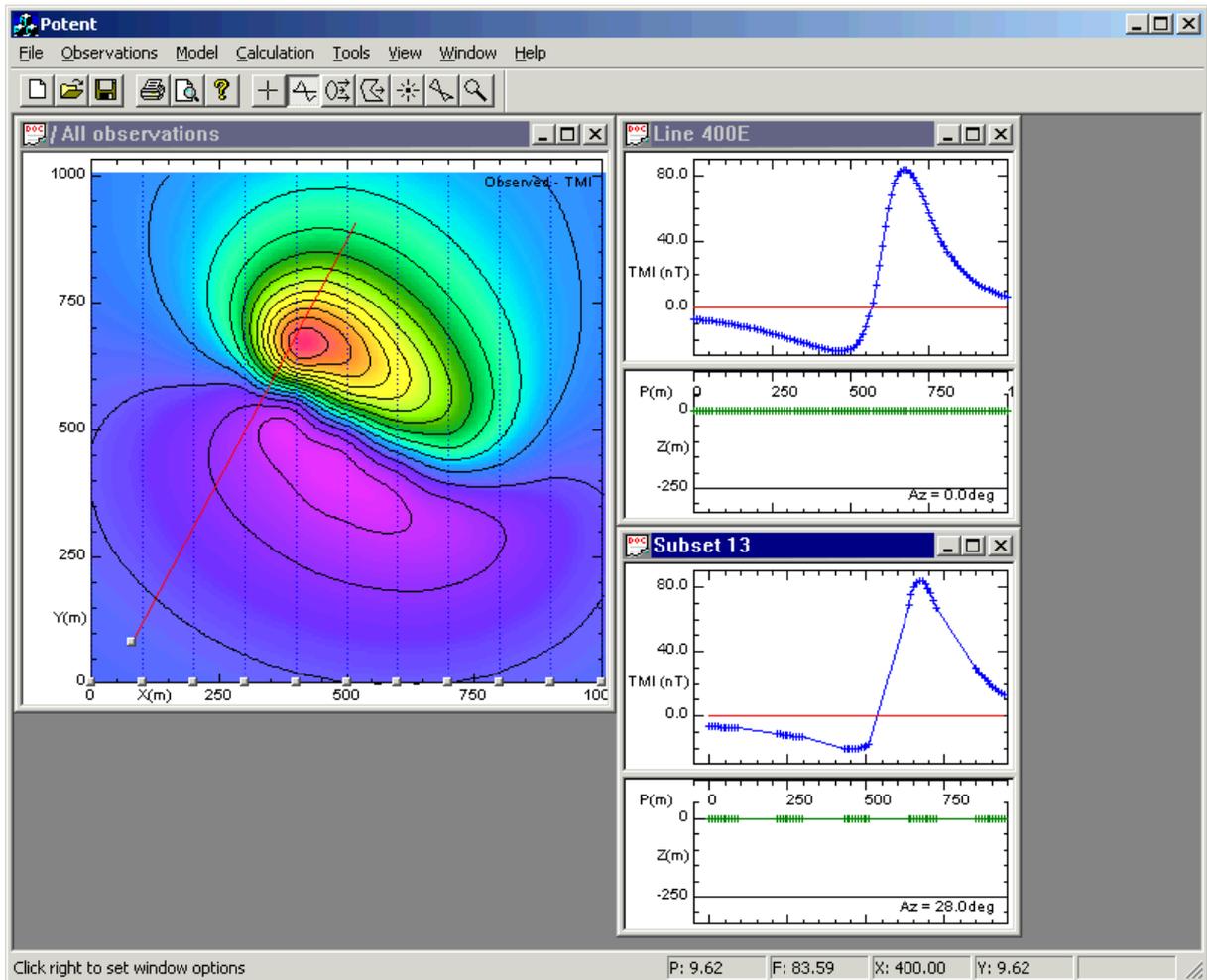
Using this cursor, drag a line diagonally across the Plan as shown. (Press the left mouse button at the start of the line. While holding the button down, drag the cursor to the other end of the line and then release the button.) The **Create subset** dialog box is displayed. If you make a mistake just cancel the dialog box and try again.



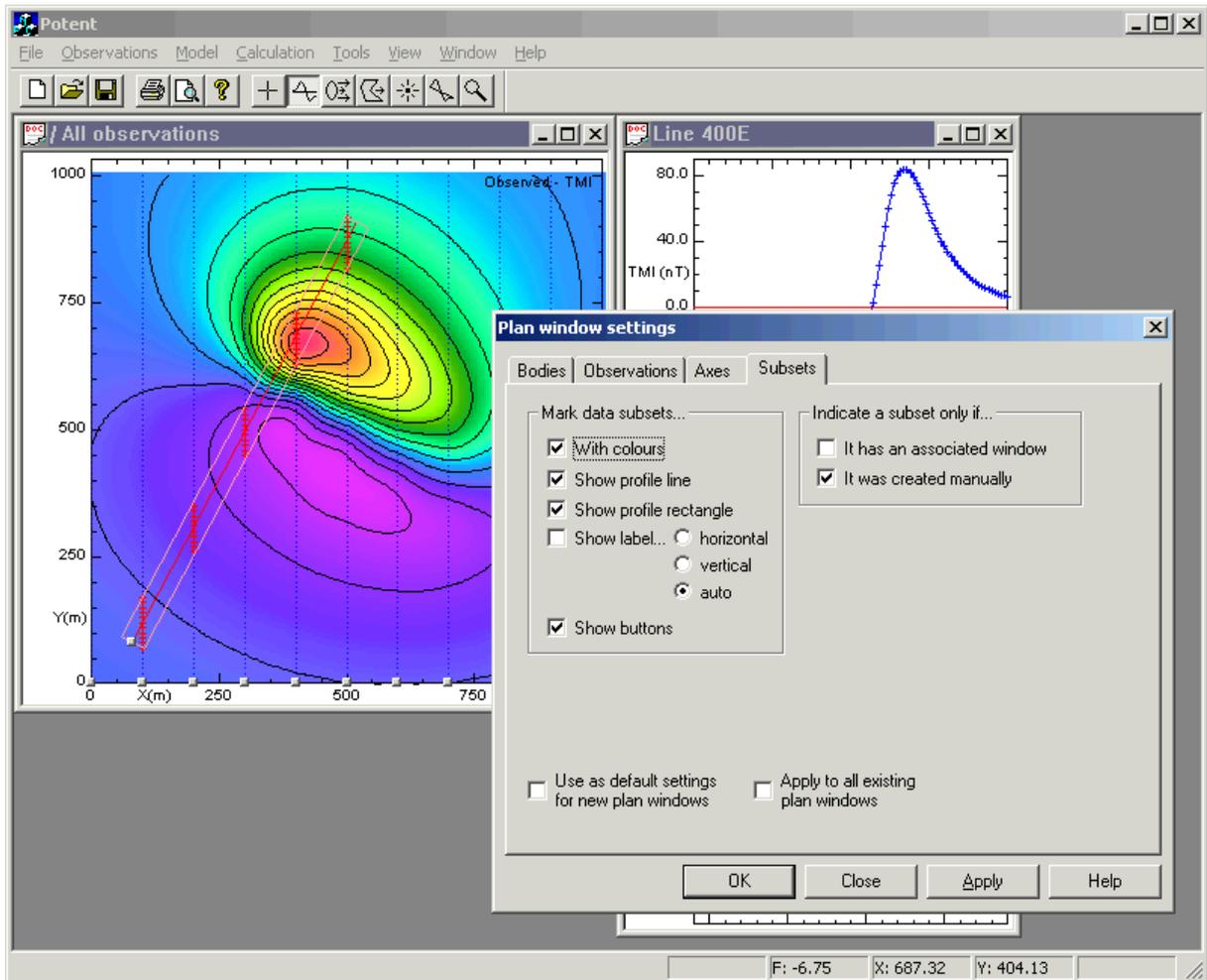
You define a subset by specifying a rectangular selection of the data. Click on the small UP arrow attached to the edit box labelled **Width of enclosing rectangle**. The line you dragged with the mouse expands into a rectangle. For the purpose of this exercise make the width 50m. Your screen should look like this.



In this case the subset is going to be long and narrow. The subset is simply a sample of the observed data (not the interpolated points underlying the image). You can choose to display it in a **Plan**, **Profile** or **Down-hole** window initially, or not display it at all, by choosing the appropriate button in the dialog box. In this case we want a profile window, so click on the button labelled **Create subset and open a profile window**.

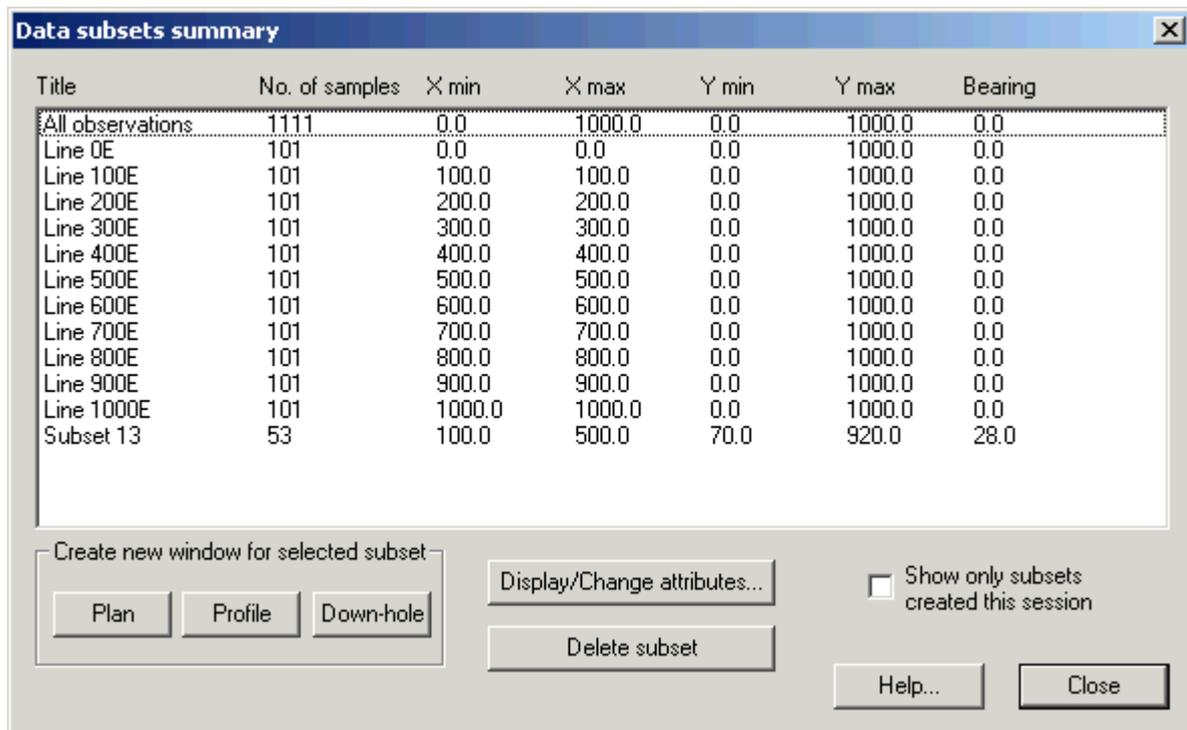


The new window should have positioned itself neatly below the previous profile window. It contains clusters of points from each of the lines of observations that were covered by the rectangular area you created. You can see this more clearly by altering the plan slightly. Right click anywhere on the Plan window and click **Options** on the popup menu. Select the **Subsets** tab of the **Plan Window Settings** dialog box. In the **Mark data subsets** group select **With colours** and **Show profile rectangle**. Click the **Apply** button and the Plan window should look like this.



(Click **Close** to dispose of the dialog box.) The rectangle on the Plan corresponds to the one you created when you defined the subset. The red crosses lie within it and represent the observation points that are included in the subset. You can see that the clusters of points in the new profile window result from the projection of the points sampled from each data line onto the plane of the profile.

Click **Subset management** on the **Observations** menu to display the **Subset management** dialog.



You can use this dialog box to view all the subsets for the current dataset. The first subset is always **All observations** (i.e. the entire dataset). In this example there are also 11 predefined subsets corresponding to each line defined in the XYZ file. The last subset (Subset 13) is the one we created. Note that the first profile window did not create a new subset; it merely displayed the predefined **Line 400E** subset. Using the **Plan**, **Profile** and **Down-hole** buttons you can display a subset in as many different windows as you wish. Click **Close** to dispose of the dialog box.

[Back to A Simple Tutorial](#)

## The Model

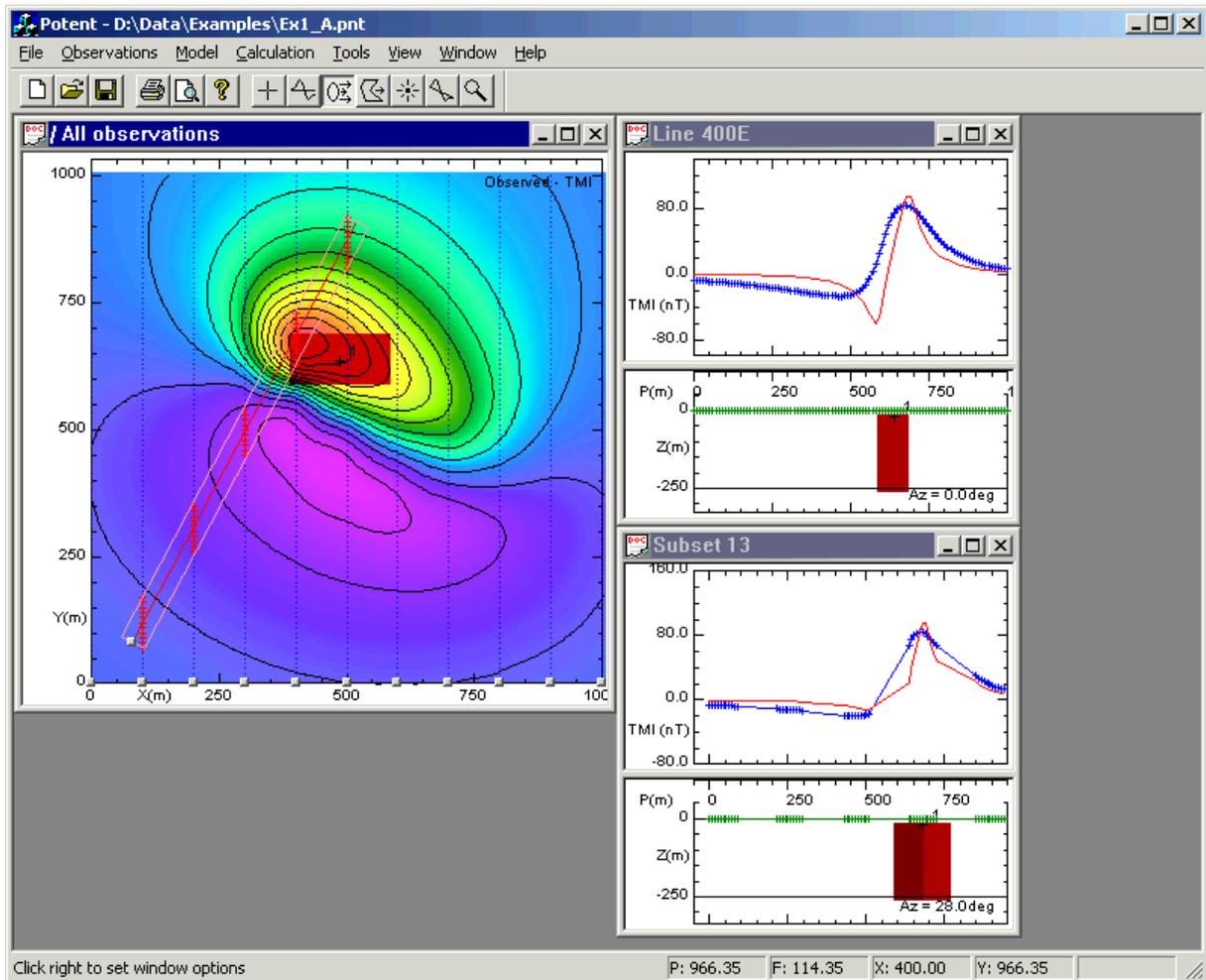
We shall now create a model that is consistent with the observed data. Remember that the observations are synthetic and that no geological realism is suggested!

(The example given here uses a tabbed dialog box to edit the model. Note that the **Composite Edit Body** dialog box provides identical functionality in a single dialog box. However, it does require a large display, or a dual monitor system. Refer to the description of the Show composite Edit Body dialog command in the Reference Manual for more information.)

Ensure the Plan window is the **active window** by clicking on its title bar. On the **Bodies** toolbar, click **Rectangular prism**.

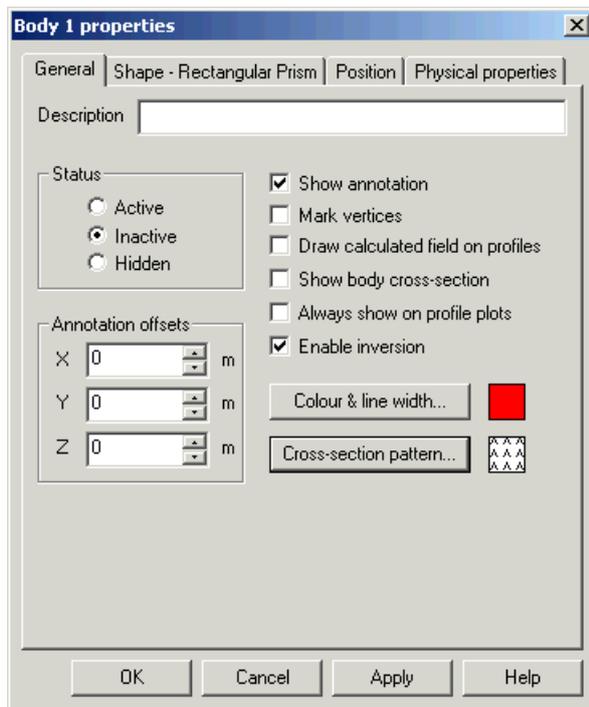


This creates a body with the shape of a rectangular prism near the centre of the Plan window. Notice that you can also see the body on the profile windows.

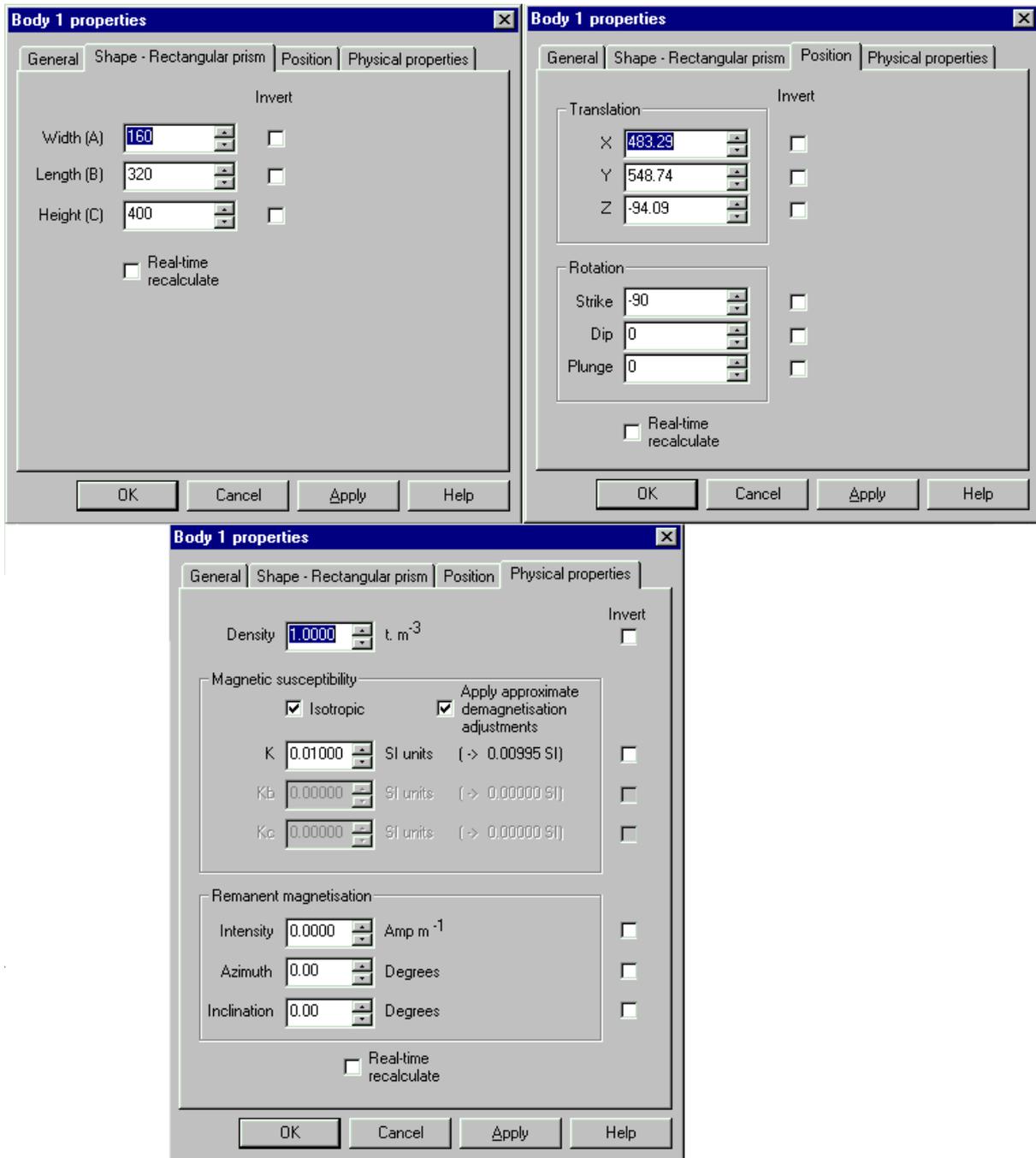


*Potent* assigns the body a default set of parameters (shape, position, physical properties). The first thing you must do when you create a body is to make these parameters more sensible. Also note that the calculated field is still zero. The body is **inactive**, and you must explicitly make it **active**.

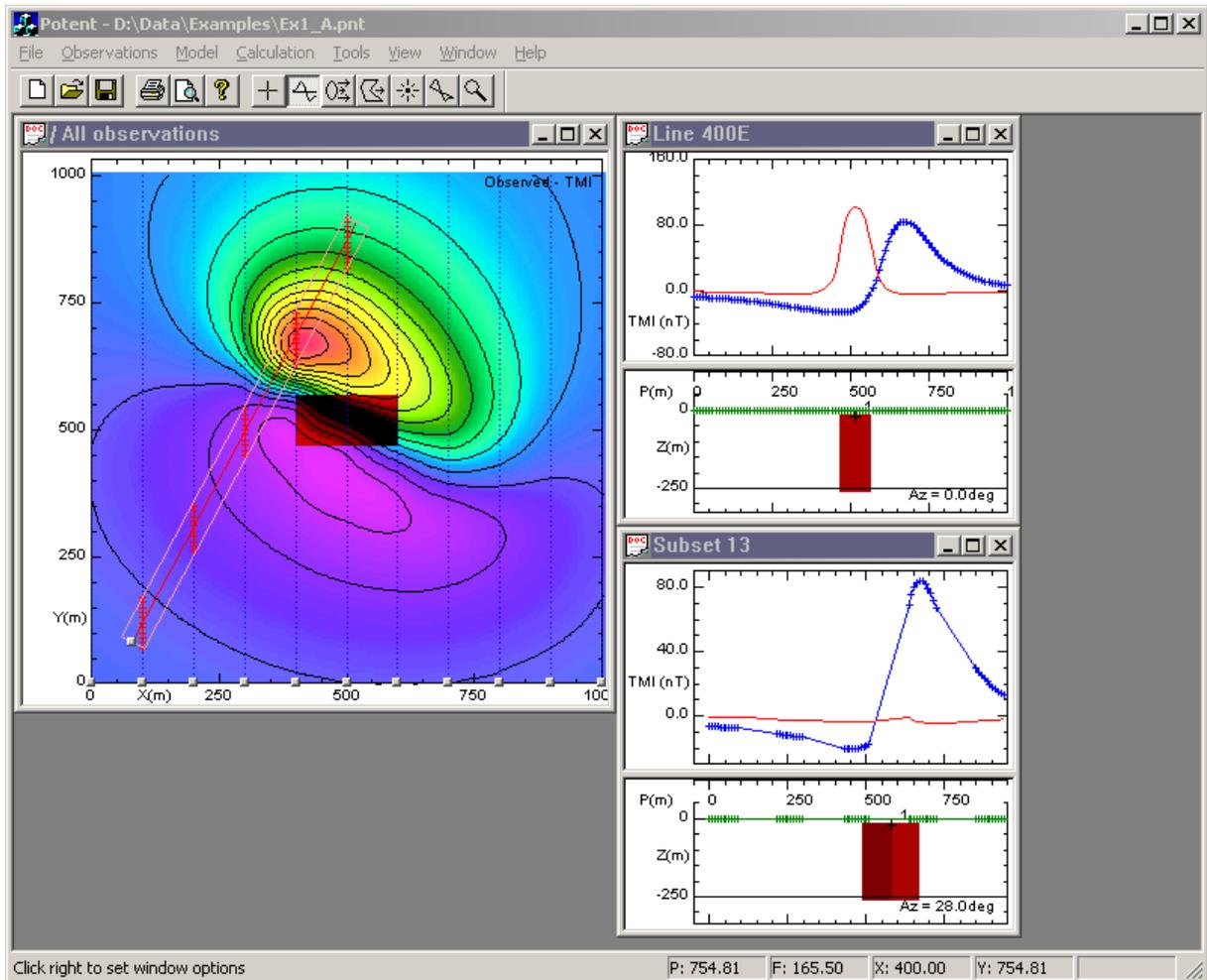
Edit the body either by choosing the **Edit model** command from the **Model** menu or, more easily, by clicking the right mouse button on the small cross that marks the top centre of the body (any window will do). The **Edit body** tabbed dialog box is displayed.



Notice that this dialog box has tabs for **Shape** (which depends on the type of the body), **Position** (including rotation), and **Physical properties** (including remanent magnetisation), giving you complete control over the body parameters. Although we will not use them just now, the dialog boxes corresponding to these three tabs look something like this.



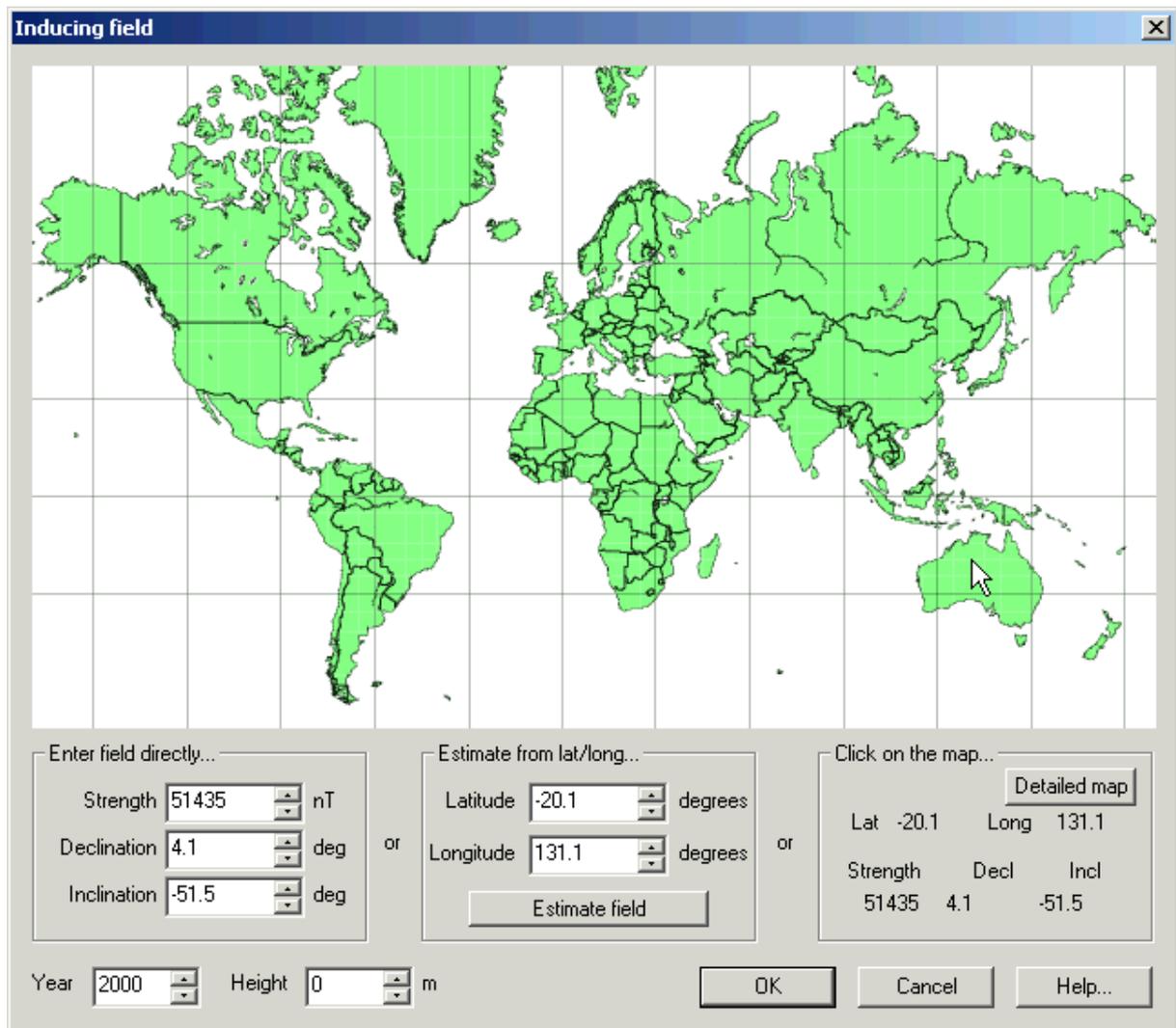
In the **General** tab, set the body's **Status** to **Active** and click **OK**. The body is drawn in its assigned colour, and its calculated field is displayed on the profiles.



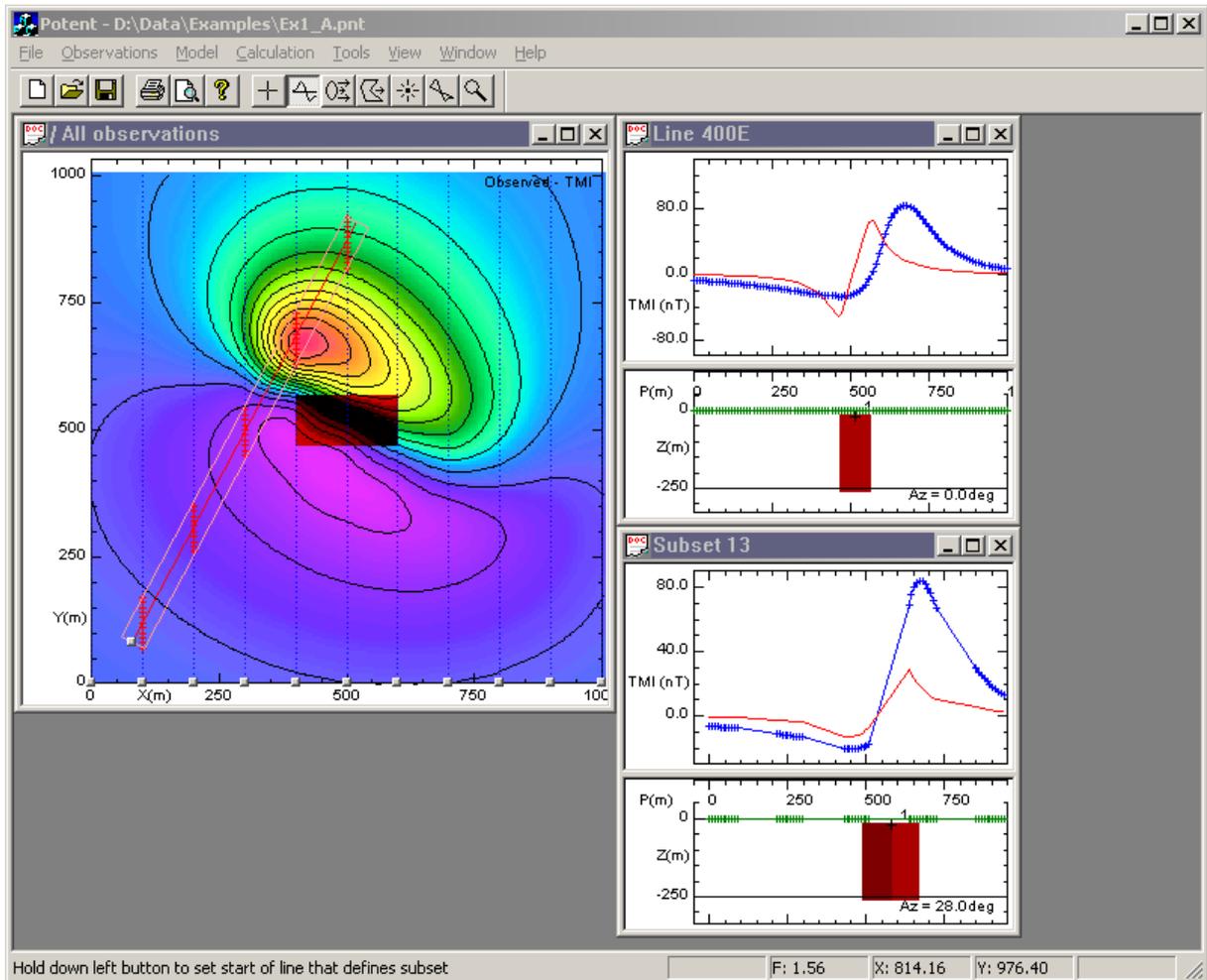
(Some systems have problems rendering the bodies as solid objects. If at this point the body is not drawn as a solid object as shown above, click **Always draw bodies as wireframe** on the **Model** menu. The solid body will be replaced by a wireframe drawing that will always be drawn correctly.)

Magnetic interpretation experts will observe that the calculated field displayed on the profiles is that for the north magnetic pole. However the synthetic dataset was generated for the centre of the Northern Territory of Australia, latitude 19S, longitude 134E. We obviously need to set the appropriate inducing field.

Click **Inducing field** on the **Model** menu to display the **Inducing field** dialog box, where you can set the IGRF for your survey area.



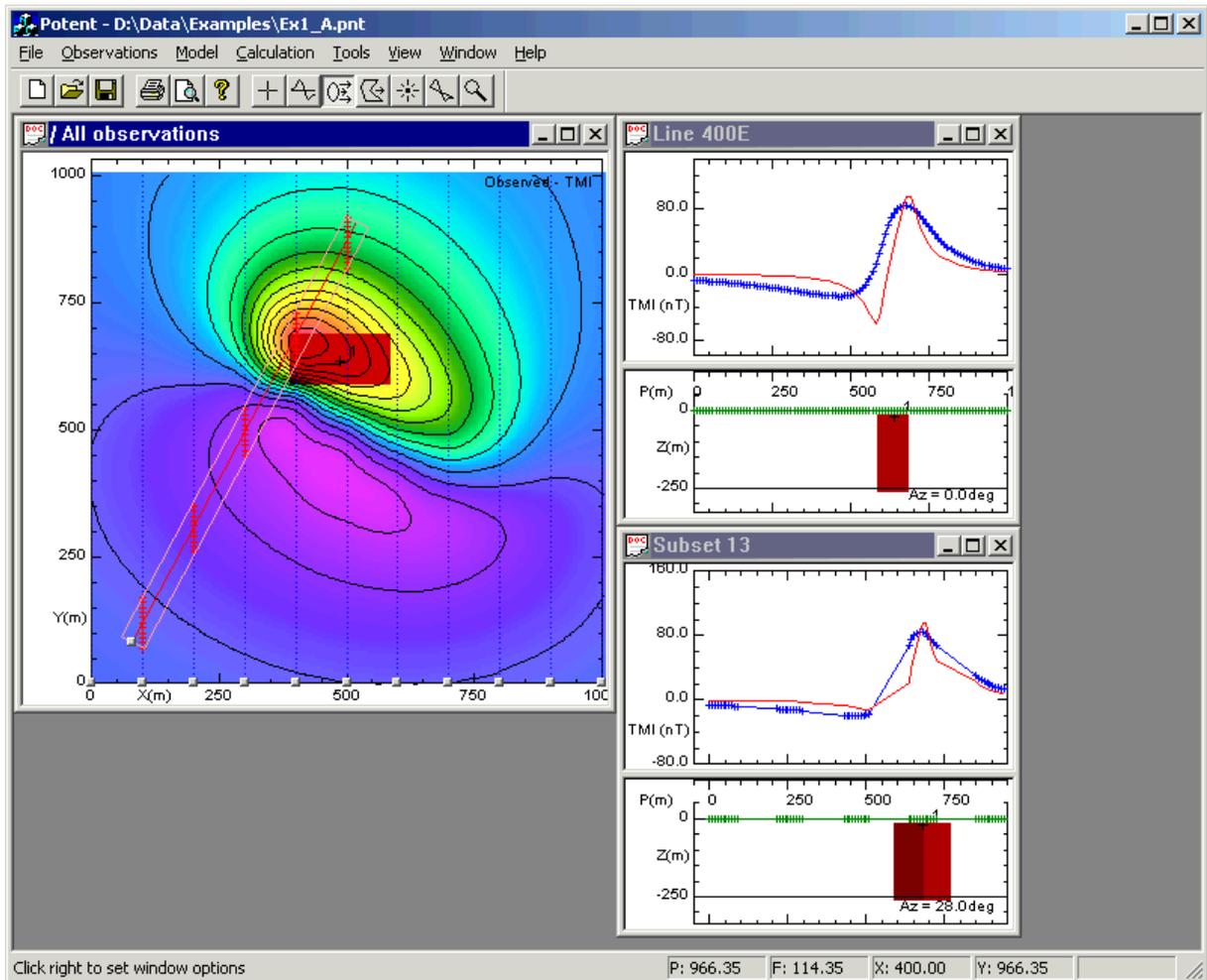
You can set the parameters in various ways (refer to the **Model | Inducing field** command in the reference Manual for details). For the purpose of this example, click on the map in northern Australia (roughly where the cursor is shown) to put the field values in the **Strength**, **Declination** and **Inclination** boxes. Then click **OK** and the calculated field is updated on the profiles.



Click **Move bodies** on the **Tools** menu or the **Tools** toolbar.



On a profile window, position the new mouse cursor over the small cross that marks the top centre of the body. The cursor changes to a small square. Hold down the left mouse button and drag the body around the window. Notice that with a Profile window you can move the body vertically, and horizontally along the line of the profile. Notice also that as you drag the body around its coordinates are displayed in the status bar. Drop the body at a depth of around 100m (ie.  $Z = -100$ ). Try dragging the body around on the Plan. This time you can alter its X and Y position, but not its depth. Put the body into a position something like this.



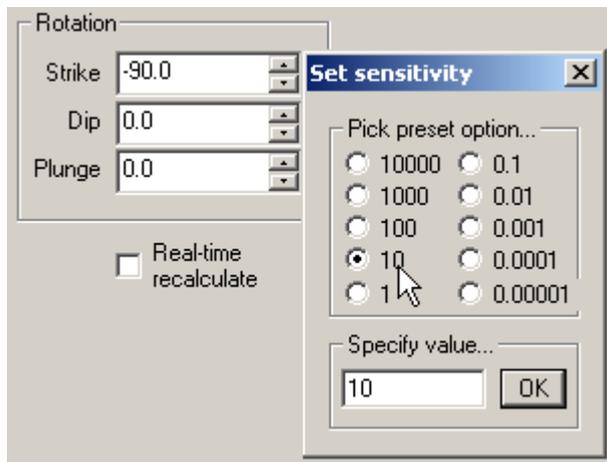
Display the **Edit body** dialog box again (by right clicking on the cross marking the top of the body). Select the **Position** tab.

The screenshot shows the 'Body 1 properties' dialog box, specifically the 'Position' tab. The dialog box has four tabs: General, Shape - Rectangular prism, Position, and Physical properties. The 'Position' tab is active, showing the following settings:

- Translation:**
  - X: 483.29
  - Y: 548.74
  - Z: -94.09
- Rotation:**
  - Strike: -90.0
  - Dip: 0.0
  - Plunge: 0.0
- Invert:** Three checkboxes are present, all of which are unchecked.
- Real-time recalculate:** An unchecked checkbox.

Buttons at the bottom include OK, Cancel, Apply, and Help.

Experiment with adjusting the translation and rotation parameters with the small arrows attached to the edit controls. Note that you can change the sensitivity of the arrows by right clicking them. In this example the sensitivity of the Strike adjustments will be set to 10 degrees per click.

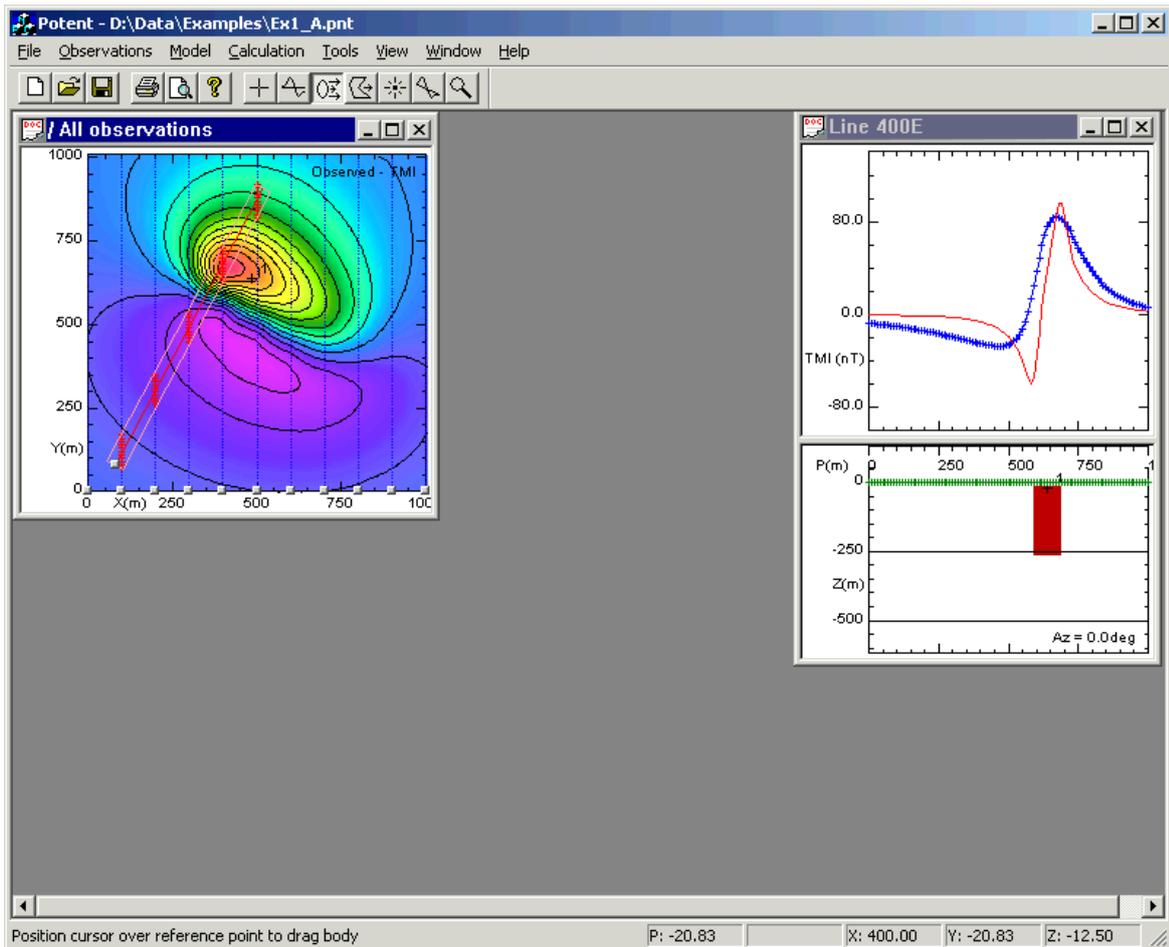


Back to [A Simple Tutorial](#)

### More about visualisation

Visualisation is an inherent part of using *Potent* – you can't use the program without doing some visualisation even if it is just displaying a single profile. In the next exercise we'll look at some more advanced ways of visualising your modelling session.

First of all make some room on the screen by closing the diagonal profile window in the usual Windows way and then reorganising the other windows so that they look something like this.



On the **Observations** menu, click **Subset management**.

The screenshot shows the "Data subsets summary" dialog box. It contains a table with the following data:

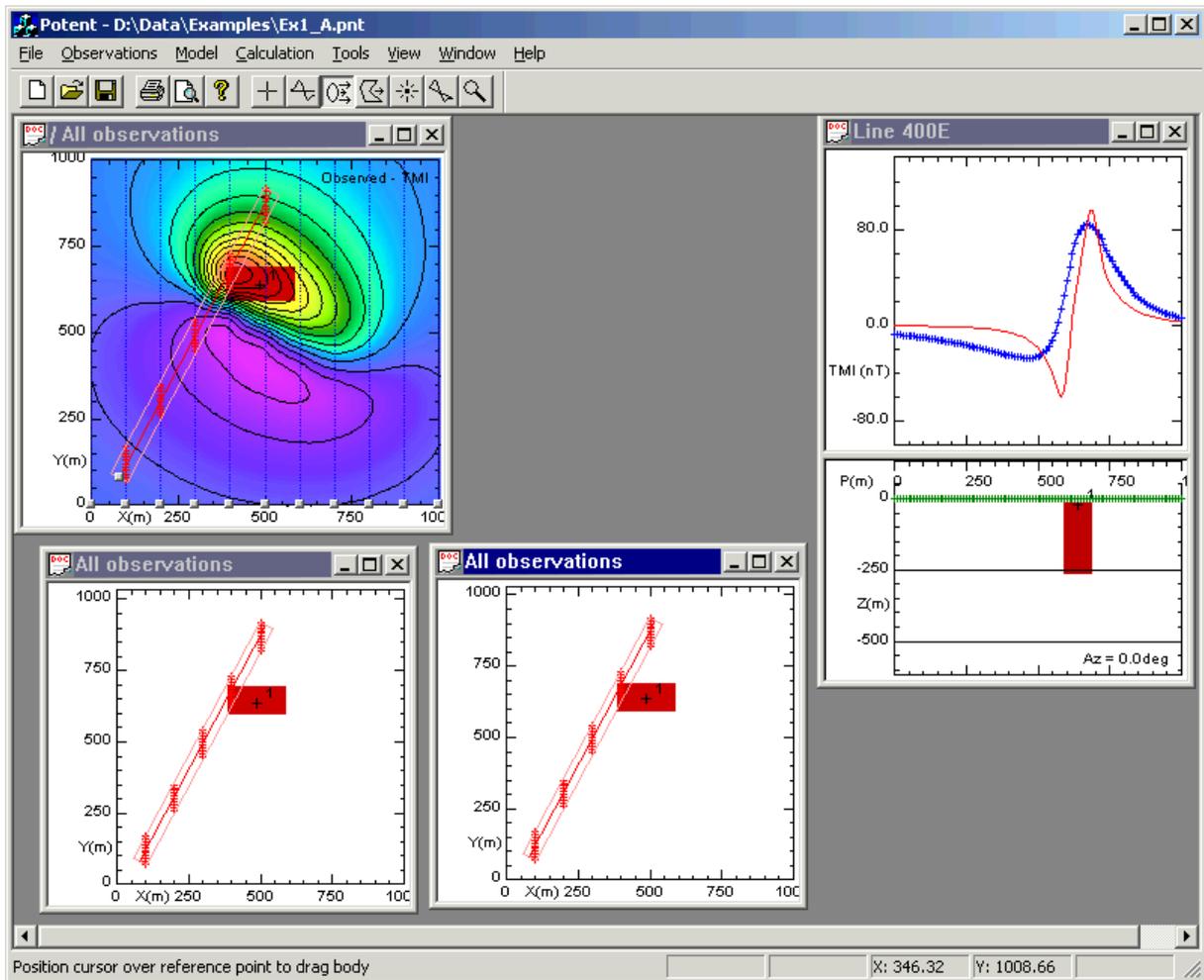
Title	No. of samples	X min	X max	Y min	Y max	Bearing
All observations	1111	0.0	1000.0	0.0	1000.0	0.0
Line 0E	101	0.0	0.0	0.0	1000.0	0.0
Line 100E	101	100.0	100.0	0.0	1000.0	0.0
Line 200E	101	200.0	200.0	0.0	1000.0	0.0
Line 300E	101	300.0	300.0	0.0	1000.0	0.0
Line 400E	101	400.0	400.0	0.0	1000.0	0.0
Line 500E	101	500.0	500.0	0.0	1000.0	0.0
Line 600E	101	600.0	600.0	0.0	1000.0	0.0
Line 700E	101	700.0	700.0	0.0	1000.0	0.0
Line 800E	101	800.0	800.0	0.0	1000.0	0.0
Line 900E	101	900.0	900.0	0.0	1000.0	0.0
Line 1000E	101	1000.0	1000.0	0.0	1000.0	0.0
Subset 13	53	100.0	500.0	70.0	920.0	28.0

Below the table are several control buttons and options:

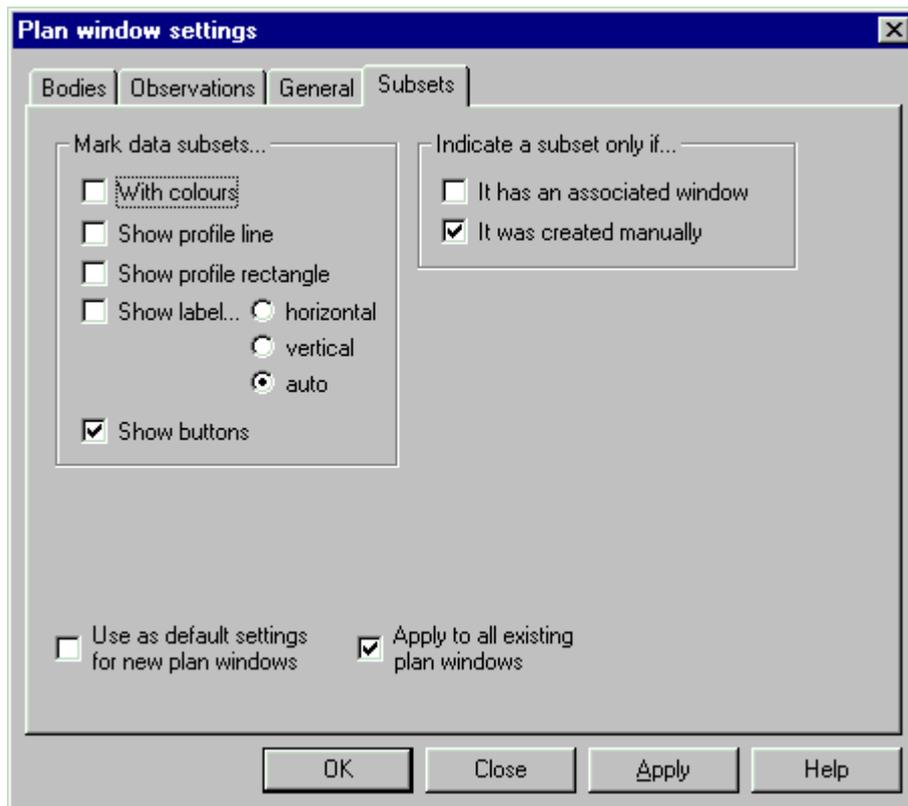
- Buttons: Plan, Profile, Down-hole, Display/Change attributes..., Delete subset, Help..., Close.
- Checkbox:  Show only subsets created this session.
- Text: Create new window for selected subset.

Remember that the first subset represents the entire dataset. Select it and then click **Plan**

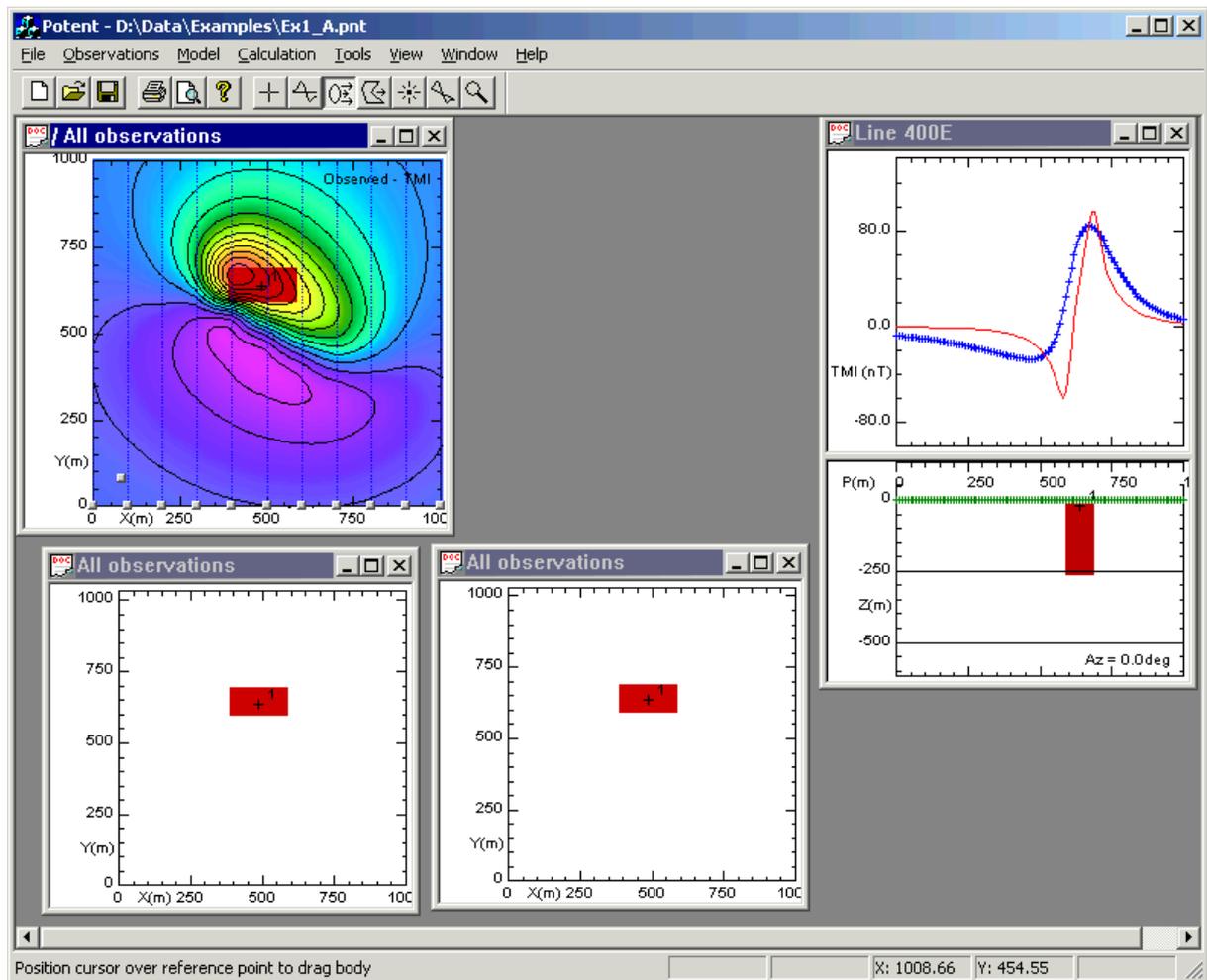
twice to create two new Plan windows. Click **Close** to dismiss the dialog box. Reorganise the two new Plan windows like this.



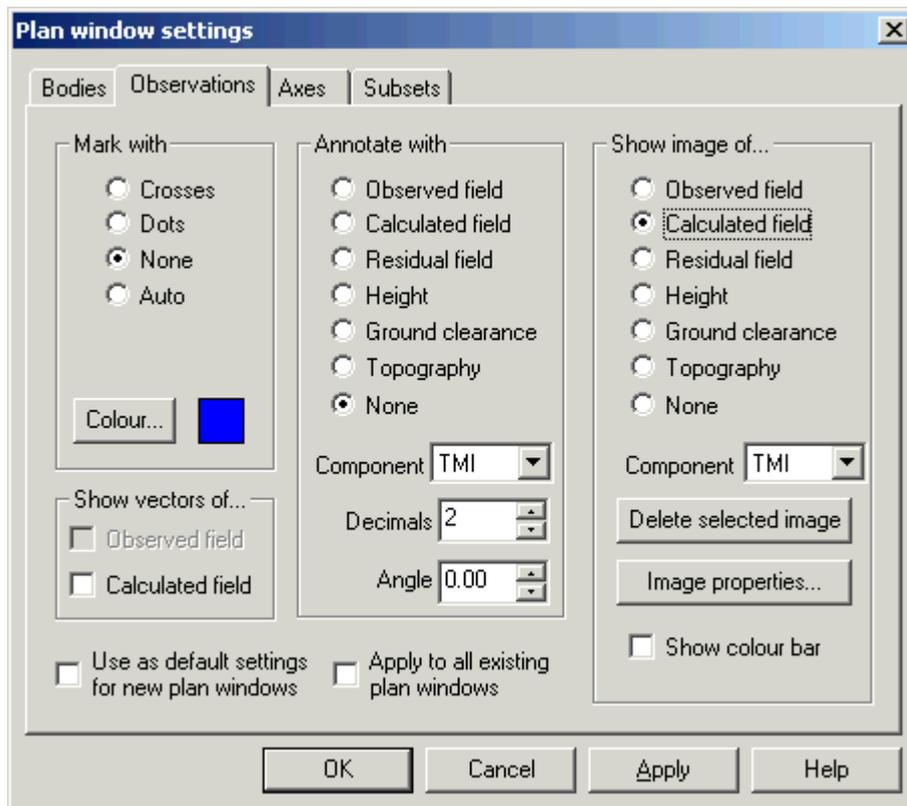
Right click on one of the new Plan windows and click **Options** on the popup menu to display the **Plan Window Settings** dialog box. Set up the **Subsets** tabbed page as follows (don't forget to select **Apply to all existing plan windows**):



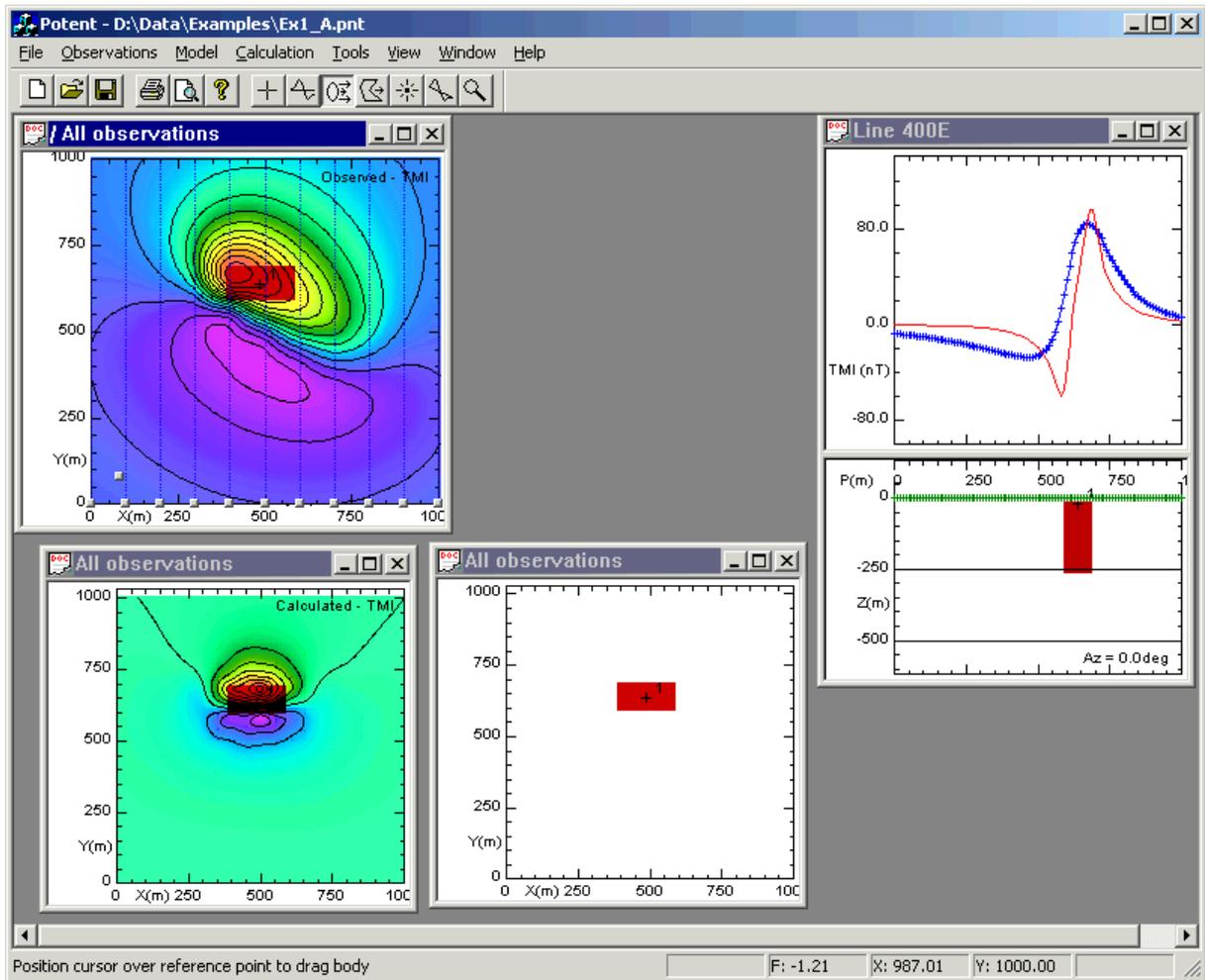
Click **OK**. *Potent* applies the settings you have specified to all the plan windows. In this case it means that the subsets are not marked.



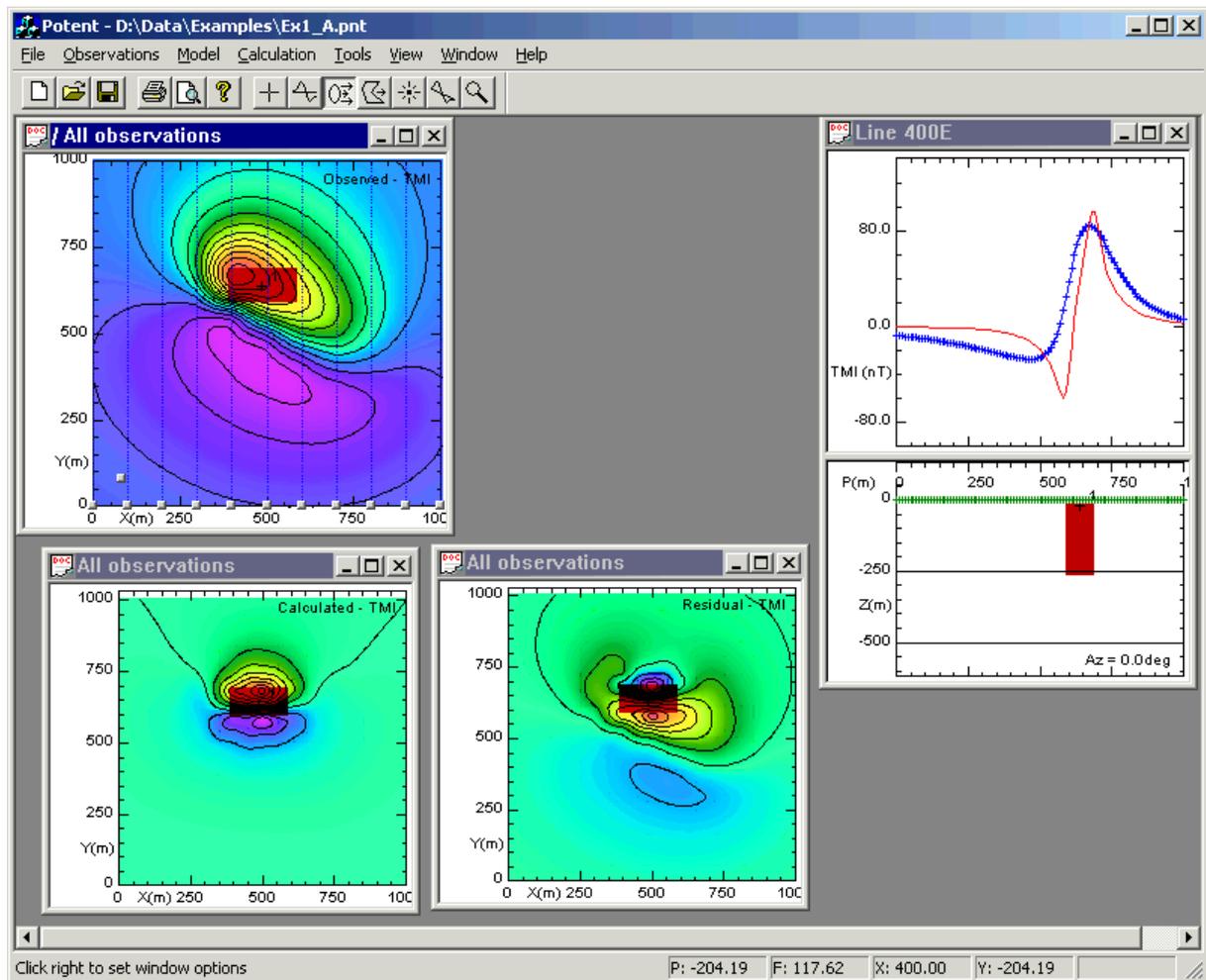
Right click on the new Plan window at the bottom left and click **Options** on the popup menu to display the **Plan Window Settings** dialog box. Click the **Observations** tab, then click **Calculated field** in the **Show image of** group.



Click **OK** and the image of the calculated field appears on the plan.



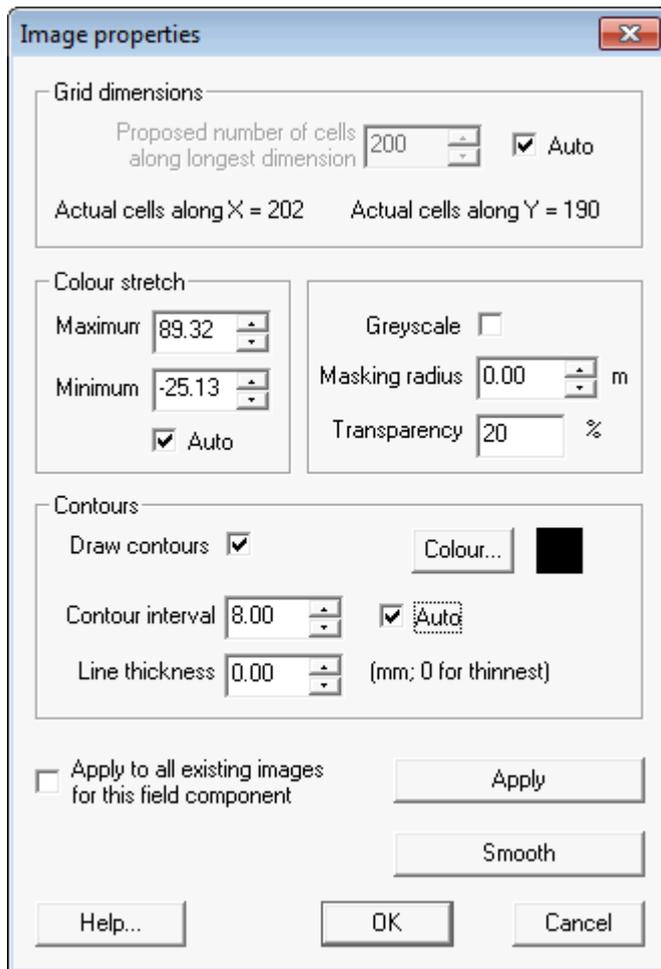
Using the same technique, display the image of the residual field (observed minus calculated) in the remaining plan window.



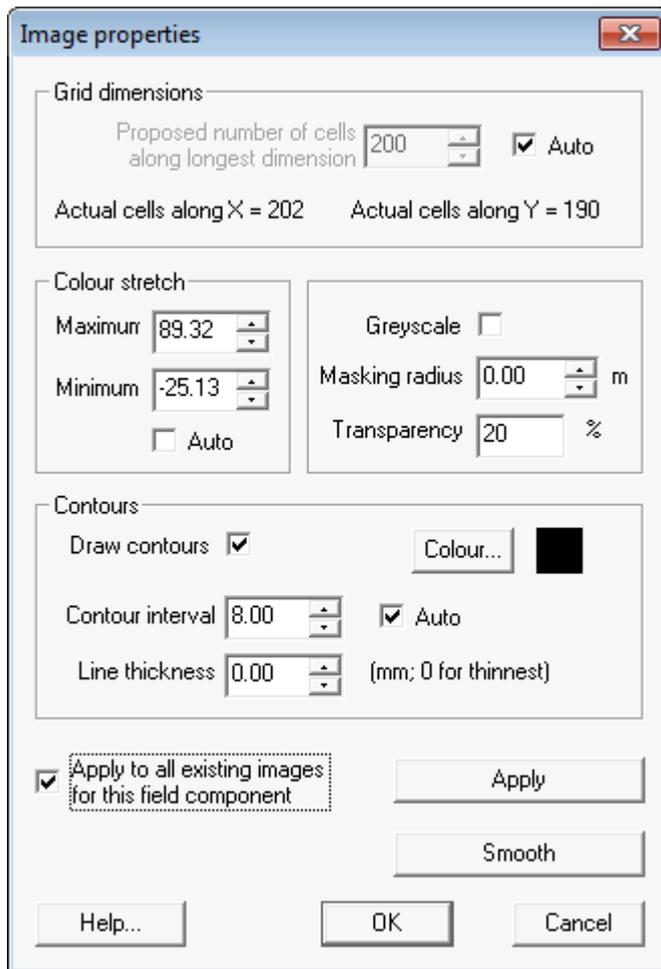
You now have three images visualising the observations and the model. Experiment with manipulating the model so as to make the observed and calculated images look similar. The residual image highlights the places where the match is poor.

One problem with the images is that by default *Potent* stretches the colour range so that the minimum value is purple and the maximum red. Therefore even when there is good correlation between the observed and calculated fields the residual will still look unsatisfactory (although numeric values as displayed on the status bar would be seen to be small). It would be better to display the residual image using the same colour stretch as the observed field. You can do this as follows:

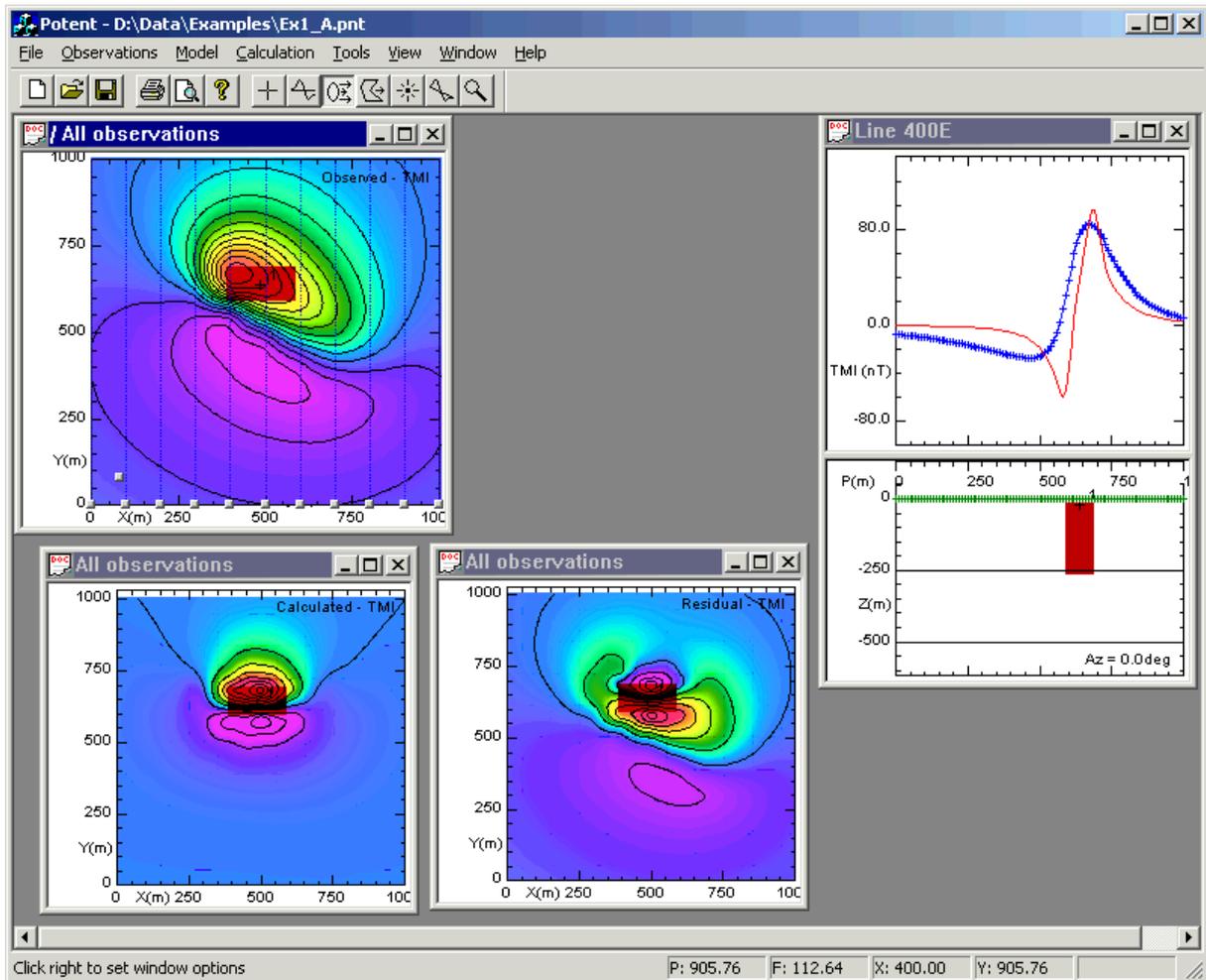
Right click on the plan window that displays the *observed* field and click **Options** on the popup menu to display the **Plan Window Settings** dialog box. Click the **Observations** tab, then click **Image properties**.



In the **Colour stretch** group, the **Maximum** and **Minimum** values correspond to red and purple on the image. *Potent* has calculated these automatically because **Auto** is selected. Clear **Auto** and select **Apply to all existing images....**



Click **OK** to apply the changes and exit this dialog box, and **OK** again to exit the **Observations** dialog box. The three images will all be displayed at the new colour stretch. Clearing **Auto** for the colour stretch of each image has prevented *Potent* from automatically recalculating it.



Now, as your model improves, the residual image will tend to a flat shade of blue.

(As this window configuration will be used in the inversion exercise in a later topic, consider using the **File | Save workspace as** command to save the entire workspace for future use. Assign a name *pre-invert.pnt*.)

[Back to A Simple Tutorial](#)

## Inversion modelling

Inversion modelling is a mathematical process that automatically adjusts model parameters so as to improve the fit between the calculated field and the observed field. The important word here is “mathematical”. The inversion algorithms used in *Potent* take no account of geological issues; it is up to you, the interpreter, to ensure that the inversion starting conditions are sensible, and to reject any inversion results that are not geologically plausible.

*Potent*'s inversion scheme is very flexible. You can invert on:

- any combination of parameters from any number of bodies;
- multiple datasets (e.g. regional gravity and aeromagnetic);
- multiple components from within the same dataset such as three-component down-hole magnetic data (which you can invert in conjunction with, for example, ground TMI data).

In this exercise we shall perform a very simple inversion using a single body and a single-

component dataset.

What's needed for inversion

Preparation for inversion

Inversion exercise

### What's needed for inversion

There are two key items needed for inversion:

- A set of model parameters that you want the inversion process to adjust;
- A sample of observed data (X,Y,Z,F values, where Z is height and F is the observed field). This has two functions. Firstly, the data points in the sample provide the geographic (X,Y,Z) coordinates at which *Potent* will calculate the field due to the model. Secondly, they provide the observed field values, F, against which the calculated field values will be compared. The inversion algorithm (a *mathematical* process) will attempt to minimise the root-mean-square (RMS) difference between the observed and calculated values.

Back to [Inversion modelling](#)

### Preparation for inversion

Preparation for inversion involves the following steps:

- Identify the model parameters that you want to adjust by selecting the appropriate **invert flags** in the **Edit body** tabbed dialog boxes. (When setting the Invert flag you also have the opportunity to set a range within which the value will be allowed to vary.)  
(For **polygonal bodies** you will probably choose to use the **Invert tool** to set up inversion on body vertices.)
- Identify the observations by displaying them in windows. Close or minimise any windows that contain parts of your datasets that are not relevant to the inversion.
- Consider using the **Inversion weights** tool to set the relative importance of sections of observations.
- Extract any regional background field by using the **Regional tool**.
- Set up a reasonable starting model by adjusting parameters manually.  
"Reasonable" means that your starting model is both geologically sensible and has a calculated field that is similar to the observed field.

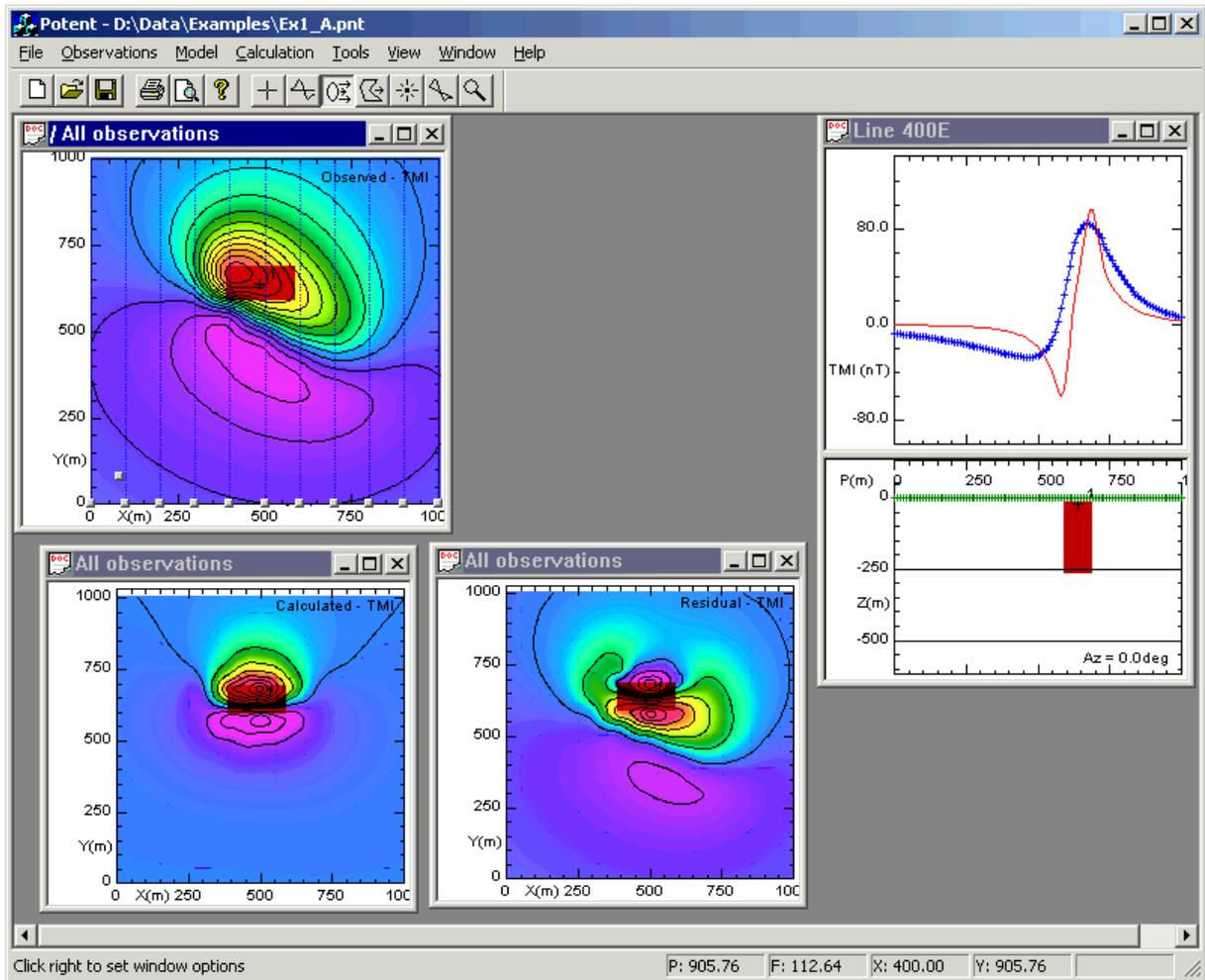
Back to [Inversion modelling](#)

### Inversion exercise

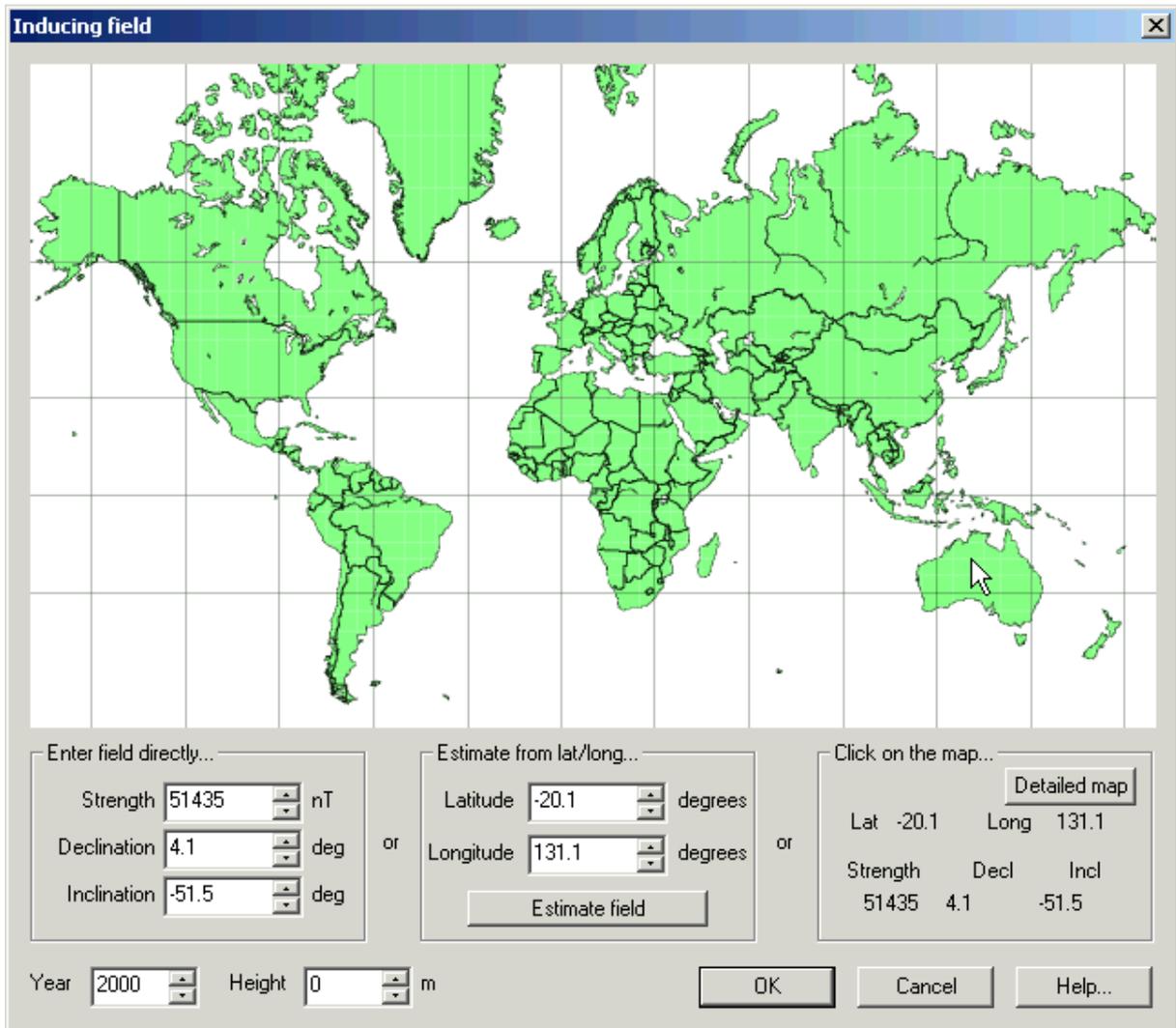
This part of the tutorial assumes that your screen is set up as at the end of [More about visualisation](#).

(If you followed the advice at the end of that topic you will have saved the workspace as a file named *pre-invert.pnt*. Use the **File | Open workspace** command to reload the workspace.)

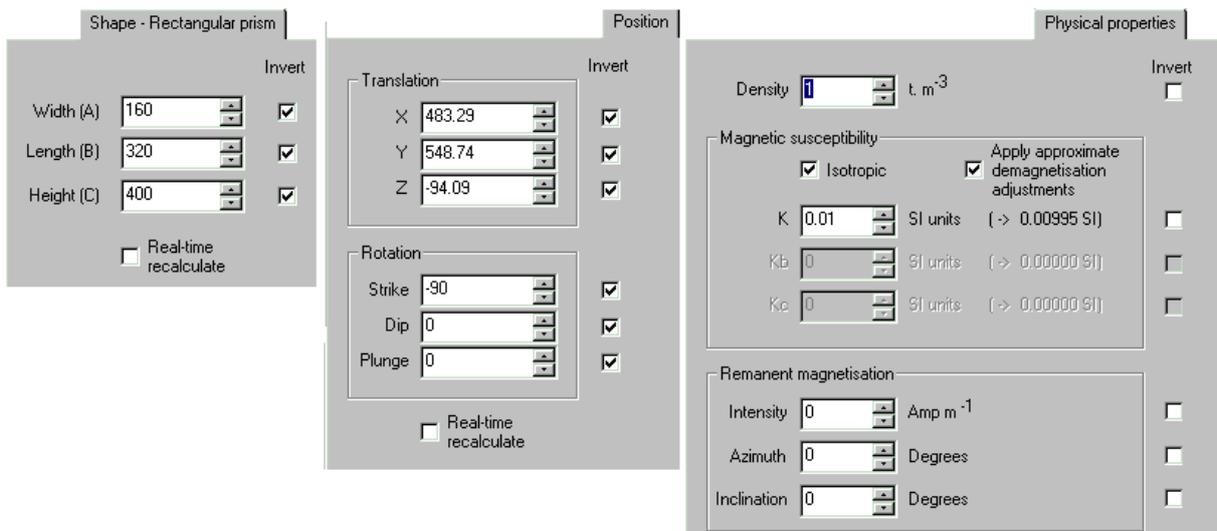
The dataset is *Example1.xyz*.



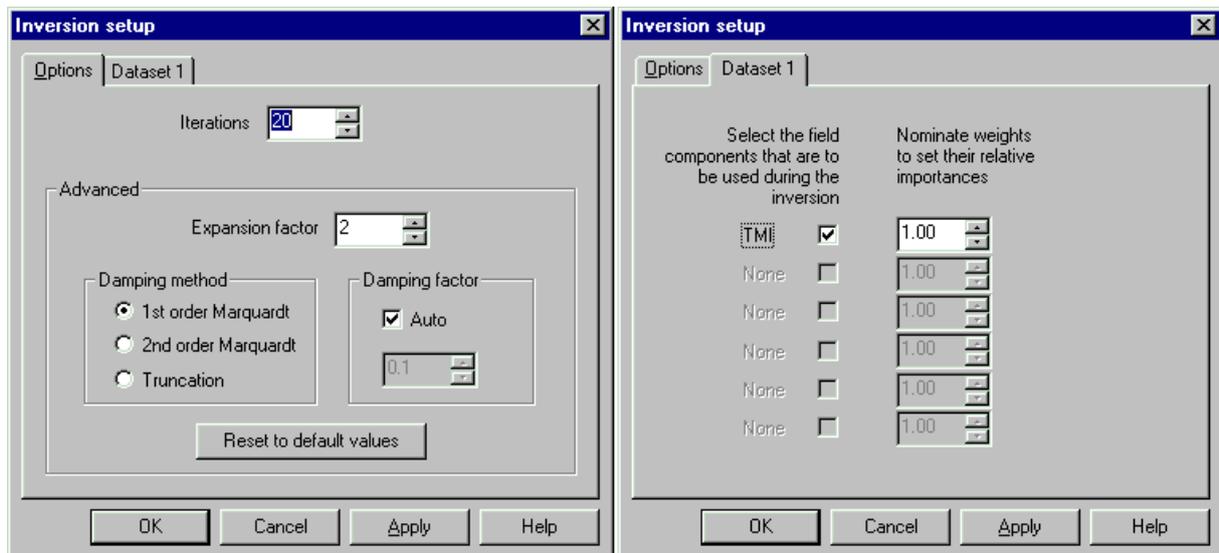
Remember that the inducing field (IGRF) must be set for northern Australia (near Darwin). If necessary use the **Model | Inducing field** command.



Set up a model that consists of a single rectangular prism with parameters similar to the following. Use the **Edit body** command to view and edit the model (which is a single body in this case). Note the **Invert** check boxes. You are telling *Potent* to adjust all the Shape and Position parameters during inversion, but none of the Physical Properties parameters.

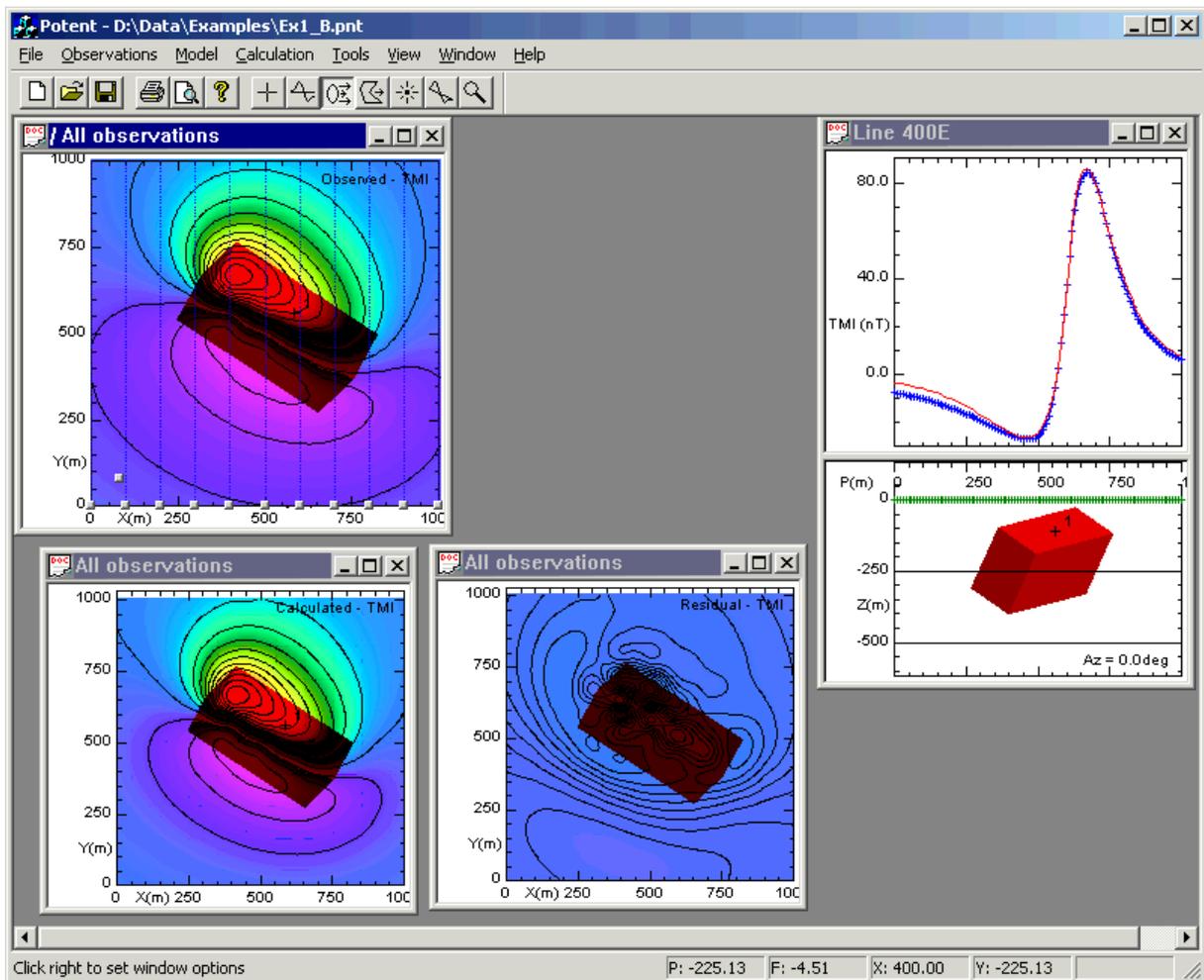


Click **Invert options** on the **Calculation** menu to display the **Inversion Setup** dialog box . In this case it has two tabbed pages.



The **Options** tab allows you some control over the inversion algorithm. Generally you will only change the number of iterations. For this example, set **Iterations** to 20. (Apart from the **Options** tab, there is one other tab for each dataset. Each of these tabs allows you to specify the components from each of the datasets that you wish to contribute to the inversion, and their relative weights, or importance. In this example we have only a single dataset consisting of a single component, so there are no choices to be made.)

Click **OK** to close the dialog box. Click **Invert** on the **Calculation** menu (or **Invert** on the **Inversion** toolbar). *Potent* will proceed to adjust the model parameters to improve the fit between the observed and calculated fields. The RMS error is displayed on the status bar as the inversion proceeds. After about 14 iterations you should end up with a screen like this.



As the data is synthetic we end up with an almost perfect fit.

If you carried out the inversion exactly as described then it would have proceeded relatively slowly. This is because the grids for the images in the Calculated field and Residual field windows must be recalculated at each iteration. Try using the **Restore from snapshot** command from the **Calculation** menu to restore the pre-invert workspace. Then *minimise* (rather than close) the Calculated field and Residual field windows before running the **Invert** command. The inversion will iterate much more quickly. Once it is finished you can restore the other two windows.

Note that you *must* keep open at least one window that contains the whole anomaly. We chose the first (Observed field) plan window in this case. Any of the plan windows would have done, but would have incurred the calculation overhead mentioned earlier.

Notice that we chose to have a profile window open. This was entirely for visual effect; the inversion would have proceeded perfectly well without it.

Back to [Inversion modelling](#)