Reference Manual

Toolbars

There are four toolbars, which you can choose whether or not to display via the **View** menu:

Standard

This is the standard Windows document toolbar. In the context of Potent, clicking the icons has the following effect:

 – Run the Make command from the Observations menu.

 – Run the Open workspace command from the File menu.

 – Run the Save workspace command from the File menu.

 – Run the Print command from the File menu.

 – Run the Print preview command from the File menu.

 – Run the About Potent command from the Help menu.

Tools

The Tools toolbar provides shortcuts to all the commands in the **Tools** menu.

See also: Tools in the User's Guide.

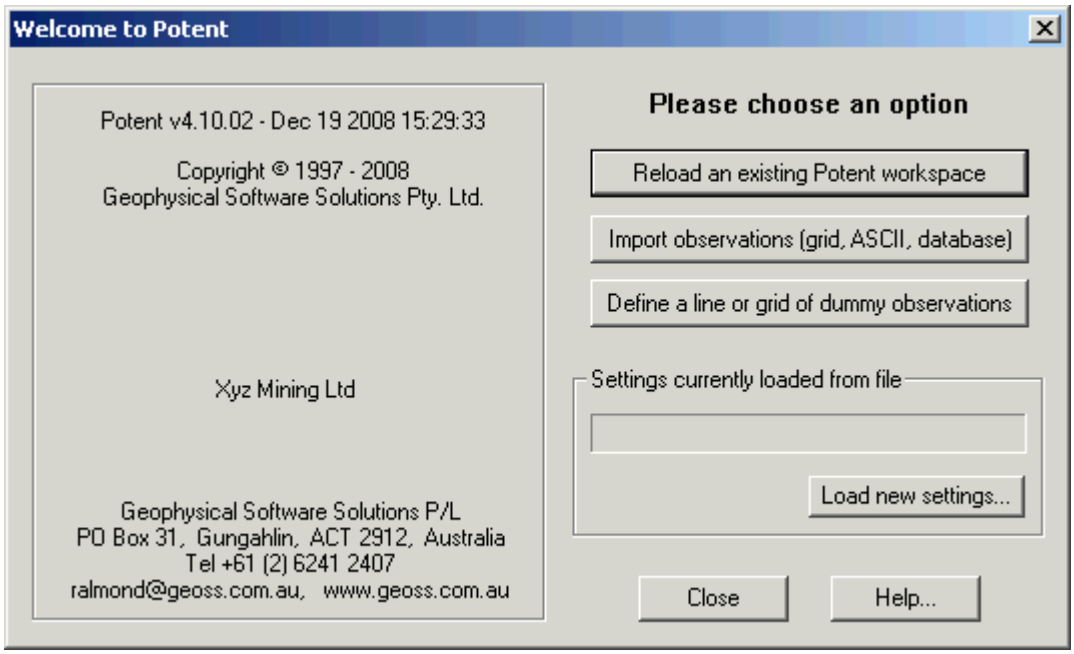
Bodies

The Bodies toolbar provides shortcuts to all the commands in the Bodies submenu of the **Model** menu.

Inversion

The Inversion toolbar provides shortcuts to the inversion-related commands of the **Calculation** menu.

The Welcome dialog box



This dialog box appears either when you first run *Potent*, or when you click Help | About. As well as providing information about your licence, the version of *Potent*, and contact information for Geophysical Software Solutions, it has buttons for the following options:

- Reload an existing *Potent* workspace – Invokes the File | Open workspace command.
- Import observed data from a grid or XYZ file – Invokes the Observations | Import command.
- Define a line or grid of dummy observations – Invokes the Observations | Make command.

Each of these options disposes of the welcome dialog. The **Load new settings** button invokes the File | Load settings command. It does not dispose of the dialog box, enabling you to load settings prior to choosing one of the other options.

The **Close** button closes the dialog box without doing anything. You are presented with an empty *Potent* window.

File menu

Workspace commands:

Open workspace

Save workspace

Save workspace as

Close workspace

Printing commands:

Print

Print preview

Page setup

Page layout

Drawing export commands:

Draw to metafile

Configuration commands:

Load settings

Save settings

Preferences

Licence commands:

Licence - View

File | Open workspace

Use this command to restore a workspace that has been saved previously with the File | Save workspace or File | Save workspace as commands. A dialog box displays the title that was assigned to the workspace when it was saved, along with the version of *Potent* that was used to create it. Click **OK** to load the new workspace, or **Cancel** to retain the current workspace.

See also: Working with workspaces in the Potent User's Guide

--- Return to the File menu.

File | Save workspace

Use this command to update your current workspace file. If you have not previously saved the workspace, *Potent* runs the File | Save workspace as command. In either case you are prompted for a title to assign to the workspace file. This title is displayed when the file is loaded again.

See also: Working with workspaces in the Potent User's Guide

--- Return to the File menu.

File | Save workspace as

Use this command to save your entire session to a file. *Potent* will prompt you for a file name, and for a title to assign to the workspace file. This title is displayed when the file is loaded again.

See also: Working with workspaces in the Potent User's Guide

--- Return to the File menu.

File | Close workspace

Close the current workspace so that the main *Potent* window is empty. *Potent* will ask whether you want to save the current workspace.

Click **Yes** to save the workspace. *Potent* automatically runs the File | Save workspace command. If you cancel this command, the workspace is not closed.

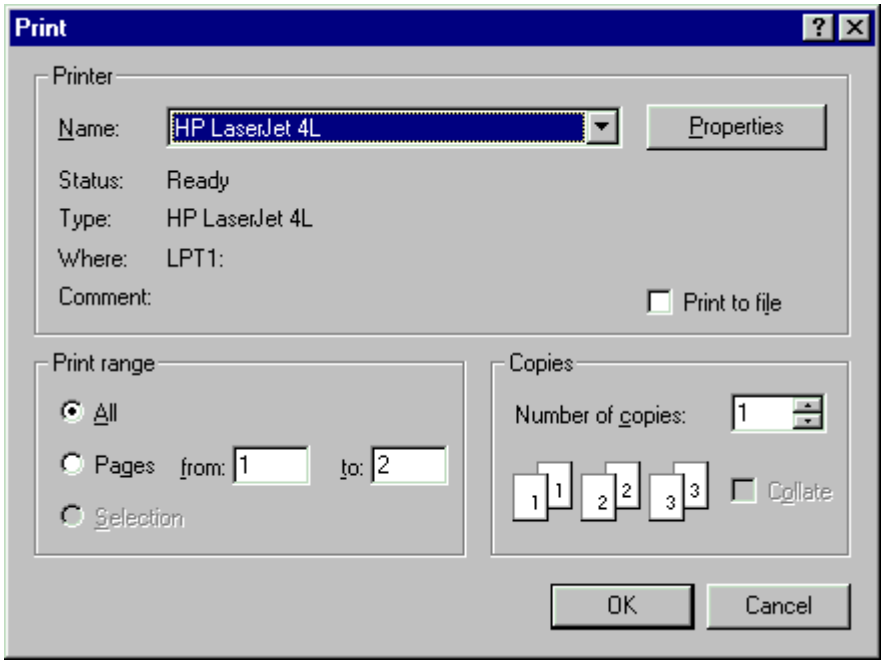
Click **No** to close the workspace without saving it.

Click **Cancel** to retain the current workspace.

--- Return to the File menu.

File | Print

Choose this command to print the current layout. A standard Windows Print dialog box is displayed.



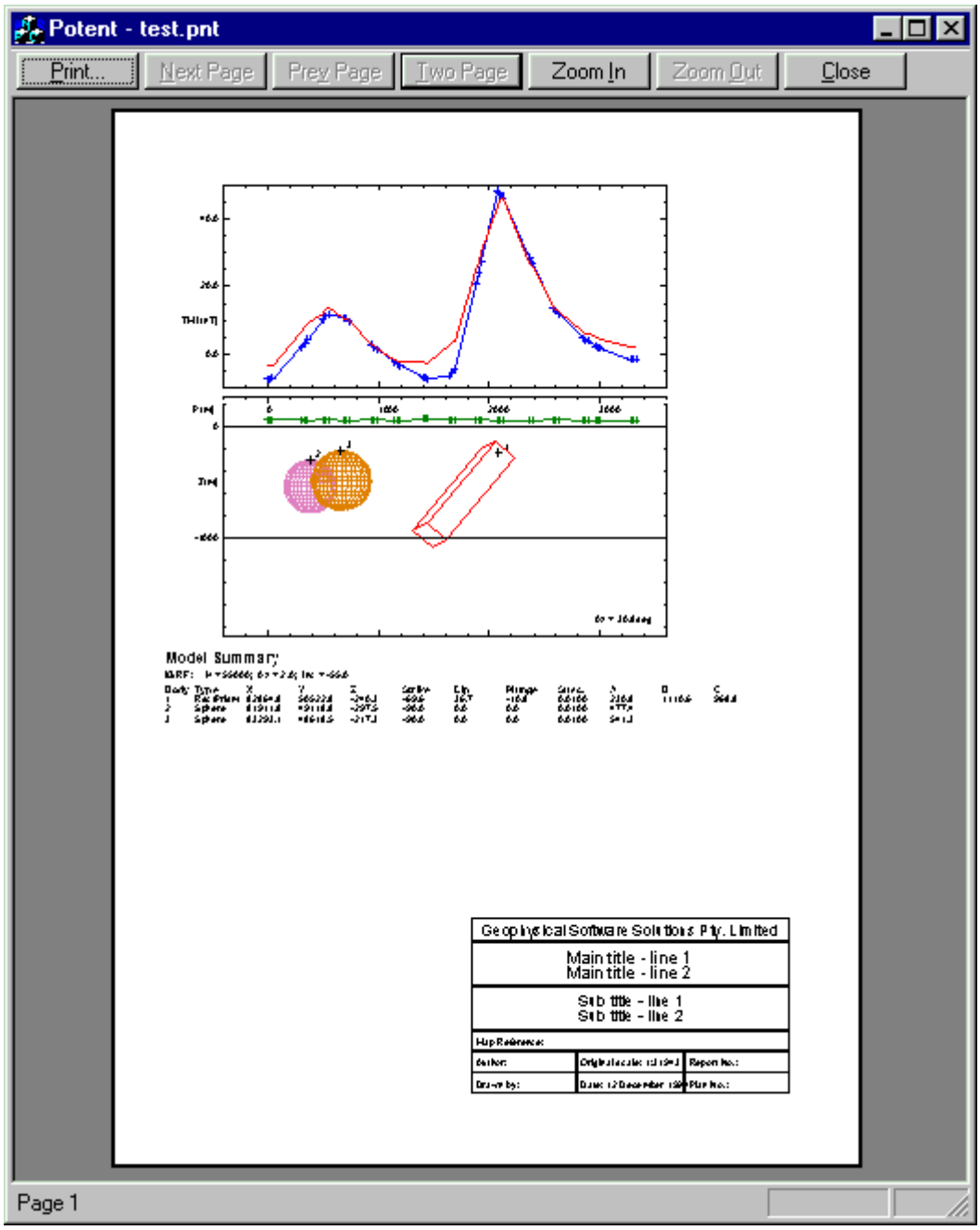
Note that bodies currently are always drawn as wireframe when printing, even if **Solid** has been specified in the **Plan window settings – Bodies** or **Profile window settings - Cross-section** dialog boxes.

See also: File | Page setup, File | Page layout

--- Return to the File menu.

File | Print preview

Use this command to check the layout of your printed output, before sending it to the printer. A standard Windows Print Preview window is displayed.

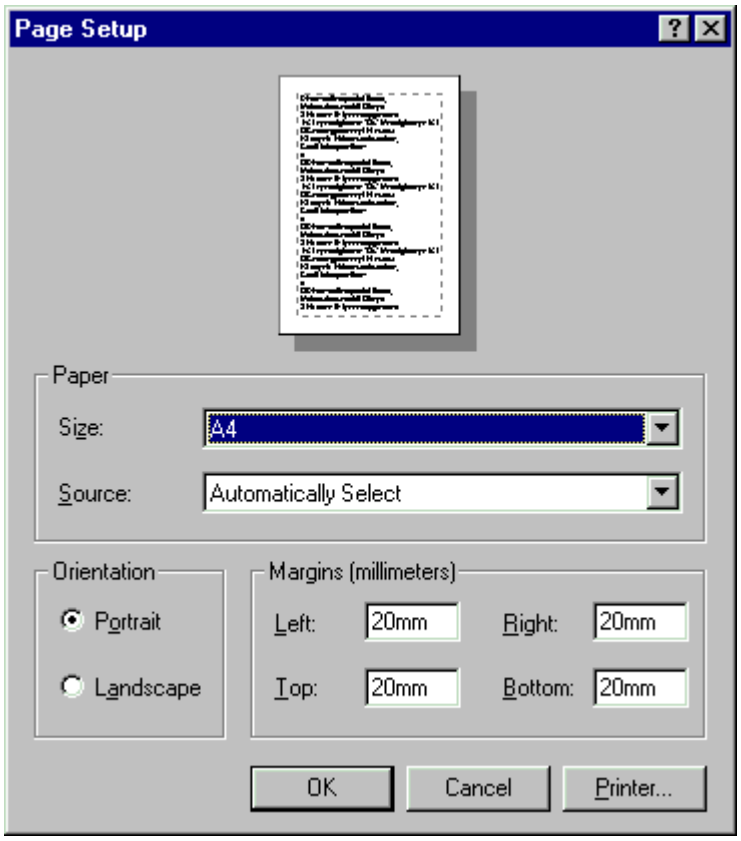


See also: File | Page setup, File | Page layout

--- Return to the File menu.

File | Page setup

This command displays a standard Windows Page Setup dialog box, which allows you to specify paper size, paper orientation, and margins.

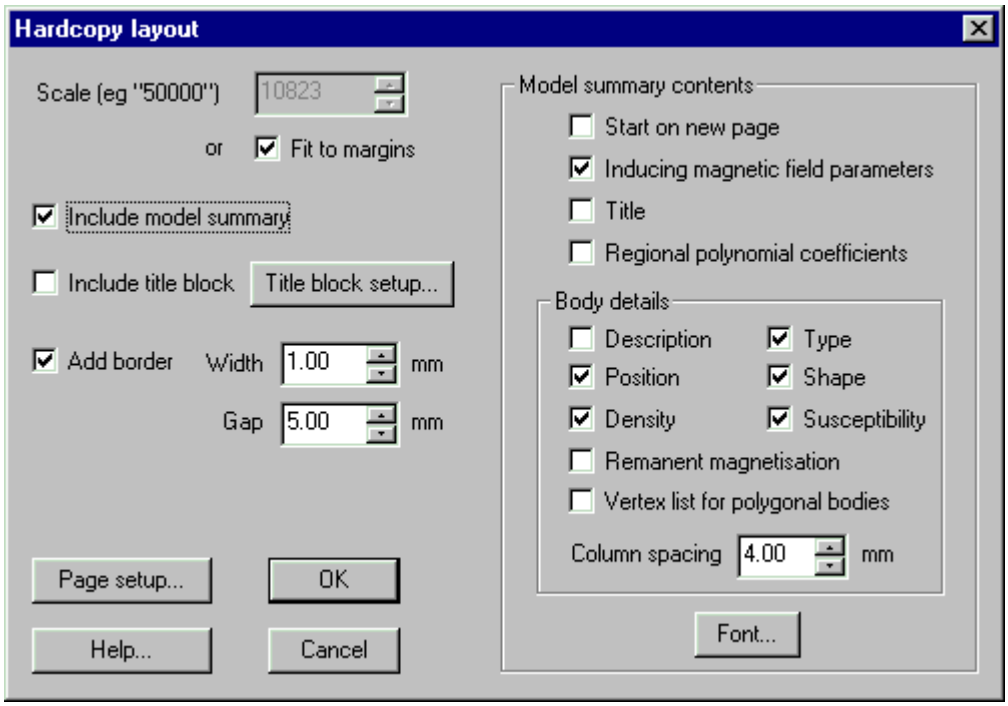


See also: File | Page layout

--- Return to the File menu.

File | Page layout

This command displays a dialog box that allows you to control the contents and layout of the printed page.



Scale – This option allows you to specify the scale of the plotted window. If you are unsure what scale to use, try running **File | Print preview** with **Fit to margins** selected. *Potent* will adjust the scale so that the plot fits between the margins, while preserving the proportions of the window. When you next run the File | Page layout command, you will see the scale that *Potent* has calculated. This will generally be a strange number such as the 10823 in the above example. In this case you could clear **Fit to margins** and safely round the scale up to 12000 – the plot would be slightly smaller and so would still fit within the margins.

Include model summary – Select this if you want Potent to include a model summary, as defined in the **Model summary contents** group (below).

Include title block – Select this to include an information block in the bottom right corner of each page of the output, as shown in the example in File | print preview.

Title block setup – Click the Title block setup button to specify the contents of the title block.

Add border – Draw a rectangular border around the plot at the position set for the margins (see File | Page setup). You can also set the **width** of the border (in millimetres), and the **gap** that is left between the border and the contents of the drawing. (In effect **Gap** applies an extra margin inside the border.)

Page setup – Click this button to display the Page Setup dialog box.

Model summary contents group:

Start on new page - Clear this to have *Potent* position the model summary block underneath the plan or profile window on the paper, as shown in the example in

File | print preview. Otherwise it will be on the top of the second page.

Inducing magnetic field parameters – Select this to include a block containing the strength, declination and inclination of the inducing field.

Title - Include the model title.

Regional polynomial coefficients - If this is selected the model summary lists the coefficients that specify the regional background field, as specified using the Tools | Regional command.

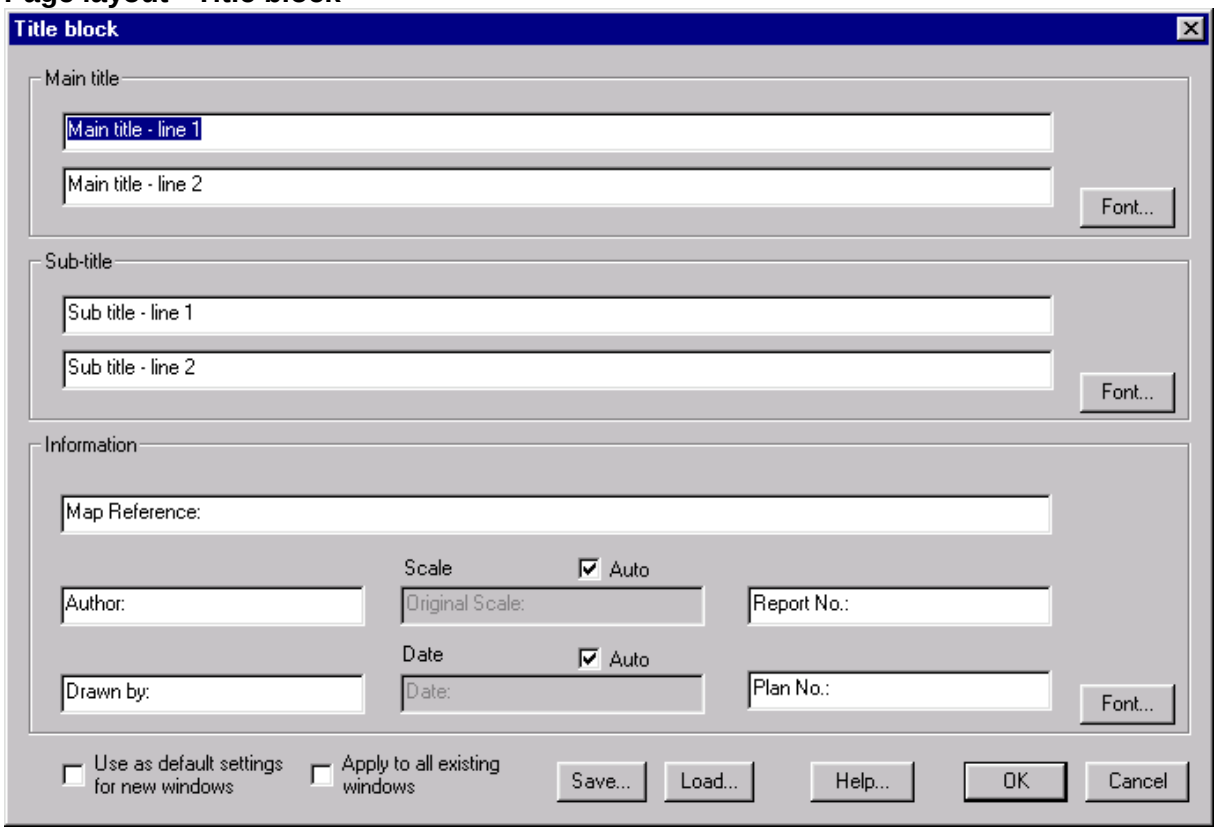
Body details - Each box you select becomes a column in the model list.

Column spacing - Specify the number of millimetres between the columns of the Model Summary block.

Font - Click this button to choose the font that will be used for the Model Summary.

--- Return to the File menu.

Page layout - Title block



This dialog box allows you to specify the contents of an information block that will be included in the bottom right corner of each page of printed output for the active window, as shown in the example in File | print preview. Include the title block by checking the appropriate option in the File | Page layout dialog box.

The layout of the dialog box reflects that of the title block. The default entries, such as "Author", are suggested usage – replace them with your own content as required. However, be aware of the space available to each field and adjust the font and the length of the entry accordingly.

If **Auto** is selected alongside the **Scale**, then *Potent* will automatically include the scale of the printed output. If it is cleared then you can use the field for your own text.

If **Auto** is selected alongside the **Date**, then *Potent* will automatically include the current date. If it is cleared then you can use the field for your own text.

Use as default settings for new windows - Select this and click **OK** if you want these settings to be used for any new windows created for the current dataset.

Apply to all existing windows - Select this and click **OK** if you want to apply these settings to all windows of the current dataset.

Use the **Save…** and **Load…** buttons to save and restore the title block to and from a text file. The format is that of a Windows initialisation (".ini") file.

See also: Creating a global default Title Block.

--- Return to the File | Page layout command.

Creating a global default Title Block
To create a global default title block (that will be used as the title block for all windows and all datasets), run the File | Page layout command from an empty workspace (perhaps by using the File | Close workspace command). Set up the title block as described in Page layout – Title block.

--- Return to the File | Page layout command.

File | Draw to metafile

A standard "Save As" dialog box is displayed.

Use this command to draw the active window to a Windows enhanced metafile (file extension EMF). The enhanced metafile can then be pasted into other Windows applications such as word processing and GIS packages.

Note that bodies currently are always drawn to the metafile as wireframe, even if **Solid** has been specified in the Plan window settings – Bodies or Profile window settings - Cross-section dialog boxes.

Enhanced metafiles preserve their precision when resized.

--- Return to the File menu.

File | Load settings

Use this command to load settings that were saved with the File | Save settings command.

--- Return to the File menu.

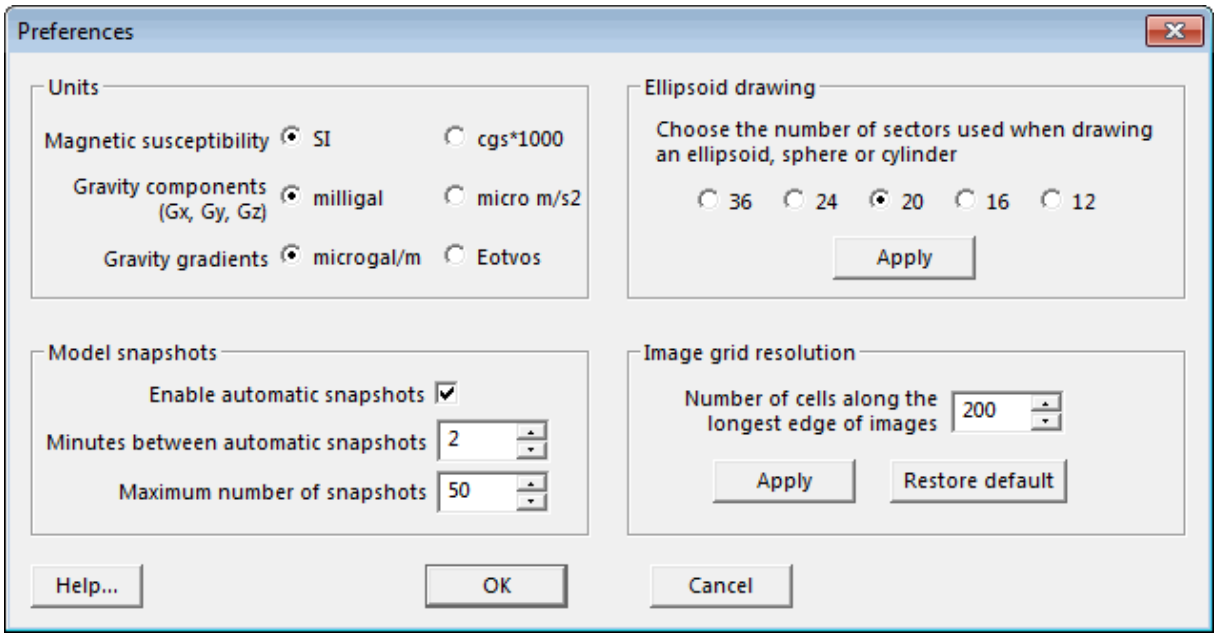
File | Save settings

Use this command to save most *Potent* settings. The settings are saved in a Windows ".ini" file. This method is used rather than the system registry as it is useful to be able to move settings files around, or to maintain several settings files for different purposes.

Note that loading a workspace file will replace most settings with those that were saved with that workspace. Consider reloading the settings after the workspace is opened.

--- Return to the File menu.

File | Preferences



This dialog box allows you to specify your own defaults for various settings.

Units

Specify your preferred units for susceptibility and gravity gradient.

For magnetic susceptibility:

1 SI unit is equivalent to 4*PI cgs units

For gravity components:

1 milligal is equivalent to 10 micrometre/sec^2

For gravity gradients:

1 microgal/m is equivalent to 10 Eotvos

Notes on units:

These settings apply to all datasets in your workspace.

Changing a setting does not scale the data. Rather, it causes Potent to interpret the current field values of the observations as being in the specified units.

Model snapshots

The use of snapshots is described in Undoing changes in the model in the User's Guide.

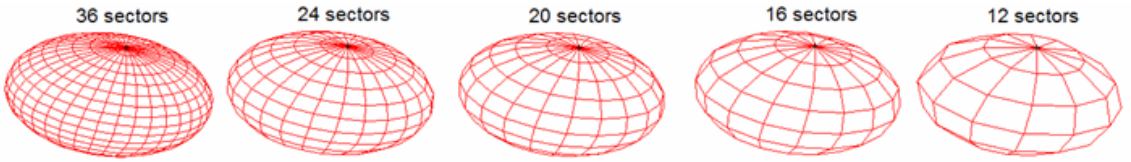
Select **Enable automatic snapshots** to activate time based snapshots.

If automatic snapshots are enabled, enter the **Minutes between automatic snapshots**. Snapshots are not necessarily taken at this interval. Rather, when this time has elapsed a snapshot will be taken next time the model changes.

Enter the **Maximum number of snapshots** that Potent will store. This must be a number between 1 and 100. (Snapshots are stored as files in your TEMP or TMP folder, so reduce this number if you need to conserve disk space.)

Ellipsoid drawing

Choose the number of sectors that will be used to draw the wireframe mesh framework of an ellipsoid as "latitude" and "longitude" lines. The larger the number, the higher the resolution of the drawing. For example, 36 sectors will cause lat/lon lines to be drawn every 10 degrees, while 12 sectors would draw them every 30 degrees. These images show how an ellipsoid wireframe mesh might look for each option:



By default, the number of sectors is set to 36. If you have many ellipsoids in your model, setting sectors to a smaller value can speed up redrawing and make the window less cluttered.

Note that the mesh is used only for drawing. Calculations are always done for the true ellipsoid.

The number of sectors specified here also affects the drawing of the cross-sectional ellipse of the Cylinder type of body.

See also - Plan window settings – Bodies
Profile window settings – Cross-section

Image grid resolution

When Potent creates a grid, by default it sets the number of cells along the greater of the X and Y data extents to be nominally 200. (The actual number of cells might be changed slightly from this figure by the gridding algorithm.) As grid cells are square, the number of cells along the other dimension is usually less than 200, in proportion to the aspect ratio of the data.

The **Image grid resolution** group allows you to change this default number of cells. The new setting applies to all current grids that have **Auto** grid dimensions set (see the **Image properties** dialog box), and becomes the default setting for new grids.

Click **Restore default** to restore the default value used by Potent.

Click **Apply** to see the effect of a new value without closing the dialog box.

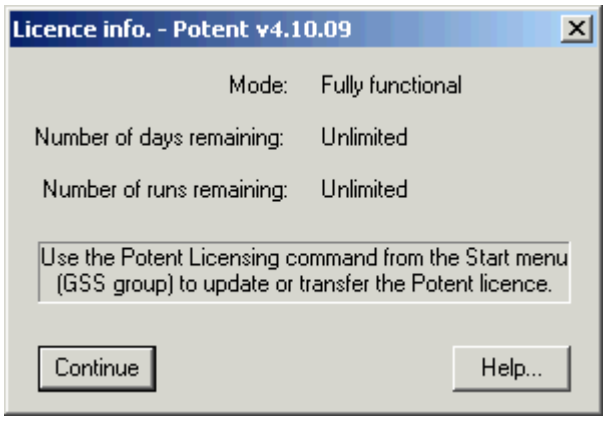
Note: For PotentQ, the default is 100 rather than 200.

See also – The **Grid dimensions** group in **Image properties**

--- Return to the File menu.

File | View licence

This command allows you to view your licence. If you have a full Potent licence then the dialog box would appear as follows:



Refer to the Licensing section in the User's Guide for information regarding updating and transferring the licence.

--- Return to the File menu.

Observations menu

This menu contains commands for obtaining and managing observations.

Import

Merge

Options

Close dataset

Dataset title

Save subset(s) as

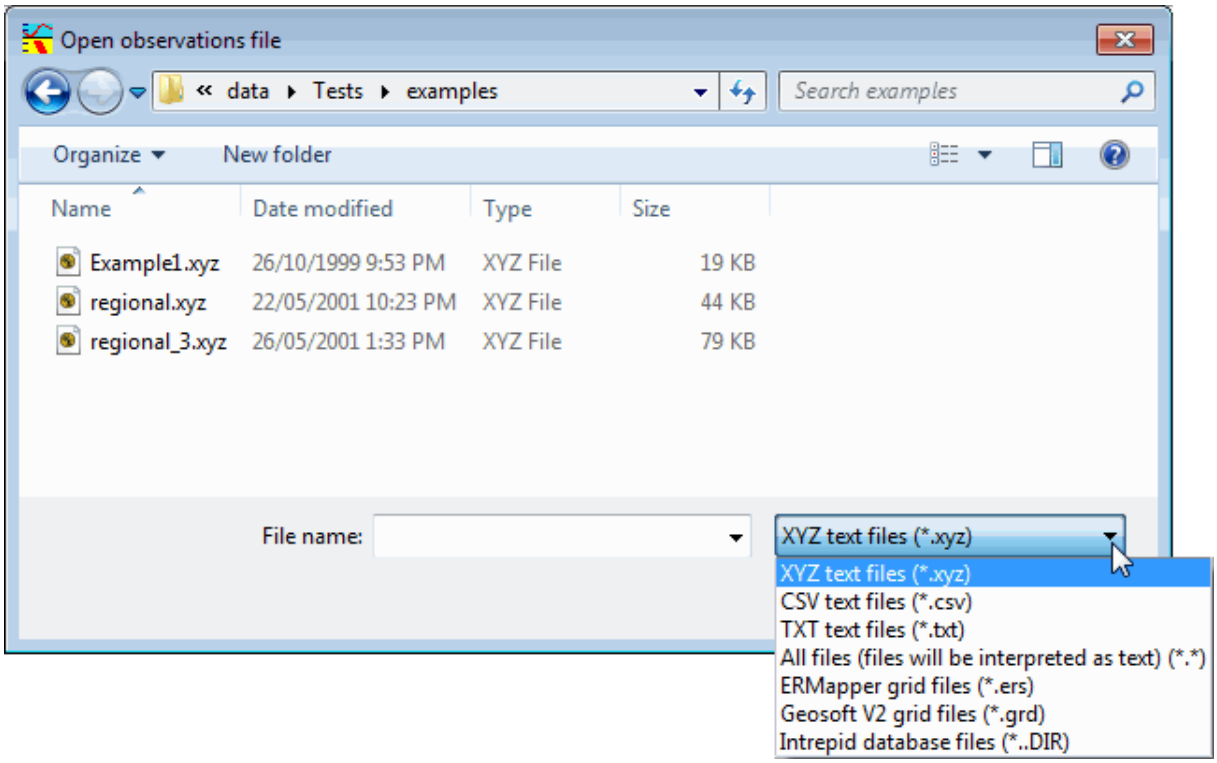
TMI & Mz sensor spacing

Make

Subset management

Observations | Import

This command displays a standard Windows file-open dialog box.



Use this command to import observations from several types of file, which you specify by selecting the appropriate option from the drop-down list:

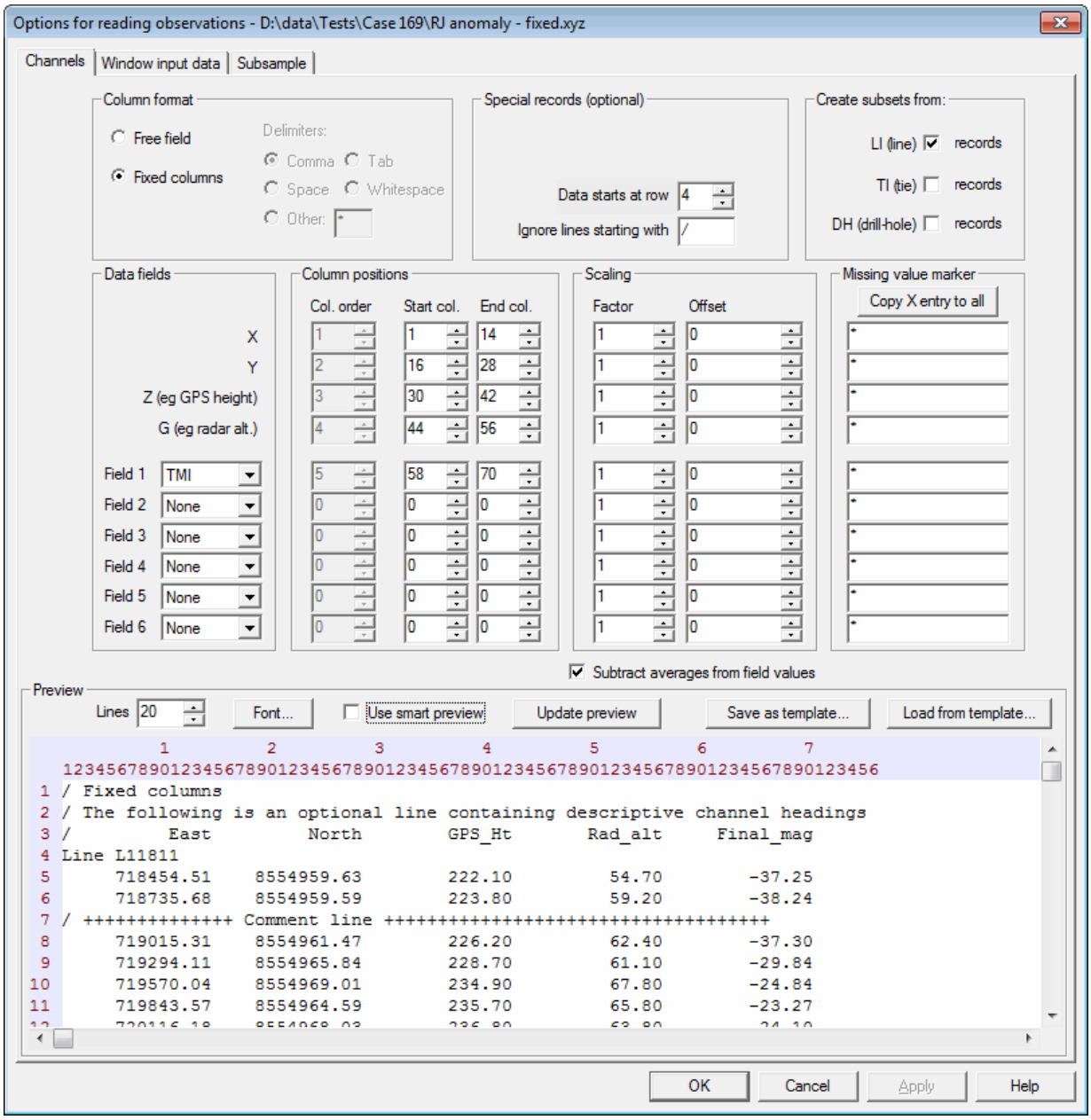
- Text files (for point-located data in text format)
- ER Mapper grids
- Geosoft grids
- Intrepid databases

See also: Importing observations in the User's Guide.

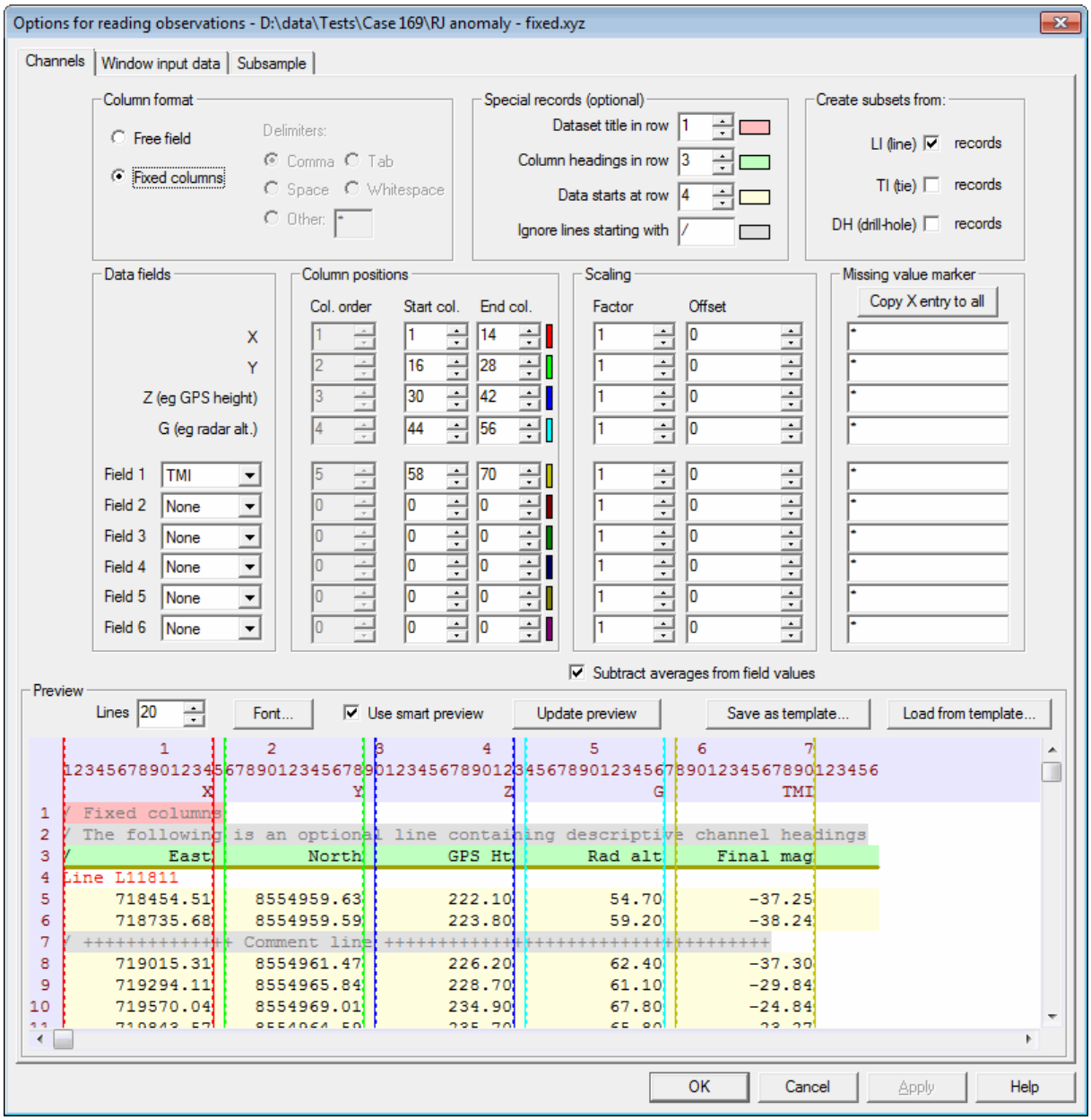
--- Return to the Observations menu.

Point-located data (text format)
When you attempt to read an observations file consisting of lines of text *Potent* displays the **Options for reading observations** tabbed dialog box. The **Channels** tab allows you to specify the format of the input file and the columns from which you want to import. The bottom of the dialog box is a preview section that contains a row of controls and a scrolling **Preview** window that displays the first few lines of the file (20 lines by default, up to a maximum of 99).

If **Use smart preview** is cleared then the **Preview** window simply reproduces the lines from the file, except that any TAB characters will be represented by the caret symbol, ^ . Use the **Preview** window as a guide to filling in the **Data fields** and **Column positions** groups.



If **Use smart preview** is selected then the **Preview** window contains features that help you interpret the file contents, as in this example:



Refer to Importing Observations from Text Files in the User's Guide for a detailed account of reading data from a text file using the **Use smart preview** option.

See also - Observations | Options - Channels command for details of the essential fields in the dialog box.

--- Return to the Observations menu.

Grids – General comments

When you open a grid file you are presented with a dialog box that displays the grid header and gives you certain grid-specific options. However, some options provided by the Observations | Options dialog box also apply to grids. These are:

- Scaling. Offsets and factors for X, Y, Z, G and Field 1 are applicable. Z and G, which cannot be inferred from the grid, are set to zero before applying the standard

scaling formula

value => (value - offset) * factor

- Windowing.

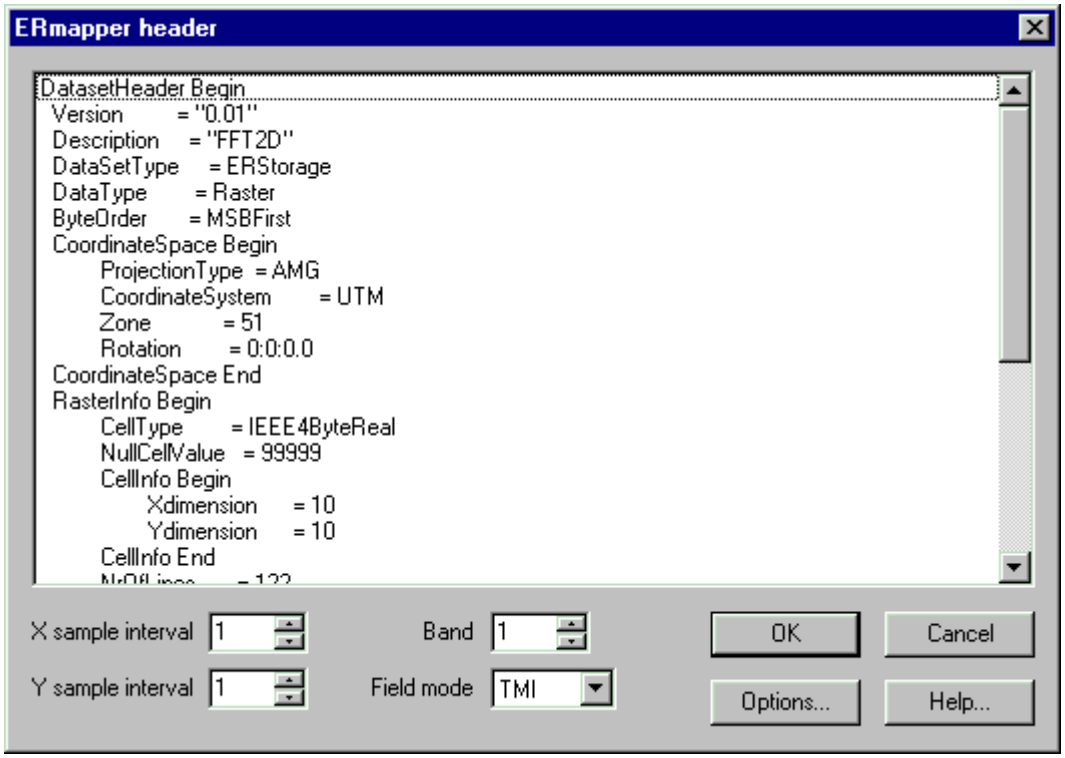
For example, if the grid is of data from an airborne survey for which the ground clearance was 100m and you wanted to use this as the observation height, Z, you could set the offset for the Z channel to -100, which will cause the Z value for all observations to be 100m.

You may run the Observations | Options command before opening the grid file, or use the **Options** button that is provided in each of the grid dialog boxes as a shortcut.

Refer to Grids in the User's Guide for further comments.

--- Return to the Observations menu.

ER Mapper grids



This dialog box allows you to inspect the grid header. You can also specify the X and Y (i.e. column and row) sample intervals, as described in Grids in the User's Guide.

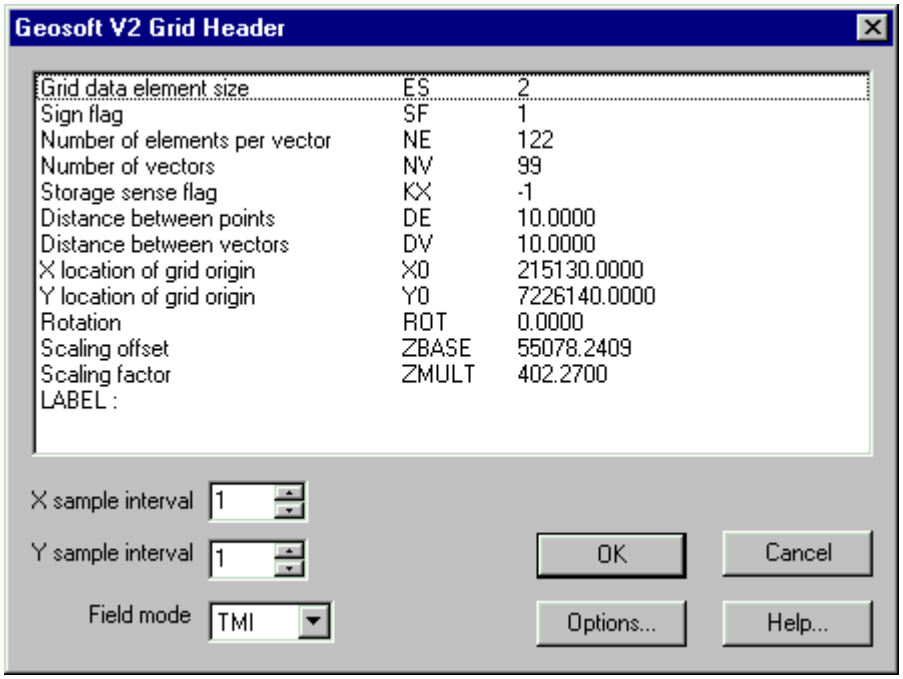
The **Band** list box contains an entry for each band in the grid. You must select one.

Use the **Field mode** list box to specify the field component that corresponds to your selected band.

The **Options** button is a shortcut to the Observations | Options command. Refer to Grids – General comments, for details.

--- Return to the Observations menu.

Geosoft grids



This dialog box allows you to inspect the grid header. You can also specify the X and Y (i.e. column and row) sample intervals, as described in Grids in the User's Guide.

Use the **Field mode** list box to specify the field component that the grid represents.

The **Options** button is a shortcut to the Observations | Options command. Refer to Grids – General comments, for details.

Note:
Potent will not read a Geosoft compressed grid file. If you have the Oasis montaj basic processing system you can convert the grid to the uncompressed format.

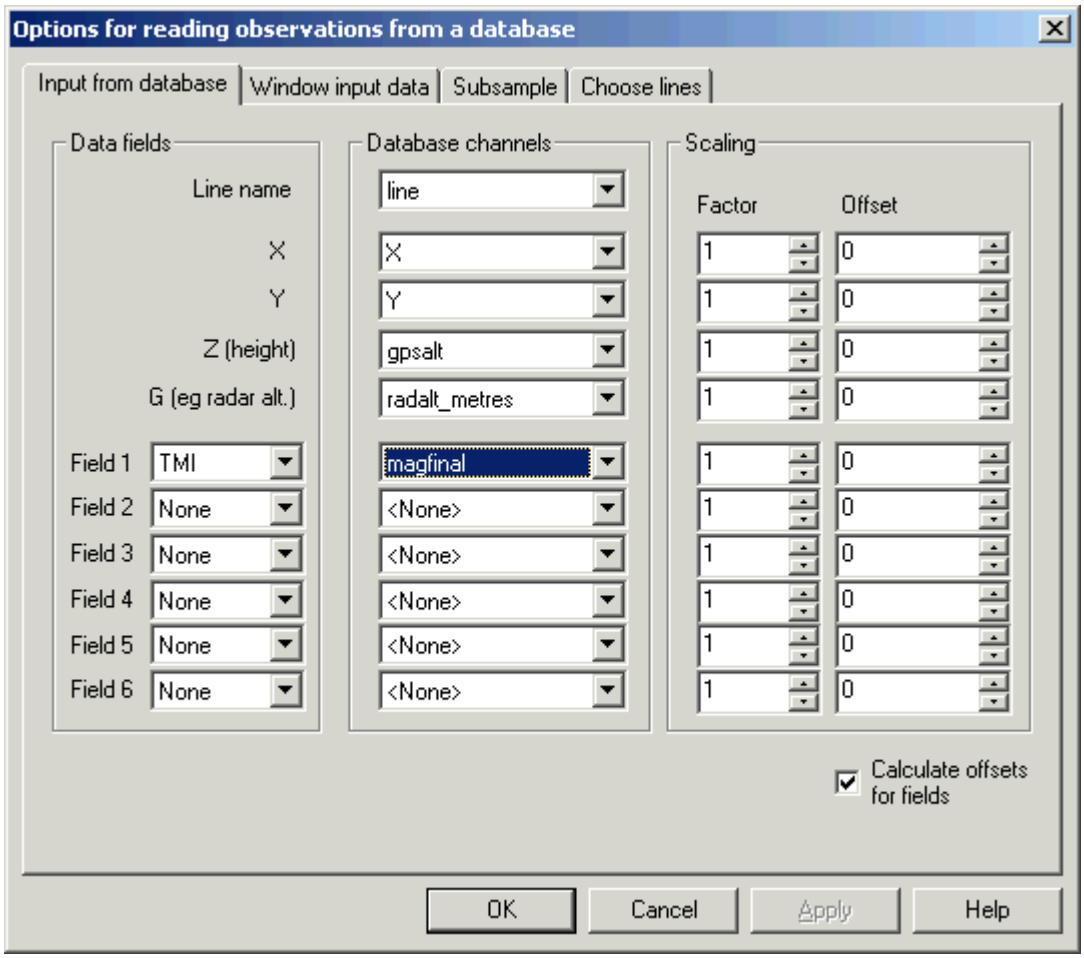
--- Return to the Observations menu.

Intrepid databases

If the *Intrepid* database is large consider using the **Window input data**, **Subsample** and **Choose lines** (see below) tabs to thin out the data. Refer to Working with large datasets in the User's Guide for related information.

Input from Database tab

Use the controls on this page to specify the channels.



Data fields group

A **Data fields** setting identifies the item that a row applies to. The first five items are fixed. The **Line name** field is optional; if present its values are used to label lines and to allow you to select lines for input using the **Choose lines** tab (see below).

The remainder (Field 1 through Field 6) identify field components to be input from the file. Drop down the list box and select a field component. Unused fields must be set to **None**. The fields that *are* specified must occupy adjacent boxes, starting at **Field 1**.

Database channels group

For each data field choose the corresponding database channel.

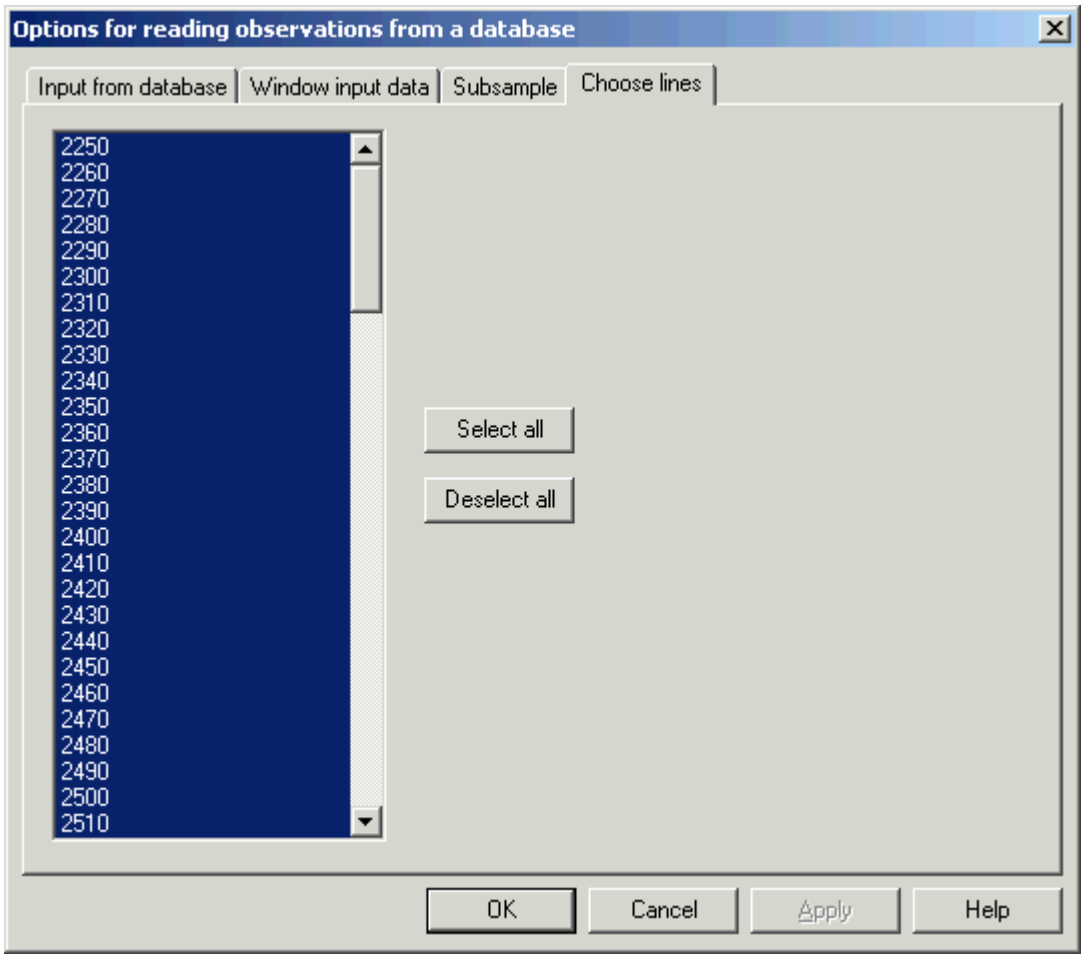
Scaling group

Factor / Offset – Use these to scale the field values as they are read from the file. The value stored by Potent is calculated according to the scheme

value = (value - offset) * factor

Calculate offsets for fields – Select this if you want *Potent* to automatically assign values for the offsets associated with the six field values. For each field channel, *Potent* sets the offsets to be equal to the mean of the values read from the file.

Choose Lines tab



If you specified the **Line name** field in the Input from database page, the **Choose lines** page lists all the lines that were read from the database. By default all lines are selected, but you can change the selection to specify which lines *Potent* will read.

See also - Window input data, Subsample

--- Return to the Observations menu.

Observations | Merge

This command is similar to the Observations | Import command. However, instead of creating a new dataset, the observations are added to the active dataset. Note that the data fields specified in the Options for reading observations dialog box must be the same as the data fields of the active dataset.

--- Return to the Observations menu.

Observations | Options

There are three pages to the Observation | Options dialog box:

Observations | Options - Channels

Observations | Options - Window

Observations | Options - Subsample

--- Return to the Observations menu.

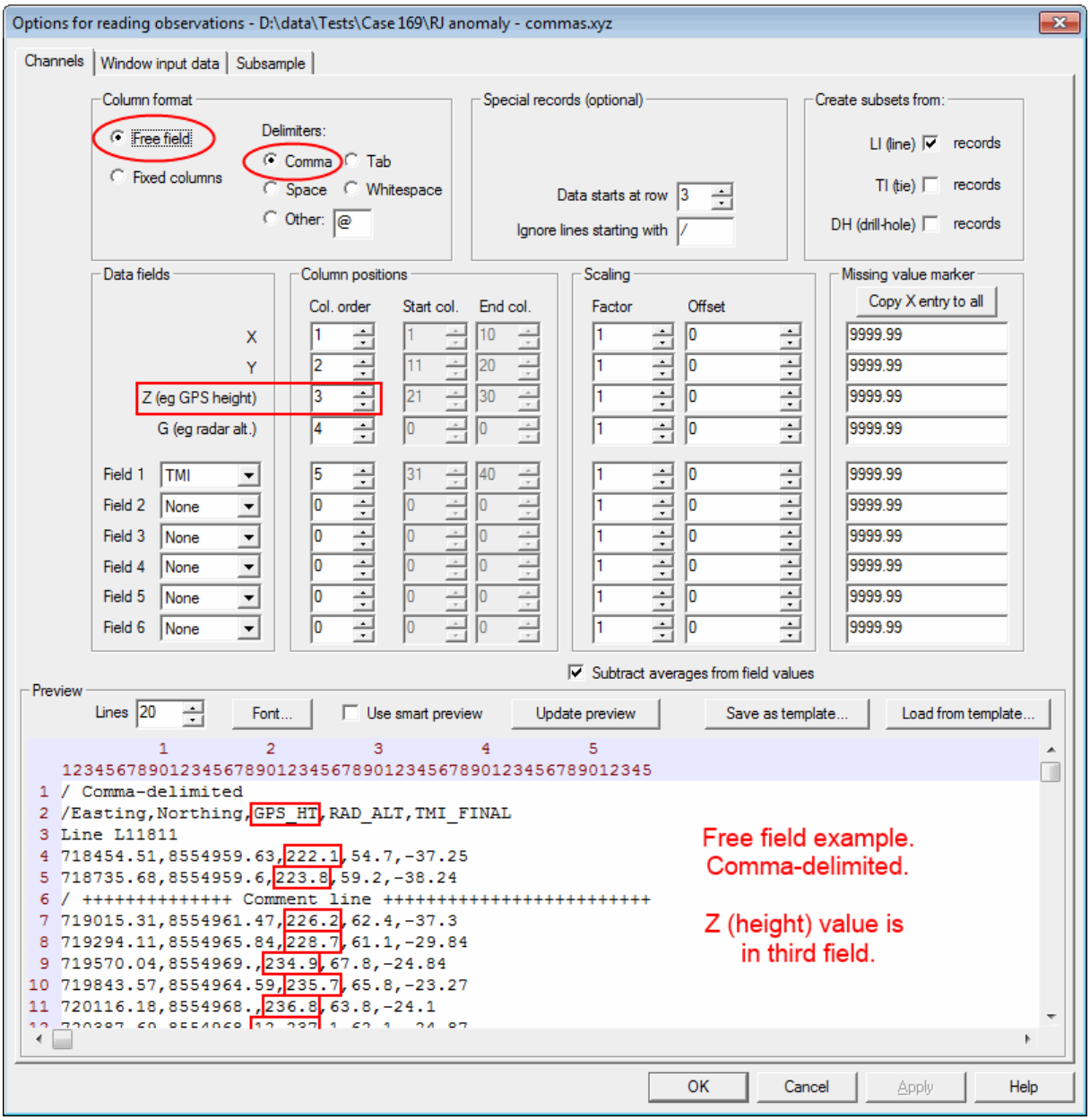
Observations | Options - Channels
(This dialog box works in one of two modes, depending on the setting of the **Use smart preview** checkbox. Here we assume this option is cleared as this exposes only the essential elements of the dialog box. Refer to Importing Observations from Text Files in the User's Guide for a description of the rich set of features available when **Use smart preview** is selected.)

Use this dialog box to specify how data will be decoded from a text file. The bottom third of the dialog box contains a preview area that displays the first few lines of the file (20 lines by default, up to a maximum of 99).

Column format group

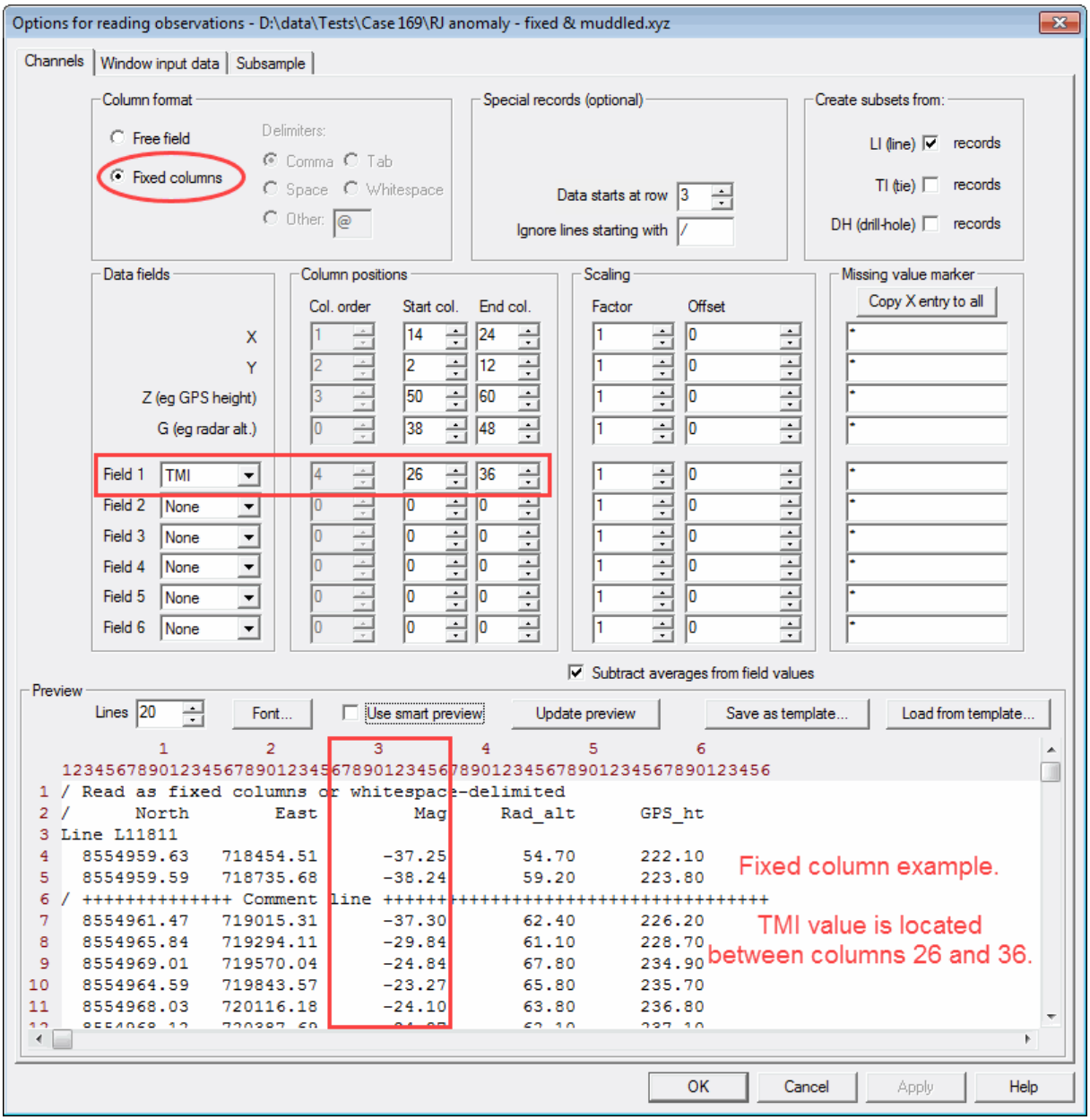
- **Free field** – A line of the data file contains values separated by a delimiter. **Delimiter** options are:
  Comma – a single comma separates values, as in the example shown above.
  Tab – A single tab character separates values.
  Space – A single space character separates values.
  Whitespace – Values are separated by any number of space and tab characters.
  Other – Specify any other single character to use as a delimiter.

- This example highlights how the **Z** (height) value would be extracted from a comma-delimited file:



The second line of the file (the “headings line”) indicates that the GPS height is in the third column (after Easting and Northing).Therefore in the **Column positions** group the **Col order** for **Z** is set to 3. The **Start col** and **End col** controls are disabled as these pertain to **Fixed columns** format.

- **Fixed columns**:
  In a line of data the values occupy specific ranges of columns. The following example shows how the **Field 1** value might be extracted:



Notice that the headings line shows that in this example the values are in an unusual order. However the column indicators in the top margin allow you to choose the appropriate ranges for **Start col** and **End col**. The **Col order** controls are disabled as these pertain to **Free field** format.

Special records group

When **Use smart preview** (see below, under **Preview** group) is switched off this group contains only two entries:

- **Data starts at row** - Specify the line at which actual data starts.
- **Ignore lines starting with** - Potent will ignore any lines whose leading characters exactly match this string.

Refer to Using smart preview in the User's Guide for information about **Use smart preview**.

Create subsets from group

This is a naming system borrowed from Geosoft's XYZ file format. Records starting with the characters "Li", "Ti" or "DH" (not case-sensitive) signify the start of groups of data. These groups typically are flight lines (for "Li"), tie lines (for "Ti") or drill holes for "DH"). In *Potent* terminology these lines become *subsets*. Predefining subsets in this way can greatly reduce the amount of work you need to do to visualise your data in Potent.

Data fields group

A Data fields setting identifies the item that a row in the main area of the dialog box applies to. The first four (spatial) items, X, Y, Z and G, are fixed. The remainder, Field 1 through Field 6, identify field components to be input from the file. Drop down the list box and select a field component. Unused fields must be set to None. The fields that *are* specified must occupy adjacent boxes, starting at Field 1.

Column positions group

- **Col. order** – This option applies when the column format is Free field (see above). The number in each row specifies the column from which the field is to be read. (Refer to the example under General text files in the User's Guide.)
  Enter 0 if a field is not to be input. Values will be set to 0.
- **Start col. / End col**. – This option applies when the column format is Fixed column (see below). The data fields occupy fixed ranges of columns in each record of the file. No delimiters are required. Enter 0, 0 if a field is not to be input. Values will be set to 0.

Scaling group

Factor / Offset – Use these to scale the field values as they are read from the file. The value stored by *Potent* is calculated according to the scheme

value => (value - offset) * factor

Missing value marker group

For each component you can enter a text string that the file metadata might state will indicate a missing value. Usually the missing value marker is the same for all channels, in which case you can simply enter the value for the **X** channel and click **Copy X entry to all** to populate the remaining channels with the same value.
If a missing value marker is found in any data field that Potent is trying to populate when reading the file, the entire record will be skipped.

This feature is most useful when missing values in the file are flagged with a string that looks like a number (perhaps in exponent form) such as 99999 or 1E8. Unless directed otherwise, Potent will interpret the string as a number and accept it as a valid value

If missing values are flagged with a string that cannot be interpreted as a number, such as an asterisk (*), Potent will skip the entire record anyway when it fails to recognise a required field as a number. In this case it is not strictly necessary to enter any **Missing value marker** strings.

Subtract averages from field values

If selected, then for each of the field channels Potent will subtract the average field from each value. For example if, in a magnetic survey data file, the GRF has not been removed during processing then the anomalous field of interest for interpretation will typically be a

small ripple on the much larger GRF. Subtracting the average is a first-order background field removal. Detailed background removal can then be done using the Extract regional tool.

Preview group

- **Lines** - The number of lines to preview, up to a maximum of 99.
- **Font** - Choose the font used to display the preview. Potent will attempt to list only fixed pitch fonts (i.e. fonts in which all characters are the same width).
- **Use smart preview** - Select this to activate a rich set of features to assist you in decoding the file. (Use of this feature is recommended. It is cleared by default so that you have a simple initial view of the file.)
  Refer to Importing Observations from Text Files in the User's Guide for a detailed description of reading data from a text file using the **Use smart preview** option.
- **Update preview** - Click this button if you think that the **Preview** window has not been updated automatically. It is used mainly when **Use smart preview** is selected.
- **Save as template** - When you have set up the **Options for reading observations** dialog box correctly for a particular data file you can save the settings to a Potent Input Template file (which by default is assigned the extension **.pit**).
- **Load from template** - Populate the dialog box from a previously saved Potent input template file.
- **Preview window** - This window displays (scrolling if necessary) the specified number of lines from the file. If **Use smart preview** is cleared then its contents should be an exact reproduction of the text from the file, except that a Tab character will be shown as a single caret, ^. If **Use smart preview** is selected then refer to Importing Observations from Text Files in the User's Guide for more information.

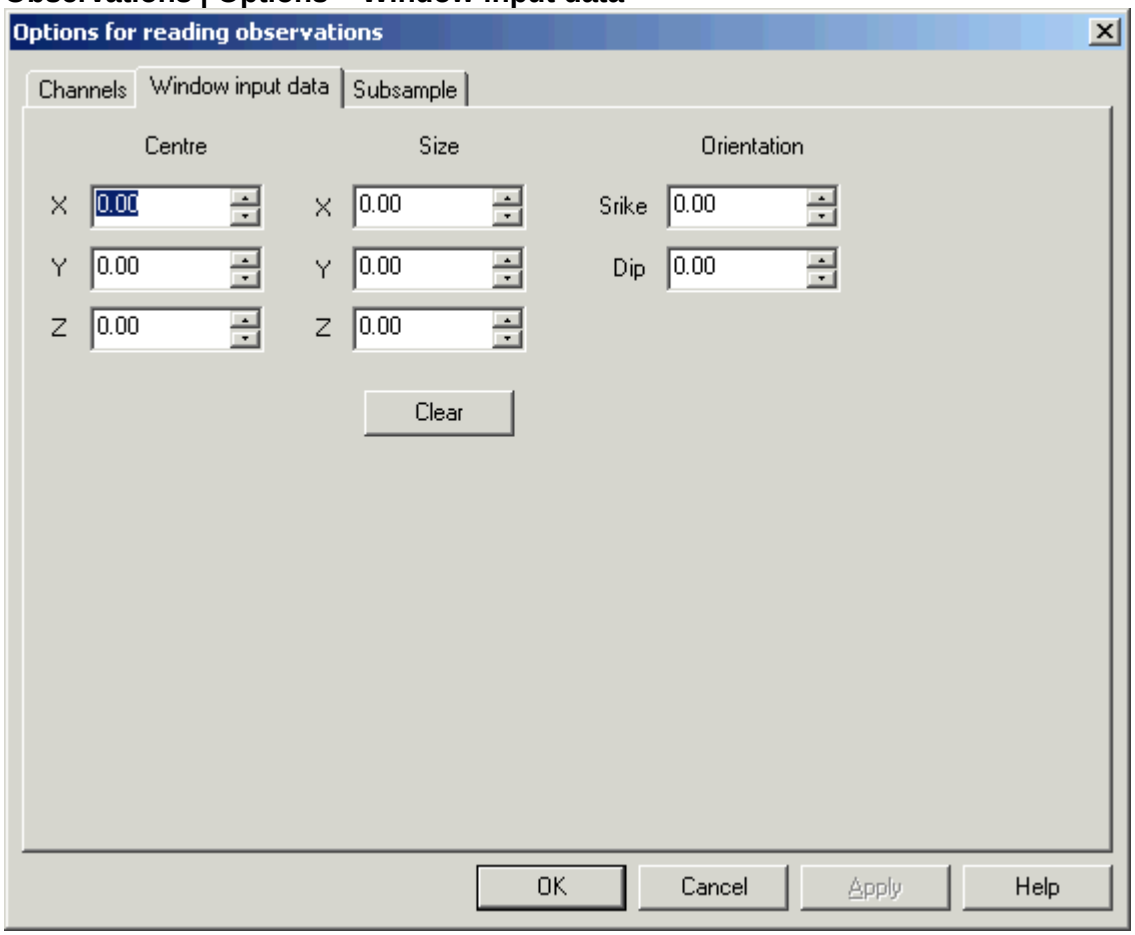
  The window has vertical and horizontal scroll bars, which will function if the text exceeds the bounds of the displayable area.

  The left and top margins contain row and column markers.

See also - Importing Observations from Text Files in the User's Guide

--- Return to the Observations menu.

Observations | Options – Window input data



Use this dialog box to specify a rectangular "box" that forms a 3D window for the data. Only observations that lie within the window will be imported.

Although you can enter values manually, if you are working in the (X,Y) plane consider using the Window observations tool populate the fields graphically.

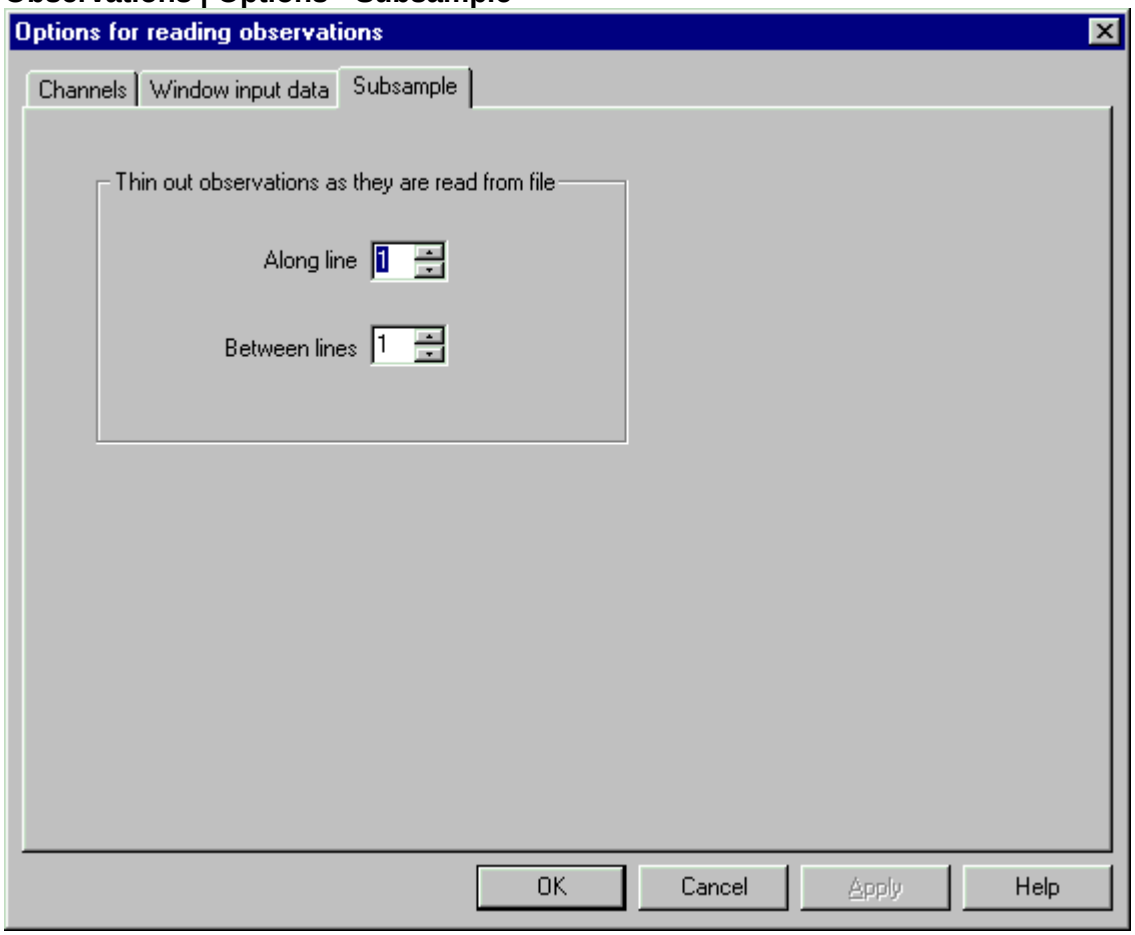
- **Centre** – Specify the centre of the box.
- **Size** – Specify the dimensions of the box (in metres). Note that "0" implies "infinite". Therefore if your window is horizontal, and dip and plunge are zero, setting Z to 0 will make the window insensitive to the vertical coordinate.
- **Orientation** – Specify the orientation of the central axis of the box. Dip is zero if you are windowing surface or airborne observations. Set it to an appropriate value if you are windowing down-hole data (perhaps to separate observations from multiple holes).

Click **Clear** to set all values to "0", which disables windowing.

See also - Working with large datasets in the User's Guide.

--- Return to the Observations menu.

Observations | Options - Subsample



- **Along line** – When importing observations, thin out by this factor. For example, an along-line sample interval of 3 would cause *Potent* to import only every third observation point.
- **Between lines** – This option is of significance only for XYZ files. A between-line sample interval of 3 would cause *Potent* to import only every third line.

See also - Working with large datasets in the User's Guide.

--- Return to the Observations menu.

Observations | Close dataset

Use this command to remove the active dataset from the workspace, and to close all windows associated with it.

--- Return to the Observations menu.

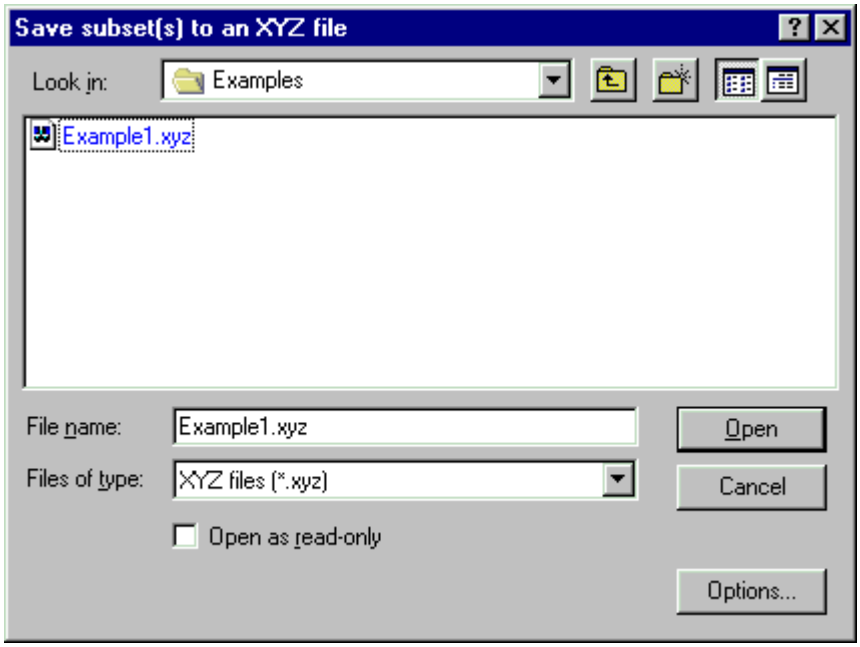
Observations | Dataset title

Use this command to set a title for the active dataset.

--- Return to the Observations menu.

Observations | Save subset(s) as

Use this command to export observations belonging to one or more subsets to an XYZ file. You are presented with a file-open dialog box that has an **Options** button.



Click the **Options** button to display the "Options for saving observations" tabbed dialog box. There are three tabs:

- Column order
- Output format
- Subsets to save

Set the required options and click **Open** to write the data to the file. The first record of the file is the dataset title. The second record identifies the contents of each column. "(C)" appended to the column name indicates that it contains calculated values. The remainder of the file contains each of the subsets you have specified (as described under Subsets to save). Here are the top few lines from a file in which the calculated TMI and Gz components have been written.

/ Magnetic and gravity ground survey data

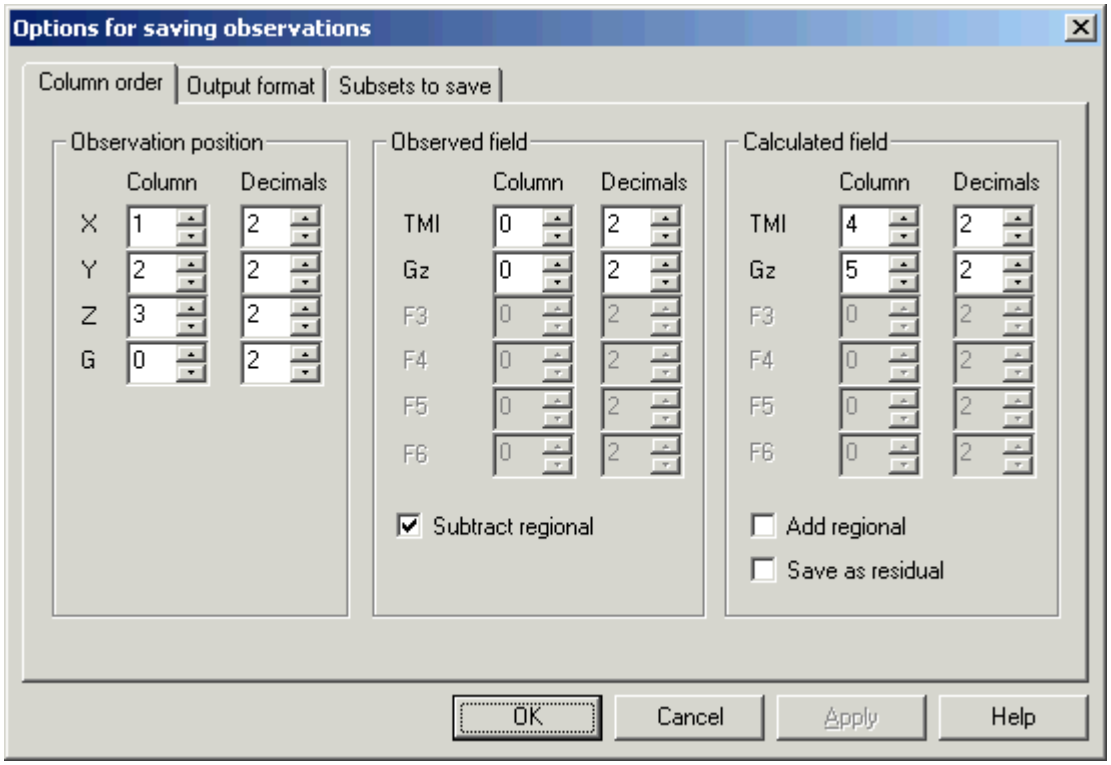
/ X	Y	Z	TMI(C)	Gz(C)
Line 500				
500.00	0.00	0.00	-1.08	0.050
500.00	10.00	0.00	-1.15	0.052
500.00	20.00	0.00	-1.23	0.055
500.00	30.00	0.00	-1.31	0.058
500.00	40.00	0.00	-1.40	0.061

See also: Exporting observations, in the User's Guide.

--- Return to the Observations menu.

Save subsets as - Column order

The output file will contain one line of ASCII text for each observation. Use this dialog box to specify the order of the fields within each record.



If TAB delimiters are specified on the Output format tab then this example would produce an output file looking as follows…

/ Title

/ X Y Z TMI(C) Gz(C)

Line 500

500.00 0.00 0.00 -1.08 0.050

500.00 10.00 0.00 -1.15 0.052

500.00 20.00 0.00 -1.23 0.055

500.00 30.00 0.00 -1.31 0.058

500.00 40.00 0.00 -1.40 0.061

The **Observed field** and **Calculated field** groups contain a row for each component in the dataset. Set the column position and number of decimal places as required. Set the column position to 0 for any fields that you do not want to write to the output file.

Select **Subtract regional** if you are writing one or more observed field components and you want the regional background to be subtracted from the values before they are written. Note that you should first have defined a regional background by using the Tools | Regional command, otherwise the option has no effect.

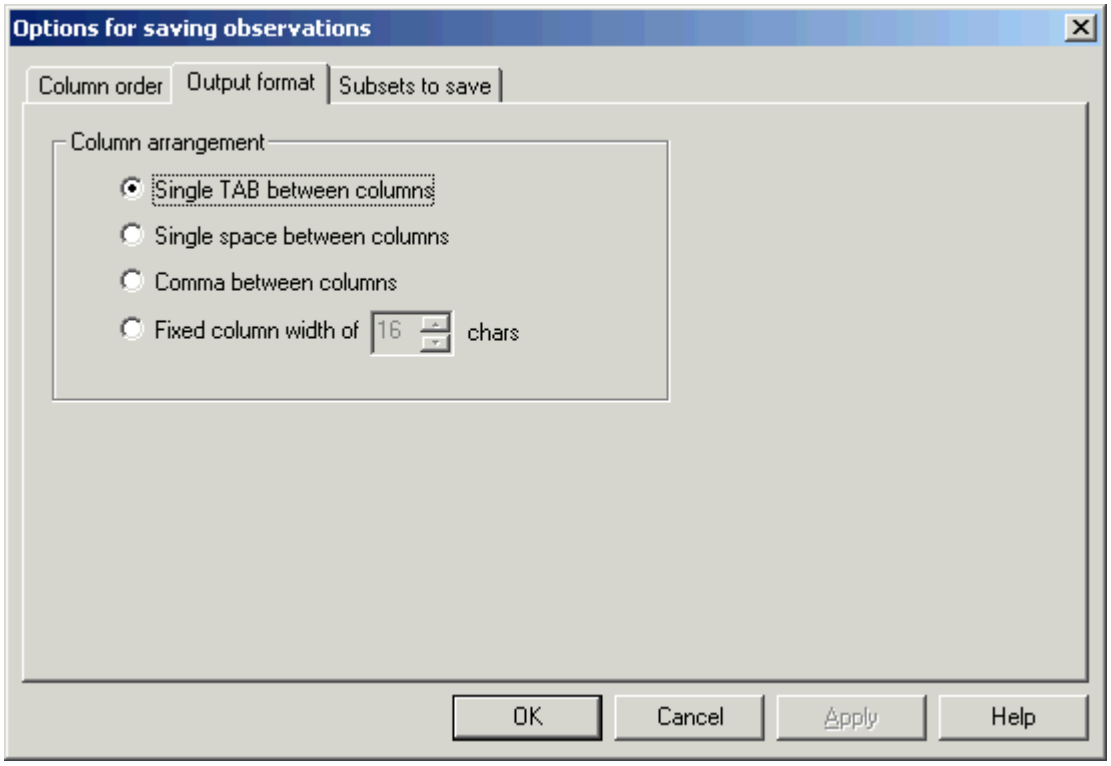
Select **Add regional** if you are writing one or more calculated field components and you want the regional background to be added to the values before they are written. Note that you should first have defined a regional background by using the Tools | Regional command, otherwise the option has no effect.

Select **Save as residual** if you are writing one or more calculated field components and you want to save the residual field rather than the whole calculated field. This feature is useful

for stripping. You might model some major features, save the residual field, then use the Observations | Import command to load the residual field for modelling of the finer features.

--- Return to Observations | Save subset(s) as.

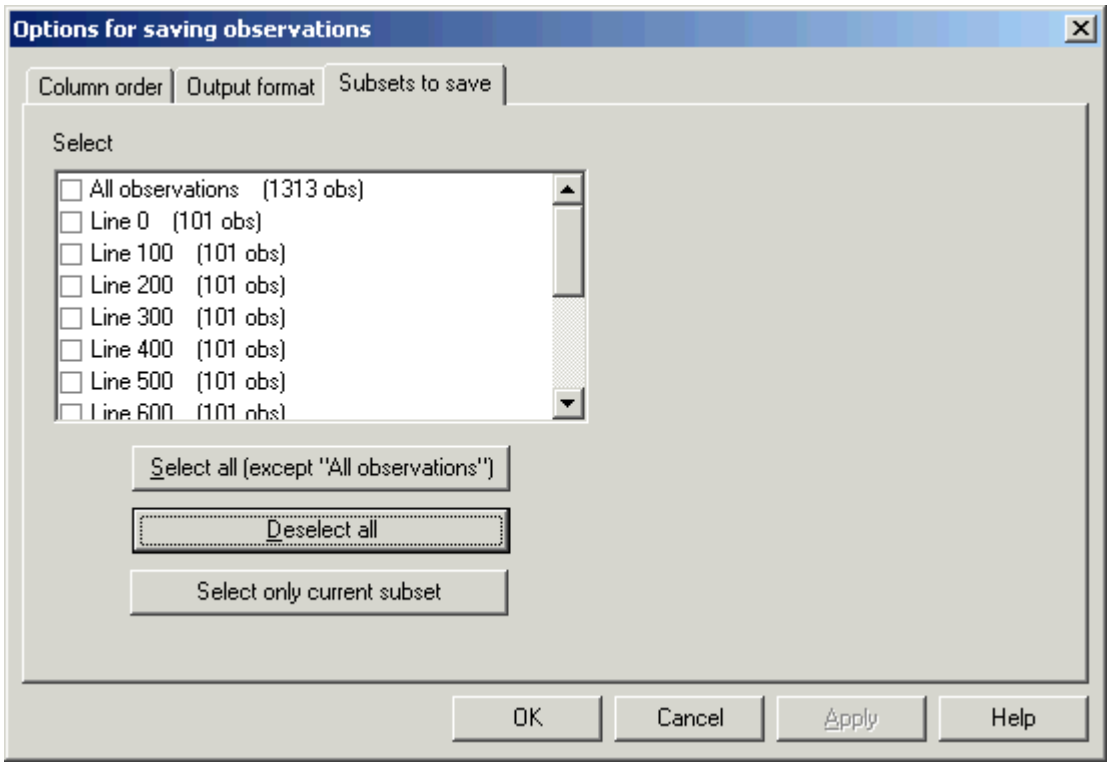
Save subsets as - Output format



Use this dialog box to define the spacing of the fields within each record.

--- Return to Observations | Save subset(s) as.

Save subsets as - Subsets to save



Use this dialog box to specify exactly which subsets are written to the output file. Use the three buttons below the list-box to speed the selection process. Subsets are written to the output file according to the following criteria:

- If no subsets are marked (the default) then the subset corresponding to the active window is written. If the active window is the main plan, then the "All observations" subset (as described in the Observations | Subset management command) is written.
- If one or more subsets are marked then each one is added to the output file. If the subset title does not begin with the letters "LI" (for "line") or "TI" (for "tie") then the string "Line – " is prepended to the title at the start of each subset. This ensures that every subset will be recognised as a Line or Tie when the file is read by *Potent* or other programs that are aware of the XYZ file structure.

Use the three buttons below the list-box to speed the selection process.

Select all (except "All observations") – This does what its label implies. Use this feature to write out all the subsets. When the file is imported again *Potent* will automatically create an "All observations" subset, so it is often unnecessary to include "All observations" on output.

Deselect all – Use this to start again from scratch. If you leave all the subsets deselected then *Potent* will output only the subset corresponding to the active window.

Select only current subset – This is mainly intended as a way of identifying the subset associated with the active window. It is functionally equivalent to having nothing selected.

--- Return to Observations | Save subset(s) as.

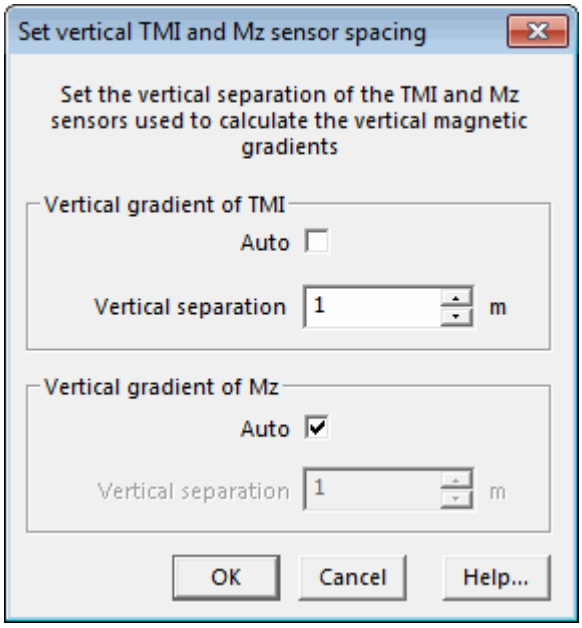
Observations | TMI & Mz sensor spacing

By default, Potent calculates the gradient of a field component by performing a simple numerical differentiation. This involves calculating two closely spaced values (where "close"

is deduced from the spatial extent of the dataset) and dividing their difference by this distance. We will refer (loosely) to this calculation method as the *analytic* method.

However, it is common practice in detailed ground magnetic surveys to use two vertically separated sensors to measure the vertical gradient of the TMI or Mz (the MAGZ component). As the spacing of the sensors is not always "small" relative to the distance from the magnetic sources this gradient might differ significantly from the analytic gradient that is calculated by Potent.

The **TMI & Mz sensor spacing** command is available when the observed data includes either the TMI_dz or Mzz component. The command displays a small dialog box that allows you to enter the actual separation of the sensors used for calculating the gradient. Select **Auto** (the default) to have Potent calculate the gradient analytically. Otherwise set **Vertical separation** to reflect the actual geometry of the instrument used in the survey.



To use this command you first input the data; Potent will proceed to calculate TMI_dz or Mzz analytically. Then with any window of that dataset active run the **TMI & Mz sensor spacing** command from the **Observations** menu. The sensor spacing will be applied only to TMI_dz or Mzz calculations for that dataset.

Note 1: The position of the observation is assumed to be the position of the upper sensor. If F1 is the field calculated at the upper sensor, and F2 the field calculated at the lower sensor, then:

Calculated gradient = (F1 – F2) / sensor spacing

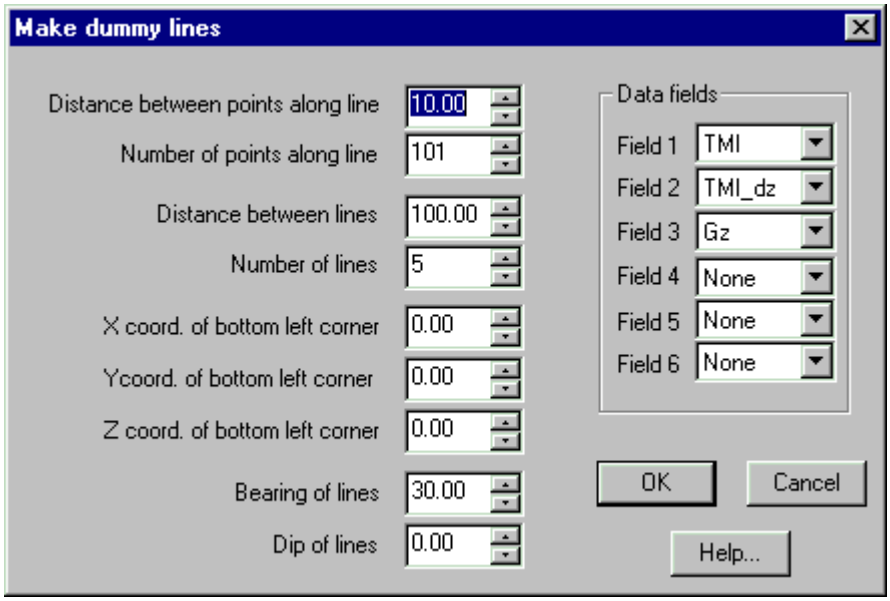
If your observed data is acquired on the basis of some other geometry you should be able to convert it by setting suitable scaling in the **Options for reading observations** dialog box.

Note 2: If it is not practicable to arrange the sensors vertically during a survey (for example, in the case of a bore-hole magnetometer in an inclined hole) then consider not modelling the gradient as a single component. Rather, model the two TMI or Mz measurements by inputting the data from the two sensors as two separate datasets. (This involves reading the data file twice.) In one of the datasets, in the Observations | Options for reading observations | Channels dialog box, specify offsets for the X, Y and Z channels that reflect the actual spatial arrangement of the sensors.

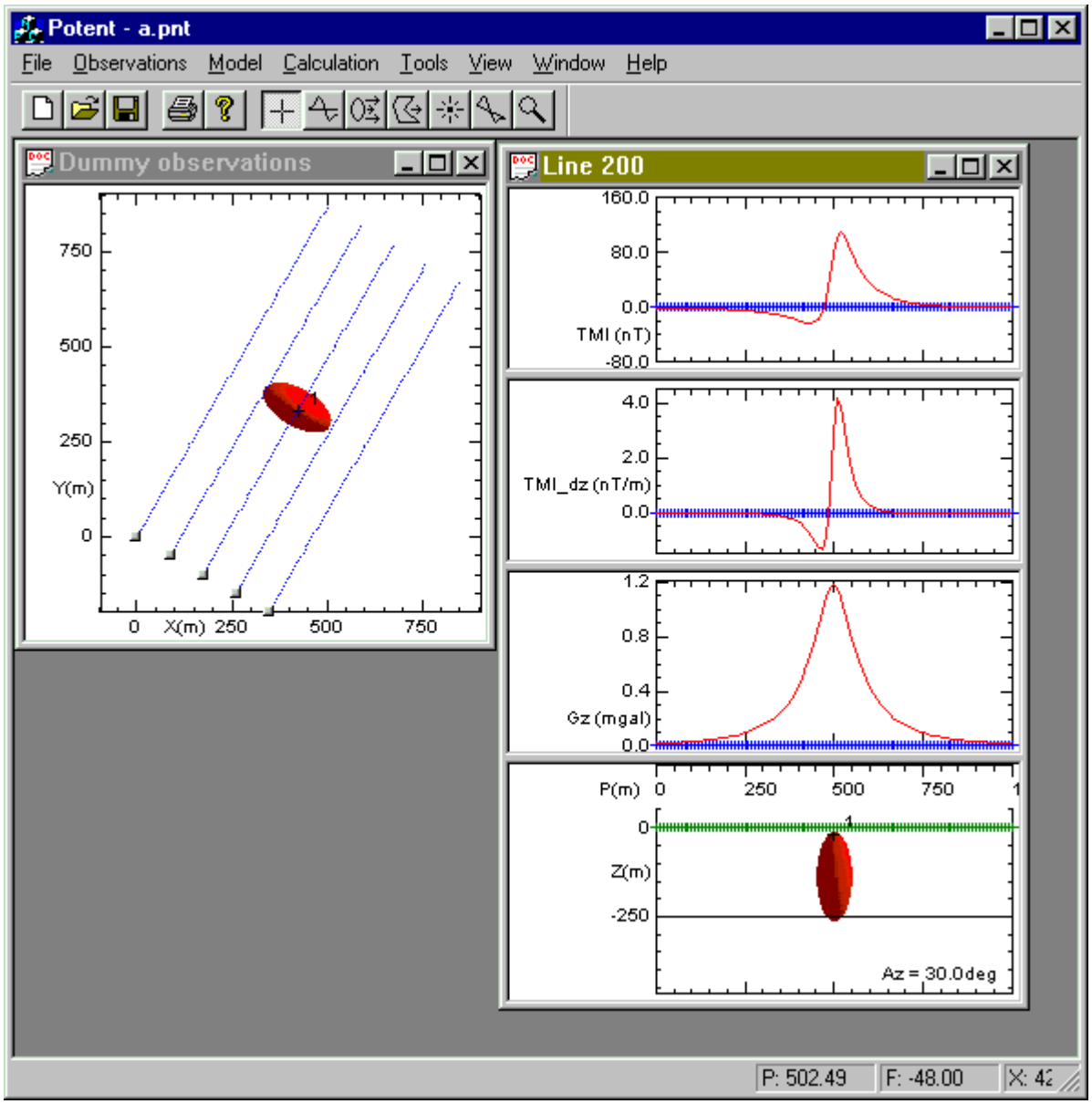
Note 3: It is not practicable to use this technique for gradients other than vertical as Potent has no way of knowing the horizontal arrangement of sensors. As described above, consider inputting data from the two sensors as two separate datasets.

Note 4: This Sensor spacing feature might impose a significant calculation overhead if the number of observations in the dataset is large. This is because Potent usually recalculates only when the model has changed, which greatly speeds up window redrawing. However, for programmatic reasons, when using the Sensor spacing feature Potent does not store the second field value that is required to calculate the gradient and so calculates it whenever a window associated with that dataset is redrawn. Consider using one or more of the calculation reducing methods described in Working with large datasets in the User's Guide.

Observations | Make



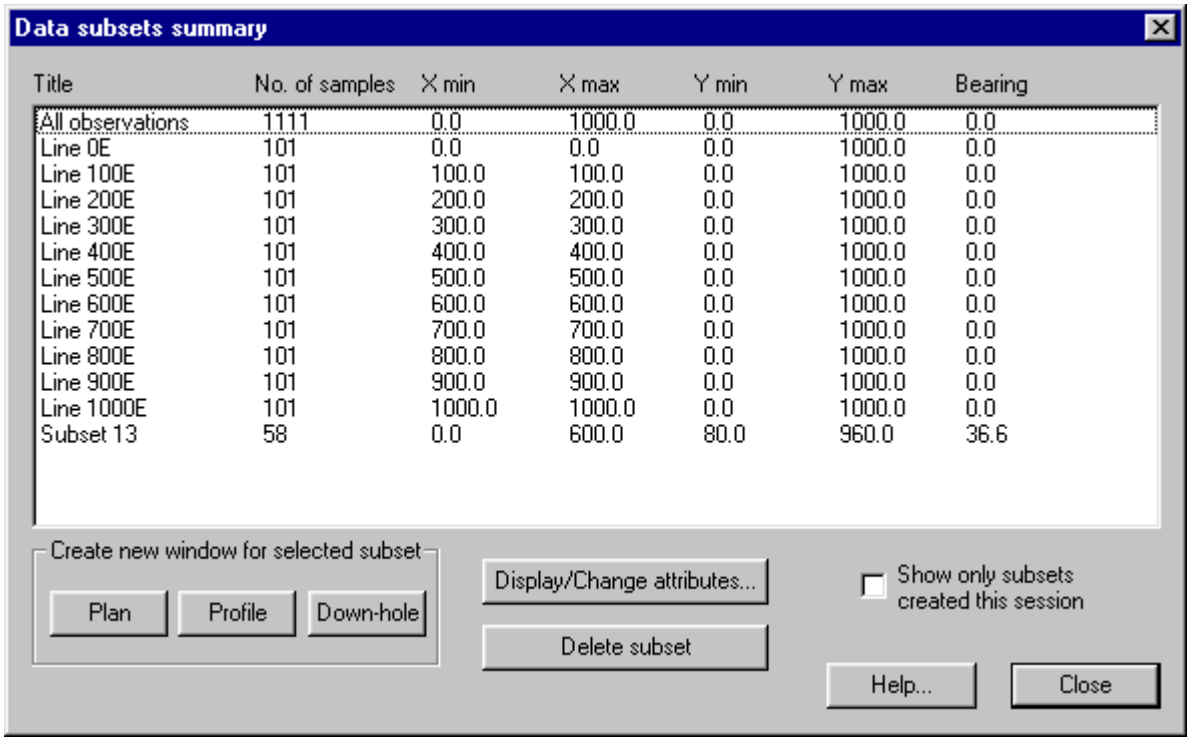
Use this command to create a set of dummy observations. The observations are created as a set of parallel lines, as specified by the left-hand column of edit-controls. Use the **Data fields** group to specify the components that you wish to model. The following example shows the observations that were created from the dialog box shown above. The ellipsoidal body was added subsequently.



The **Dip of lines** field is intended to simulate a drill-hole. To create a dummy drill-hole, set **Number of lines** to 1, **Bearing of lines** to the drill-hole azimuth, and **Dip of lines** to its inclination from the horizontal.

--- Return to the Observations menu.

Observations | Subset management

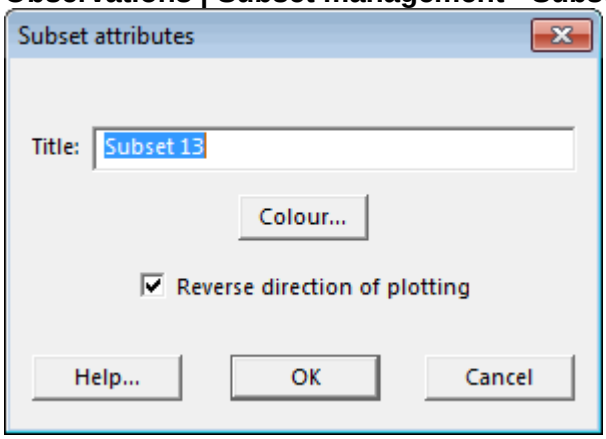


This dialog box displays a list of subsets for the active dataset. The first subset is always "All observations".

- **Plan** – Create a new plan window for the highlighted subset.
- **Profile** – Create a new profile window for the highlighted subset.
- **Down-hole** – Create a new down-hole window for the highlighted subset.
- **Display/change attributes** – Display the Subset attributes dialog box that allows you to change such things as the name of the subset and the colour with which it is represented in plan windows.
- **Delete subset** – Delete the highlighted subset. (Note that this is the only way to delete a subset. Closing the plan or profile windows in which a subset is displayed does not actually delete it; the subset continues to exist and can be re-displayed at any time by using the **Plan** or **Profile** buttons as described above.)
- **Show only subsets created this session** – When you load observations from an XYZ file using the Observations | Import command, a subset is created for each embedded "line" record. Similarly, when you create observations using the Observations | Make command, a subset is created for each line. Selecting the **Show only subsets created this session** box means that only the "All observations" subset and those subsets you have created manually, using the subset tool, are listed.

--- Return to the Observations menu.

Observations | Subset management - Subset attributes



This dialog box is displayed by clicking the **Display/change attributes** button in the Observations | Subset management dialog box, or by right-clicking on a subset button on a Plan window.

- **Title** – The title of the subset. The title is used as the window caption, and to label various types of output.

- **Colour** – The colour associated with the subset is used when drawing it in plan windows. (Refer to the **Subsets** tab of the Window | Options command for details.)

- **Reverse direction of plotting** – By default Potent will configure a subset so that when it is viewed in section form (i.e. using a Profile window) the viewer's eye is at a distant point normal to the profile and in the SW or SE semi-plane. This corresponds to profiles with bearings ranging from -45°, through 45°, to 135°. Therefore a profile will have West or South on the left.

  Usually this provides an intuitive viewpoint. However, consider a number of profiles that have bearings close to NW (-45°), such as those from an airborne survey that has been flown with SE-NW lines. Profiles with bearings slightly North of NW (-44.9°, say) will be drawn with SE to the left. However, profiles with bearings of slightly South of NW (-45.1°, say, which is interpreted as +134.9°) will be drawn with NW to the left.

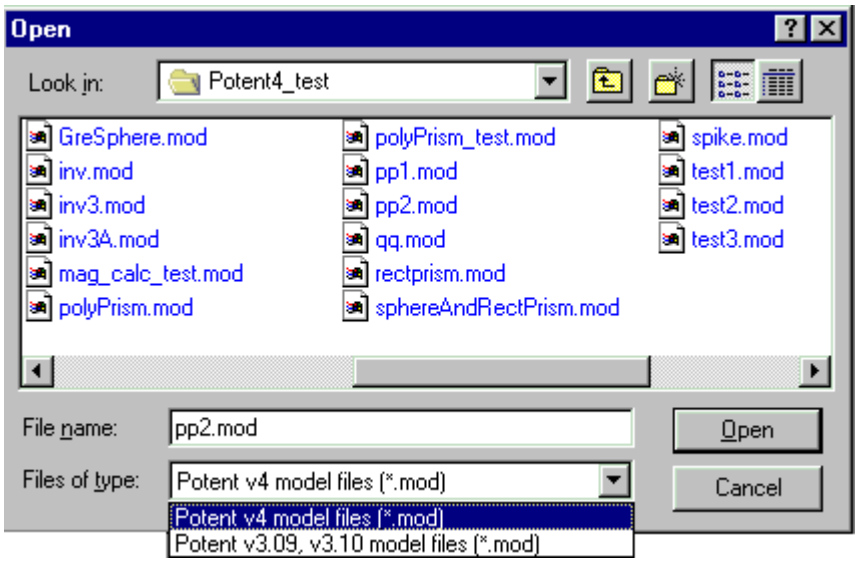
  When modelling a number of adjacent sub-parallel lines this can result in confusing changes in direction when drawing the profiles. Select **Reverse direction of plotting** to cause subsets to be drawn in the opposite direction to the default.

--- Return to the Subset management dialog box.

Model menu

Model | Open

Load a model from a disk file that was created by the Model | Save command.



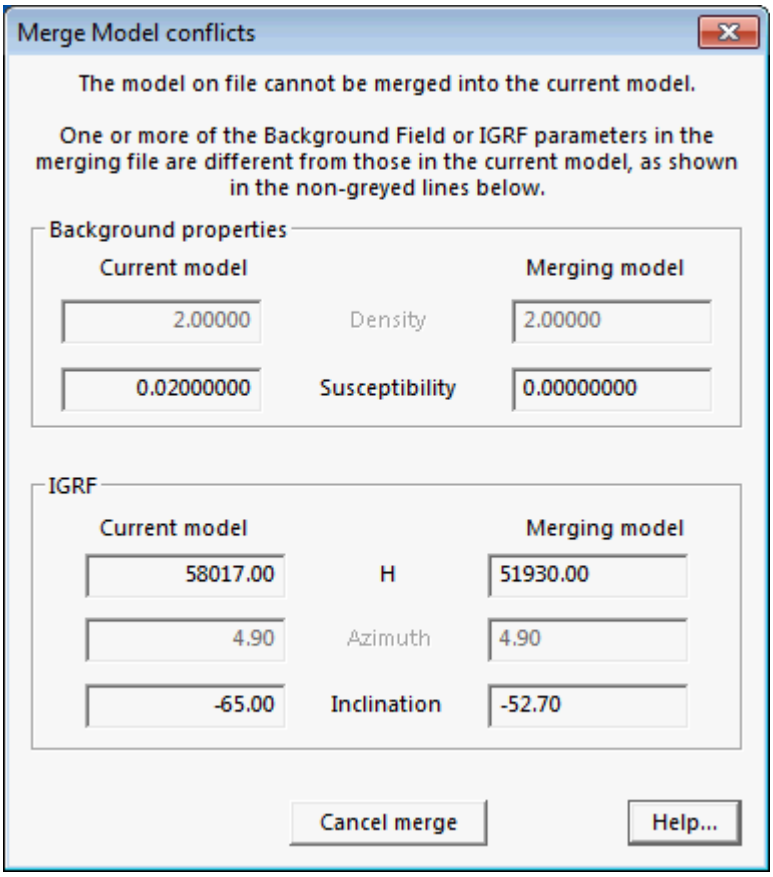
--- Return to the Model menu.

Model | Merge

Use this command to merge the contents of a model file previously created by the Model | Save as command with the model already in your Potent workspace. Note the following issues:

- If the ID number of a body to be merged clashes with that of a body in the current model then the merged body will be silently re-numbered with the first unassigned body ID. (The search starts at ID 1 and continues until the first unassigned ID is found.)

- As well as the bodies, a model includes the background properties and the IGRF. If the versions of any one of these in the merging file differs from that in the current model then the models cannot be merged. The following information box is displayed:



Any conflicts are active (not greyed) and must be resolved in either the current model or the model file before the merge can succeed. Small differences in the IGRF usually can be made without significantly affecting the calculated field. However if you change a background property then you will need to edit each body in the model and add the difference in background values to the appropriate physical property.

Model | Save

Use this command to save your model to disk as a text file. All bodies are saved, along with the current inducing field parameters (as set by the Model | Inducing field dialog).

If you have already saved the model, you are prompted for permission to update the current file; otherwise you are prompted for a file name. If you have not specified a title for the model, you are given the opportunity to provide one just before the model is saved.

--- Return to the Model menu.

Model | Save as

Use this command to save the model to disk as a text file with a name you specify. If you have not specified a title for the model, you are given the opportunity to provide one just before the model is saved.

--- Return to the Model menu.

Model | Clear model

This command deletes all bodies from the model. This is done in memory; the disk file, if any, associated with the model is unaffected.

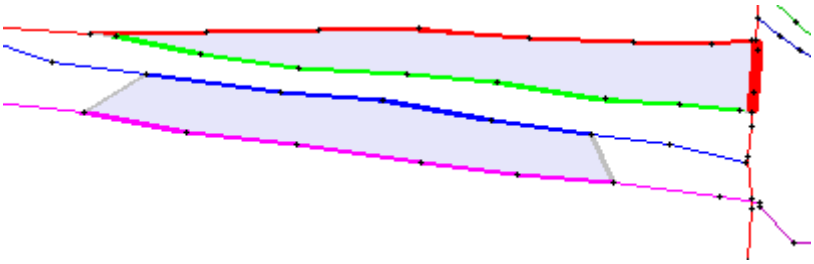
--- Return to the Model menu.

Model | Import as interfaces

This command displays a window that allows you to build layered model structures from stratigraphic interface data. Two data formats are supported.

Note: *This feature was introduced at the request of two specific Potent users, hence the two formats. Please contact GSS if you would like to make use of the feature but need to import data in a new format. However, be aware that we* might *charge a fee for the work involved in developing the software to decode the new format.*

An interface is a boundary between two rock types and as such does not constitute a solid object that can be used for modelling purposes, as is required by Potent. However in the simplest case a stratum can be created as a polygonal prism body by using two interfaces to define the top and bottom surfaces. This diagram shows a detail from a stratigraphic sequence consisting of some sub-horizontal strata and a vertical fault.



Each interface is defined by a series of vertices, shown as small black crosses. Two bodies have been constructed by combining sections of interfaces. If the interfaces don't touch then Potent closes the body by connecting the gaps, as shown in the lower body. (In this image the bodies have been filled with light grey for clarity, using a drawing tool.)

The **Import as interfaces** feature provides a method to create polygonal prism type bodies rapidly from interface data. A manual approach was chosen as experience showed that attempts to create bodies automatically generally failed because of unforeseen complexities in the structures.

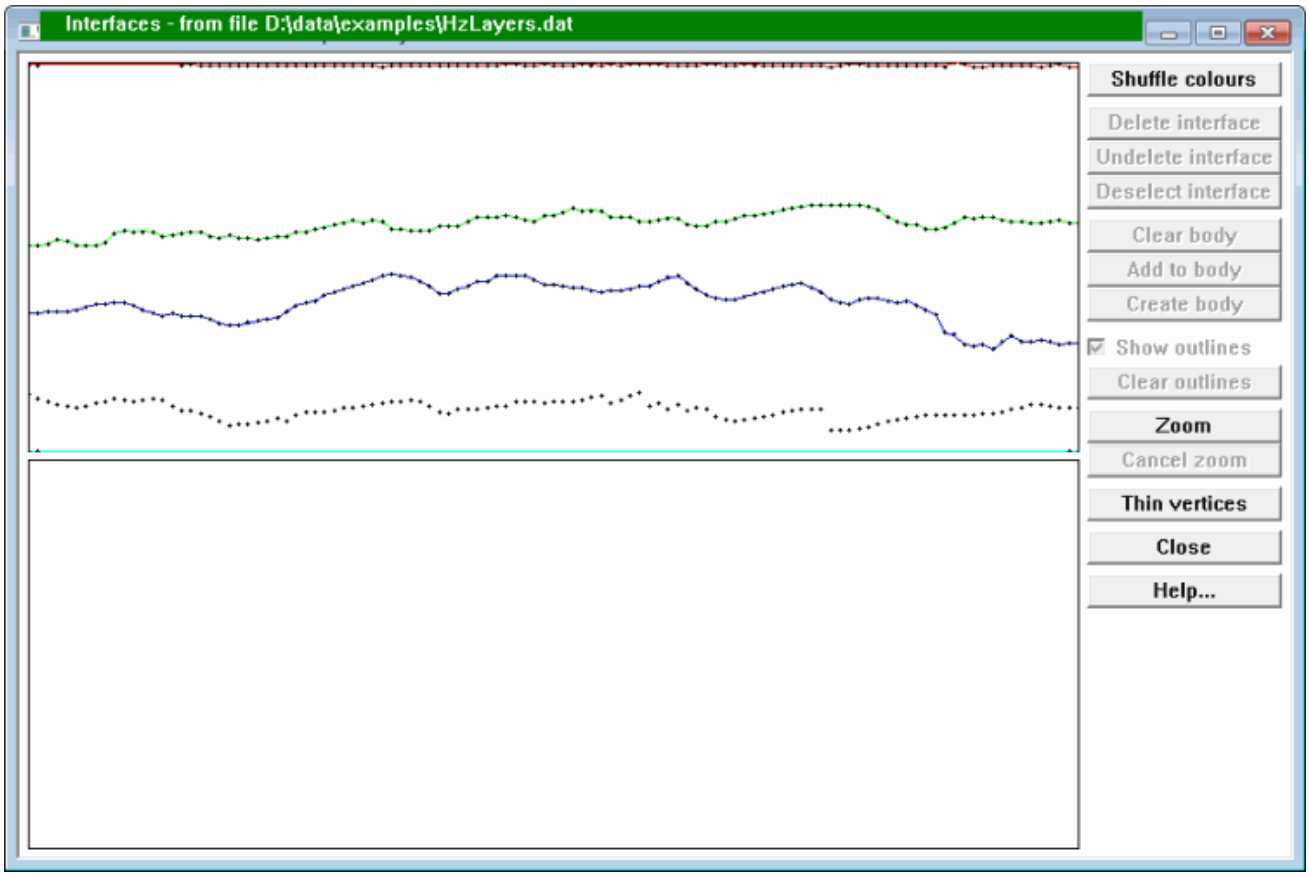
For more information refer to these topics:

The Interfaces window
Interfaces tutorial
Data formats

The Interfaces window

The following window is a typical Interfaces window that was created when some digitiser data was input (see data formats).

(**Note**: This example shows the **Thin vertices** button. Sometimes the density of vertices from the digitiser is far greater than needed for modelling purposes. Click this button to display a dialog box that allows you to enter a factor by which to thin out the vertices. Repeated applications of the **Thin vertices** button are cumulative. Once accepted the thinning cannot be undone except by reloading the file. Subsequent images in this section were created using an older version of *Potent* that did not contain the **Thin vertices** feature.)



The top pane of the window is the plotting area for the interfaces. The data is scaled automatically to fill the frame; if you want to change the vertical exaggeration of the plot then simply change the aspect ratio of the entire window by dragging one of its corners.

The right hand part of the window is occupied by a column of buttons. These provide the functionality for creating polygonal prism bodies from the interfaces.

The interfaces are coloured automatically using a palette of ten colours. If two or more adjacent interfaces have confusingly similar colours then choose the **Shuffle colours** button to redistribute the colours across the interfaces. The vertices of the interfaces (i.e. the points read from the input file) are marked by small black crosses.

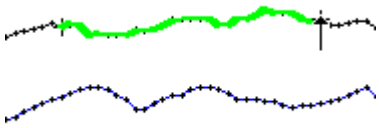
The default mouse cursor in the Interfaces window is the normal Windows pointer. When it is close to a vertex it changes to an Up arrow, ↑, pointing to the vertex. Throughout the following discussion the term "point to an interface" means "point to any vertex belonging to the interface" (so that the cursor changes to an Up arrow).

- **Select an interface** by pointing to it and clicking the left mouse button. The selected interface is drawn in black and the mouse cursor ceases to be sensitive to other interfaces.

- **Deselect an already selected interface** by choosing the Deselect interface button.

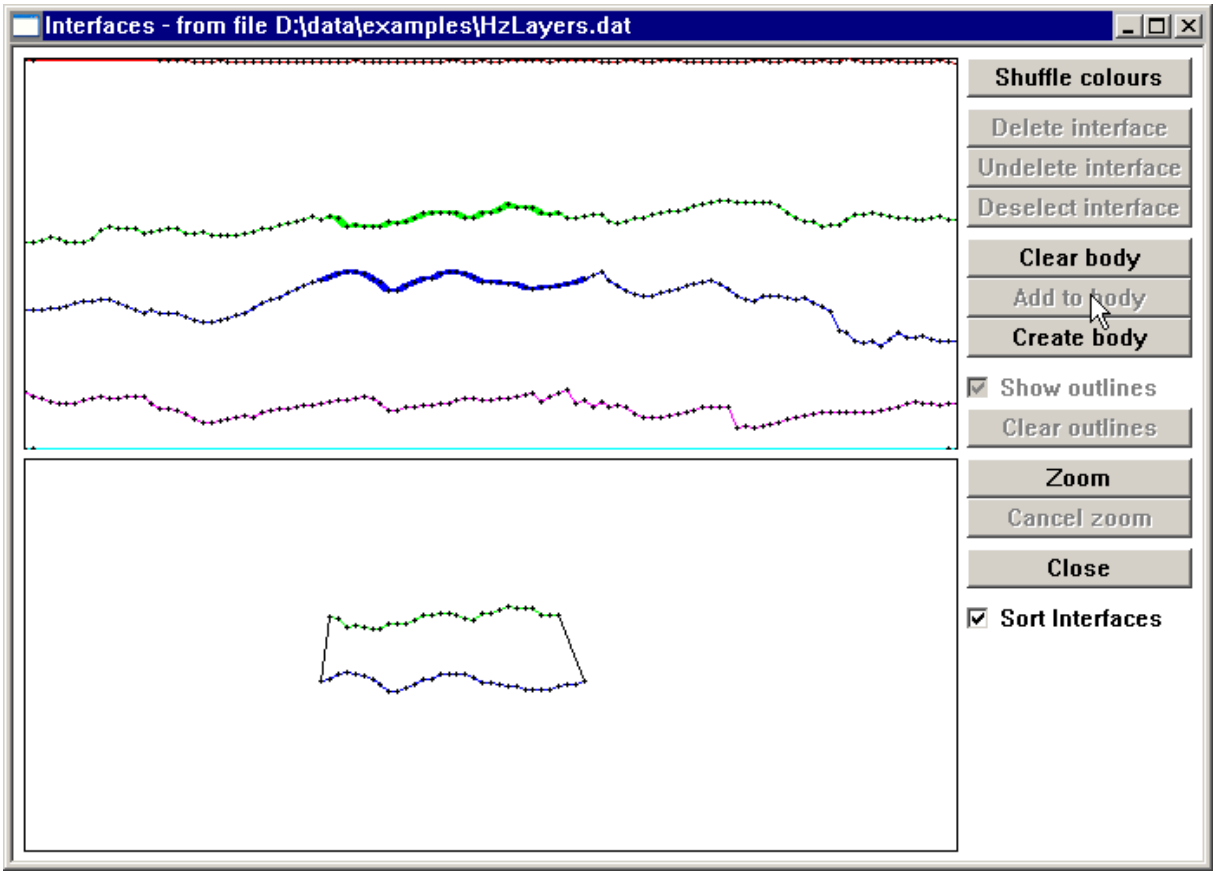
- **Query an interface** by pointing to it and pressing (and holding down) the right mouse button. The queried interface is drawn in its complementary colour and statistics are displayed below the buttons in the bottom right corner of the window. Release the mouse button to return to normal operation.

- **Mark a section of an interface** (for possible addition to a body under construction) by selecting it and then clicking the left mouse button on the two vertices that bracket the required section. The selected part of the interface is drawn with a wide pen and its ends are marked by black crosses.

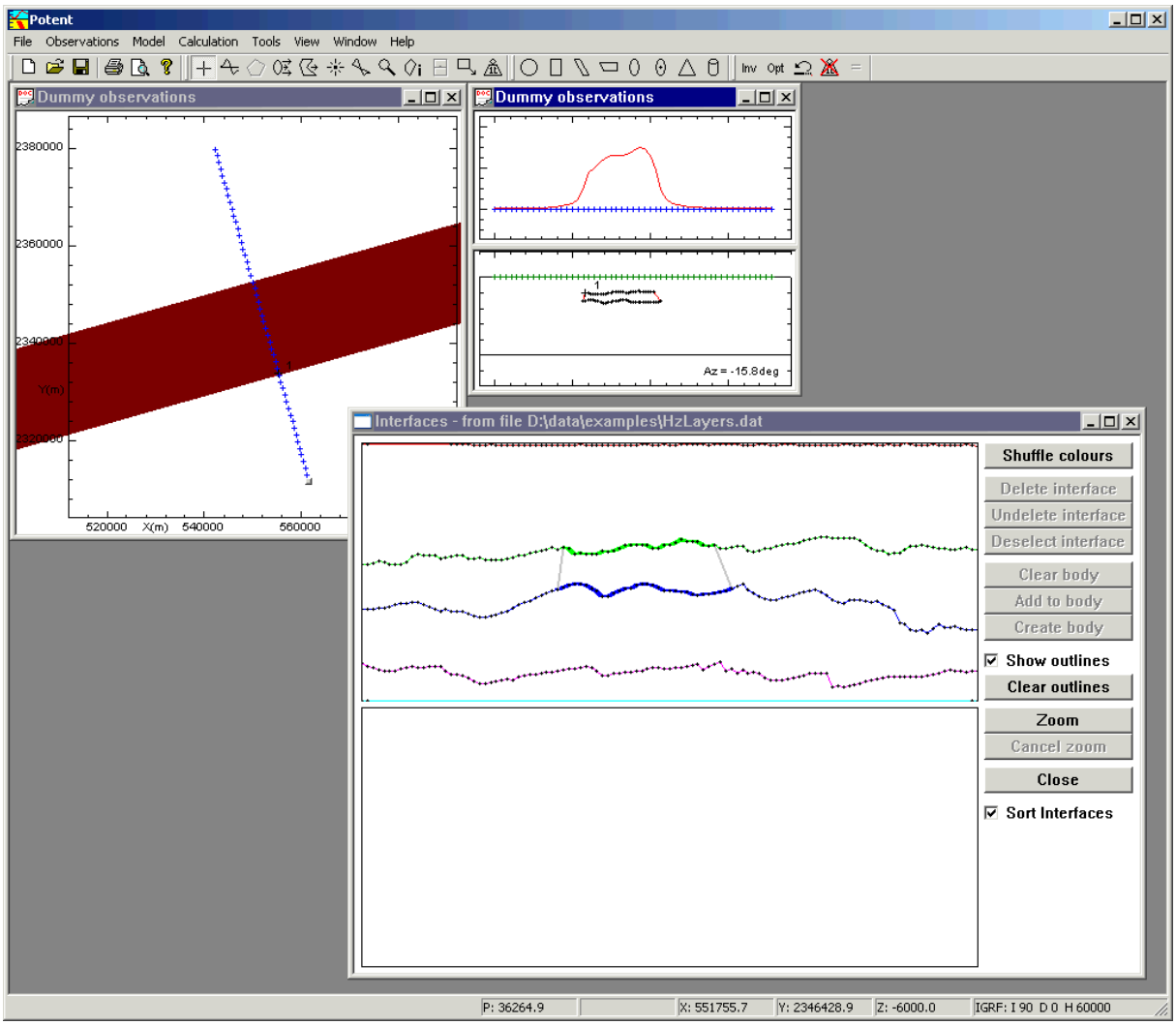


Cancel the selection by choosing the **Deselect interface** button.

- **Build a new body** by marking sections of interfaces (as described in the previous paragraph) and choosing the **Add to body** button for each one. The body under construction is drawn in the bottom pane of the window. You must be consistent in the direction in which sections are marked - either clockwise or anticlockwise - otherwise the lines connecting vertices of the body will cross. Abort the current body under construction by choosing the **Clear body** button.



- **Create a new body** by constructing it from interface sections (as described in the previous paragraph) and choosing the **Create Body** button. Potent creates a new polygonal prism type body and adds it to the model. It is removed from the bottom pane of the Interfaces window.



If **Show outlines** is selected bodies that have been created continue to be highlighted in the top pane. Clear **Show outlines** to temporarily stop this highlighting. Choose the **Clear outlines** button to permanently remove the stored outlines of bodies that have been created so far. Note that neither of these operations has any effect on the actual bodies, which are now part of Potent's model assemblage.

- **Delete an interface** by selecting it and then choosing the **Delete interface** button. The interface is not permanently deleted but is added to a delete list and ceases to be plotted or sensed by the mouse cursor. Restore deleted interfaces sequentially by repeatedly choosing the **Undelete interface** button.

- **Zoom** on the interfaces (top) pane by choosing the **Zoom** button. The cursor changes to a magnifying glass and the name of the button changes to **Stop zooming**. Click the left mouse button to zoom in by a factor of two, placing the area under the cursor at the centre of the zoomed window.

  When the interfaces pane is zoomed you can click the right mouse button to zoom out by factors of two.

  Continue zooming up to a cumulative maximum of five times.

  Choose the **Stop zooming** button to return to normal mouse operation, preserving

any zoom effect that might exist.

Choose the **Cancel zoom** button to return to an unzoomed state.

Interfaces tutorial

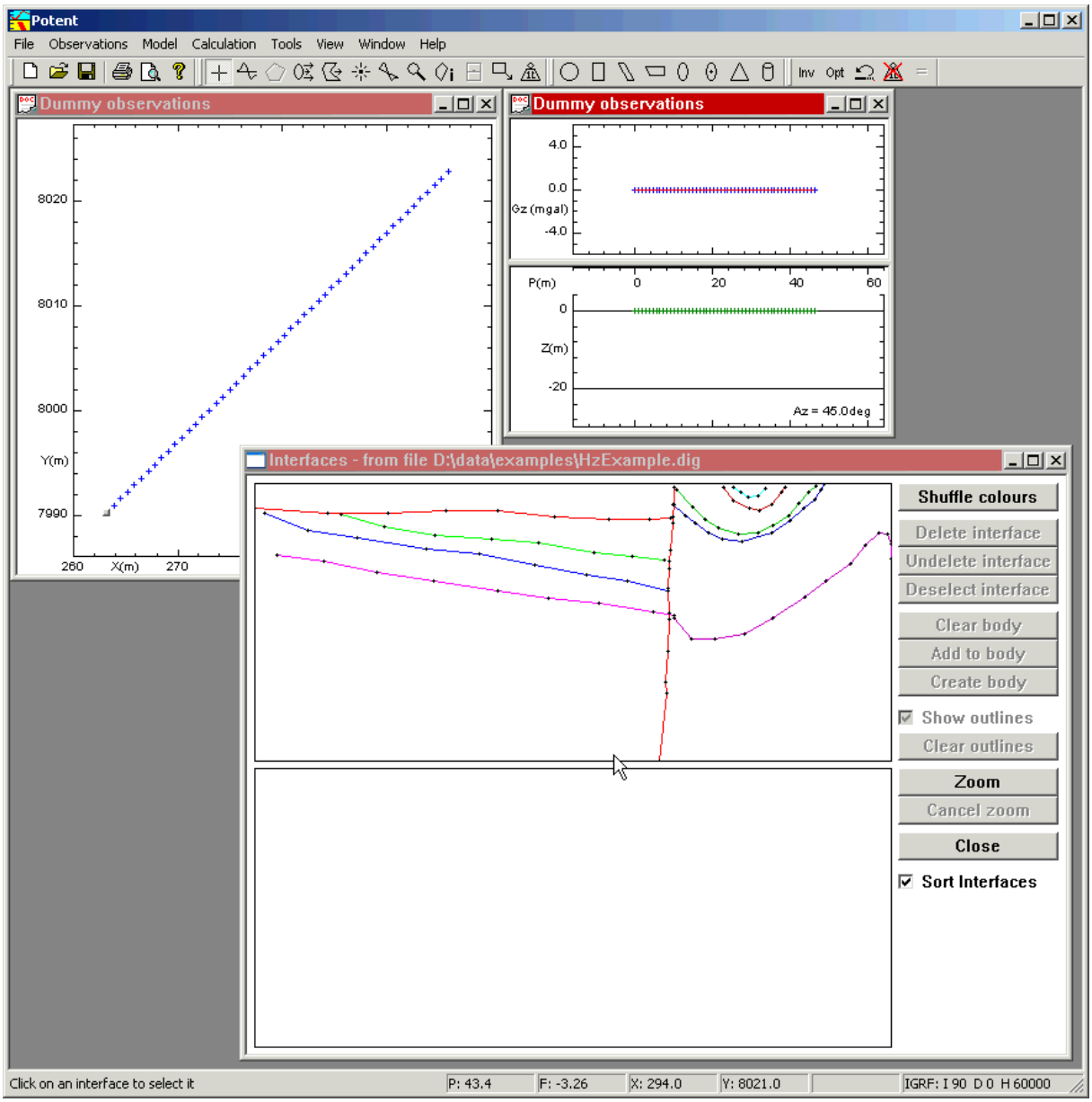
This tutorial is a step by step account of creating a simple layered model from the example data file HzLayers.dat.

If you are already running Potent we suggest that you exit the program and run it again to ensure you are starting with a clean setup.

1. Choose the **Import as interfaces** command from the **Model** menu and open the file HzExample.dig. Potent inputs the interface data and draws the interfaces in the interfaces (top) pane of the interfaces window. A set of dummy data points is created, which can be seen in a Plan and a Profile window. The bearing of the line of observations is along the plane of the digitised section.

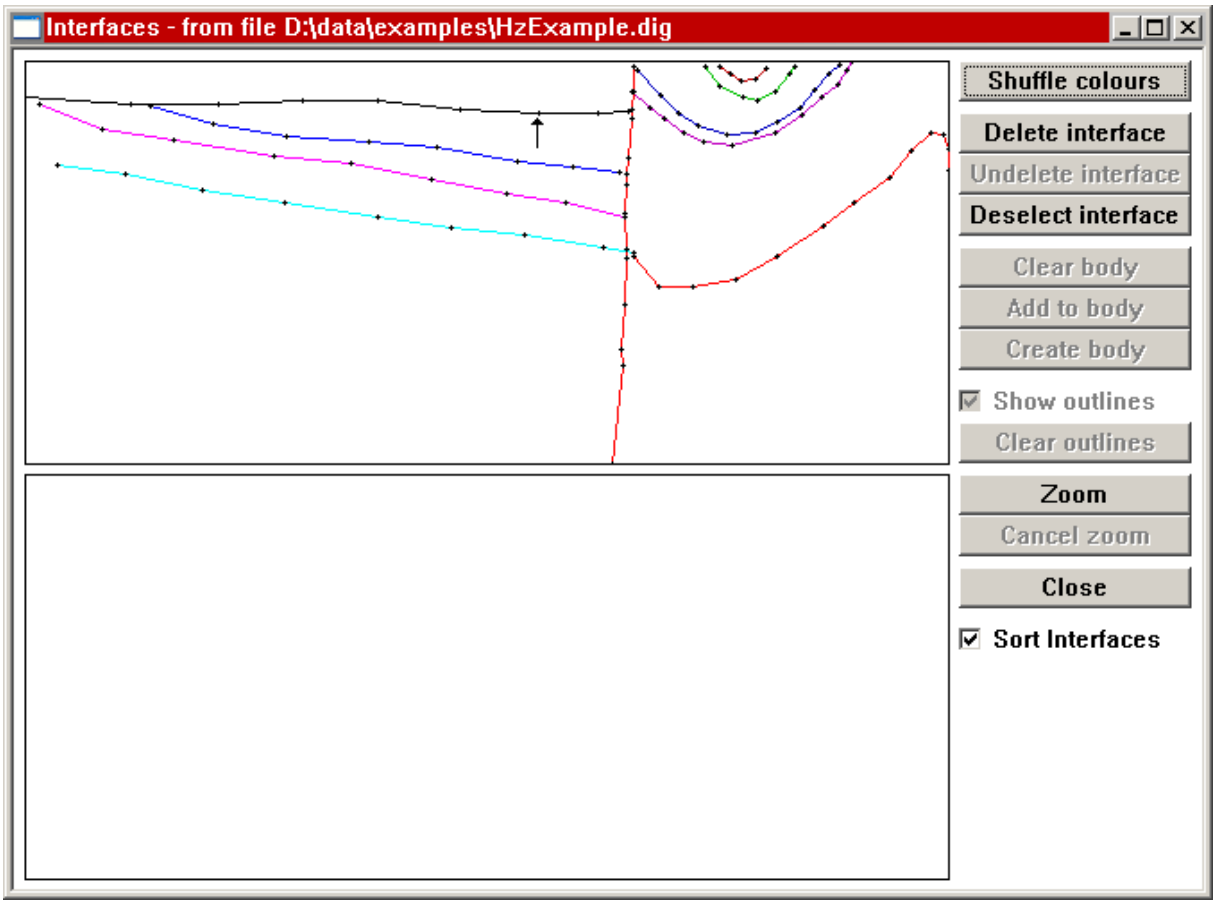
   Note*: The Plan and Profile windows are merely to help visualise the model construction, and the field component is set to Gz (vertical gravity). Ultimately you will save your model and load it into a workspace that uses your actual data, be it any mix of gravity and magnetic components.*

   Re-arrange the windows so that they look something like this example:

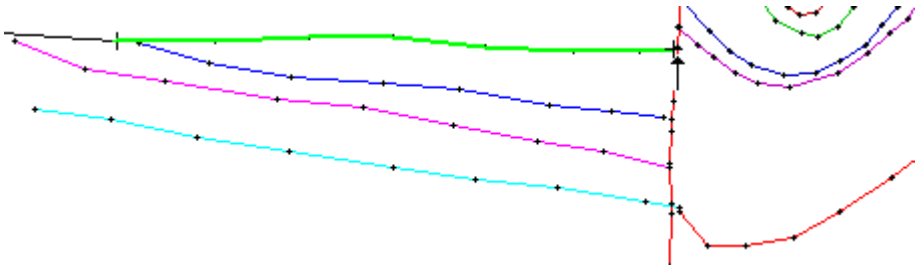


1. Move the cursor around the interfaces pane and observe that when it is close to a vertex it changes to an Up arrow. When the Up arrow is present, press and hold the right mouse button. The interface is highlighted and statistics are displayed in the bottom right of the window. Release the mouse button to return to normal operation.

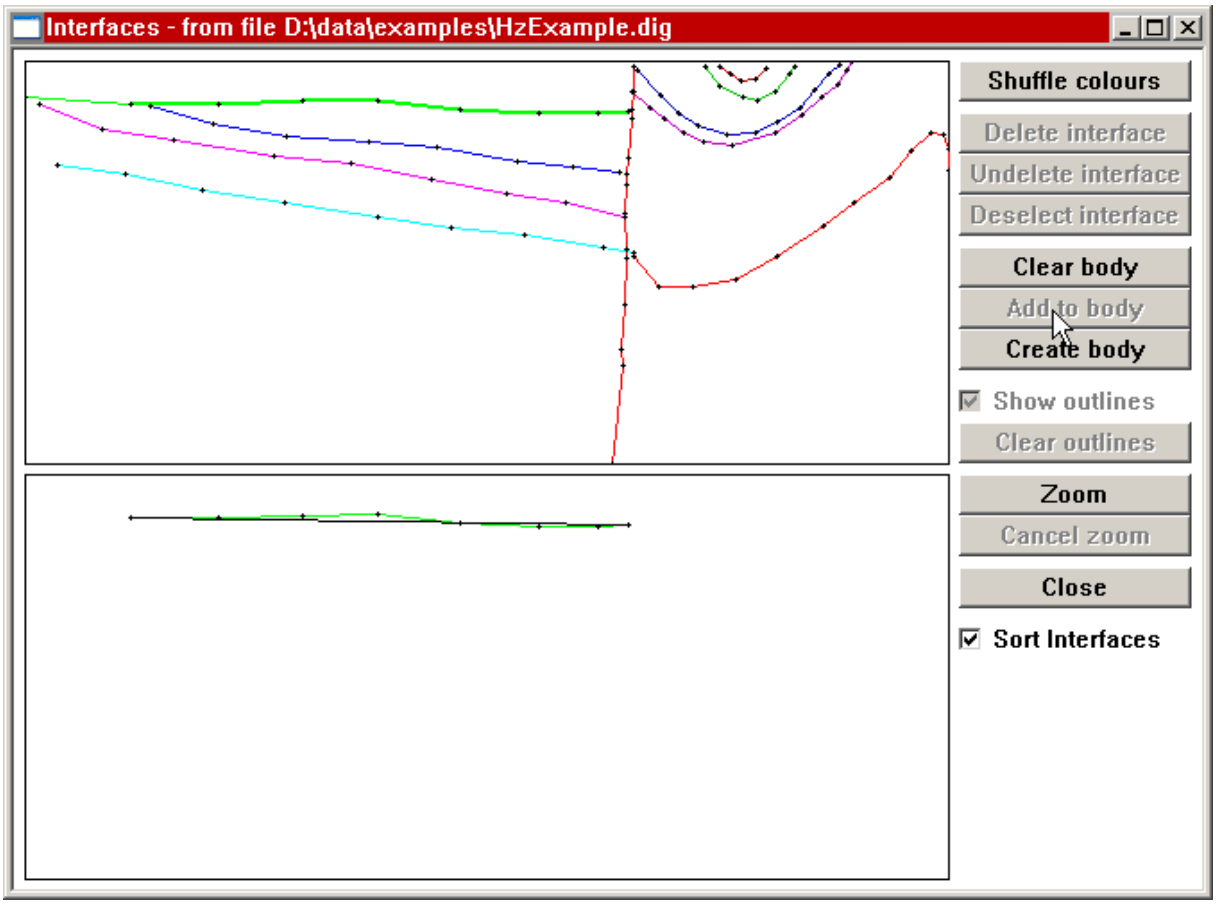
   The first objective of this example is to create a body representing the top wedge shaped area to the left of the fault. Select the top interface on the left side of the fault by pointing to one of its vertices and clicking the left mouse button. The interface is redrawn in black and the mouse cursor ceases to respond to other interfaces. Note also that the Delete interface and Deselect interface buttons become active.



2. With the interface selected click the left mouse button on the vertex that is second from the left edge of the interfaces pane. A cross is drawn to mark the vertex. Then click the left mouse button on the vertex that is closest to the fault plane. A second cross is drawn and the section of the interface between the two crosses is redrawn with a wide pen. Note that the Add to body button becomes active. (Also note that by selecting the vertices in this order we have specified that we are creating the body in a clockwise sense.)

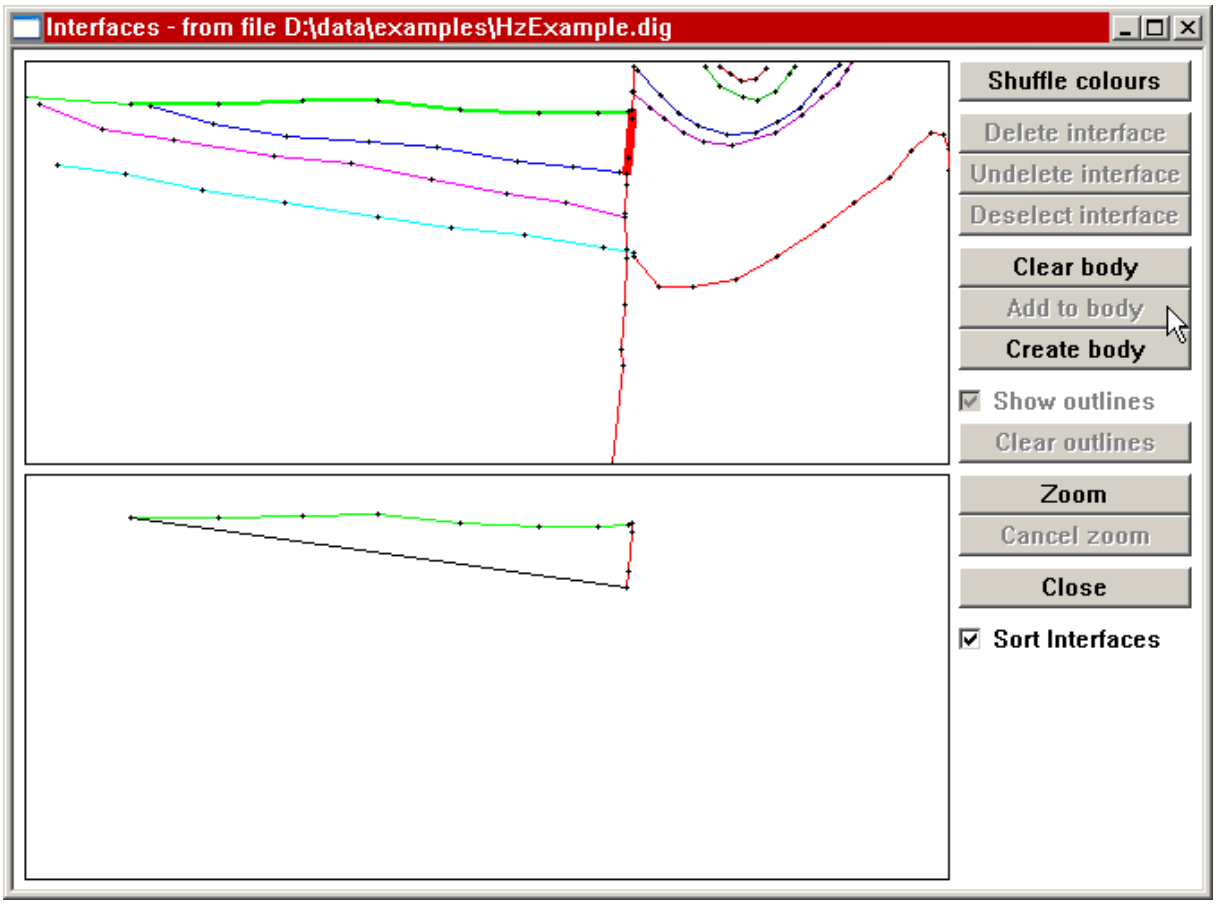


3. Choose the **Add to body** button. The highlighted section of the interface is copied to the body (lower) pane, where the body under construction is drawn, and the interface is automatically deselected. As there is currently only a single interface section the polygon is closed by joining the two ends by a black line. Note that the Create body button becomes active.

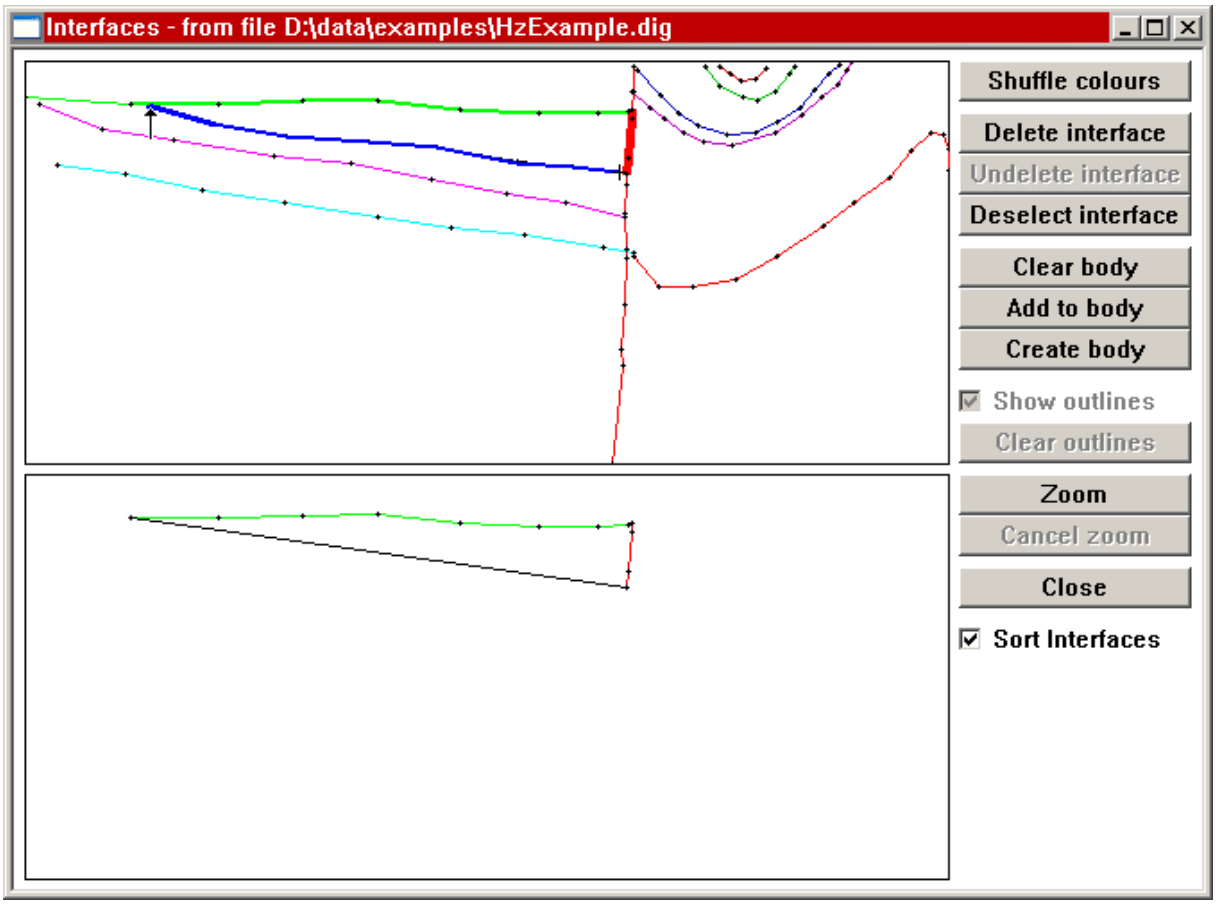


4. As you are creating the body in a clockwise sense you must now add the part of the body bounded by the fault...
   - Select the fault plane by clicking on any vertex.
   - Select the vertex that is closest to the end of the highlighted section of the first interface.
   - Select the vertex that is closest to the end of the second interface (not yet highlighted).

   The section of the fault plane between the first and second interfaces should now be highlighted. Choose the **Add to body** button to add this section to the body under construction. The interfaces window should look like this:

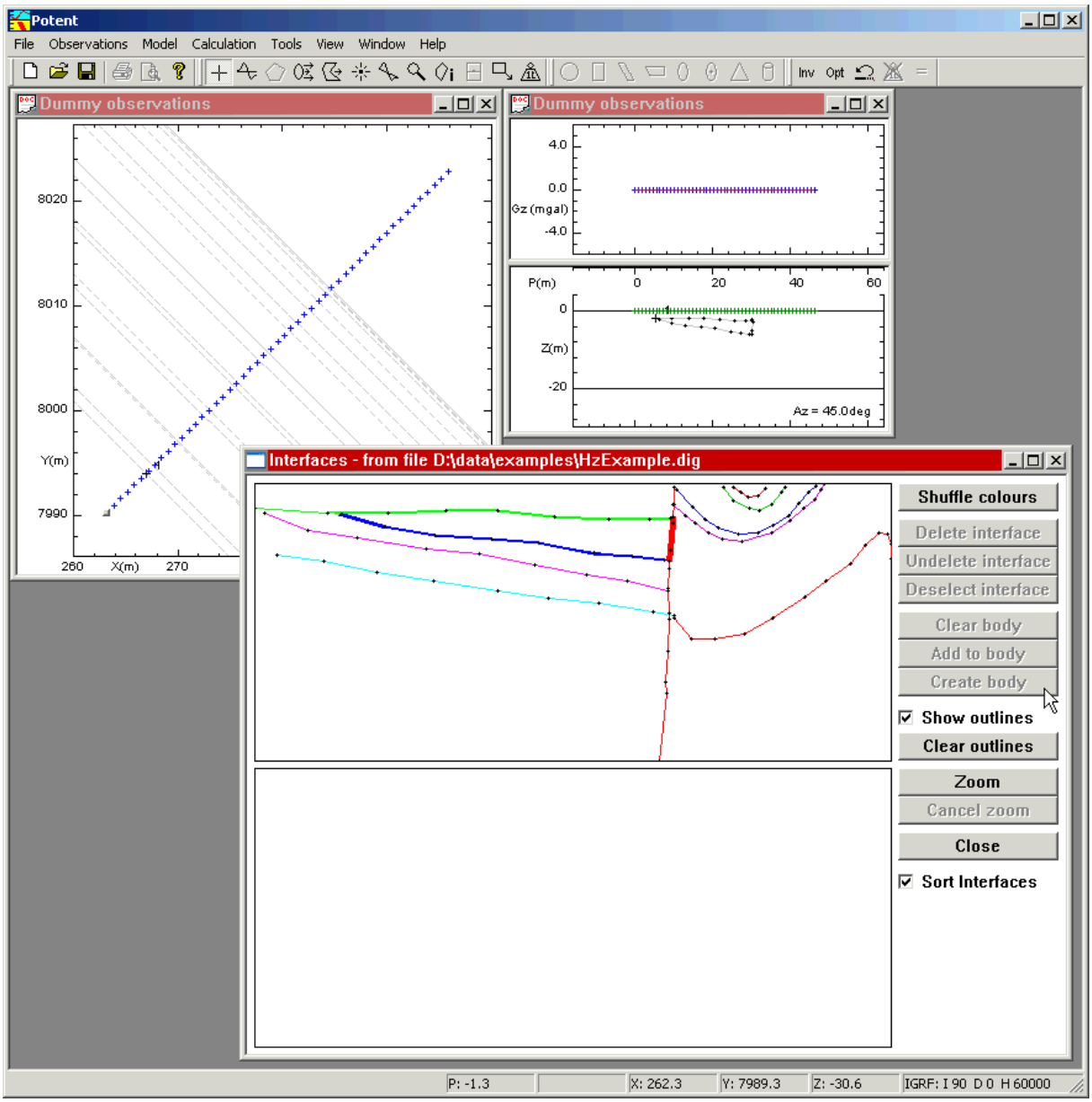


5. Now select the second interface and add to the body the section that constitutes the bottom of the wedge. As the body is being defined in a clockwise sense you must choose the vertex at the fault plane first, followed by the vertex that is closest to the point of the wedge. After choosing the **Add to body** button the body pane should show a wedge shaped body...

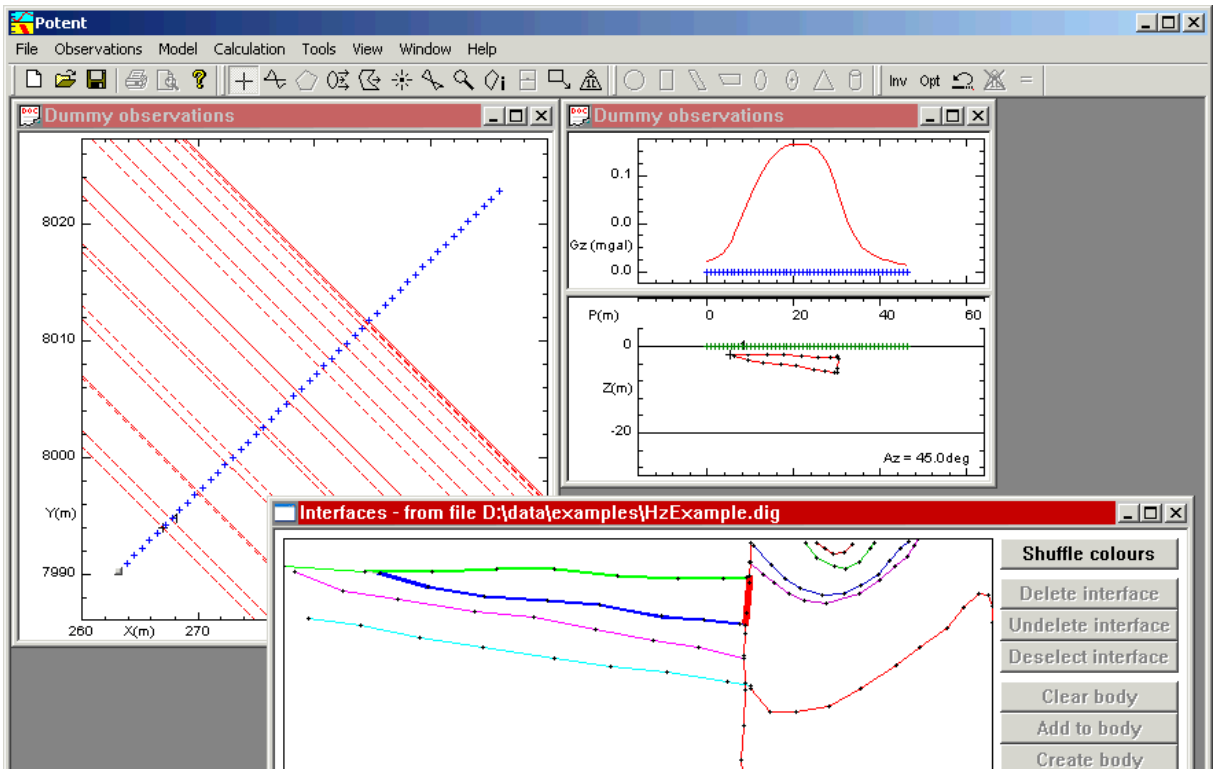


6. Choose the **Create body** button. Potent creates a polygonal prism body using the constructed body cross-section as a template. The new body has infinite strike extent and its strike is normal to the plane of the section, as can be seen in the Plan and Profile windows. The body pane of the interfaces window is cleared.

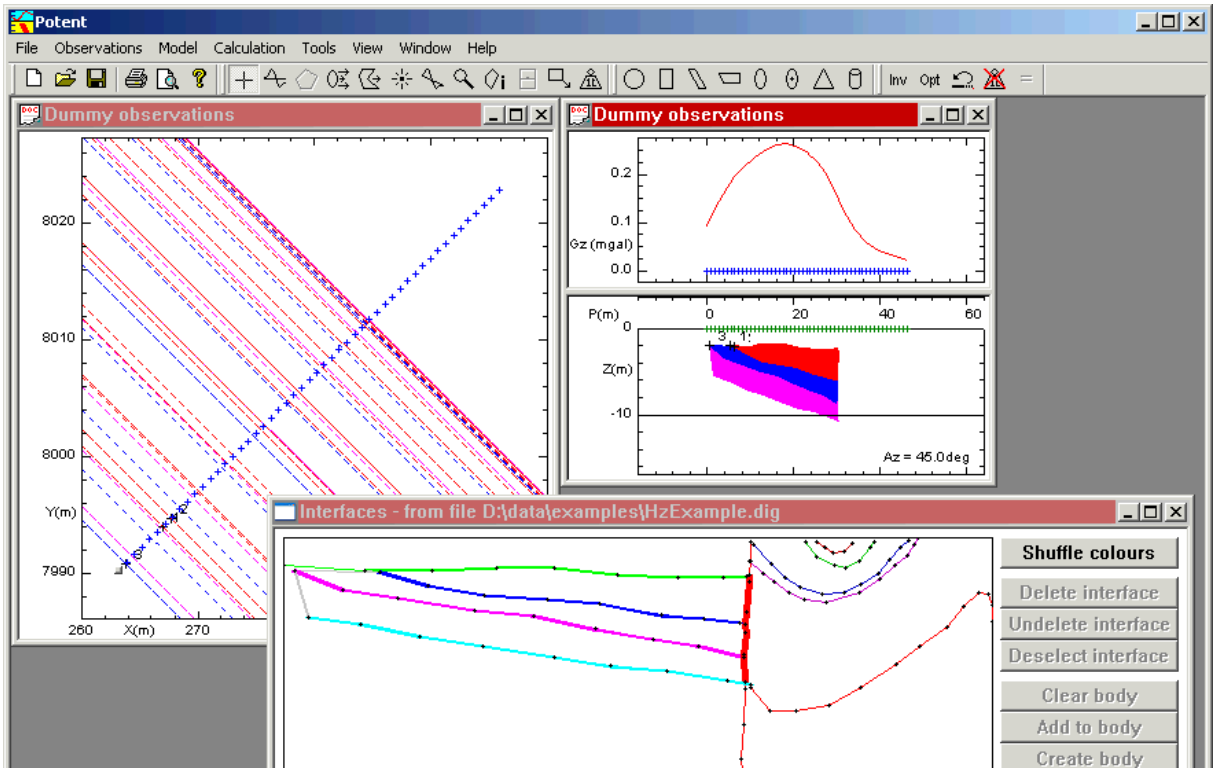
   The outline of the new body is drawn in the interfaces pane using a medium width pen. You can clear the **Show outlines** to temporarily suppress the drawing of body outlines in the interfaces pane. (The **Clear outlines** button is used to permanently remove the outlines of bodies that have been created so far.)



7. You can now manipulate the new body using normal Potent functions. In the following example the body has been made **active** so that its calculated field is displayed.



8. Generate a layered structure by creating two more bodies. As each is created, add it to the model using the **Create body** button, then make it **active** in the usual way. After making all bodies **active** and setting the vertical exaggeration in the Profile window to 2:1, the final result looks something like this...

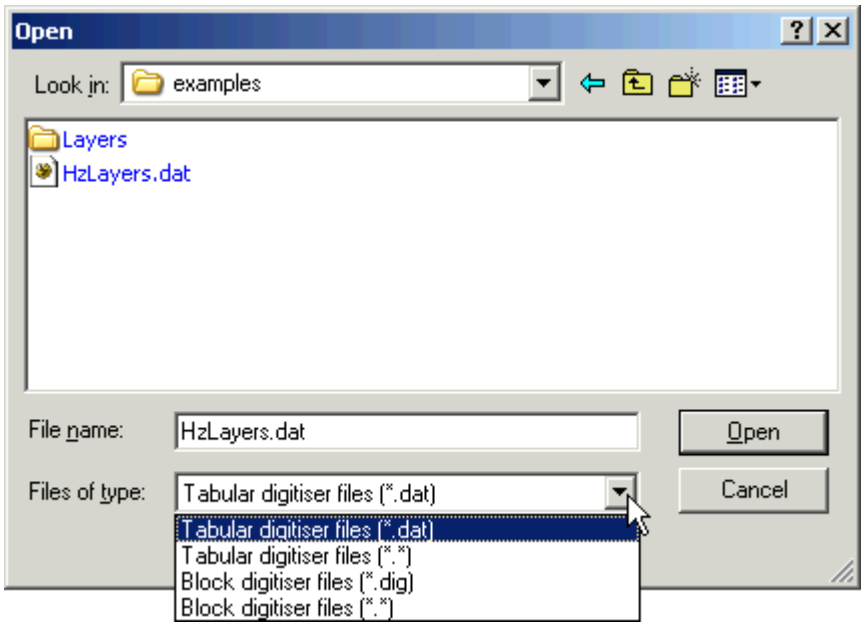


9. Use the **Save as** command from the **Model** menu to save your model. Eventually you will import your field data into a new workspace, then use the **Open** command from the **Model** menu to load the new model.

Data formats
Currently only two formats are supported:

Tabular digitiser format
Block digitiser format

Both are simple forms of digitiser output. When you choose the **Import as interfaces** command from the **Model** menu an **Open** dialog box is displayed:



The two types of file can be chosen from the **Files of type** list. We suggest you use the extension “dat” for tabular format files and “dig” for block format files, in which case you can use the filters provided in the list. Otherwise use the appropriate *.* option, which allows you to choose files of the required type but with any particular extension.

Tabular digitiser format

Interface data are in free-field columns, separated by any combination of spaces, tabs and commas. The first line contains column headings; the first two columns must be X and Y. NULL values (not for X or Y) contain a single “0” or “*”. (To assign an actual zero value the field should contain “0.0 rather than “0”.) This example is a fragment of the file HzLayers.dat, which is located in the Examples subfolder of your Potent installation folder (typically C:\Program Files\Potent):

X	Y	Surface	Formation_1	Formation_2	Formation_3	Basement
561659.48	2311542.51	-376.93	1201.13	1794.81	2506.18	3000.
561482.92	2312164.44	-376.93	1198.35	1788.74	2540.12	3000.
561129.81	2313408.30	0	1155.55	1784.73	2590.26	0
560776.69	2314652.16	0	1198.42	1761.54	2615.01	0
560423.58	2315896.02	0	1197.53	1720.60	2584.27	0
560070.46	2317139.88	0	1104.15	1699.41	2542.39	0
559717.35	2318383.75	0	1080.18	1733.92	2566.73	0
559364.24	2319627.61	0	1085.22	1791.80	2543.53	0
559011.12	2320871.47	0	1108.81	1788.83	2602.73	0
558834.56	2321493.40	-376.93	1103.82	1817.22	2649.12	0
558658.01	2322115.33	-377.61	1080.31	1816.31	2647.90	0
557951.78	2324603.05	-369.67	1116.29	1902.33	2771.38	0
557775.22	2325224.98	-372.73	1137.11	1899.89	2767.55	3000

```
   557598.67   2325846.91    -376.13     1141.82       1879.03       2764.58      3000
```

When the interfaces have been input Potent creates a set of observations in the form of a profile that is along the direction along which the interfaces have been digitised. This simplifies the viewing of the model that is created from the interfaces.

Block digitiser format

Interface data are in blocks as shown in the following fragment from file HzExample.dig, which is located in the Examples subfolder of your Potent installation folder (typically C:\Program Files\Potent).

```
***Digitized section
*
* Comment lines begin with * in column 1
* Start of interface indicated by # in col 1, optionally followed by interface name
* Map coordinates of start and end of line allow full (X,Y,Z) coordinates to be retrieved
*
* These four lines are mandatory
Easting Start   263.0
Northing Start 7990.0
Easting End     273.0
Northing End   8000.0
*
# Interface one
     0.342100122     1.545481795   1
     5.611577827     1.992251416   1
     9.953481332     1.957931413   1
    14.157192266     1.810476401   1
    17.942569998     1.777256398   1
    22.066067702     2.260541438   1
    26.023299646     2.495689375   1
    28.916283560     2.529248544   1
    30.452968724     2.358578947   1
# Interface 2
     6.576385860     2.127546844   2
     9.759524157     3.098026508   2
    13.399093599     3.831873236   2
    17.466837704     4.146697846   2
    20.901074590     4.504330792   2
    24.948978664     5.388112116   2
    27.721713699     5.673860056   2
    30.020571414     5.959539247   2
```

Model | Export as Geosoft polygon file

This command allows you to export body outlines (Plan view only) as Geosoft polygon files for import by Oasis montaj™. An associated text file provides annotation.

After exporting the files you can use the PotDraw and PotText "GX" applications from within Oasis montaj to add the outlines and annotations to a map. Refer to the on-line Help included with these GXs for details.

See also - Model | Export as DXF file. A 2D DXF file could be a more versatile way of exporting to Oasis montaj.

--- Return to the Model menu.

Model | Export as DXF file

This command allows you to export model and data in Autodesk's Drawing Interchange and File Format, Release 12 ("DXF"). The resulting DXF file can be in either 2D or 3D format. 2D DXF files can be imported into a wide range of applications, including ER Mapper, Geosoft's Oasis montaj, and Microsoft Word (with appropriate graphic filter). Oasis montaj will also

display 3D DXFs, as will more specialised viewers such as GeoExpress and Autodesk's free DWG TrueView.

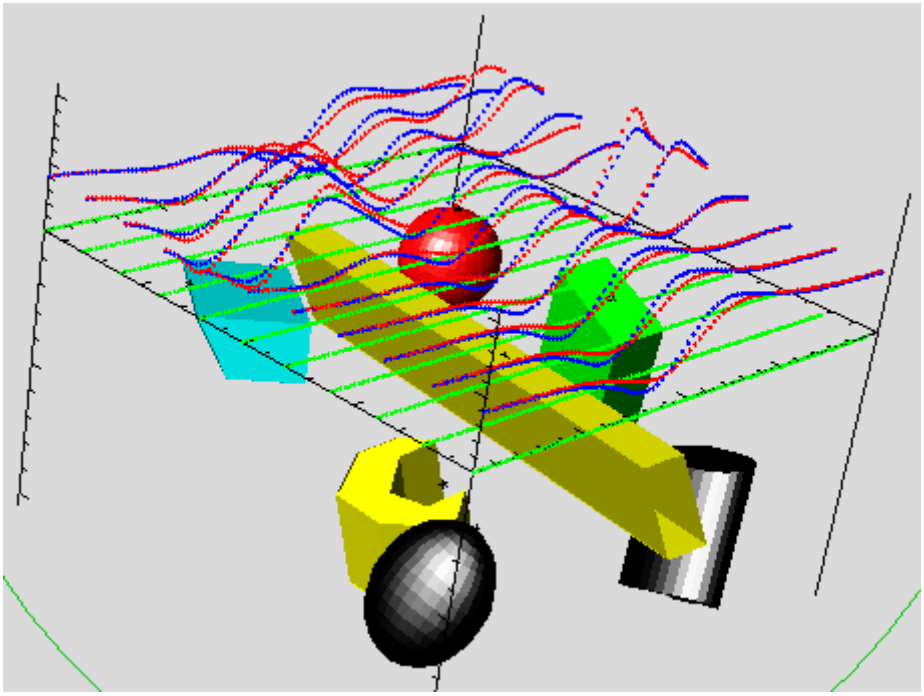
The command is available only when a Plan window is active. The X and Y axes of the Plan window define the horizontal boundaries of the contents of the DXF file. When you choose the command, the **DXF export options** dialog box is displayed:

The options in this dialog box are as follows:

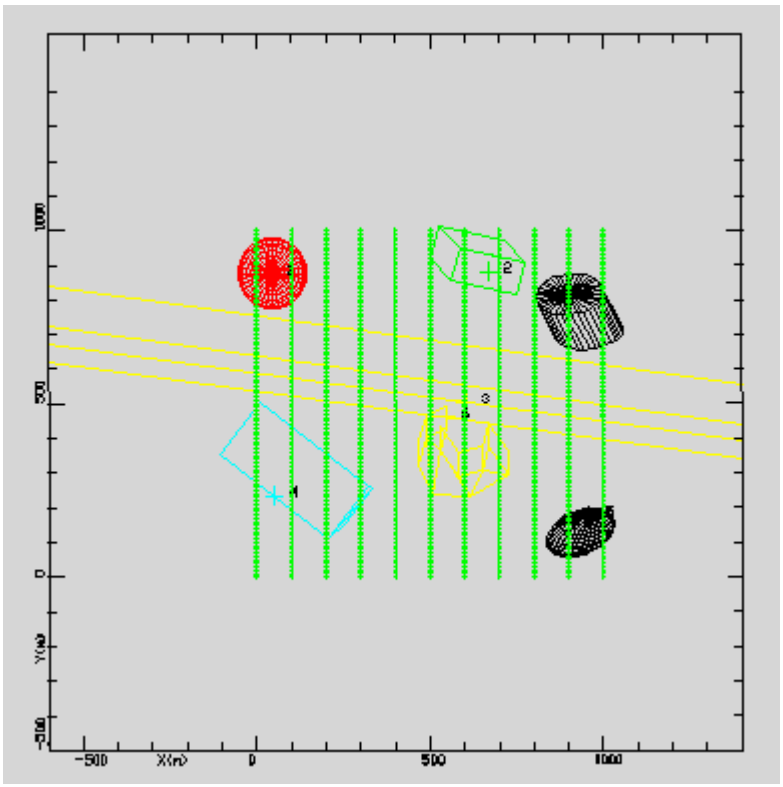
- **3D** – The resulting DXF file potentially contains 3D entities as well as 2D.

  In order to generate a 3D DXF file the dataset must have at least one profile or down-hole window visible (i.e. non-minimized). This is because the Z axis of this window is used to define the Z dimension of the DXF file. (If more than one such window is visible then Potent will use the first one it locates. If you are unsure which one will be used then minimize all except the one you prefer.)

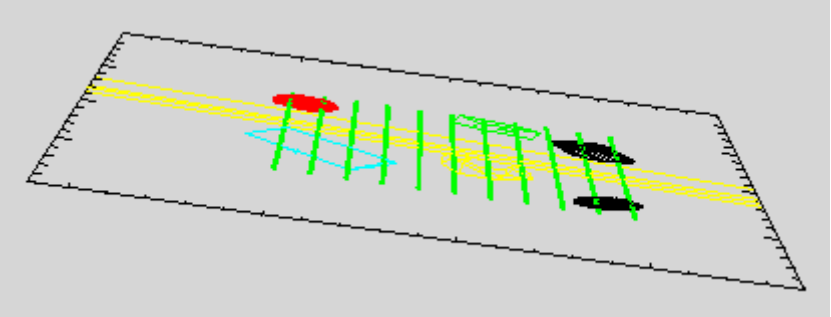
  This example was viewed using Volo View Express (now superseded by DWG TrueView). It shows model, observation points (green), observed field (blue) and calculated field (red).



- **2D** – The resulting DXF file has no 3D entities. In this case the three “field” options are not available. The previous example would look like this:



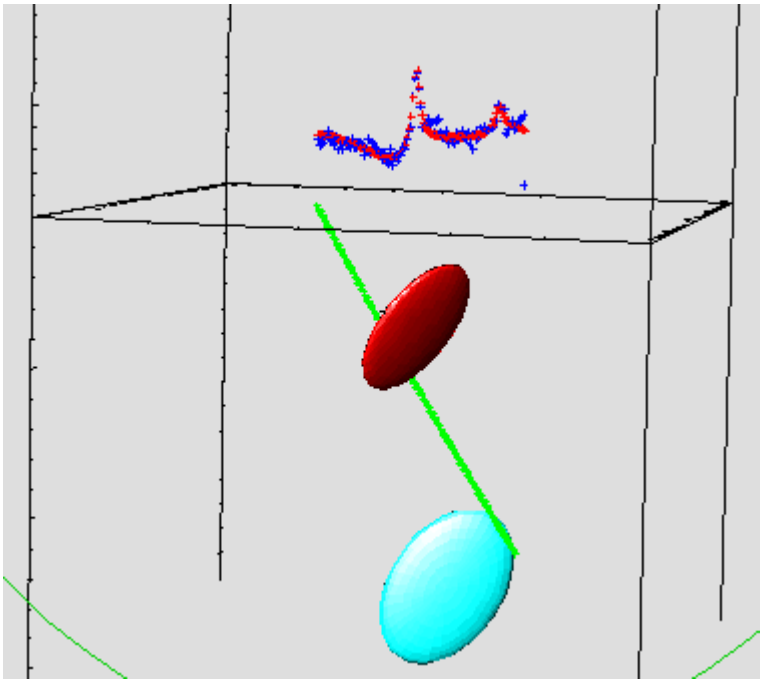
Rotating the picture shows how all the entities are confined to a plane:



- **Model** - Select this to have the model included in the file.

- **Clip to axes** – Select this to have Potent clip the exported model to the X,Y axes of the active Plan, and the Z axis of a profile or down-hole window (see **3D**, above).

- **Observations** – Select this to have the observation locations included in the DXF file. If the **3D** option is also selected, the observation points are plotted with reference to their height. This is demonstrated in this down-hole example, where the track of the hole is shown in green:



- **Axes** – Select this to include axes in the DXF file.

- **With labels** – If **Axes** is selected, you can also select this to add labels (e.g. “X”) and values to the axes.

- **Observed, Calculate and Residual field** – Select any of these to add the appropriate field value to the DXF file.

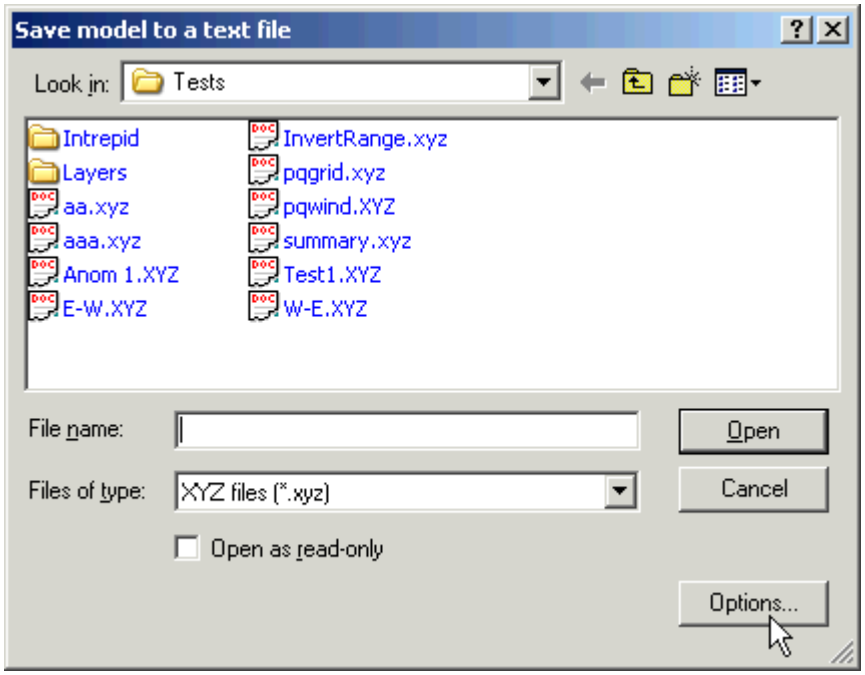
- **Component** – The drop-down list contains all the field components in your dataset. Select the one that you want to be used when plotting.

In addition to the above options, any body annotations that are shown on the Plan (Window | Options command) will be included in the DXF file.

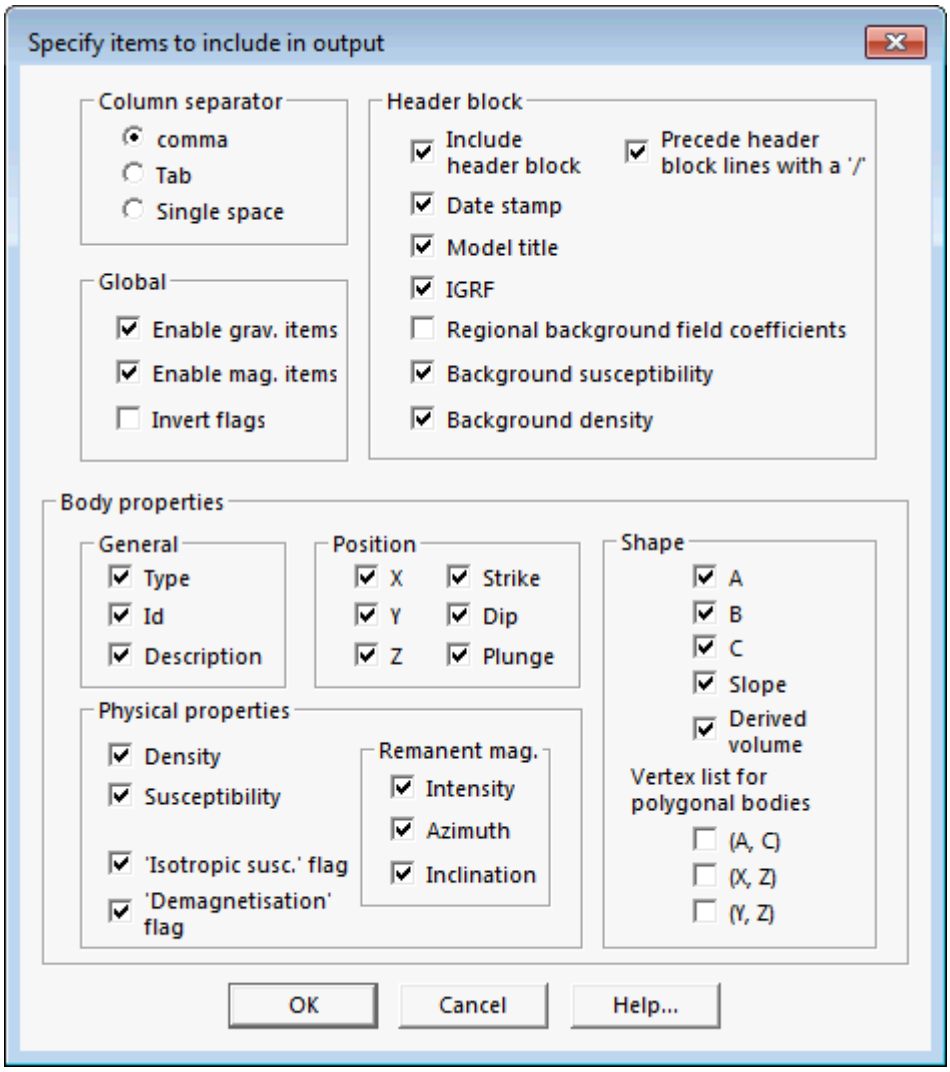
--- Return to the Model menu.

Model | Export as text file

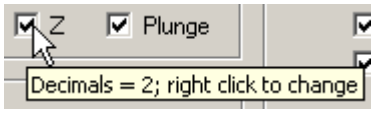
This command allows you to create a text file that contains all model information, one line per Active body.



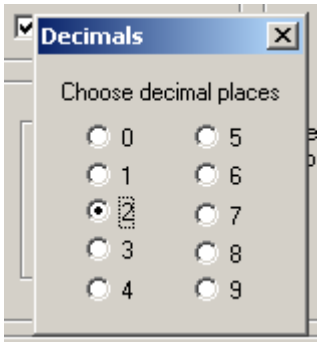
Click the **Options** button to display a dialog box (below) that gives you detailed control over the content and format of the output file. For example, tab-delimited format is useful for importing into a word processing program, while comma-delimited format facilitates import into a spreadsheet.



If a check box is associated with a decimal value then you can right-click on it to change the number of decimal places to be used. A tool-tip style prompt is displayed when the mouse cursor hovers over the check box, as for **Z** in this example:



Right-clicking displays the **Decimals** pop-up dialog box.



The **Derived volume** is the volume in cubic metres calculated from the shape of the body. (If a body is 2D then the value is given in terms of “cubic metres per along-strike metre”, which is equivalent to the cross-sectional area in metres.)

When modelling at the geological scale the volume of a body can be a very large number. Therefore it is output in scientific (exponent) notation, such as 1.23e+007.

Model | Save snapshot

This command saves a snapshot of your current model. Refer to Undoing changes in the model for details.

See also – Restore from snapshot.

Model | Restore from snapshot

This command restores a previously saved snapshot of your current model. Refer to Undoing changes in the model for details.

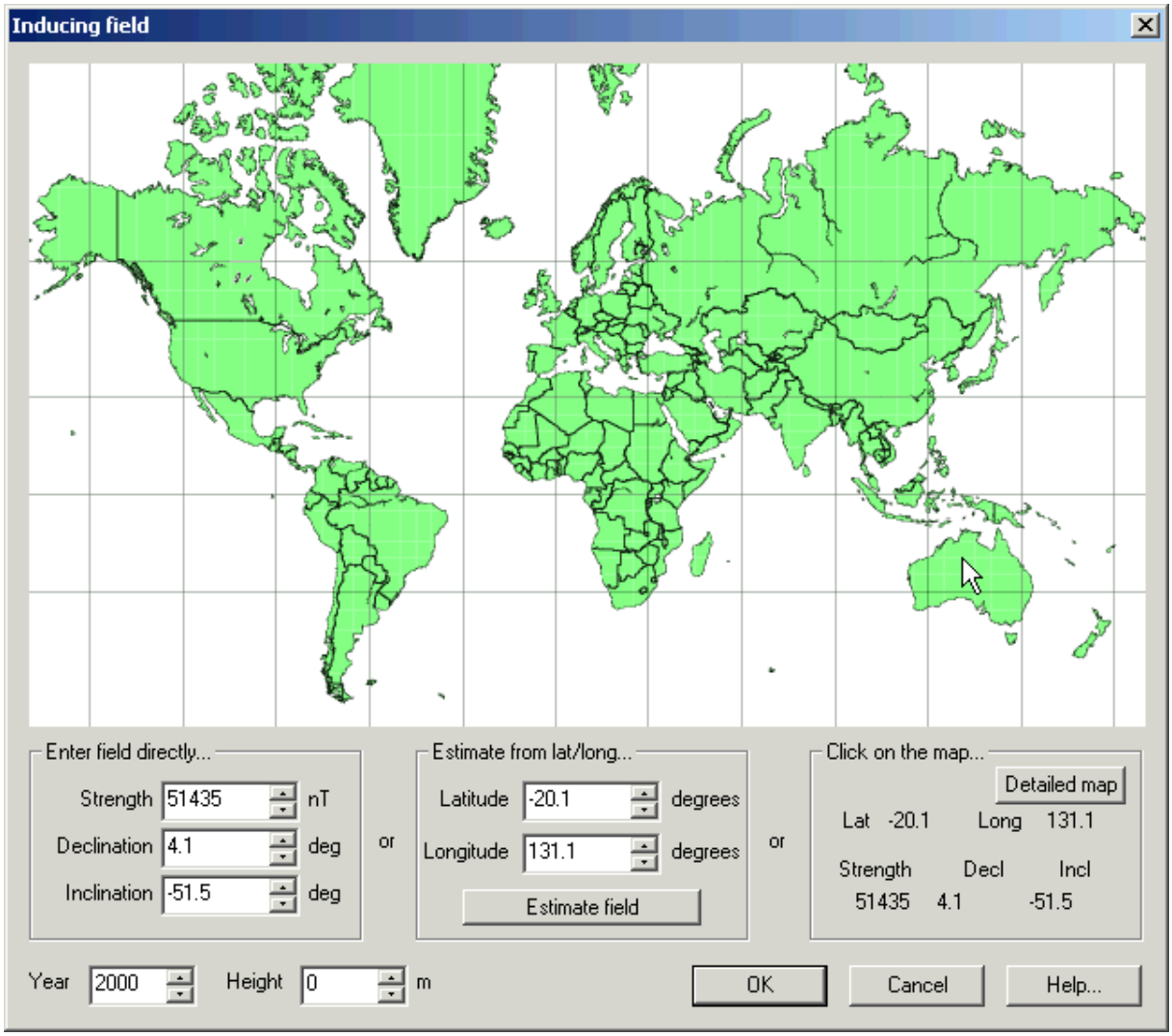
See also – Save snapshot.

Model | Clear all invert flags

Clear all invert flags for all bodies.

--- Return to the Model menu.

Model | Inducing field



Use this command to set the inducing magnetic field (IGRF) that will be used for calculating induced magnetisation when you are working with magnetic models.

You can enter the field parameters in any of three ways:

- Using the **Enter field directly…** group. Simply enter the field values that you already know and click **OK**.

- Using the **Estimate from lat/long…** group. Enter the latitude and longitude of the survey area, the year and height (see below), and click the **Estimate field** button. The field values are calculated and inserted into the **Strength, Declination and Inclination** boxes. Click **OK**.

- **Click on the map…** group. First, set the year and height (see below). As you move the mouse cursor over the map, the current latitude, longitude and field values are displayed. Click on the map to update the edit boxes with the values. (The graphic shows the values for central Australia.) Click **OK**.

When using method 2 or 3 you should set the year of the survey in the **Year** edit box and (if the altitude is extreme) the average height (metres above mean sea level) of the survey area in the **Height** box.

Click the **Detailed map** button (or double-click on the map) to zoom in.

Algorithm

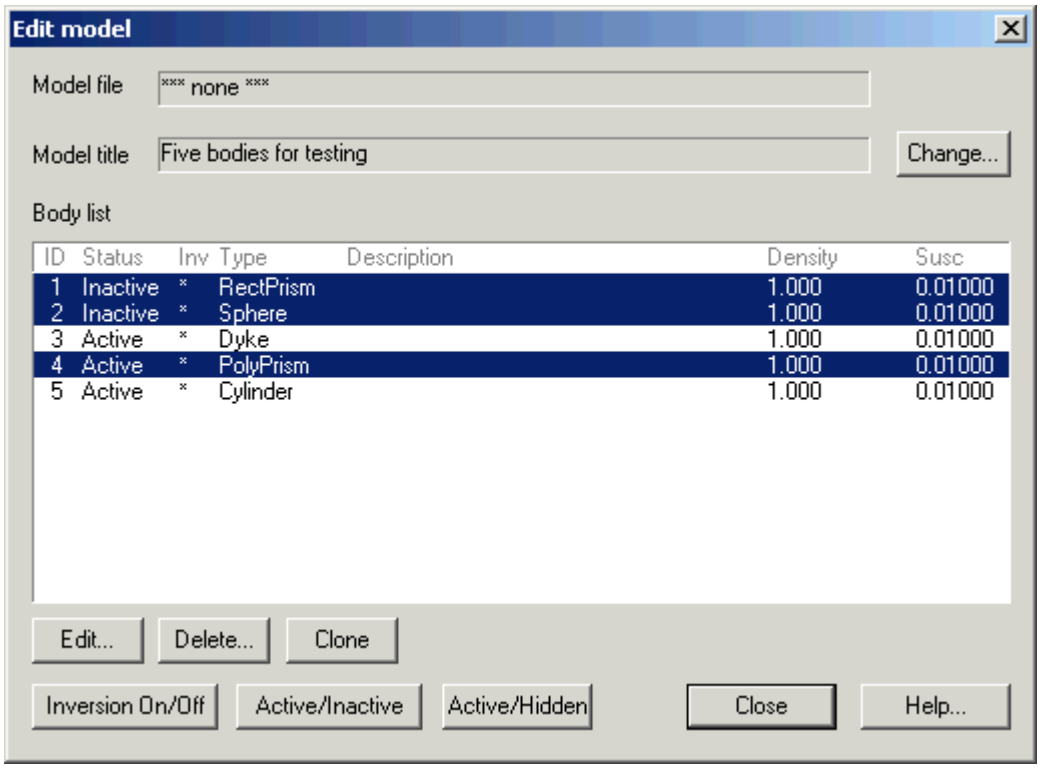
The IGRF algorithm is adapted from modelling code provided by the National Geophysical Data Center. It uses the spherical harmonic coefficients provided in the file IGRF12.COF.

IGRF12 is the twelfth generation standard main field model adopted by the International Association of Geomagnetism and Aeronomy (IAGA). This is a degree and order 10 model from 1900 to 1995 and a degree and order 13 model from 2000 to 2020, providing estimates of the main field for dates between January 1, 1900 and January 1, 2020. For more information on the IGRF and IAGA, visit the IAGA Working Group V-MOD Web site at http://www.ngdc.noaa.gov/IAGA/vmod/.

See also: View IGRF on status bar.

--- Return to the Model menu.

Model | Edit model



This dialog box allows you to edit any body of the model, including those that are not visible on the screen.

(Note that Potent always uses the compact form of the Edit body dialog box while you are using the **Edit model** dialog box. If you have the Composite Edit body dialog box displayed then it is temporarily closed while the **Edit model** dialog box is displayed.)

Model file – The name of the file, if any, currently associated with the model. The **Model | Save** command updates the contents of this file. Use the **Model | Save as** command to change the name of the model file.

Model title – A title that is used to describe the model. It is saved as part of the model file, and is listed as part of the Model Summary on printed output (see the **File | Page layout** command).

Body list – A list of all the bodies in the model. You can choose multiple bodies from the list, as shown in the above image. Double-clicking a body is equivalent to selecting a single body and clicking the **Edit** button.

Edit – Display the **Compact Edit body dialog** box for the highlighted body. If multiple bodies are selected, the Edit button presents each body in sequence for editing.

Delete body – Remove the selected bodies from the model.

Clone body – Create a new body that is identical to the highlighted body except that its initial status is **Inactive**. As soon as the new body is created, you should change its position using the **Edit body dialog (Position tab)**, as it is created with exactly the same position as the original body and so will overlay it on the screen.

The following commands are shortcuts to the settings in the **General** page of the **Edit body dialog box**.

Inversion On/Off – Toggle the inversion state of all selected bodies.

Active/Inactive – Toggle the state of all selected bodies between the Active and Inactive states.

Active/Hidden – Toggle the state of all selected bodies between the Active and Hidden states.

--- Return to the Model menu.

Compact Edit body dialog box

This dialog box is a compact alternative to the **Composite Edit body dialog box**. To use it, you must clear **Show composite edit model dialog** on the **Model** menu.

The compact Edit body dialog box can be displayed either by right-clicking on a body's reference point, or by editing the body from the **Edit Model** dialog box.

There are four tabs:

General

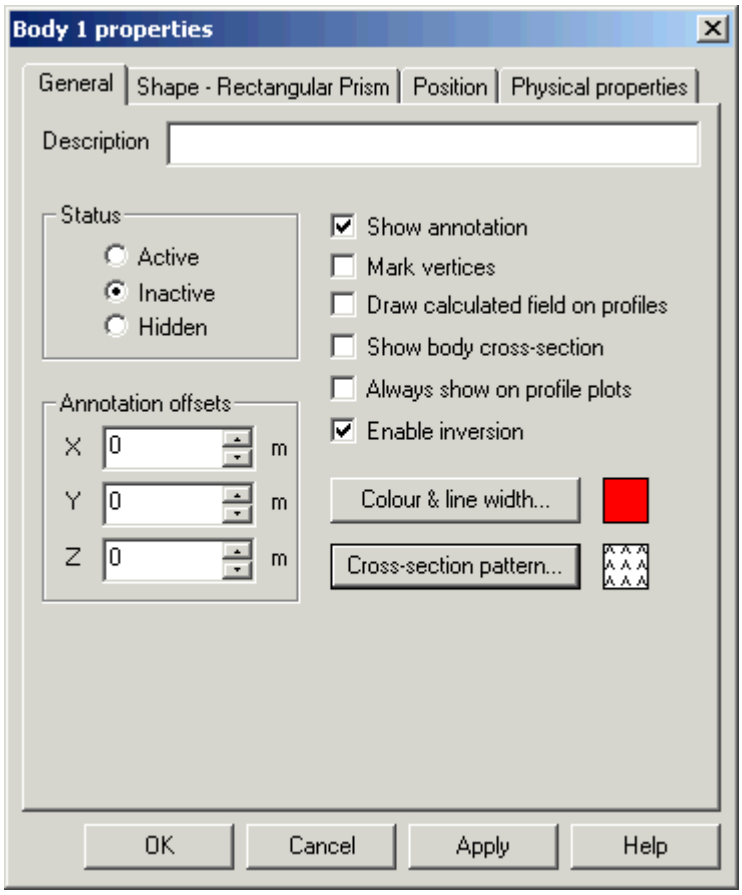
Shape

Position

Physical properties

--- Return to the Model menu.

Edit body – General tab



Status - When a body is created its status is set to **Inactive**. *Potent* will not calculate the body's field until you set its status to **Active**.

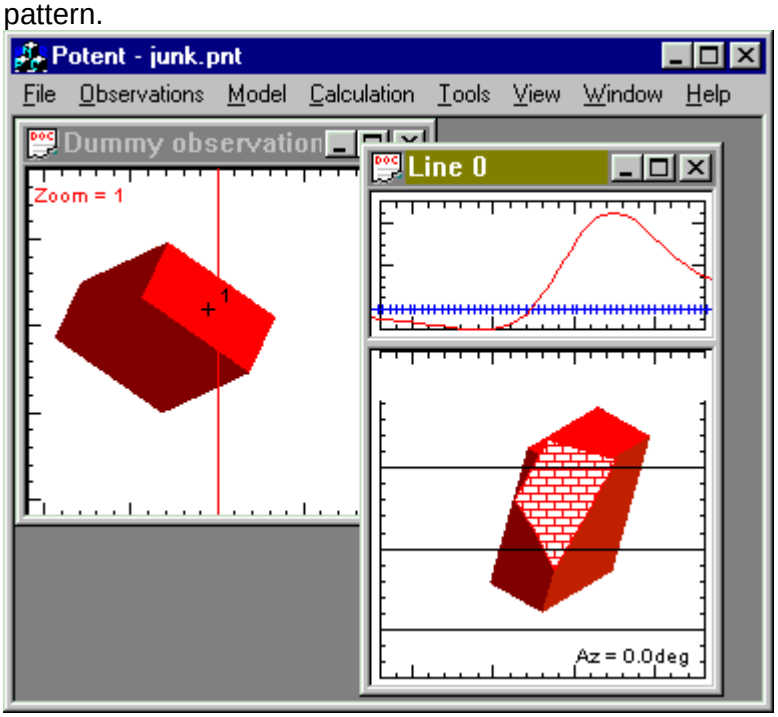
Show annotation - Clear this if you do not want the body to be labelled with any annotations specified by the **Annotate bodies with** group in the **Profile window settings** and **Plan window settings** dialog boxes.

Mark vertices - This option applies only to polygonal prism body types. When this is selected, small crosses are used to mark the body vertices in plan and cross-section windows. This occurs even if the equivalent option is cleared in the **Profile window settings** and **Plan window settings** dialog boxes.

Draw calculated field on profiles - When this is selected the calculated field due to the individual body is plotted in grey on profile panes. (Usually only the calculated field due to the entire model is plotted.)

Show body cross-section - When this is selected, the cross-section of the body in the plane of the profile is drawn in cross-section windows. This occurs even if the equivalent option is cleared in the **Profile window settings** dialog box. The cross-section is filled with any pattern you have specified with the **Cross-section pattern** button. The pattern is drawn with the colour specified using the **Colour** button. In the following example the plan window shows the profile line going over the eastern corner of the body. In the cross-section pane of the profile window the section of the body in the plane of the profile is filled with a "brick"

pattern.



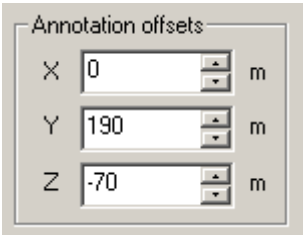
Always show on profile plots - Select this to over-ride the **Don't draw bodies…** option in the **Profile window settings** dialog box. The body is drawn on cross-section panes regardless of its distance from the line of the profile.

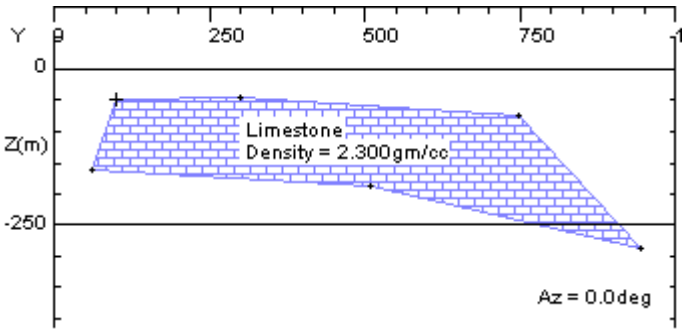
Enable inversion – Clear this to exclude the body from inversion, regardless of the setting of any of its Invert flags.

Colour & line width - Click this button to set the colour and line width (in mm) of the Active body. (Inactive bodies are always drawn as grey.)

Cross-section pattern - Click this button to set the pattern that is used to fill the body cross-section (if this has been selected either by the **Show body cross-section** option, above, or by the equivalent option in the **Profile window settings** dialog box).

Annotation offsets - Specify the position of the body's annotation (see above) relative to the reference point of the body. The distances are expressed in metres in (X,Y,Z) space. In this example the Y and Z offset values have shifted the annotation from the top left corner to a central position within the body outline:





See also - Free format annotation boxes in the Users' Guide.

--- Return to the Model | Edit body dialog.

Edit body – Shape

The contents of this dialog box depend on the type of the body that you are editing. Available body shapes are:

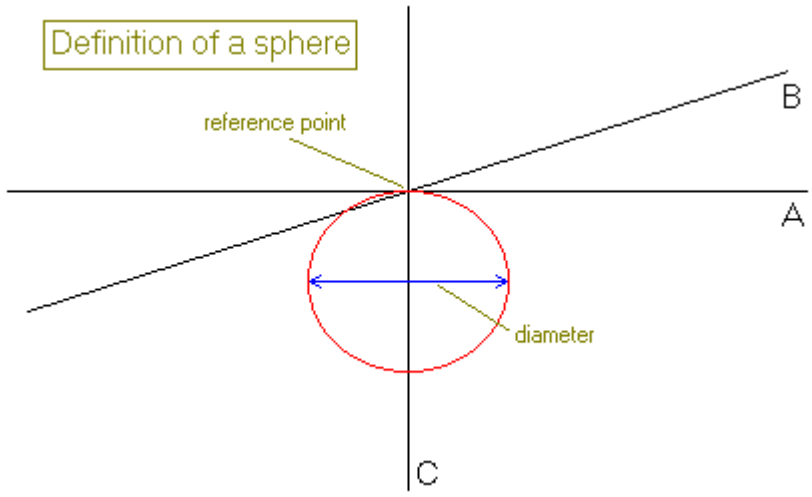
- Sphere
- Ellipsoid
- Lens
- Rectangular prism
- Cylinder
- Polygonal prism
- Dyke
- Slab

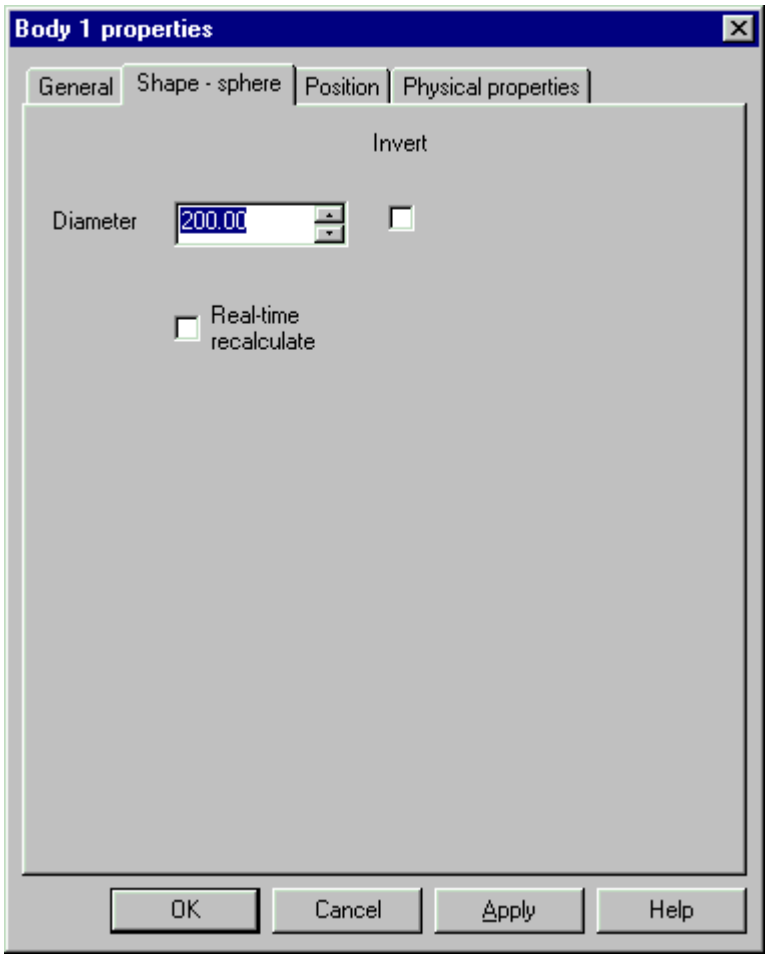
See also Body shape in the User's Guide.

Note: The volume of a body can be viewed by checking the **Derived volume** checkbox in the Model | Export as text file | Options dialog box.

--- Return to the Model | Edit body dialog.

Sphere





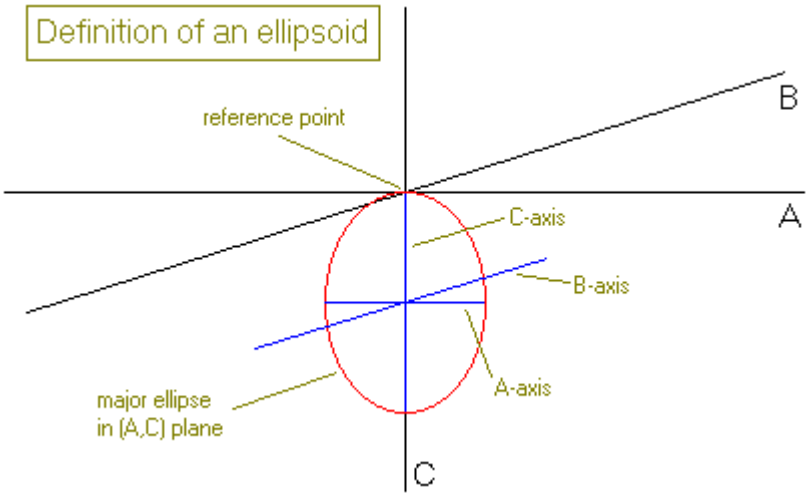
Diameter – The diameter of the sphere.

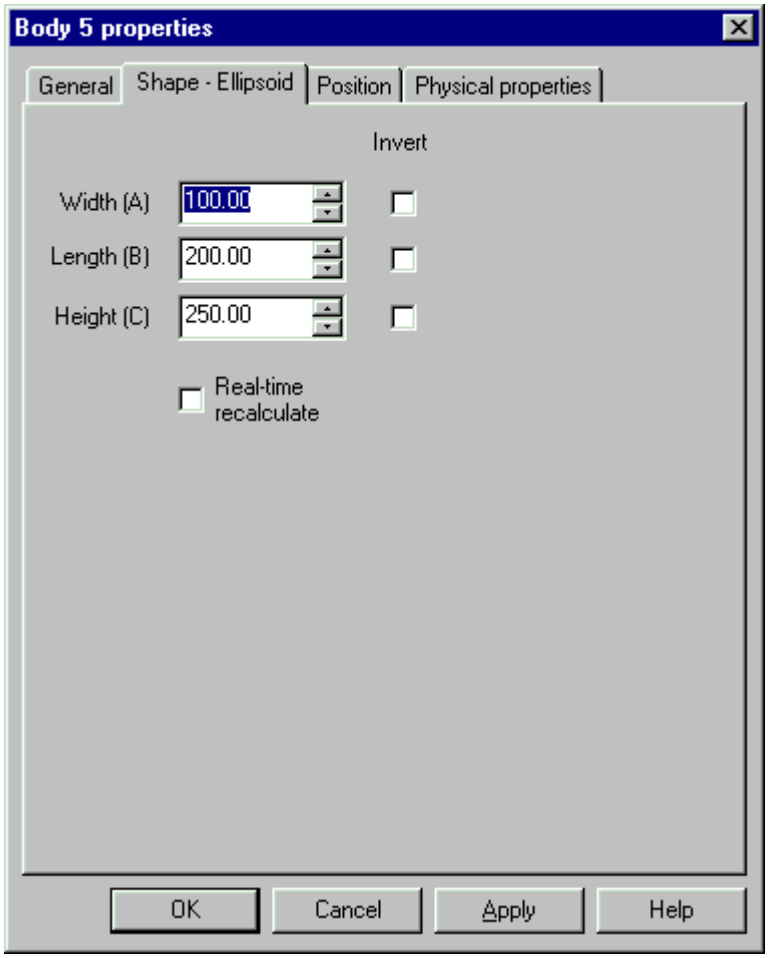
Invert Select a parameter's Invert box to set its invert flag.

Real-time calculate If this is selected then *Potent* automatically recalculates the field due to the model as values are adjusted using the arrows attached to the edit controls.

--- Return to the Model | Edit body dialog.

Ellipsoid





A, B, C – The full axis (i.e. not semi-axis) lengths of the ellipsoid.

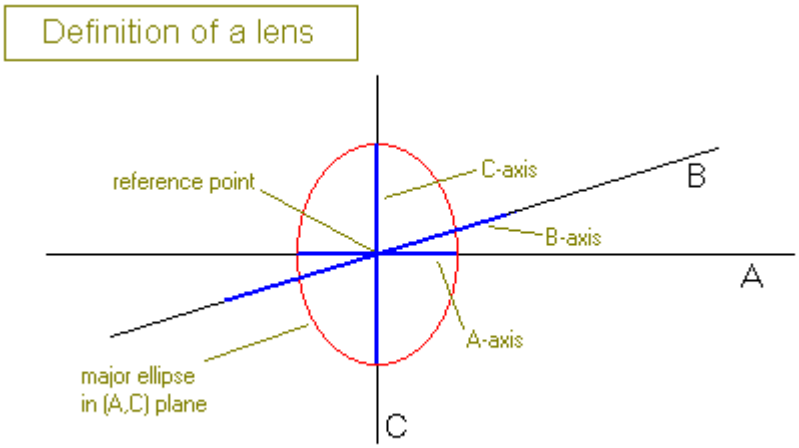
Invert Select a parameter's Invert box to set its invert flag.

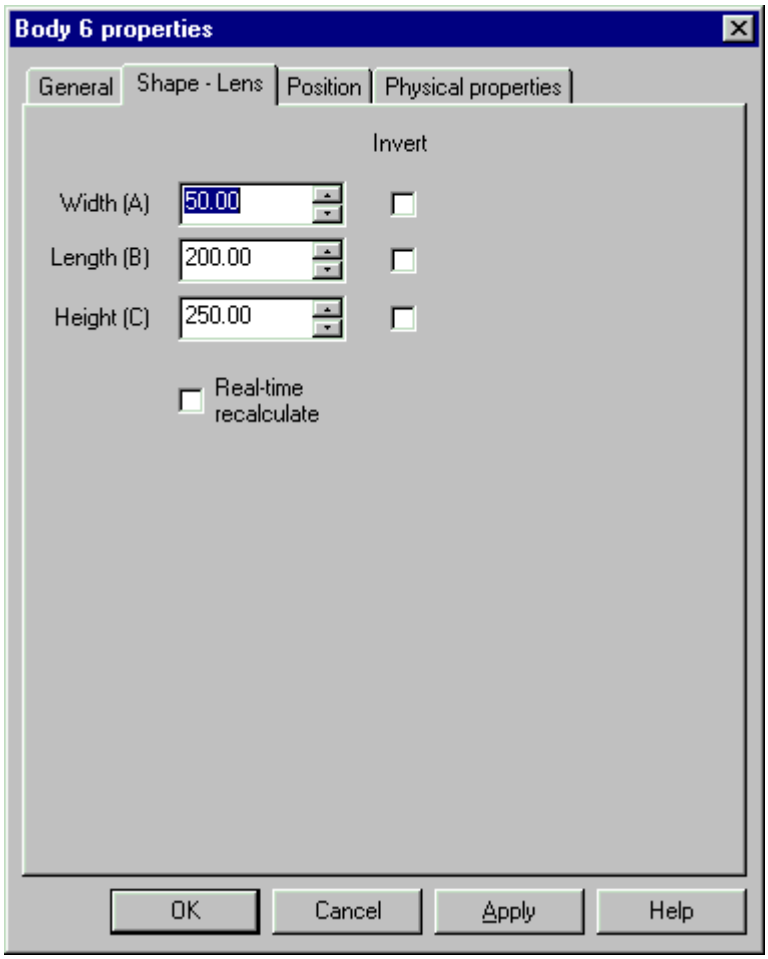
Real-time calculate If this is selected then *Potent* automatically recalculates the field due to the model as values are adjusted using the arrows attached to the edit controls.

--- Return to the Model | Edit body dialog.

Lens

The lens is identical to the ellipsoid except that its reference point is at its centre rather than its top. This makes it behave differently when rotated.





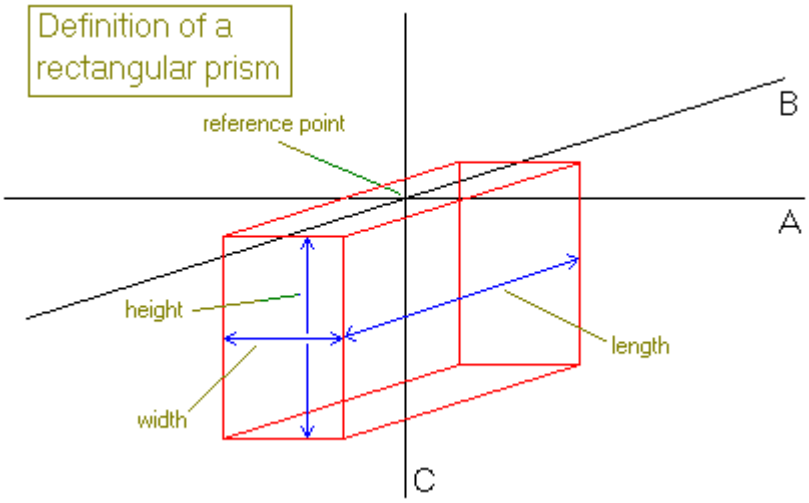
A, B, C – The full axis (i.e. not semi-axis) lengths of the ellipsoid.

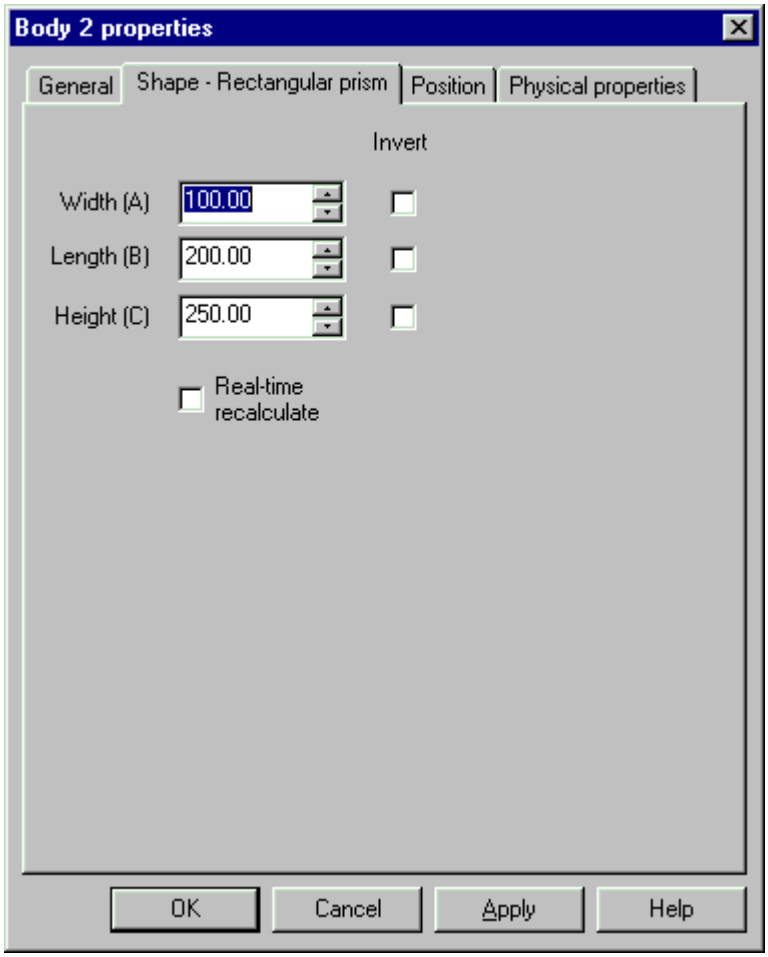
Invert Select a parameter's Invert box to set its invert flag.

Real-time calculate If this is selected then *Potent* automatically recalculates the field due to the model as values are adjusted using the arrows attached to the edit controls.

--- Return to the Model | Edit body dialog.

Rectangular prism





A, B, C – The full axis (i.e. not semi-axis) lengths of the prism.

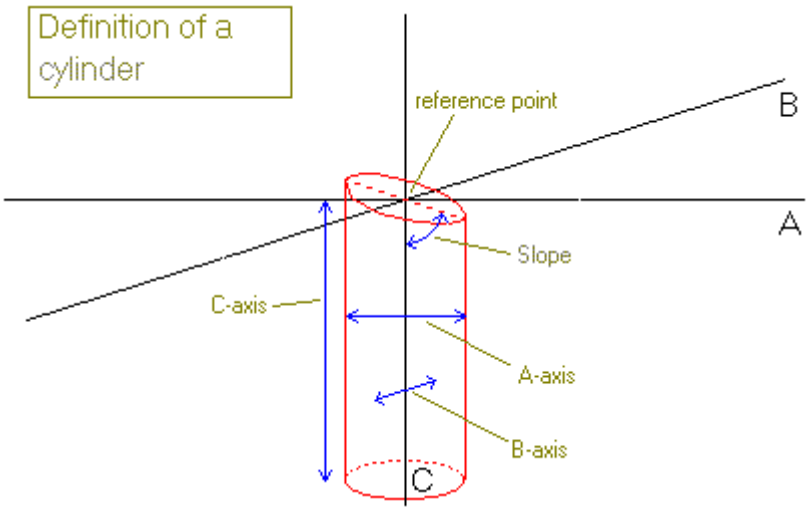
Invert Select a parameter's Invert box to set its invert flag.

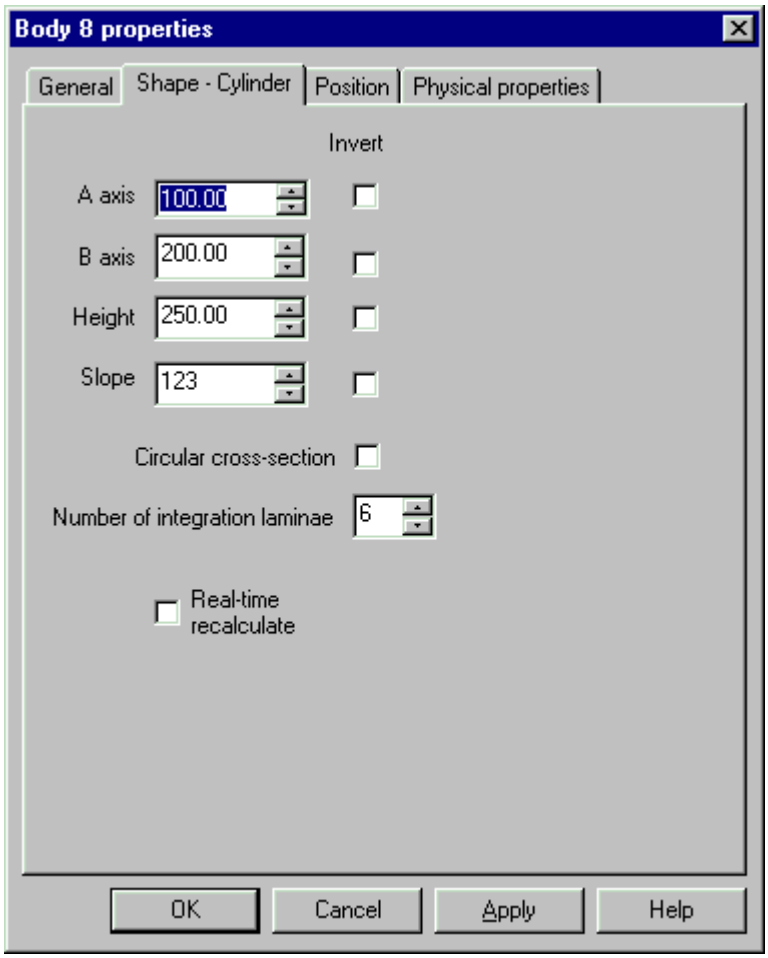
Real-time calculate If this is selected then *Potent* automatically recalculates the field due to the model as values are adjusted using the arrows attached to the edit controls.

--- Return to the Model | Edit body dialog.

Cylinder

This body type is an ellipsoidal cylinder with one inclined end.





A, B – The full axis (i.e. not semi-axis) lengths of the cross-sectional ellipse.

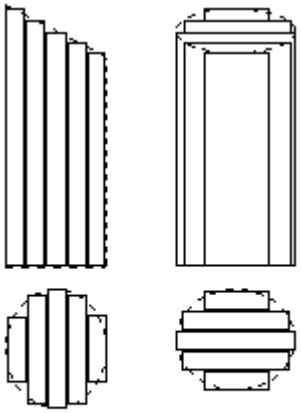
C – The height of the cylinder.

Slope – The angle of the top face of the cylinder relative to its C-axis.

Circular cross-section – Select this to force the A and B axes to be of equal length.

Number of integration laminae –*Potent* calculates the effect of the cylinder as though it were several adjoining rectangular prisms, or "laminae". This is a crude form of

numerical integration but provides fast calculation when the number of laminae is small. Use the **Number of integration laminae** edit box to control the precision of the calculation. Use a small number of laminae (perhaps 5, as shown) for initial modelling, and increase this to a larger number for final models.



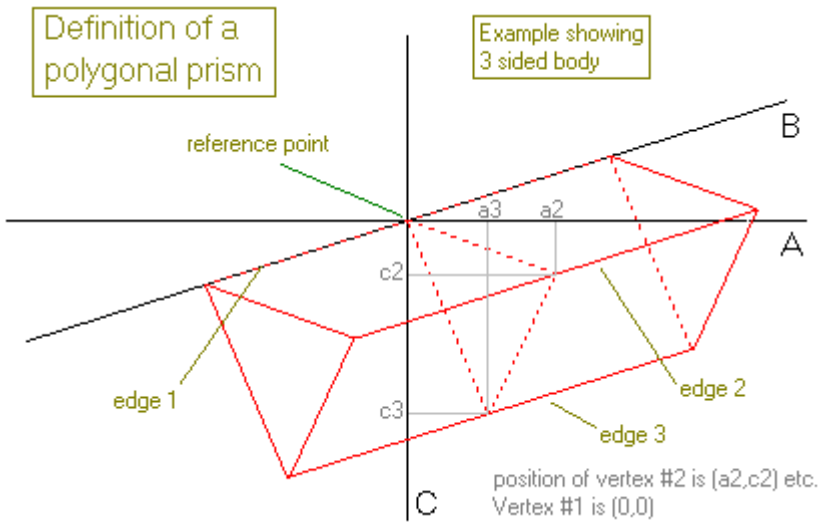
A cylinder with a sloping end is approximated by several rectangular prisms

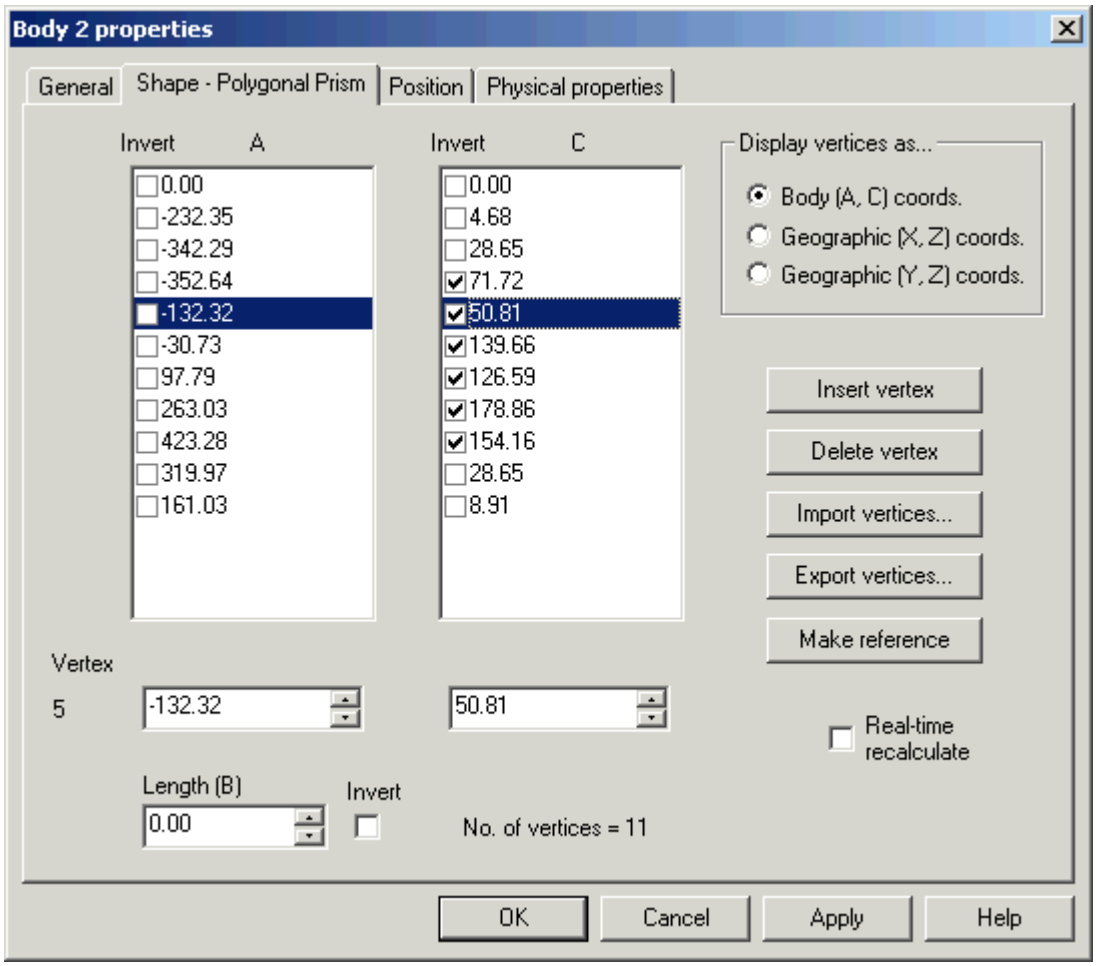
Invert Select a parameter's Invert box to set its invert flag.

Real-time calculate If this is selected then *Potent* automatically recalculates the field due to the model as values are adjusted using the arrows attached to the edit controls.

--- Return to the Model | Edit body dialog.

Polygonal prism





This dialog box allows you to set the shape of a polygon prism explicitly. It provides the only way of changing the length (B-axis) of the prism. However, consider closing the dialog box and using the Body shape tool to set the polygon vertices graphically.

Zero implies infinity when applied to the length, in which case the body is 2-dimensional.

The two list-boxes contain a list of body vertices. The actual contents depend on the **Display vertices as** setting (see below). You can set the invert flags for any vertex. Alternatively, close the dialog box and use the Invert tool to set the invert flags graphically.

In order to edit a vertex, you select it in the list-box. Its contents then become available for editing in the two edit controls below the list-boxes. The selected vertex is marked by a small box in plan and cross-section windows.

Display vertices as – The vertices of the body are stored internally as the (A,C) coordinates relative to the first vertex, as shown in the diagram. However, it is possible to display the corresponding (X,Z) or (Y,Z) coordinates instead. The option you choose would depend on the strike of the prism. *Potent* does the internal conversion back to (A,C) coordinates as required.

When showing X/Y and Z coordinates in the list boxes, the two edit boxes continue to show A and C values. However, as you adjust these A and C values the corresponding X/Y and Z values appear in the list boxes.

Insert vertex – Insert a new vertex midway between the highlighted vertex and the following one.

Delete vertex – Delete the highlighted vertex. This button is greyed if the number of vertices equals three, the minimum.

Import vertices – This button allows you to import a list of vertices from a text file. All existing vertices are discarded. The file must have EXACTLY the following format:

First record – The number of vertices;
Subsequent records – Vertices as A, C pairs separated by spaces or tabs.

The cross-sectional polygon must be specified in an anti-clockwise sense. Note that (A,C) coordinates are relative to the first vertex of the polygon, which therefore is always (0,0). If you specify non-zero values for the first coordinates, then they will be subtracted from all the coordinates in the list. For example:

```
4
0  0
50  100
150  100
100  0
```

Export vertices – This button exports the (X,Y,Z) coordinates of the vertices of each end-face of the prism to a text file. In the case of a 2-dimensional body the two faces are coincident, and pass through the reference point.

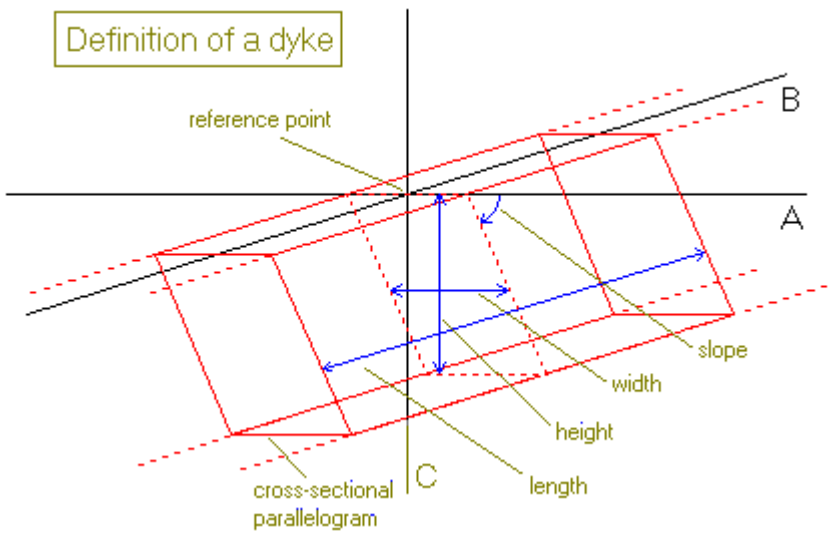
Make reference – Click this button to position the body's reference point on the currently selected vertex. The coordinates of the vertices are adjusted so that the outline of the prism stays fixed on the screen, so there is no change to the calculated field.

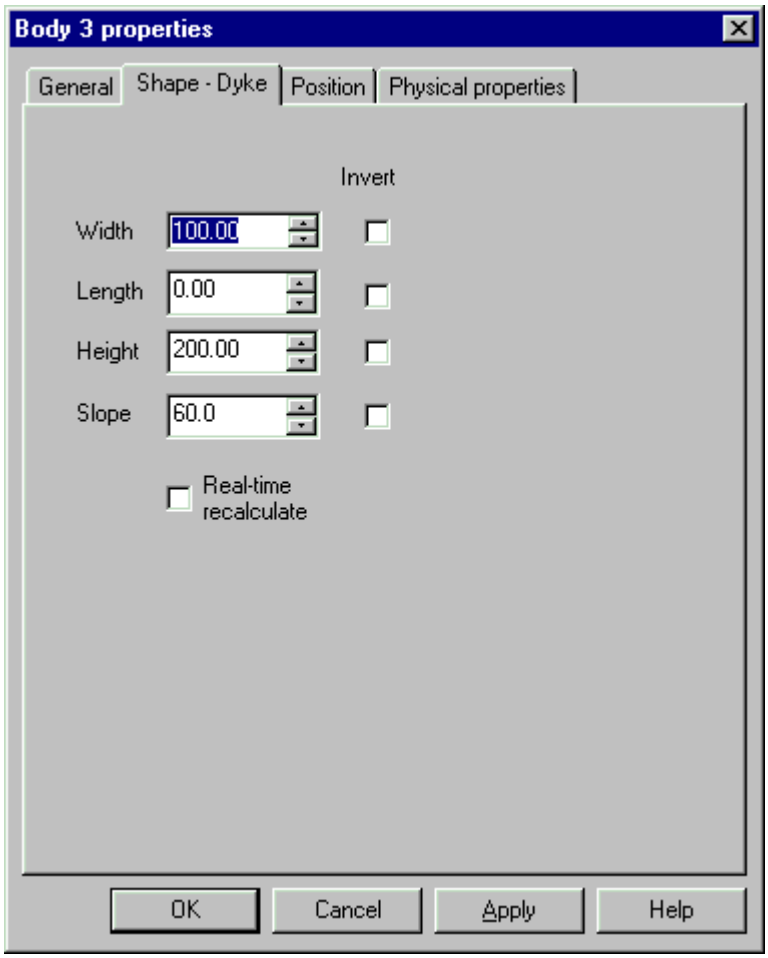
Invert Select a parameter's Invert box to set its invert flag. Alternatively, close the dialog box and use the **Invert tool** to set the invert flags graphically.

Real-time calculate If this is selected then *Potent* automatically recalculates the field due to the model as values are adjusted using the arrows attached to the edit controls.

--- Return to the Model | Edit body dialog.

Dyke





The dyke is a variety of polygonal prism in which the cross-section polygon is constrained to be a parallelogram.

The four shape parameters are as shown in the diagram. Don't confuse "slope", a shape parameter, with *Potent*'s use of the term "dip", which is a position parameter. Changing the slope alters the shape of the cross-sectional parallelogram. Changing the dip rotates the entire parallelogram about its top centre.

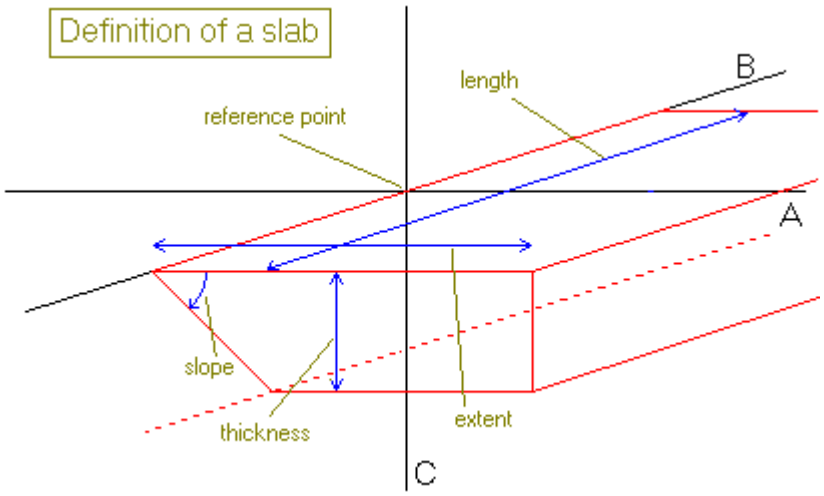
Zero implies infinity when applied to the length, in which case the body is 2-dimensional.

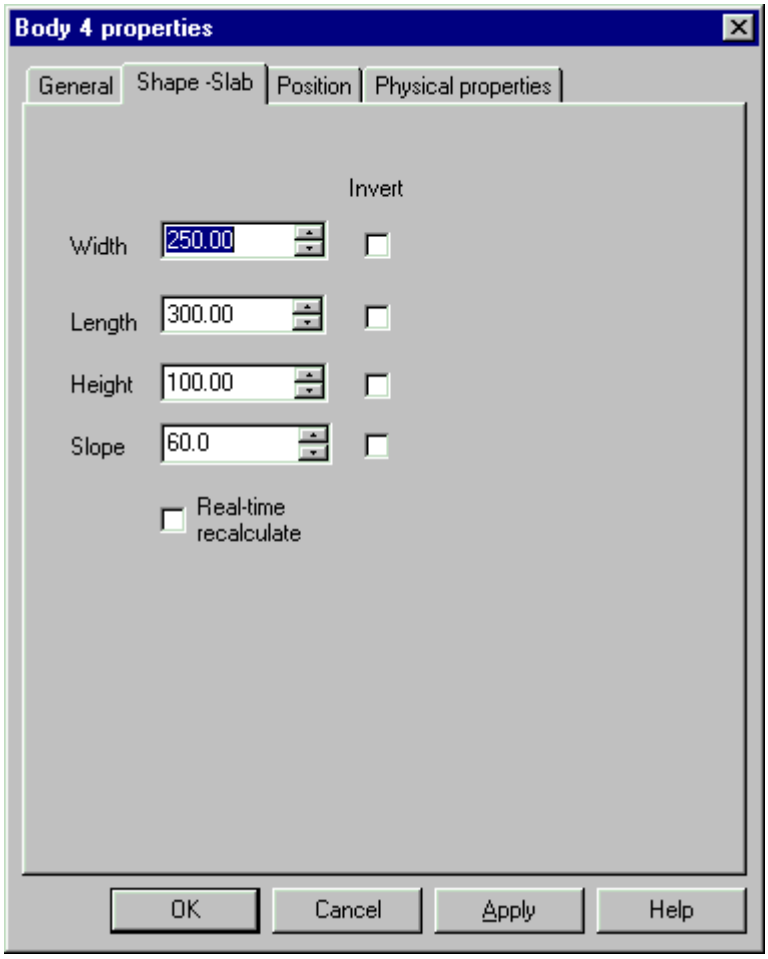
Invert Select a parameter's Invert box to set its invert flag.

Real-time calculate If this is selected then *Potent* automatically recalculates the field due to the model as values are adjusted using the arrows attached to the edit controls.

--- Return to the Model | Edit body dialog.

Slab





The slab is a variety of polygonal prism in which the cross-section polygon is constrained to be a trapezium.

The four shape parameters are as shown in the diagram. Don't confuse "slope", a shape parameter, with *Potent*'s use of the term "dip", which is a position parameter. Changing the slope alters the shape of the cross-sectional trapezium. Changing the dip rotates the entire trapezium about its top left point.

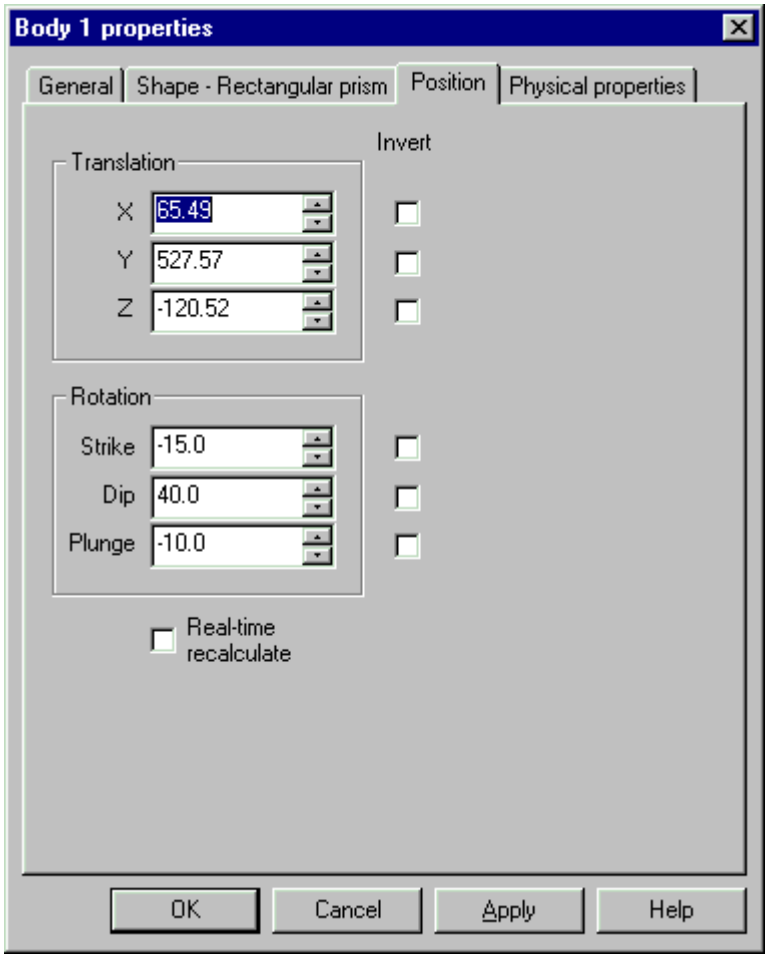
Zero implies infinity when applied to the length, in which case the body is 2-dimensional.

Invert Select a parameter's Invert box to set its invert flag.

Real-time calculate If this is selected then *Potent* automatically recalculates the field due to the model as values are adjusted using the arrows attached to the edit controls.

--- Return to the Model | Edit body dialog.

Edit body – Position



This dialog box allows you to set the translational and rotational position of a body by defining the position of its reference point and the orientation of its (A,B,C) axes relative to the main (X,Y,Z) coordinate system.

X The X coordinate of the body's reference point

Y The Y coordinate of the body's reference point

Z The Z coordinate of the body's reference point. Remember that Z is height, and so the depth of a body is the difference between its Z coordinate and the elevation of the ground surface vertically above it.

Strike The body's rotation about its C axis.

Dip The body's rotation about its B axis.

Plunge The body's rotation about its A axis.

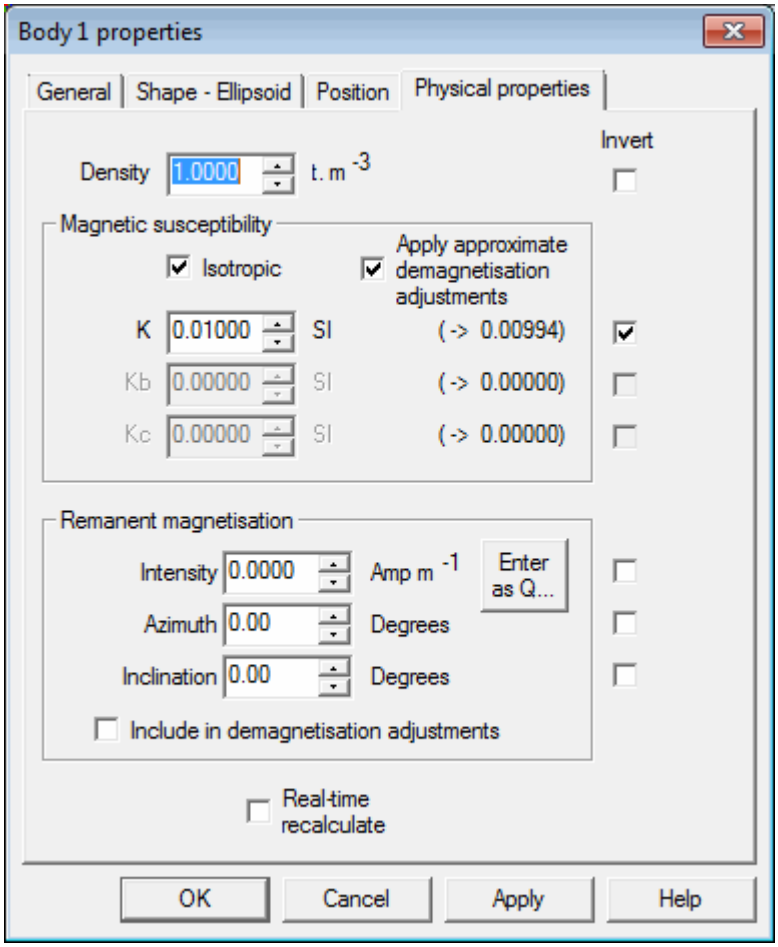
Invert Select a parameter's Invert box to set its invert flag.

Real-time calculate If this is selected then *Potent* automatically recalculates the field due to the model as values are adjusted using the arrows attached to the edit controls.

See also – Position in the User's Guide.

--- Return to the Model | Edit body dialog.

Edit body – Physical properties



Use this dialog box to set the physical properties of a body. Set as many properties as possible, even those that are not relevant to the current calculation mode, as you might want to use the model in a different mode in the future. In particular, try to set the remanent magnetisation, which often is overlooked in magnetic models.

All parameters are expressed in SI units by default. You may change the susceptibility to units of cgs*1000 by using the File | preferences command.

By default the density and susceptibility are specified as the contrast relative to surrounding rocks. If you want to specify the actual density then use the Model | Background properties command to set the background value relative to which calculations will be made.

Density – The density in tonnes m^{-3} (which is identical to gms/cc).

Isotropic – If this is selected then susceptibility is a single (scalar) value. If it is cleared, then susceptibility is defined independently along the three body axes A, B and C.

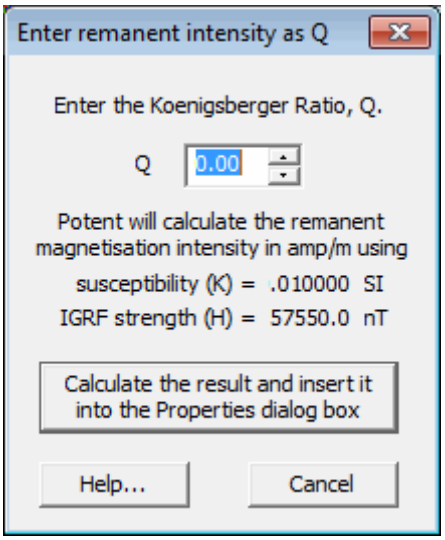
Apply approximate demagnetisation adjustments – If this is selected then *Potent* applies demagnetisation corrections to the susceptibilities of some types of body.

Refer to Calculation algorithms in the User's Guide for details.
As you change the susceptibility with the spinners, the corrected susceptibility (i.e. that which will be used in calculations) is displayed alongside the value you have set. (Only the effect of induced magnetisation is included in this display. If remanent magnetisation is included in the demagnetisation calculation (see below) its effect is not displayed here.)

Susceptibility – The magnetic susceptibility, which is dimensionless. Select **Isotropic** if the body's susceptibility is isotropic, in which case you enter only a single susceptibility value. Otherwise select **anisotropic**, in which case you must enter three susceptibility components. The components are directed along the body's A, B and C axes.

Remanent intensity – The intensity of remanent magnetisation, in amp m-1

(Although Potent requires the remanent intensity to be entered in Amps/m, if you prefer to work with the Koenigsberger ratio (Q) then click **Enter as Q**…



Enter your Q value and click **Calculate…**. Potent will calculate the remanent intensity using the formula

MRem(amp/m) = **Q** * **K**(SI) * **F**(nT) / (**400***PI)

and automatically enter the result in the **Body properties** dialog box.)

Remanent azimuth – The azimuth of remanent magnetisation, measured in degrees clockwise from the +Y axis.

Remanent inclination – The inclination of remanent magnetisation, measured in degrees positive downwards from the (X,Y) plane.

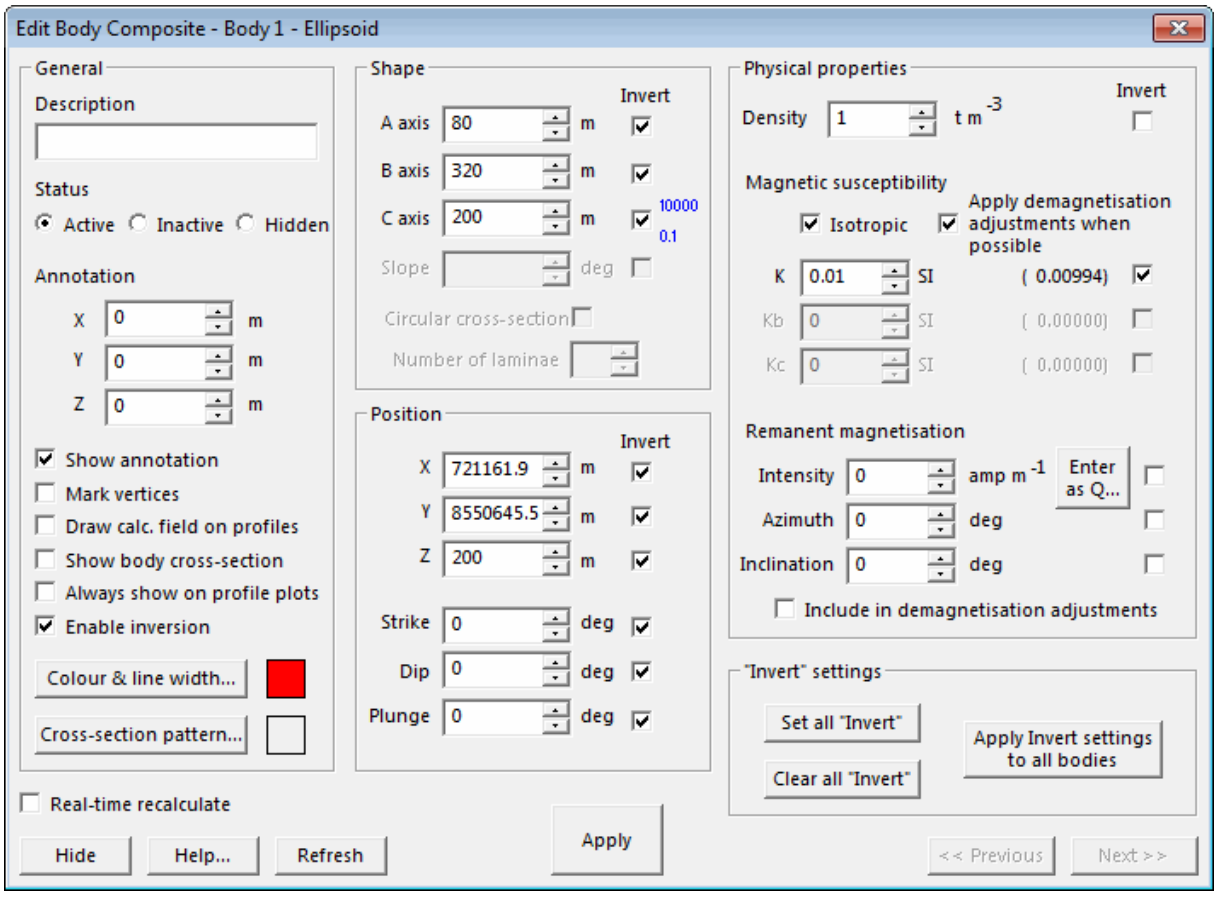
Invert Select a parameter's Invert box to set its invert flag.

Real-time calculate If this is selected then *Potent* automatically recalculates the field due to the model as values are adjusted using the arrows attached to the edit controls.

Remanent "Include in demagnetisation adjustments" – Select this box to have the effect of remanent magnetisation included in the demagnetisation adjustments (see above).

--- Return to the Model | Edit body dialog.

Composite Edit Body dialog box
This dialog box is displayed whenever the Show Composite Edit Body dialog command is selected. It is a large dialog box that works best on a system with a wide display or dual monitors.



The dialog box provides the same modelling functionality as the Compact Edit Body dialog box, but with some significant advantages:

- It remains displayed while you perform other Potent operations.
- It has no tabs – with the exception of the vertices of polygonal prisms, all model parameters are accessible at once. (To edit the vertices of polygonal prism, use either the Body Shape tool or the Compact Edit Body dialog box.)
- It has Previous and Next buttons for cycling through the bodies.
- It has buttons for intelligently setting and clearing inversion flags.

General

This group contains the same commands as the General page of the Compact Edit Body dialog box.

Shape

For all body geometries apart from the Polygonal Prism, this group contains the same commands as the Shape page of the Compact Edit Body dialog box.

Position

This group contains the same commands as the Position page of the Compact Edit Body dialog box.

Physical properties

This group contains the same commands as the Properties page of the Compact Edit Body dialog box.

Set all "Invert"

This button (in the **"Invert" settings** group) causes Potent to intelligently set the Invert flags for the body. For example, if you are not modelling any gravity components then this button will not set the invert flag for Density.

Clear all "Invert"

Clear all the Invert flags for the body.

Apply Invert settings to all bodies

Click this button to attempt to apply the Invert settings for this body to all the other bodies of the model. This is done as intelligently as possible. For example, if this body were a Dyke, which has a Slope parameter, then the flag cannot be set for a body such as an Ellipsoid, which has no such parameter.

If you use this button then you should check each body (perhaps using the **Previous** and **Next** buttons as described below) to confirm that the result is what you want.

Previous / Next

Use these buttons to cycle through the bodies in the model.

Real-time calculate

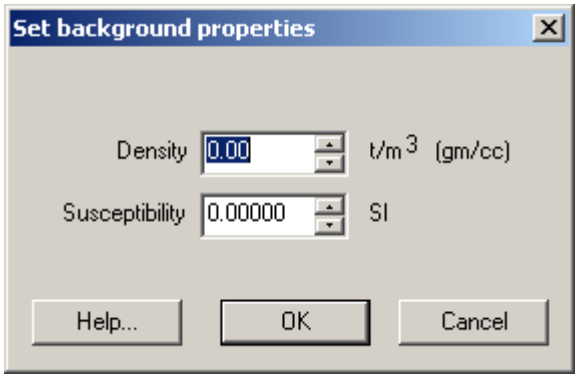
If this is selected then *Potent* automatically recalculates the field due to the model as values are adjusted using the arrows attached to the edit controls.

Apply

If you type values using the keyboard, or don't have **Real-time recalculate** selected, then you must click the Apply button to cause changes to values to take effect. (Note that this does not apply to Invert flags; changes to these are always applied immediately.)

--- Return to the Model menu.

Model | Background properties



This command displays a dialog box that allows you to specify background physical properties that are subtracted from the physical properties of the model during field calculations.

Potent always calculates the *anomalous* field due to the model. For example, if you were gravity-modelling a block of density 2.8 $t.m^{-3}$ embedded in a country rock of 2.5 $t.m^{-3}$ then the gravity anomaly would be calculated using a density of 0.3 $t.m^{-3}$ for the block. By default you would achieve this by entering "0.3" for the block's density in the Physical properties dialog box. However, if you were to set the Background Density to 2.5 then you would enter 2.8 for the density in the Physical Properties dialog box – i.e. you could work with true densities rather than density contrasts.

Equivalent comments apply for the susceptibility. In addition, if the susceptibility is anisotropic then the background susceptibility value is subtracted from each susceptibility component.

Using background properties allows you to produce reports in which the body annotations of density or susceptibility are in actual values, rather than contrasts.

--- Return to the Model menu.

Model | Title

This command allows you to specify a title to be associated with the model. The title is saved with the model by the Model | Save command, and is used to label output generated by the File | Print command.

--- Return to the Model menu.

Model | Show Composite Edit Body dialog

If this command is selected then the Composite Edit Body dialog box remains displayed while you perform other Potent operations.

--- Return to the Model menu.

Model | Create body

- Model | Create body | Sphere
- Model | Create body | Rectangular prism
- Model | Create body | Dyke
- Model | Create body | Slab
- Model | Create body | Ellipsoid
- Model | Create body | Lens
- Model | Create body | Polygonal prism
- Model | Create body | Cylinder

--- Return to the Model menu.

Model | Create body | Sphere

This command adds a "sphere" type body to the model.

The body is created near the centre of the active window and is assigned default shape, position and physical properties. Its status is initially **inactive**. You must use the Edit body command to make it **active**, and to adjust other parameters as required.

--- Return to the Model menu.

Model | Create body | Rectangular prism
This command adds a "rectangular prism" type body to the model.

The body is created near the centre of the active window and is assigned default shape, position and physical properties. Its status is initially **inactive**. You must use the Edit body command to make it **active**, and to adjust other parameters as required.

--- Return to the Model menu.

Model | Create body | Dyke
This command adds a "dyke" type body to the model.

The body is created near the centre of the active window and is assigned default shape, position and physical properties. Its status is initially **inactive**. You must use the Edit body command to make it **active**, and to adjust other parameters as required.

--- Return to the Model menu.

Model | Create body | Slab
This command adds a "slab" type body to the model.

The body is created near the centre of the active window and is assigned default shape, position and physical properties. Its status is initially **inactive**. You must use the Edit body command to make it **active**, and to adjust other parameters as required.

--- Return to the Model menu.

Model | Create body | Ellipsoid
This command adds a "ellipsoid" type body to the model.

The body is created near the centre of the active window and is assigned default shape, position and physical properties. Its status is initially **inactive**. You must use the Edit body command to make it **active**, and to adjust other parameters as required.

--- Return to the Model menu.

Model | Create body | Lens
This command adds a "lens" type body to the model.

The body is created near the centre of the active window and is assigned default shape, position and physical properties. Its status is initially **inactive**. You must use the Edit body command to make it **active**, and to adjust other parameters as required.

--- Return to the Model menu.

Model | Create body | Polygonal prism
This command adds a "polygonal prism" type body to the model.

The body is created near the centre of the active window and is assigned default values for shape, position and physical properties. Its status is initially **inactive**. You must use the Edit body command to make it **active**, and to adjust other parameters as required.

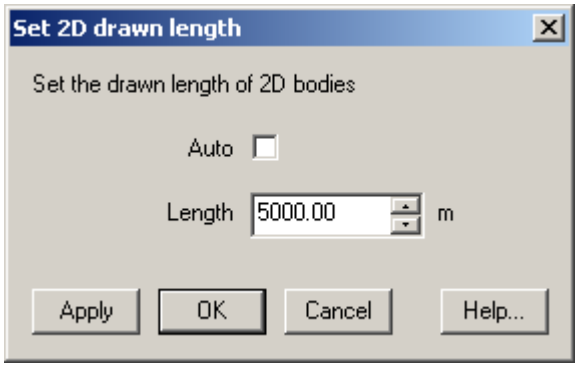
--- Return to the Model menu.

Model | Create body | Cylinder
This command adds a "cylinder" type body to the model.

The body is created near the centre of the active window and is assigned default shape, position and physical properties. Its status is initially **inactive**. You must use the Edit body command to make it **active**, and to adjust other parameters as required.

--- Return to the Model menu.

Model | Set 2D drawn length



A 2D body has infinite strike length. Therefore, when it is drawn a decision must be made as to how much of the strike length is to be represented. When drawing on-screen this is not an issue; by default *Potent* simply assigns a long strike length and relies on the clipping that is automatically applied at the window boundaries by the various Windows drawing functions.

However, when exporting to DXF and polygon files *Potent* has no way of knowing the spatial extent of, for example, a destination map, and hence of what is an appropriate strike length to draw.

The **Set 2D drawn length** command displays a dialog box that allows you to enter the length that is to be drawn, either on-screen or in exported DXF and polygon files. If **Auto** is selected then *Potent* attempts to estimate a suitable strike length on the basis of the geographical extent of the main plan window.

Model | Always draw bodies as wireframe

When this menu option is selected *Potent* switches all body drawing to wireframe, regardless of Window | Options settings. This speeds up display of complex models.

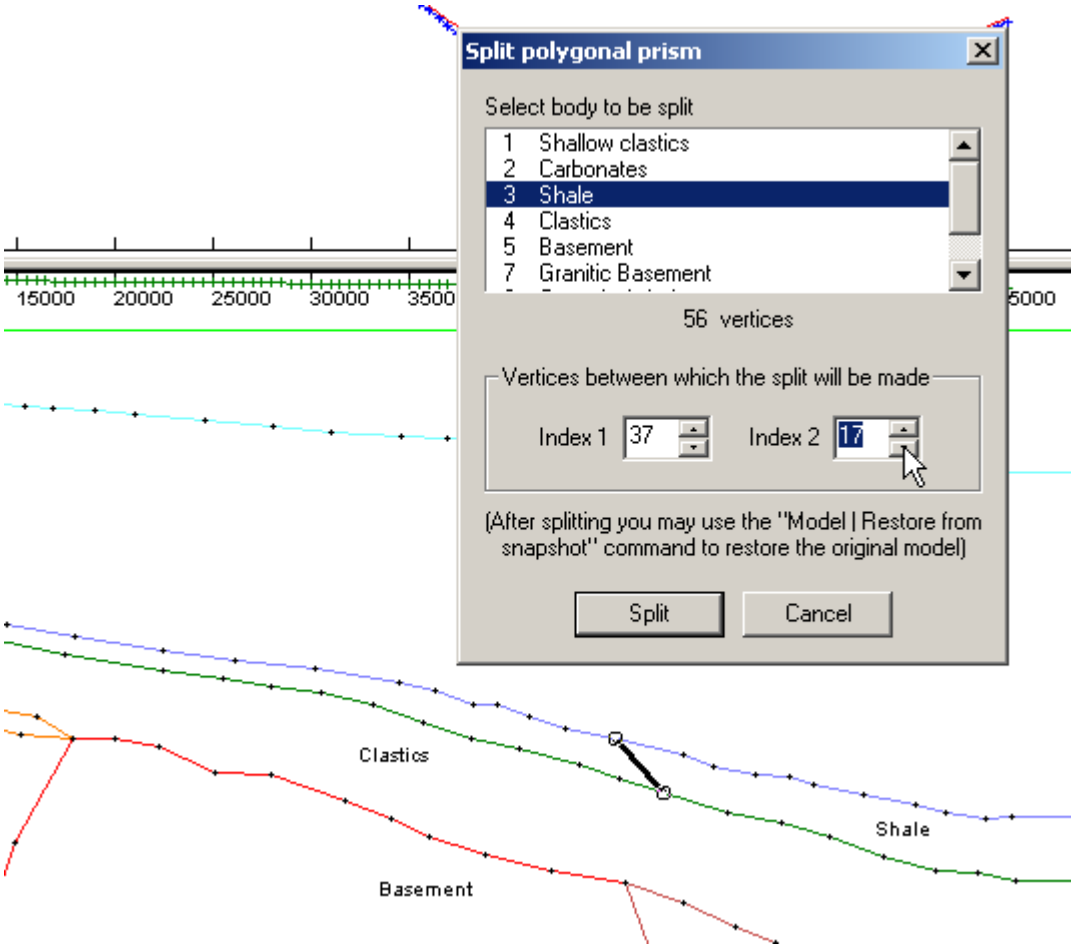
Model | Split polygonal prism

Use this command to divide a polygonal prism into two separate prisms using the **Split polygonal prism** dialog box. Note that the vertices to be joined can not include the reference point. Click **Change Reference** in the Edit body | Shape dialog box to move the reference point if necessary.

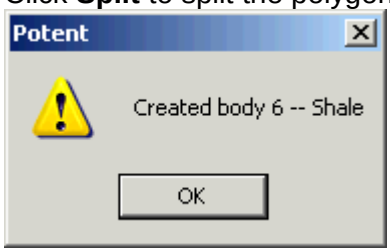
1. Select the body that you want to split.

2. Use the spinners attached to **Index 1** to highlight one of the two vertices that will define the ends of the line across which the body will be split.
3. Use the spinners attached to **Index 2** to highlight the other vertex.

A thick black line is drawn to mark the position of the boundary that will form the two new bodies, as in this example:



Click **Split** to split the polygon at the marked boundary:



The new body takes on the same properties (including description) as the original body. When you click **OK** Potent automatically displays the Edit Body dialog box for the new body so you can immediately make the desired changes.

Splitting the polygonal prisms creates a “Before splitting polygonal prism” model snapshot. Use the Model | Restore from snapshot command to restore the model to the state that existed before the split.

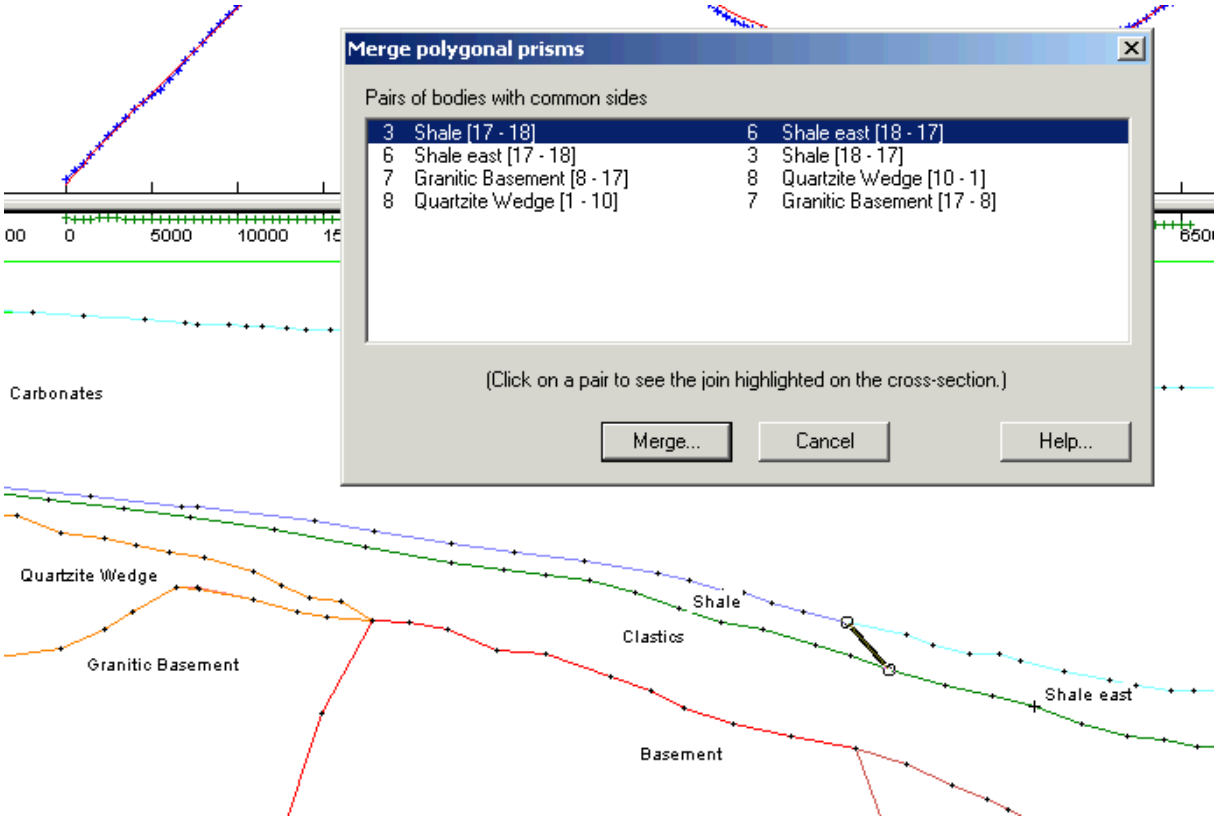
Model | Merge two polygonal prisms

Use this command to combine two polygonal prism bodies into a single body. Potent scans all combinations of pairs of bodies and lists all that have common vertex ranges if:

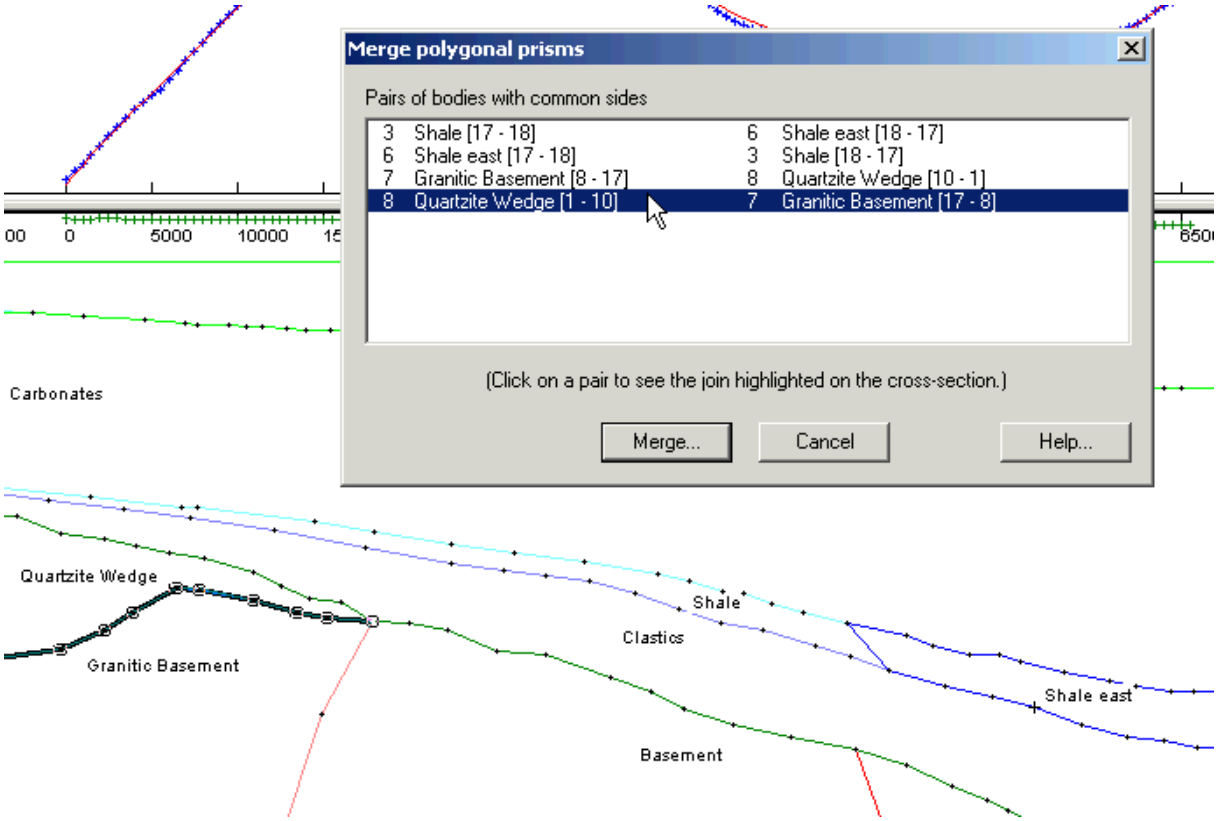
- The bodies have the same orientation (strike, dip and plunge),
- and they have the same strike length,
- and either
  both bodies are 2-D and their plunge is zero,
  or
  they are in line. (I.e. the line connecting the reference points is parallel to the B axis. See polygonal prism shape.)

In the following example only body pairs (3, 6) and (7, 8) satisfy these conditions. Each pair occurs twice (though this is not always the case), one entry in the list being the transpose of the other. This is because when the merge takes place the composite body takes on the properties of the body in the left-hand column (see below), so having the two entries allows you to choose which body persists after the merge.

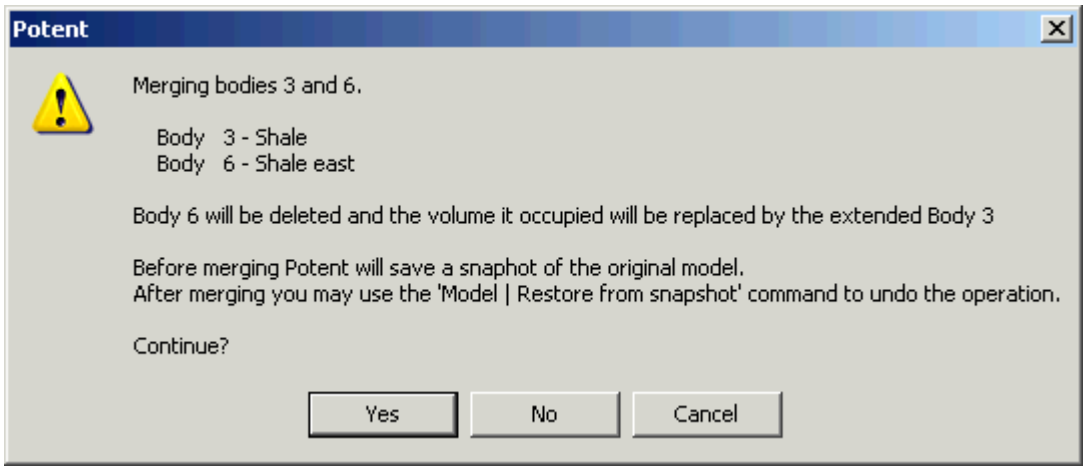
The bodies of the pair that is selected in the list box have their common sides marked by a thick black line:



This example shows the common sides of the other pair:



When you have selected the pair of bodies to be merged, click **Merge**. In this example the first pair was selected:



Click **Yes** to merge the bodies. The body in the right-hand column of the **Merge polygonal prisms** dialog box will be deleted and the other body will be extended to fill its volume.

Click **No** to return to the **Merge polygonal prisms** dialog box.

Click **Cancel** to terminate the merge operation.

Merging the polygonal prisms creates a “Before merging polygonal prism” model snapshot. Use the Model | Restore from snapshot command to restore the model to the state that existed before the merge.

--- Return to the Model menu.

Calculation menu

Automatic

Recalculate now

Dynamic

Invert

Invert options

Restore from snapshot

Reset all weights

Show previous result command

Calculation | Automatic

When this menu option is selected *Potent* automatically recalculates the field due to the model in response to changes to body parameters.

If the option is cleared, you must use the Calculation | Recalculate now command to manually calculate the field due to the model.

See also - Calculating the field due to the model in the User's Guide.

--- Return to the Calculations menu.

Calculation | Recalculate now

This command to forces *Potent* to calculate immediately the field due to the model. Use this command periodically if the **Calculation | Automatic** option is cleared.

You might also choose this command when the **Calculation | Automatic** option is selected, but you think that *Potent* has nevertheless failed to keep the calculated field up to date. (This could happen as a result of failures in the algorithms that are used to keep track of the status of model changes. Any such failure should be reported to Geophysical Software Solutions for correction in future revisions of the program.)

--- Return to the Calculations menu.

Calculation | Dynamic

When this menu option is selected *Potent* recalculates and redraws profiles dynamically when you move a body with the **Tools | Move bodies** command, or alter the shape of polygonal bodies with the **Tools | Body shape** command.

Effective use of this feature requires Potent to recalculate extremely quickly in response to changes in the position of the body. Therefore it is most effective when used with simple bodies and few observation points displayed in profiles.

See also - Calculating the field due to the model in the User's Guide.

--- Return to the Calculations menu.

Calculation | Invert

This command to starts *Potent*'s inversion calculation. Refer to the Inversion modelling topic in the User's Guide for details.

See also - Tutorial in the *Getting Started* document for a simple inversion example, and the **Calculation | Invert options** command.

You can stop the inversion at any time by pressing the **Esc** key.

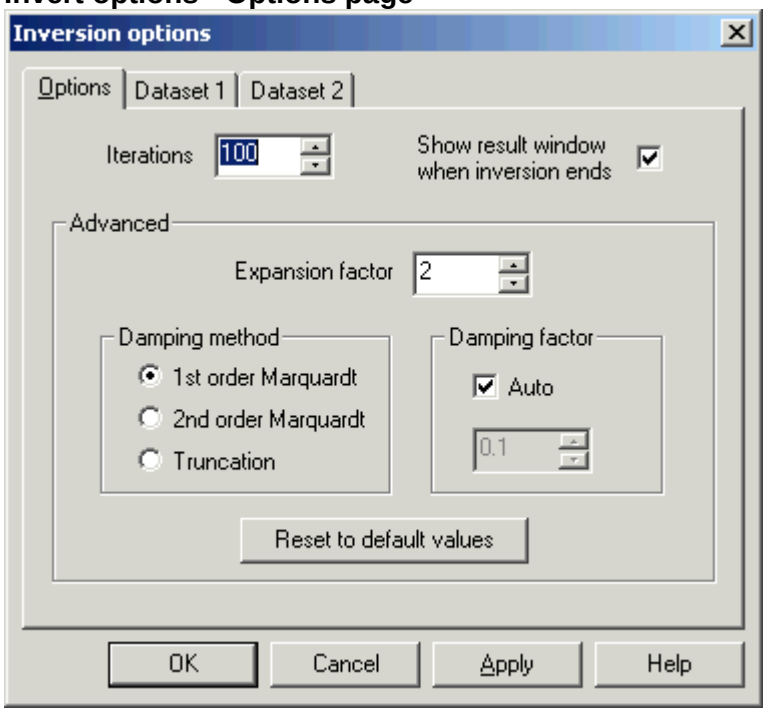
--- Return to the Calculations menu.

Calculation | Invert options

This command displays a tabbed dialog box. The first page is the Options page. This is followed by a Dataset page for each dataset.

--- Return to the Calculations menu.

Invert options - Options page



The options in this dialog box control the workings of the inversion algorithm.

Iterations - The inversion will proceed for the number of iterations you specify here, unless convergence occurs first, or the inversion is "cornered" (i.e. can't iterate any further). (You can press the **Esc** key at any time during the inversion to stop it before the iteration limit is reached.)

Expansion factor - This is the factor by which Marquardt damping is increased when convergence problems occur.

Damping method - The type of damping determines how the inversion procedure handles "unimportant" parameters. First order Marquardt is the usual method, and attenuates variations in the unimportant parameters linearly. Second order Marquardt attenuates variations parabolically, while in the truncation method they are eliminated completely.

Damping factor - This factor determines the severity of Marquardt damping applied when solving the equations. (It is ignored if the truncation option is selected.) If **Auto** is selected, Potent calculates an appropriate damping factor at each iteration based on the range of singular values obtained from the current decomposition.

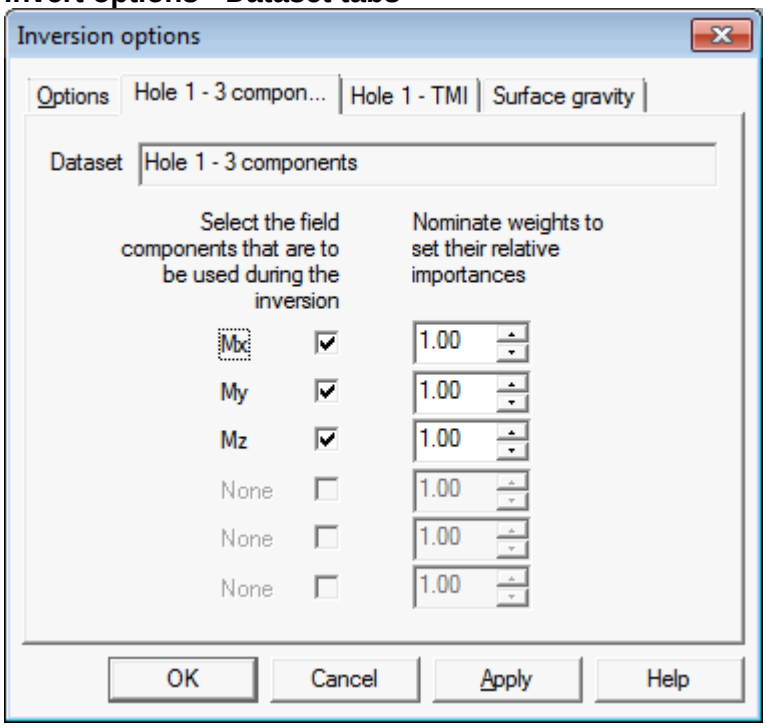
Show result window when inversion ends – If this is selected then at the end of the inversion Potent displays a small dialog box that displays the final RMS difference between the observed and calculated values. (The RMS difference also is displayed on the status bar, but this display is soon replaced by other status bar information.) Note that you can use the Show previous result command to display the last inversion result at any time.

Click the **Reset to default values** button if you want to return all options to the defaults used by Potent.

See also Jupp & Vozoff, 1975.

--- Return to the Calculations menu.

Invert options - Dataset tabs



Use this dialog box to determine how each dataset will contribute to the inversion. Each tab is labelled with the dataset title, as set by the Observations | Dataset title command. If the name of a dataset is longer than around 20 characters its tab is labelled with a truncated form, suffixed by an ellipsis (…). In this case you see the complete name in the read-only **Dataset** box at the top of the page.

There is a check box and a weight for each component in the dataset.

Select the field… - Select this if you want the component to be used in the inversion. (Each selected component becomes a column in the Jacobean matrix, as described in Jupp & Vozoff, 1975.)

Weights - Set the importance of this component relative to other components in this dataset and in all other datasets. For example, one component might be the TMI

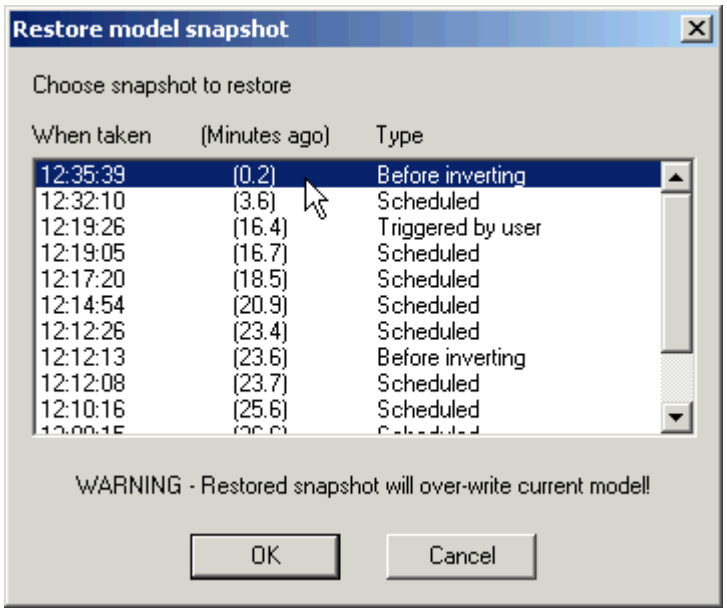
with a range of 100nT and another the vertical gravity with a range of 1mgal. In this case you could set the TMI weight to 0.01 relative to the gravity, so that the TMI values did not dominate. Similarly, if the data in one dataset is more dense than another, you might choose to reduce the weight of each component in that dataset to compensate for the greater number of values.

(Note: These weights apply to all observation points in a dataset. Use the Set **Inversion Weights tool** to weight individual observation points.)

--- Return to the Calculations menu.

Calculation | Restore from snapshot

This command allows you to quickly revert to the state the model was in prior to the Invert command. In the **Restore model snapshot** dialog box double click on the most recent "Before inverting" entry.



--- Return to the Calculations menu.

Calculation | Reset all weights

Use this command to reset all inversion weights to 1.

See also - Inversion weights tool

--- Return to the Calculations menu.

Calculation | Show previous result

This command displays the RMS difference that was returned by the previous inversion.

--- Return to the Calculations menu.

Tools menu

Potent uses a variety of tools to perform various functions. The tools are:

- Default
- Subset
- Move bodies
- Body shape
- Zoom
- Delete observations
- Regional
- Invert
- Window observations
- Backdrop image
- Inversion weights
- Query contour

Tools | Default

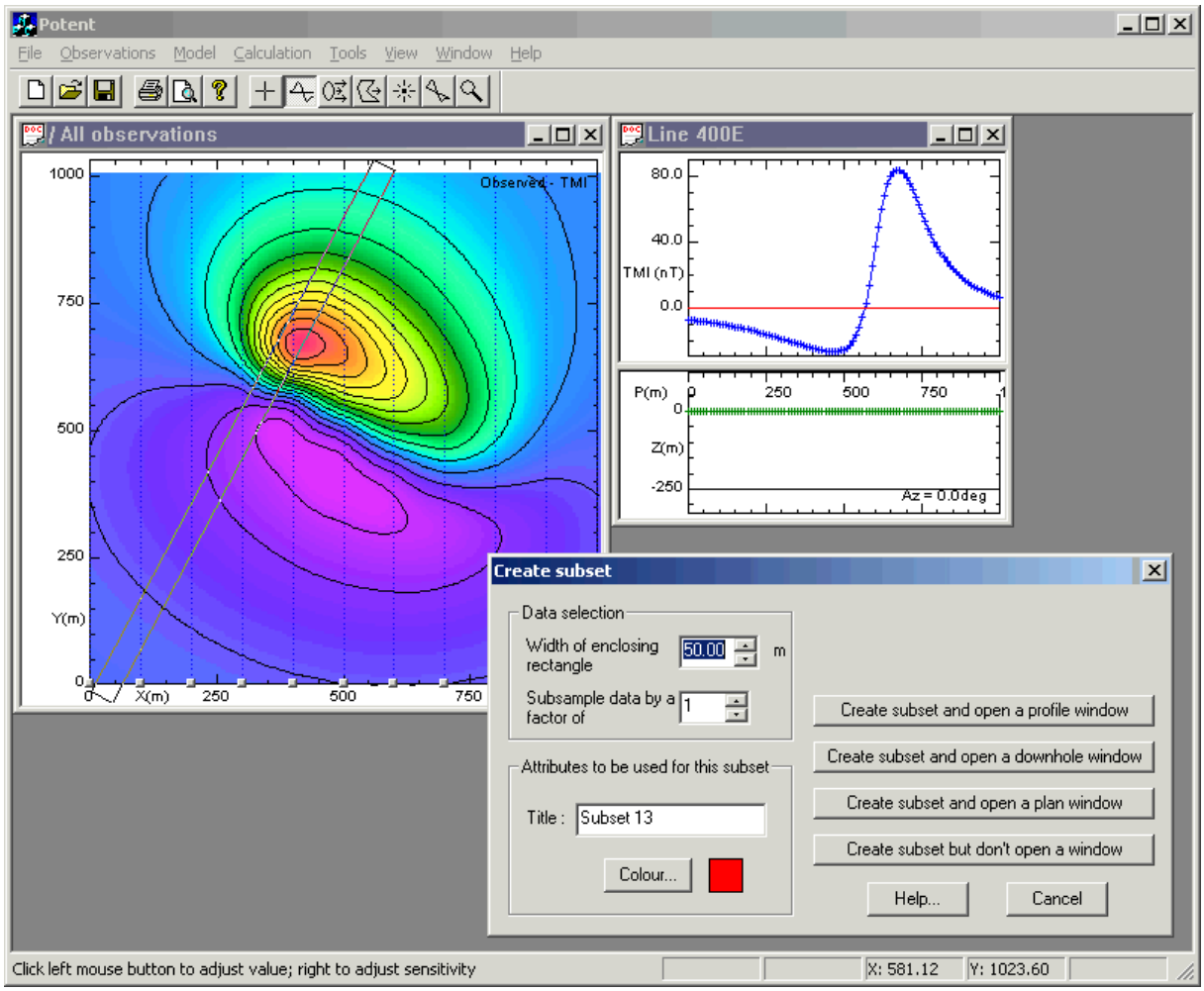
This tool provides basic functions that are shared by the other tools. These functions include:

- Display of coordinates and values on the status bar as the cursor moves over windows.
- Editing a body by right clicking when the cursor is over the reference point of the body.
- Display of the local window menu via a right click on the window.

--- Return to the Tools menu.

Tools | Make subset – box

As well as providing the basic behaviour of the default tool the subset tool allows you to create a subset of your observations, particularly if the required sample of observations is long and narrow. The strategy is to create a rectangle that encompasses the observations that you want to include in the subset, as shown in this example.



Note that you can create a subset from Plan windows (as shown above) and from the profile and cross-section panes of profile windows. Creating a subset from observations displayed in a cross-section pane is particularly useful when you are working with down-hole data.

Refer to Creating a subset in the Getting Started document for details.

See also:

- Subsets
- Why is the observed field profile spiky?
- Create subset dialog box

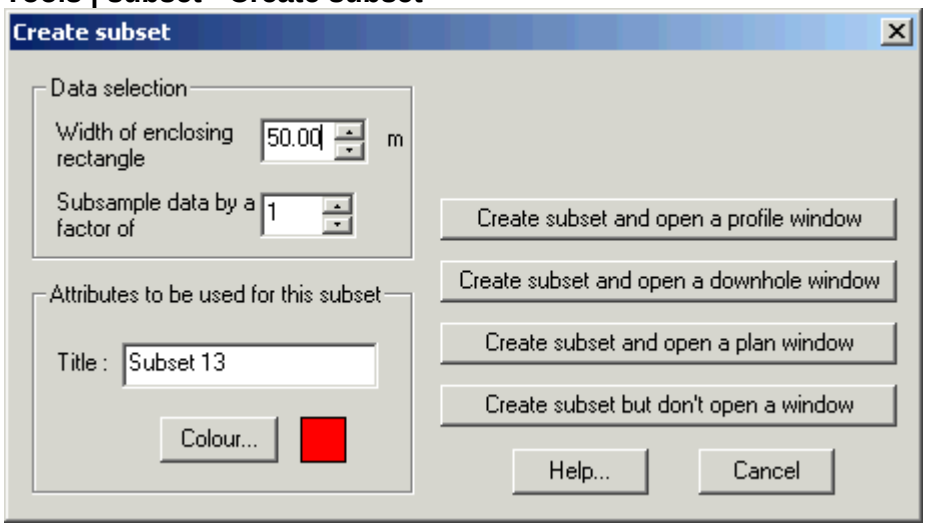
--- Return to the Tools menu.

Tools | Make subset – polygon

Use this tool to create a subset of arbitrary polygonal shape. Click the left button to define each vertex, then click right to close the polygon. The Create subset dialog box is displayed.

See also – Make subset – box.

Tools | subset - Create subset



This dialog box is displayed when you start to create a subset using the Tools | Subset – box or Tools | Subset – polygon command. The dialog box controls provide are provide the following functions:

Width of enclosing rectangle - This option is available only if the dialog box was displayed in response to the Tools | Subset – box command. When the dialog box appears you have already defined the start and end points of the centre line of a rectangle. Set the width so that the rectangle encloses the observations that you want to be included in the subset. Note that “zero” implies “infinity”, in which case all observations normal to the central line would be included in the subset. If you want to create a narrow subset (probably for displaying in profile form) then you **must** specify a non-zero width (as in the example shown in the Tools | Subset command).

Subsample data by a factor of - If you set this to 3, for example, then only every third observation in the selected area will be included in the subset. Sub-sampling data improves display and calculation performance.

Title - You may set a title that will be used to label the subset on plan windows when the appropriate options are selected in the Plan Window Settings – Subsets dialog box. It also becomes the title of all windows that are used to display the subset.

Colour - Set the colour with which the observations are drawn when the appropriate options are selected in the Plan Window Settings – Subsets dialog box.

Create subset and open a profile window - Creates the subset and immediately displays it in a new profile window.

Create subset and open a plan window - Creates the subset and immediately displays it in a new plan window.

Create subset but don’t open a window - Creates the subset but does not create an associated window. You must use the Observations | Subset management command at some stage to create a window for the subset.

--- Return to the Tools menu.

Tools | Move bodies

Use this tool to drag bodies on plan and cross-section windows. Position the cursor on the reference point of the body and hold down the left mouse button while you drag the body to a new position.

--- Return to the Tools menu.

Tools | Body shape

This tool allows you to graphically edit the vertices of polygonal cross-section bodies on plan and cross-section windows.

- To move an existing vertex - Position the cursor on the vertex and hold down the left mouse button while you drag the vertex to a new position. (When applied to dykes and slabs, other vertices are automatically adjusted so as to preserve the respective parallelogram or trapezoidal shape.)
- To delete an existing vertex - Position the cursor on the vertex and hold down the **Ctrl** key wile you click the left mouse button. (This command is ignored for dykes and slabs, which always have four vertices.)
- To insert a new vertex - Position the cursor on the line connecting two existing vertices. The cursor changes to  . Click the left mouse button to create a new vertex midway between the two existing ones. You can then drag the new vertex to its required location.

The above notes apply to a single polygonal cross-section body. If there are multiple bodies with coincident vertices (e.g. adjacent strata in a layered model) then by default *Potent* will apply the operations to all bodies that have a vertex at the same position on the screen. If you want to apply the operation to only a single body, then hold down the **Shift** key while performing the required mouse operation.

--- Return to the Tools menu.

Tools | Zoom

Use this tool to zoom in and out by multiples of two. Position the mouse cursor over the area of interest on any window and click the left mouse button to zoom in, or the right mouse button to zoom out.

Note that *Potent* zooms by automatically adjusting the scale settings of the window under the mouse cursor. In order to do this it must clear **Auto** in the appropriate Scale dialog box. Therefore, if you resize a window while in the zoomed state then the plot might appear distorted (squares appear as rectangles, etc.). Your original scale settings are restored automatically when you remove the zoom (eg by zooming out by two right clicks after you had zoomed in by two left clicks).

--- Return to the Tools menu.

Tools | Delete observations

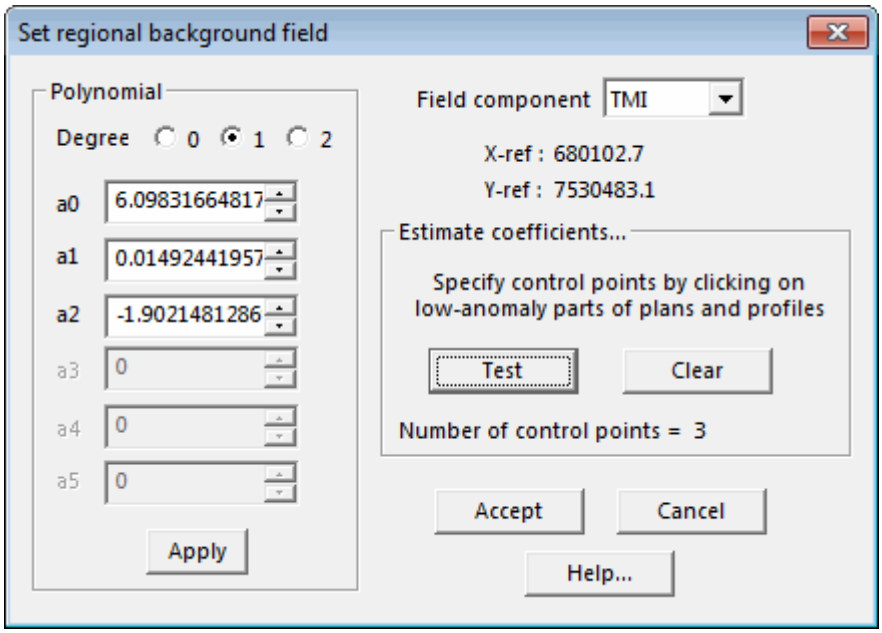
Use this tool to remove spurious observations from the subset associated with a profile or plan window. Position the mouse cursor over the observed field value (profile pane) or position (cross-section pane or plan window) of the observation to be removed and click the left button. All observations within three pixels of the cursor will be removed from the subset.

Note that the observation is not removed from the actual dataset, but only from the subset that represents a sample of the dataset.

--- Return to the Tools menu.

Tools | Regional

Use this tool to remove a regional background from the observations of the dataset that was the active dataset when the tool was selected. The dialog box is “modeless”, meaning that you can still access the underlying windows while it is displayed.



The following paragraphs provide information about the individual controls in the dialog box. Refer to the Regional Field topic in the User’s Guide for detailed guidance in using this tool.

Field component

This drop-down list box contains an entry for each field component in the dataset. Choose the component from which you wish to remove a background field.

X-ref, Y-ref

These are the X and Y coordinates of the geographical centre of the dataset. They are used as X and Y offsets in the polynomial calculation (see below) to prevent high order coefficients becoming very small.

Polynomial group

Choose a polynomial degree from the options provided:

- Degree 0 is a horizontal plane surface (i.e. a constant offset).
  $$r = a_0$$
- Degree 1 is an inclined plane surface.
  $$r = a_0 + a_1(x - x_{ref}) + a_2(y - y_{ref})$$
- Degree 2 is a second-degree curved surface (i.e. no inflexions).
  $$r = a_0 + a_1(x - x_{ref}) + a_2(y - y_{ref}) + a_3(x - x_{ref})^2 + a_4(x - x_{ref})(y - y_{ref}) + a_5(y - y_{ref})^2$$

You may set the coefficients directly or by using the **Estimate coefficients** controls (see below) to have *Potent* calculate them for you. If you alter coefficients by clicking on the spinner buttons then profile plots are dynamically redrawn to show the effect of the changed regional. Click the **Apply** button to update images of observed and residual fields (on which the regional has an effect), and to adjust profiles if you have altered coefficients by typing values in directly.

The coefficients can be included in printed reports by making the appropriate selection in the File | Page layout dialog box.

Estimate Coefficients

You can have *Potent* calculate coefficients automatically by setting control points on plans and profiles as follows:

- Click on a profile pane that shows the component selected in the **Field component** list-box. A warning will be sounded if you click on a profile showing any other component.
- Click on a plan window that shows an observed field image of the component selected in the **Field component** list-box. A warning will be sounded if you click on a plan showing any other component.

The control points are marked temporarily by red crosses. When you have set sufficient points, click **Test** to calculate the coefficients (as a root-mean-square fit to the control points) and to preview their effect on the observed values. Click **Clear** to clear the current settings prior to trying again.

Delete an existing control point by clicking close to its red cross. Click **Test** to recalculate the coefficients.

Remember that after you have calculated coefficients using **Test**, you can fine-adjust them using the controls in the **Polynomial group**, as described above.

Refer to the Regional Field topic in the User's Guide for more information.

Accept

Accept the current settings and close the dialog box.

Cancel

Close the dialog box and revert to the previous settings.

--- Return to the Tools menu.

Tools | Polygon inversion

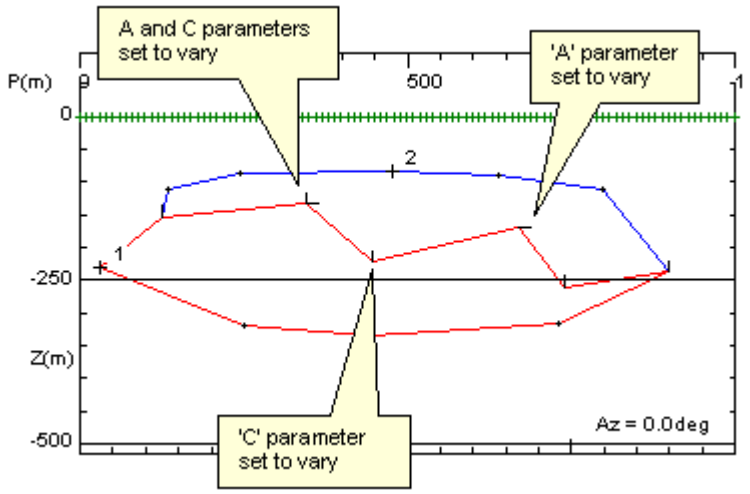
The Invert tool allows you to set graphically the inversion properties of polygonal prism type bodies.

Left click on a vertex to toggle the Invert flag of its C (vertical) coordinate. Hold down SHIFT for the A (horizontal) coordinate. Holding down CTRL will set both A and C unless both are already set, in which case it will deselect both.

A vertex that has an Invert flag set is marked as follows:

‘A’ coordinate varies – horizontal bar.

‘C coordinate varies – vertical bar.

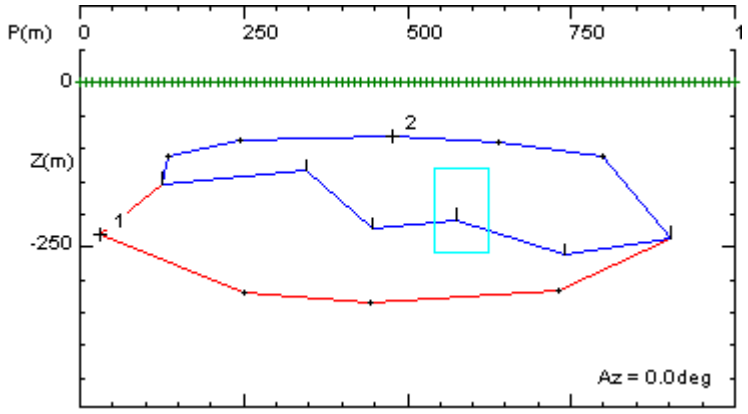


When you use the Invert tool on a vertex that is common to two or more bodies (as in an interface between two layers, as shown in the above example), the invert flags are set for each body. During inversion, *Potent* will synchronise the adjustments to the vertices to keep them together.

Refer to the Inversion modelling topic in the User’s Guide for more information.

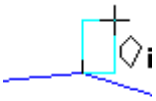
Limiting the inversion range

You can use the **Polygon inversion** tool to limit the range of movement of a vertex during inversion. When a vertex has limits imposed a cyan box is drawn around it:

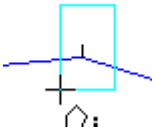


This box defines the range of possible movement of the vertex during inversion. Defining the box is a two stage graphical process. With the **Polygon inversion** tool selected:

1. Position the cursor over the vertex and press and hold the **right** mouse button. Then drag the cursor to a corner of the required box and release the button:



2. Position the cursor over the vertex again and, using the **right** mouse button, drag the cursor to the diagonally opposite corner of the required box and release the button:



--- Return to the Tools menu.

Tools | Window observations

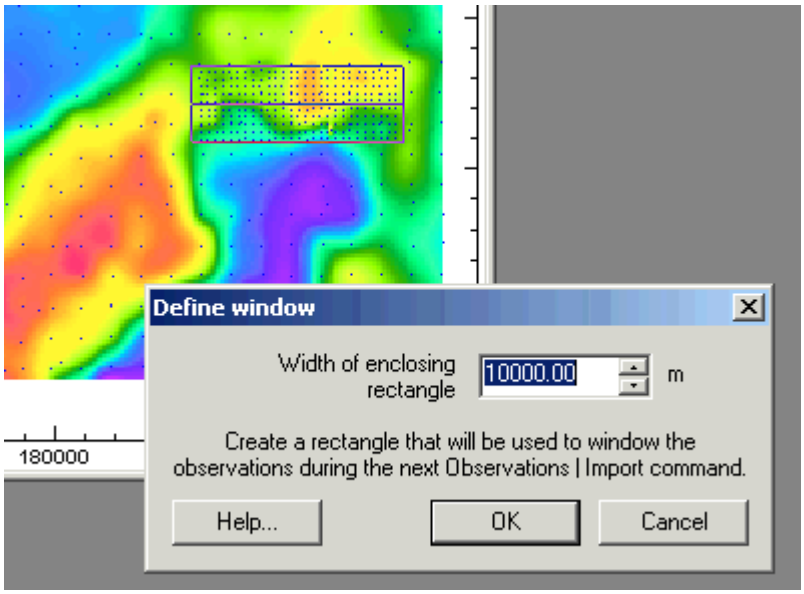
This tool simplifies the task of extracting an area of interest from a large dataset by automatically populating the **X**, **Y** and **Strike** fields in the Observations | Options - Window dialog box. To use the tool:

1 Drag a line across the centre of the rectangular area you want to define.

2 In the **Define window** dialog box, use the spinners to set the **Width of enclosing rectangle** so as to expand the line into the required rectangle.

3 Click **OK** to store the window for use the next time you use the Observations | Options - Window dialog box.

In this example a window is being prepared to re-read the area of detailed data from the gravity data file.



See also - Working with large datasets in the User's Guide.

--- Return to the Tools menu.

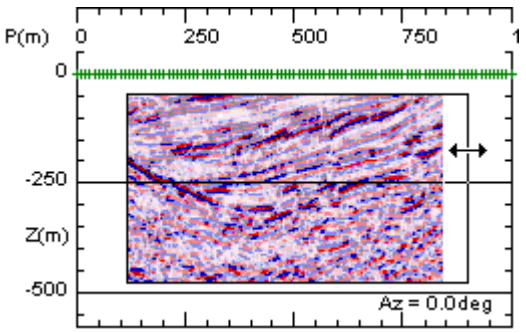
Tools | Backdrop image

Use this tool when a backdrop image has been loaded using **Load** button on the Cross-section page of the Profile window settings dialog box.

The tool works in two ways:

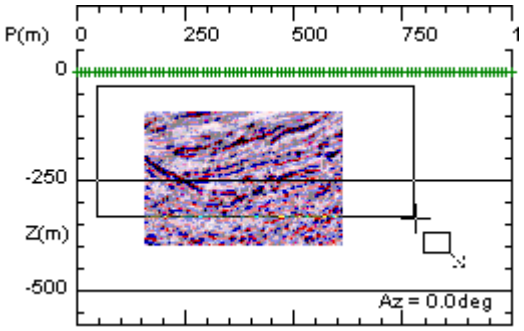
Method 1 - Click on the image to select it. A temporary black border is drawn around it. Drag the sides and corners of the border to resize the image (the cursor becomes a two-headed arrow), or drag the entire image by pressing (and holding) the left mouse button somewhere inside the border (the cursor becomes a four-headed arrow).

In this example the user is extending the right hand edge of the image:



Method 1 - If necessary, click anywhere outside the image (but still in the cross-section pane) to deselect it. Then drag a rectangular area (click and hold the left mouse button) where you want the image to be positioned. When you release the button the image will move and resize itself into the specified area.

In this example the image will be repositioned to fill the indicated rectangle:



See also - Working with stratigraphic models in the User's Guide.

-

--- Return to the Tools menu.

Tools | Inversion weights

This tool allows you to set the importance of observations in the inversion process. It operates only on profile windows.

By default, each observation has a weight of 1. You may vary this between 0 (the observation does not contribute to the inversion process at all) and 10 (the observation behaves as though there were 10 observations clumped together).

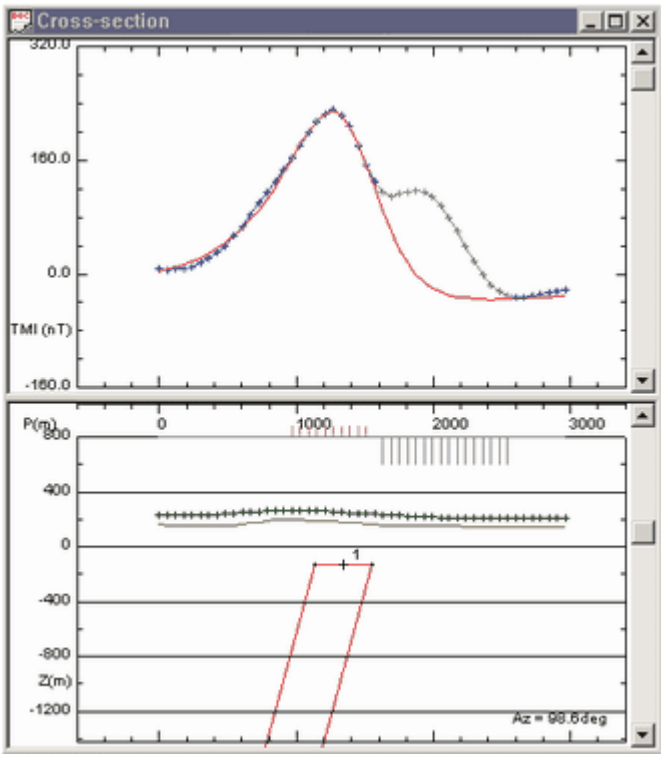
1. On a profile window (profile pane or cross-section pane) drag a line that stretches across the points that you want to weight. (The vertical positions of the ends of the line are unimportant; only the horizontal extent matters.)
2. In the dialog box, enter the weight that you want to associate with these observations and click **OK**.

On the cross-section pane a grey horizontal "weights" line is drawn at the top of the Z axis. Observations that have a weight greater than 1 have an associated light red bar extending upwards from the weights bar. Similarly, observations that have a weight less than 1 have an associated light blue bar extending downwards from the weights bar. In each case the length of the bar is related to the weight, with the maximum upwards length being for a weight of 10, and the maximum downwards length being for a weight of 0.

On the profile pane any section of the observed field plot that corresponds to observations with weight 0 is drawn in grey.

You can use this tool to set weights of 0 for spurious points on a profile, rather than deleting them using the Delete observations tool.

In this example the anomaly to the right has been weighted to 0 (grey vertical bars extending downwards from the weights line for 1 scale division, and that section of curve drawn in grey). The observations forming the peak of the anomaly have weights set to 5 (light red bars extending upwards from the weight line). The inversion has ignored the observations with 0 weight, but has produced a very good fit at the peak of the anomaly (weight 5) at the expense of the fit on the ends.



You can quickly reset all weights to 1 by choosing the **Calculation | Reset all weights** command.

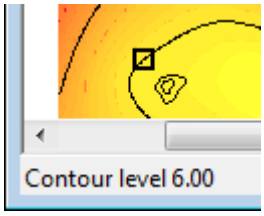
See also - Inversion modelling in the User's Guide.

--- Return to the Tools menu.

Tools | Query contour

As Potent does not label contours, this tool allows you to display the value of a contour on the status bar.

When the tool is selected, the cursor changes to  . Position the cross over a contour; the cross changes to a small square and the contour value is displayed on the status bar, as in this example:



If the Plan window shows topography contours, the functionality of the **Query contour** tool is extended to allow you to create a flat-lying polygonal body corresponding to the contour. Repeated use of this feature allows you to build up a layered model that could be used, for example, for simple gravity terrain corrections.

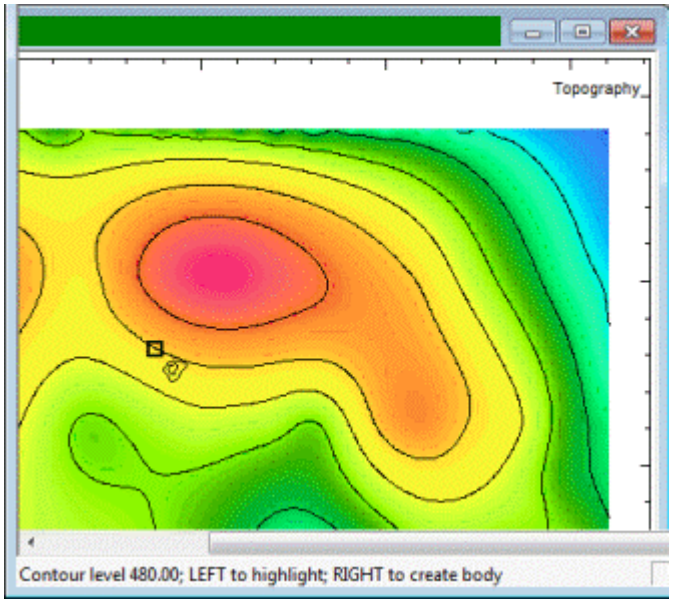
--- Return to the Tools menu.

Creating a polygonal body from a topographic contour

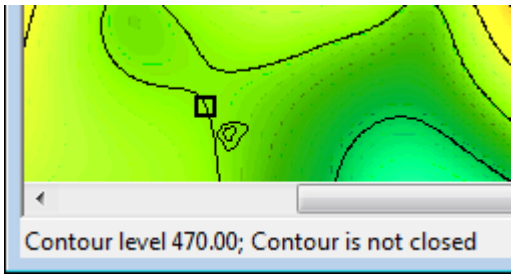
If a Plan window shows topography contours, the functionality of the Query contour tool is extended to allow you to create a flat-lying polygonal body corresponding to the contour. Repeated use of this feature allows you to build up a layered model that could be used, for example, for simple gravity terrain corrections.

Note: Due to the stepped nature of the model, some observation points might be located inside a body. Also, the steps might cause sudden local variations in the calculated field. For both these reasons this type of model should not be used for terrain corrections directly over a topographic feature, but only for points in the immediate neighbourhood of it.

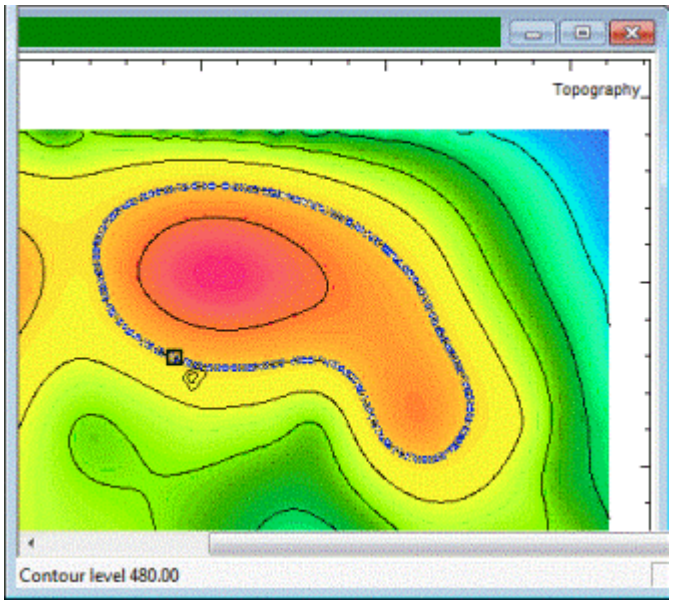
When you point to a contour, the status bar message includes hints for highlighting the contour and creating a body:



If the contour is not closed (i.e. it is truncated at the edge of the grid) then an appropriate message is displayed on the status bar:

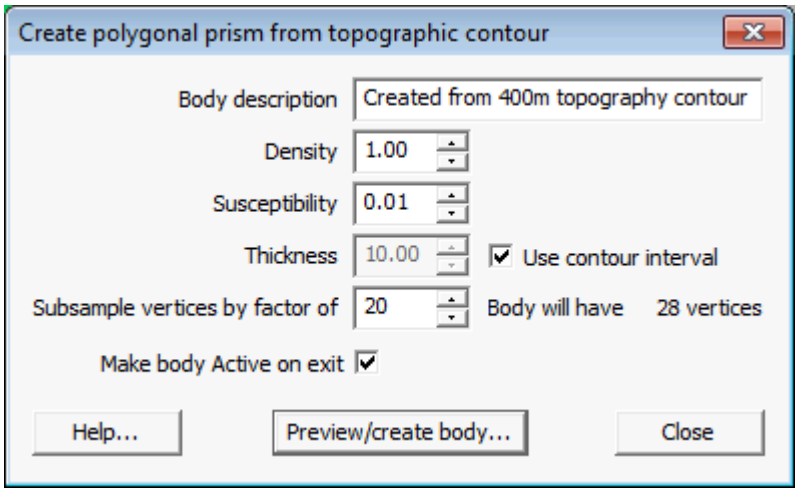


If the contour is closed you can click LEFT to highlight it:



Note: If the contour is not smooth the algorithm that follows the contour might fail, with confusing results. In this case try using one or two applications of the **Smooth** function in the Image properties dialog box.

To create a body defined by the contour, click RIGHT. The **Create polygonal prism from a topographic contour** dialog box is displayed:



This dialog box allows you to set essential properties for the body. The first time it is displayed in your Potent session the various fields contain preset default values. Any changes you make will persist either until the end of the session, or until a further change is made.

Body description - Set the description that is associated with the body.

Density - Set the density of the body.

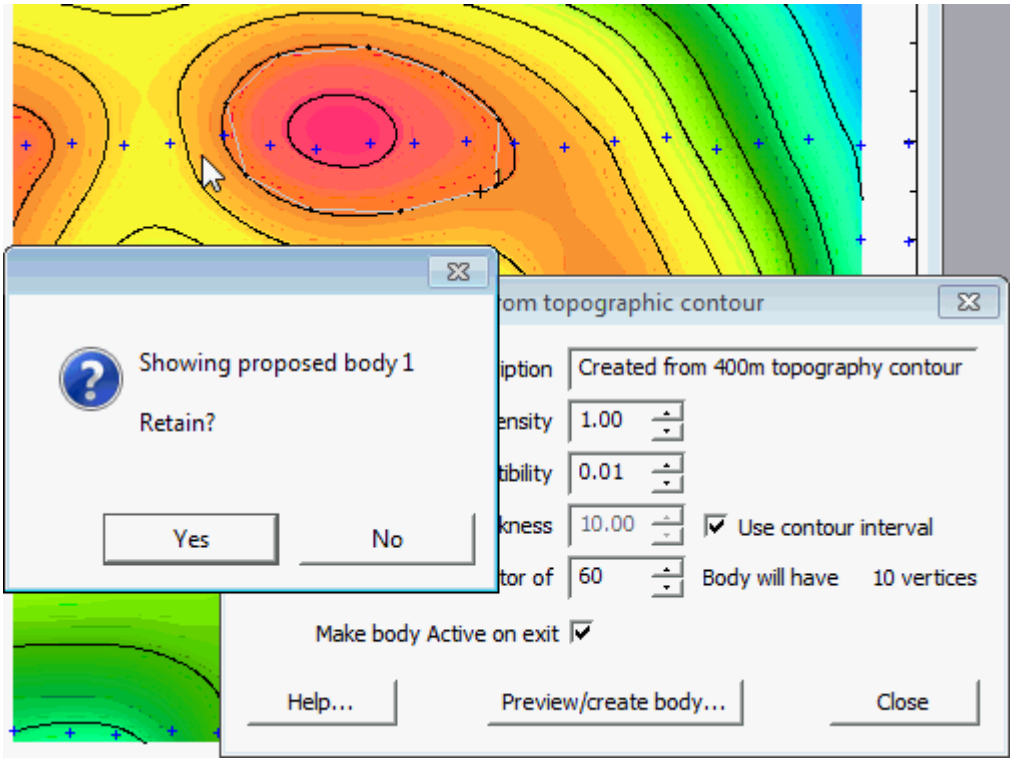
Susceptibility - Set the susceptibility of the body. The susceptibility is assumed to be isotropic.

Thickness - Set the length of the B axis of the body. If **Use contour interval** is selected, this value is ignored and the current contour interval is used.

Subsample vertices by a factor of - If this value is 1, then the polygon sides correspond to the line segments where contours cross the image grid cells. This is usually far too detailed, and would create a polygonal prism with many vertices, which would unnecessarily increase calculation time. The default value of 20 is usually a reasonable compromise, and causes a polygon side to extend over 20 grid cells. (When you click **Create body** you will have an opportunity to inspect the shape of the body before accepting it, as described below.)

Make body Active on exit - Set the body's Status to **Active**.

When the dialog box is set up as required, click **Preview/create body**. The proposed polygonal prism is drawn as a grey polygon:



Inspect the outline of the body. If it is satisfactory click **Yes** to retain it. If **Make body active on exit** is selected, the body's Status is automatically set to **Active**. If the fit to the contour is poor (as in the above example), click **No** to delete it and return to the dialog box.

--- Return to the Tools menu.

View menu

View | Toolbars
View | Status bar
View | IGRF on Status bar
View | Echo cursor
View | Annotations

View | Toolbars

These commands toggle the display of the various toolbars.

--- Return to the View menu.

View | Status bar

This command toggles display of the status bar (at the bottom of the application window).

--- Return to the View menu.

View | IGRF on Status bar

This command causes the current inducing field (the International Geomagnetic Reference Field) to be displayed at the right hand end of the status bar (at the bottom of the main window). The status bar pane displays the inclination, declination (each in degrees) and strength (in nanotesla) of the inducing field.

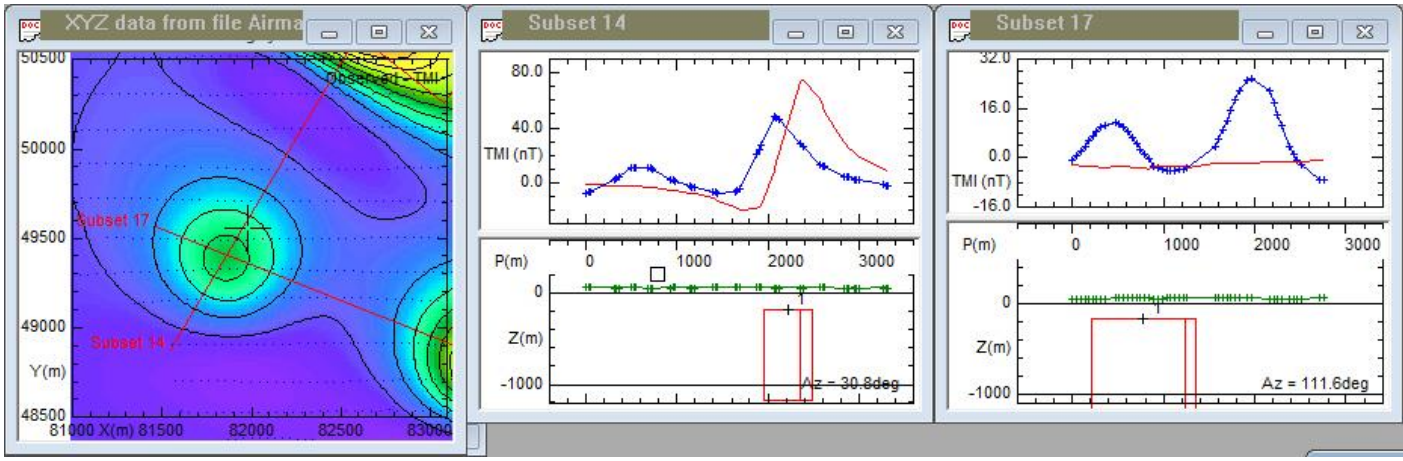
--- Return to the View menu.

View | Echo cursor

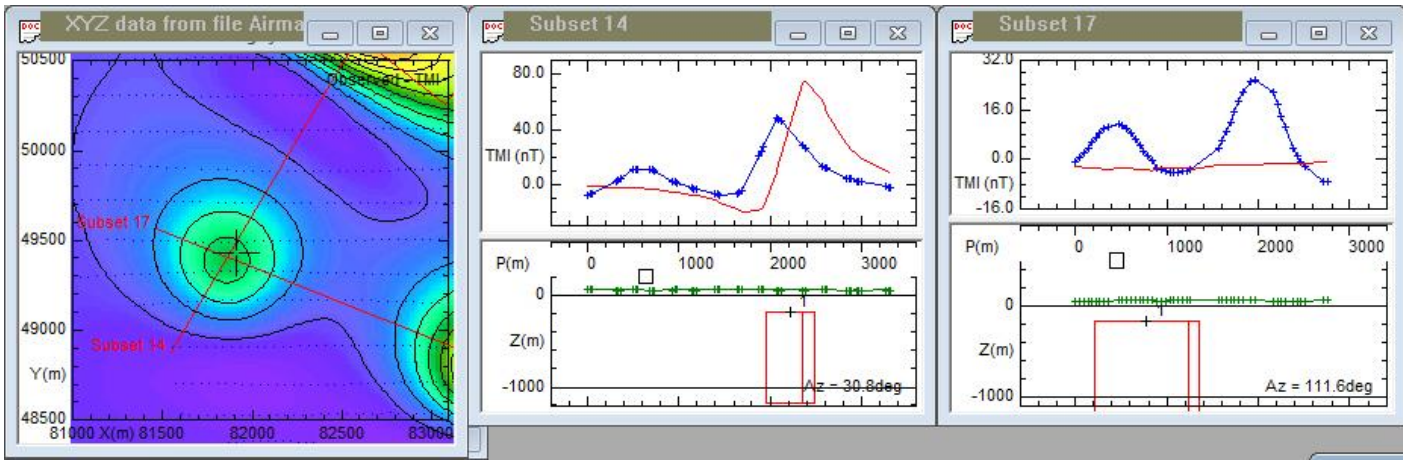
When this is selected, as the cursor is moved over a profile or plan window its position is shown as a small square on other windows that are associated with the dataset. (Note that the cursor is not echoed on down-hole windows.)

If the cursor is moving over a plan window its position is echoed on cross-section windows only if the plane of the section on the plan is close to the cursor. In this context “close” means within the length of one small X-axis division.

In this example the cursor, as seen on the plan window, is close to Subset 14 and it is echoed on the Subset 14 cross-section window (small square at around P = 800m):



If the cursor is moved slightly to the south it is also close to the section of Subset 17. The echo now appears on both cross-section windows:



--- Return to the View menu.

View | Annotations

This command toggles display of Annotation boxes.

See also – Annotating your output in the User's Guide.

--- Return to the View menu.

Window menu

Cascade

Tile

Arrange icons

Split

Redraw all

Scales

Auto scale

Options

Redraw

Annotations

Auto-fit profile panes

Window | Cascade

This command sizes and arranges all non-minimised windows within the application window so as to overlap and display the title bar of each window.

--- Return to the Window menu.

Window | Tile

This command sizes and arranges all non-minimised windows within the application window so as to take up all available space without overlapping.

--- Return to the Window menu.

Window | Arrange icons

This command arranges the icons representing any minimised windows into a neat row at the bottom of the application window.

--- Return to the Window menu.

Window | Split

This command is available only if the active window is a profile window. It enables you to resize the panes without using the mouse.

--- Return to the Window menu.

Window | Redraw all

Use this command to refresh the drawing of all displayed windows.

--- Return to the Window menu.

Window | Scales

The result of this command depends on the type of the active window.

Plan window scales

Profile window scales

--- Return to the Window menu.

Plan window scales

This dialog box has two tabs:

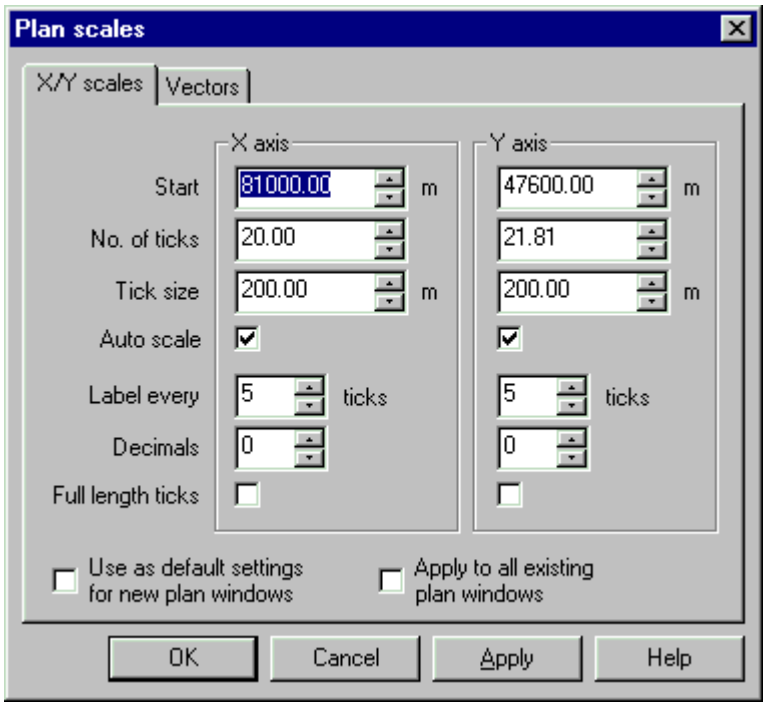
XY scales tab

Vector scales tab

--- Return to the Window menu.

XY scales

This dialog box allows you to set the (X,Y) scales of the active window, when this is a plan window. The groups of controls apply to the X-axis and Y-axis function identically.



Start – The coordinate of the left (X-axis) or bottom (Y-axis) end of the axis. (*Potent* ignores anything you enter here if **Auto scale** is selected.)

No. of ticks – The number of small axis divisions along the axis. (*Potent* ignores anything you enter here if the **Auto scale** is selected.) When auto-scaling it is normal for one of these values to be non-integer, as *Potent* adjusts the number according to the aspect ratio of the window.

Tick size – The distance in metres corresponding to a small axis division. (*Potent* ignores anything you enter here if **Auto scale** is selected.)

Auto scale - Clear this if you want *Potent* to use values you have specified in the previous edit boxes. Select this if you want *Potent* to calculate suitable values automatically.

Label every - Set this to 5, for example, if you want every 5th tick to be labelled with the axis value. (I.e. one major axis division would be equivalent to 5 small axis divisions.)

Decimals - The number of decimal places used for the labelled axis values.

Full length ticks - Select this if you want major ticks (those with labels) to extend across to the opposite axis.

Use as default settings for new plan windows – When this is selected the settings in the dialog box will be applied to any further plan windows that might be created for the current dataset.

Apply to all existing plan windows – When this is selected the settings in the dialog box will be applied to all existing plan windows for the current dataset.

Apply – Click this button to view the effect of your changes to this page (and to any other tabbed pages in this dialog box) without closing the dialog box.

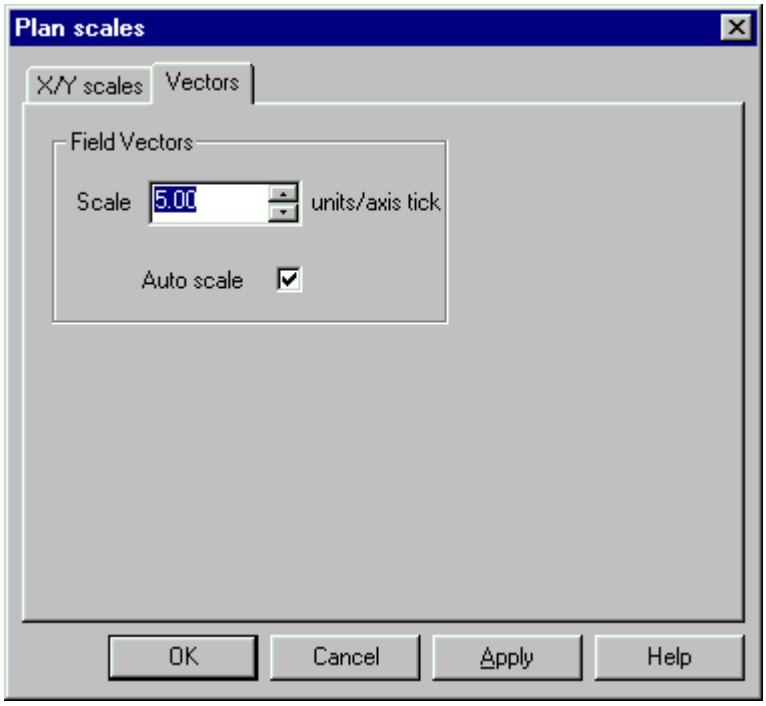
OK – Click this button to the close dialog box and apply the effect of your changes (to this page and to any other tabbed pages).

See also – Features of Potent Dialog Boxes in the User's Guide.

--- Return to the Window menu.

Vector scales

This dialog box allows you to set the scale used to draw vectors in the active window, when this is a plan window. The vectors are drawn radiating from the observation points. Refer to the Plan Window settings (Observations tab) dialog for details of adding vectors to a plan window.



Scale - Set the scale of vectors in terms of the length of a small axis division. For example, if a vector were 20nT long in the (X,Y) plane and the value in the **Field vectors** edit control was 5, then the vector would be drawn with a length of 4 minor axis division.

Auto scale - Select this if you want *Potent* to calculate a suitable scale automatically.

--- Return to the Window menu.

Profile window scales

This dialog box has four or more tabs, depending on the number of field components:

P/Z scales tab

Field scales tabs (one for each field component)

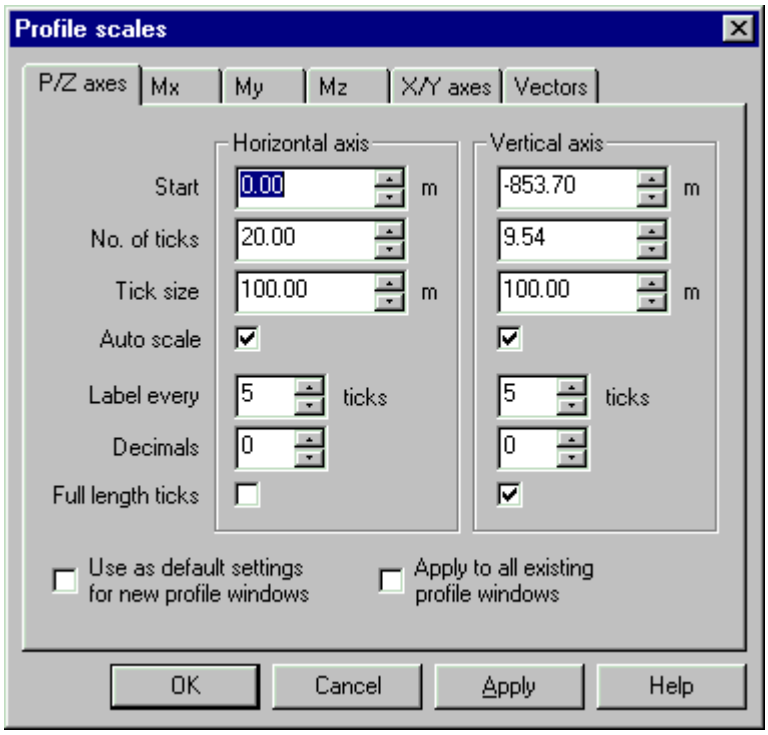
XY scales tab

Vector scales tab

--- Return to the Window menu.

P/Z scales

This dialog box allows you to set the (P,Z) scales of the cross-section pane of the active window, when this is a profile window. The groups of controls that apply to the horizontal and vertical axes function identically.



Start – The coordinate of the left (P-axis) or top (vertical axis) end of the axis. (*Potent* ignores anything you enter here if **Auto scale** is selected.)

No. of ticks – The number of small axis divisions along the axis. (*Potent* ignores anything you enter here if **Auto scale** is selected.) When auto-scaling it is normal for one of these values to be non-integer, as *Potent* adjusts the number to according to the aspect ratio of the window.

Tick size – The distance in metres corresponding to a small axis division. (*Potent* ignores anything you enter here if **Auto scale** is selected.)

Auto scale - Clear this box if you want *Potent* to use values you have specified in the previous edit boxes. Select this if you want *Potent* to calculate suitable values automatically.

Label every - Set this to 5, for example, if you want every 5th tick to be labelled with the axis value. (I.e. one major axis division would be equivalent to 5 small axis divisions.)

Decimals - The number of decimal places used for the labelled axis values.

Full length ticks - Select this if you want major ticks (those with labels) to extend across to the opposite axis.

Use as default settings for new profile windows – When this is selected the settings in the dialog box will be applied to any further profile windows that might be created for the current dataset.

Apply to all existing profile windows – When this is selected the settings in the dialog box will be applied to all existing profile windows for the current dataset.

Apply – Click this button to view the effect of your changes to this page (and to any other tabbed pages in this dialog box) without closing the dialog box.

OK – Click this button to the close dialog box and apply the effect of your changes (to this page and to any other tabbed pages).

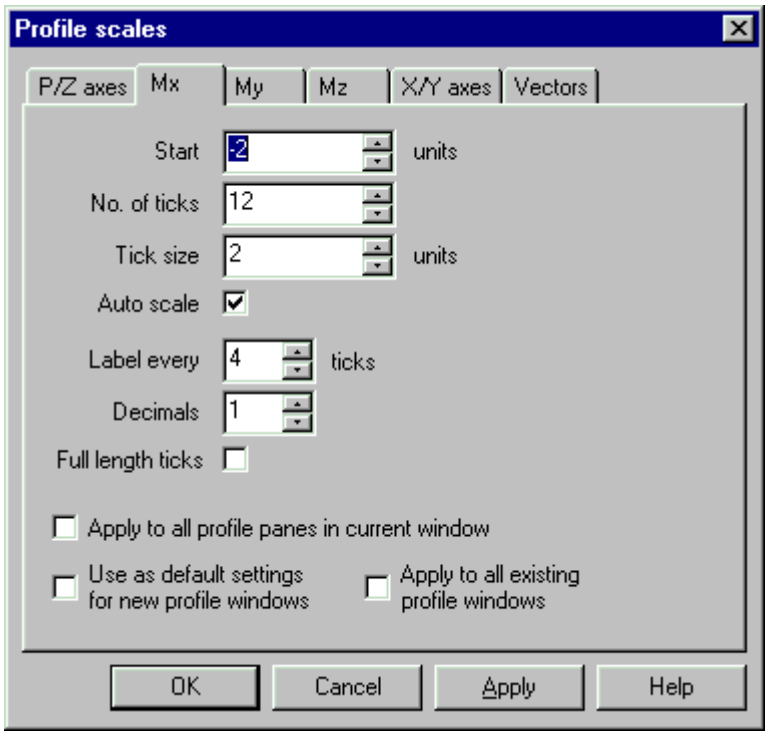
See also – Coordinate Axes and Features of Potent Dialog Boxes in the User's Guide.

--- Return to the Window menu.

Field scales

This dialog box allows you to set the scale of one field component of the active window, when this is a profile window. There is a similar dialog box for each field component.

Note that the profile panes share the horizontal (P) axis of the cross-section pane. Therefore to adjust the horizontal axis of the profiles, use the P/Z scales tab.



Start – The coordinate of the left end of the axis. (*Potent* ignores anything you enter here if **Auto scale** is selected.)

No. of ticks – The number of small axis divisions along the axis. (*Potent* ignores anything you enter here if **Auto scale** is selected.) When auto-scaling it is normal for one of these values to be non-integer, as *Potent* adjusts the number to according to the aspect ratio of the window.

Tick size – The distance in metres corresponding to a small axis division. (*Potent* ignores anything you enter here if **Auto scale** is selected.)

Auto scale - Clear this box if you want *Potent* to use values you have specified in the previous edit boxes. Select this if you want *Potent* to calculate suitable values automatically.

Label every - Set this to 5, for example, if you want every 5th tick to be labelled with the axis value. (I.e. one major axis division would be equivalent to 5 small axis divisions.)

Decimals - The number of decimal places used for the labelled axis values.

Full length ticks - Select this if you want major ticks (those with labels) to extend across to the opposite axis.

Use as default settings for new profile windows – When this is selected the settings in the dialog box will be applied to any further profile windows that might be created for the current dataset.

Apply to all existing profile windows – When this is selected the settings in the dialog box will be applied to all existing profile windows for the current dataset.

Apply – Click this button to view the effect of your changes to this page (and to any other tabbed pages in this dialog box) without closing the dialog box.

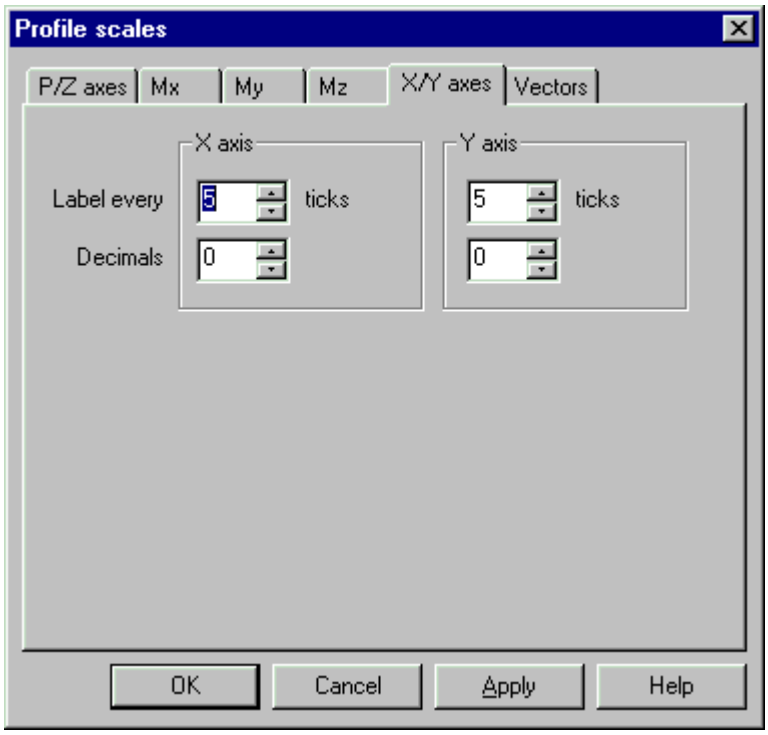
OK – Click this button to the close dialog box and apply the effect of your changes (to this page and to any other tabbed pages).

See also – Coordinate Axes, Features of Potent Dialog Boxes, and Profile windows, in the User's Guide.

--- Return to the Window menu.

XY scales

In a profile window the "natural" horizontal axis is the P-axis. However, it is possible to draw the X and Y-axes on the cross-section pane, as described in Profile window settings (Cross-section tab). These axes are essentially just a re-labelling of the P-axis, and so this dialog box does not allow you to set the actual scale; this should be done using the P/Z scales tab.



Label every - Set this to 5, for example, if you want every 5th tick to be labelled with the axis value. (I.e. one major axis division would be equivalent to 5 small axis divisions.)

Decimals - The number of decimal places used for the labelled axis values.

Apply – Click this button to view the effect of your changes to this page (and to any other tabbed pages in this dialog box) without closing the dialog box.

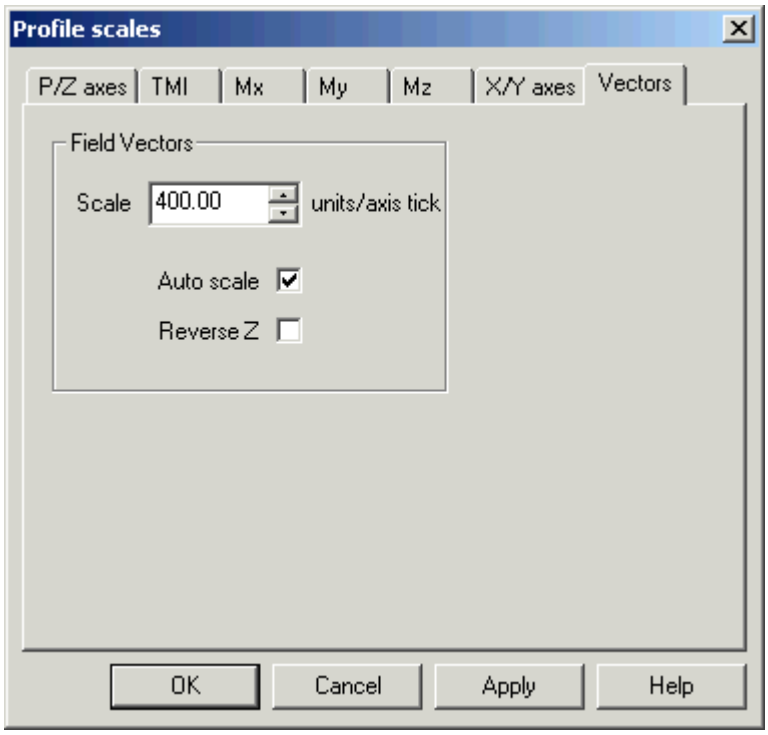
OK – Click this button to the close dialog box and apply the effect of your changes (to this page and to any other tabbed pages).

See also – Coordinate Axes, Features of Potent Dialog Boxes, and Profile windows, in the User's Guide.

--- Return to the Window menu.

Vector scales

This dialog box allows you to set the scale used to draw vectors in the active window, when this is a profile window. The vectors are drawn in the cross-section pane, radiating from the observation points. Refer to the Profile Window settings (Cross-section tab) dialog for details of adding vectors to a profile window.



Scale - Set the scale of vectors in terms of the length of a small axis division. For example, if a vector were 20nT long in the (P,Z) plane and the value in the **Field vectors** edit control was 5, then the vector would be drawn with a length of 4 minor axis division.

Auto scale - Select this if you want *Potent* to calculate a suitable scale automatically.

Reverse Z - The direction of Z is a matter of convention, and sometimes vectors drawn using one direction are more indicative of the location of the source body than those drawn using the other. Select this to reverse the value of the Z field component used when drawing vectors.

See also – Features of Potent Dialog Boxes, and Profile windows, in the User's Guide.

--- Return to the Window menu.

Down-hole window scales
Down-hole windows are controlled in the same way as profile windows, making allowance for the fact that the profile panes are arranged to the left of the cross-section pane. Refer to the Profile window settings and Profile window scales topics for details.

See also – Working with down-hole data in the User Guide.

--- Return to the Window menu.

Window | Auto scale

This command is a shortcut method of checking the **Auto scale** options in the various Plan window scales and Profile window scales dialog boxes, depending on the type of the active window.

The command also is available from the popup menu for plan and profile windows.

--- Return to the Window menu.

Window | Options

The result of this command depends on the type of the active window.

Plan window settings

Profile window settings

Down-hole window settings

This command is available from the popup menu.

--- Return to the Window menu.

Plan window settings

This dialog box is displayed when you choose the **Window | Options** command when the active window is a plan window. The dialog box has four tabs:

Plan Window settings - Bodies

Plan Window settings – Observations

Plan Window settings - General

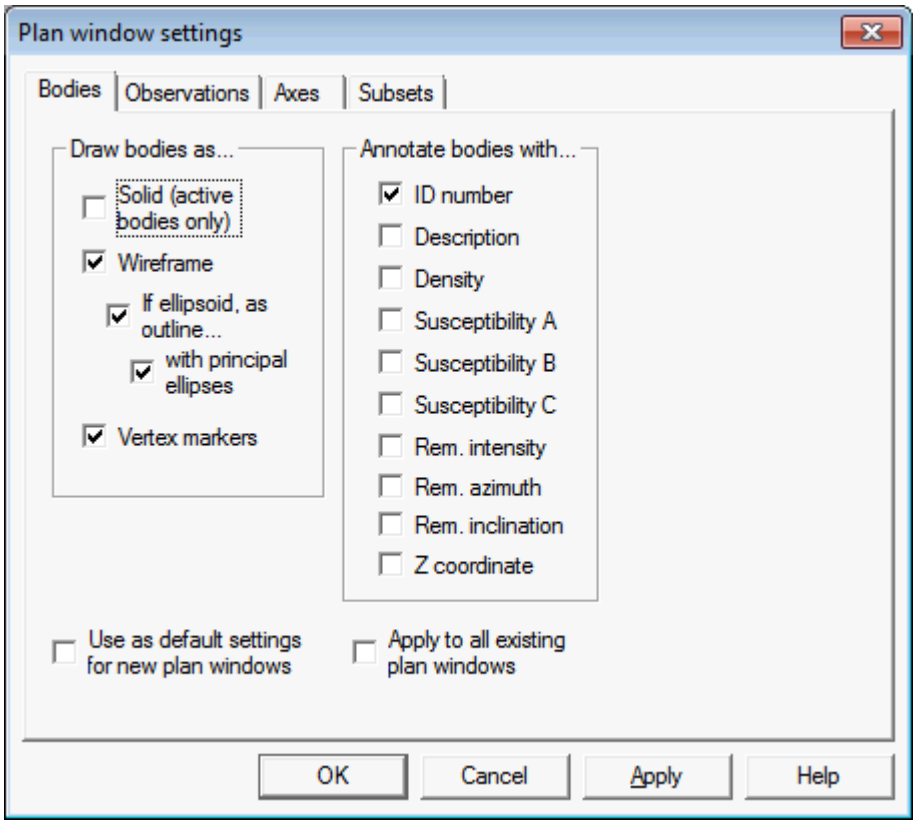
Plan Window settings - Subsets

This command is available from the popup menu.

--- Return to the Window menu.

Plan Window settings - Bodies

This dialog box is a page of the **Plan window settings** dialog box. Use it to specify how bodies are drawn in plan windows.



Draw bodies as group -

Select **Solid** to cause active bodies to be drawn as solid objects, using OpenGL.

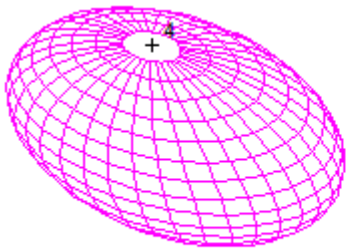
Select **Wireframe** to cause bodies to be drawn using a 3D wireframe representation. You can change the mesh size, as described in **Ellipsoid drawing** under File | Preferences.

Selecting both **Solid** and **Wireframe** has the effect of highlighting the edges of bodies. Clearing both options causes a body to be represented on plots only by its annotation (see below).

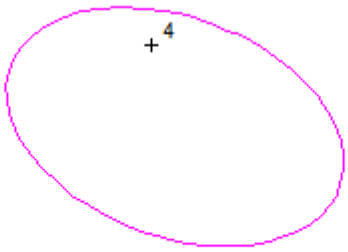
For ellipsoid or sphere body types, selecting **If ellipsoid, as outline…** causes only the outer edge of each body to be drawn on the section. For an ellipsoid, this is the ellipse that is the projection of the body onto the plane of the section. If you have many spheres or ellipsoids in your model then this option can create a clearer representation than is generated using the unqualified **Wireframe** option.

If the **outline** option is selected then you may also select the **with principal axes** option. For an ellipsoid (not a sphere), as well as drawing the outline ellipse it also draws the visible sections of the three principal ellipses (those in the ellipsoid's A, B and C planes). This adds a visual cue as to shape while, for a multi-body model, still producing a clearer result than might be obtained using the **Wireframe** option alone.

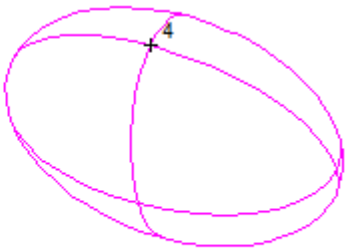
The following image shows the ways in which the three wireframe options represent an ellipsoid.



Wireframe



Outline



Principal ellipses

Select the **Vertex markers** option to cause the vertices of polygonal bodies to be marked by small crosses. This can make them easier to pick up using the **Body shape** tool.

See also **Ellipsoid drawing** under File | Preferences.

Annotate bodies with group – By default a body's position is marked by a cross, drawn at the reference point, and annotated with the body number.

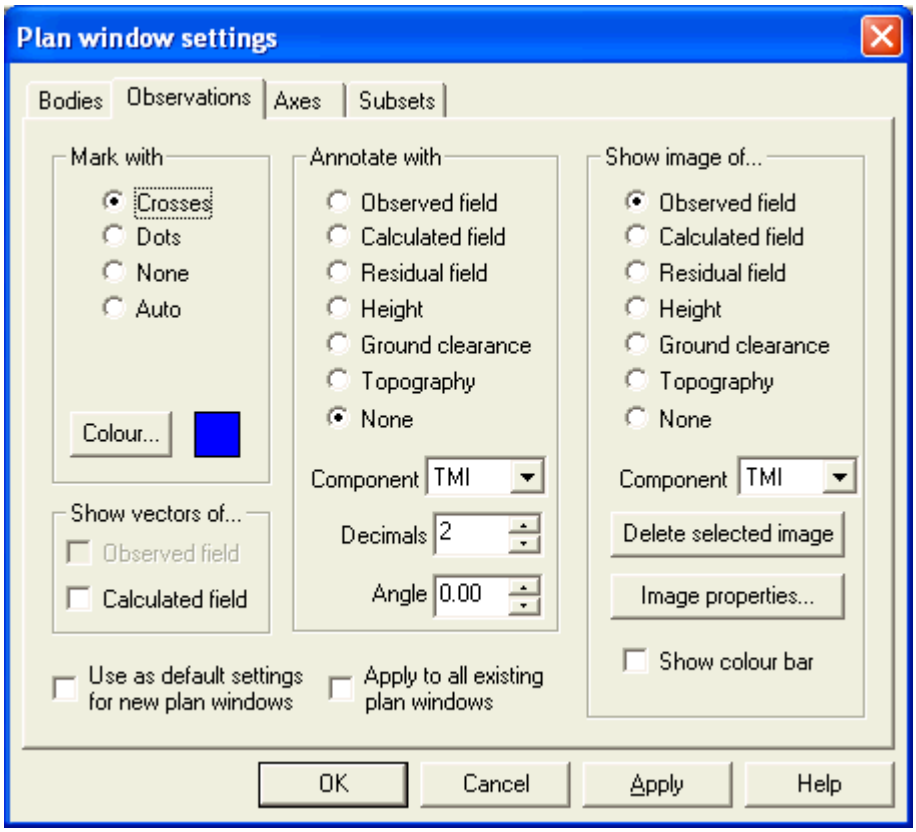
You can add further annotations by selecting other options in this group.
If you clear **all** annotation options, *Potent* also omits the cross.

See also – Features of Potent Dialog Boxes, and Plan windows, in the User's Guide.
- **Ellipsoid drawing** under File | Preferences.

--- Return to the Window menu.

Plan Window settings - Observations

This dialog box is a page of the Plan window settings dialog box. Use it to specify how observations are drawn in plan windows.



Mark with group

Crosses – Mark the position of observations by small crosses. This option is effective if the number of observations in the window is small.

Dots - Mark the position of observations by dots. A "dot" is a single pixel on the screen, but is drawn as a very small cross on hardcopy (where a single dot is usually almost invisible). Use this option if there is a large number of observations in the window.

None – Don't mark the observation points.

Auto – If the number of observations is greater than 500, use dots, otherwise use crosses.

Colour –Click this button to set the colour with which the observation point markers (dots or crosses) are drawn.

Annotate with group

Radio buttons – Choose the item the value of which is to be drawn next to each observation point. Leave the selections as **None** if there are many observation points in the window (such as when you have loaded a grid). The window can end up completely black due to the overlapping of densely packed text. Note that topography is the difference between height and ground clearance, so the **Topography** option is useful only if the observations have at least one of these defined.

Component – If you have selected one of the "field" options and your dataset has more than one component, then you can specify which component to use for the annotations.

Decimals – Specify the number of decimal places to use for the annotated values.

Show image of group

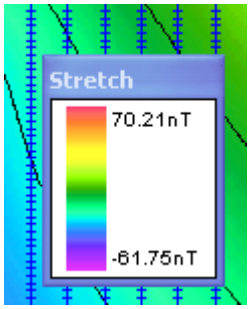
Radio buttons – Choose the item the value of which is to be used to create an image that represents an interpolation grid of the observations. Note that topography is the difference between height and ground clearance, so the **Topography** option is useful only if the observations have at least one of these defined. The image is intended to assist the modelling process and is created using a simple but fast curvature minimisation algorithm; *Potent* is not an image-processing program.

Component – If you have selected one of the "field" options and your dataset has more than one component, then you can specify which component to use for the image.

Delete selected image – If you have (say) an **Observed field** image displayed, and then change the setting to **None**, the observed field image continues to exist in memory. If you subsequently re-select the **Observed field** option then *Potent* re-displays the existing image. If you want to force *Potent* to re-grid the data, or if you want to release the memory occupied by the image, then you can use the **Delete selected image** button to delete the image from memory.

Image properties – Click this button to display the Image properties dialog box.

Show colour bar – Select this to add a colour bar window to the plan. This shows the colour stretch used for the displayed image.



You can resize and reposition the window by dragging its corners and title bar.

When printed (File | Print command) the client area (i.e. the part of the window within the frame) will be drawn in the same position on the Plan window. However, due to differences in rendering fonts on the screen and on paper, it is useful to use the File | Print preview command to help set the colour bar to the best size for printing.

The annotations on the colour bar show the field values corresponding to the top and bottom of the colour range. If you want to see neatly rounded numbers (e.g. 70.00 and -60.00 in the above example) then adjust the colour stretch from the Image properties dialog box.

Show vectors of group

Observed field – This option is available only if the observations include the three magnetic or gravity components. Select this to cause *Potent* to draw vectors of the observed field, radiating from each observation point.

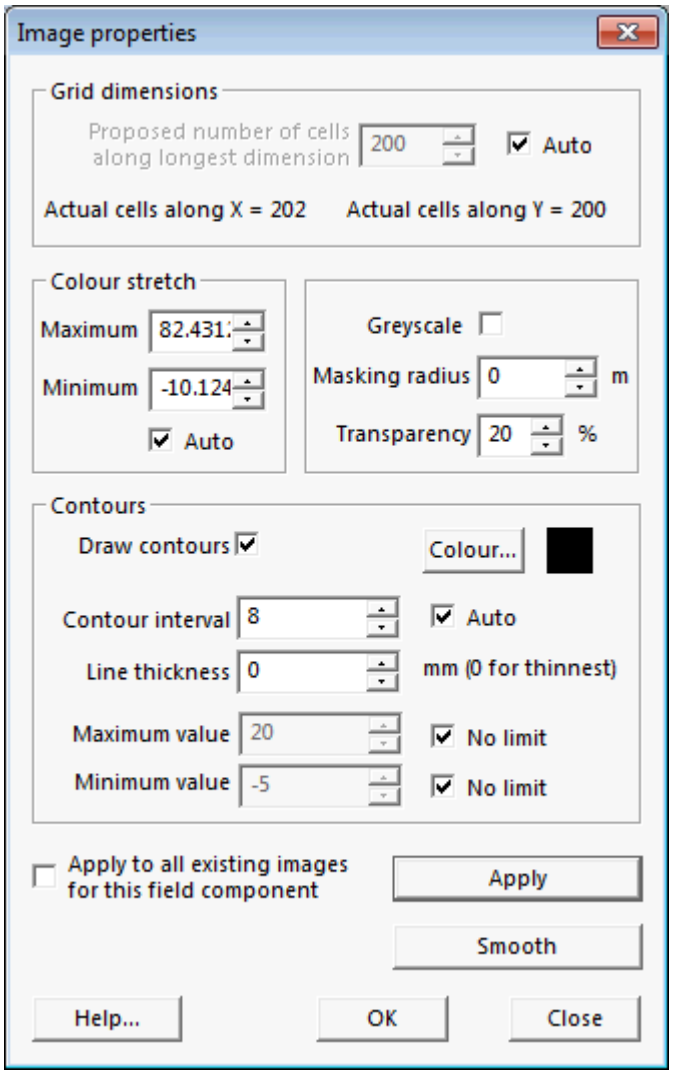
Calculated field – Select this to cause *Potent* to draw vectors of the calculated field. The vectors radiate from the observation points. If the observation components are all magnetic, then magnetic field vectors are drawn. Similarly, gravity vectors are drawn if the field components are all gravitational. If the components are a mixture of magnetic and gravitational, then magnetic vectors are drawn.

See also – Features of Potent Dialog Boxes, and Plan windows, in the User's Guide.

--- Return to the Window menu.

Plan window settings - Image properties

This dialog box is displayed from the Plan Window settings – Observations dialog box. It allows you to control the properties of the image that is currently displayed in the active plan window.



Grid dimensions – When Potent creates a grid the number of cells along the longest dimension is the value specified in the **Image grid resolution** group of the Preferences dialog box. You can over-ride this value by specifying the desired number of cells in the **Proposed number…** box. Alternatively, select **Auto** to use the number specified in Preferences.

The actual number of cells might be changed slightly from the proposed figure by the gridding algorithm. The actual numbers used along the X and Y dimensions are displayed after the grid is regenerated.

Colour stretch – *Potent* colours the image using a colour table that extends from mauve (low), through blue, green, yellow and orange, to red (high). Cells with data values equal to or greater than the maximum colour stretch value will be coloured red. Cells with data values equal to or less than the minimum colour

stretch value will be coloured mauve. Cells with intermediate values will be assigned colours that are linearly stretched over the colour range.

Auto – Select this to have *Potent* assign the two colour stretch limits to the extremes of the data values in the grid.

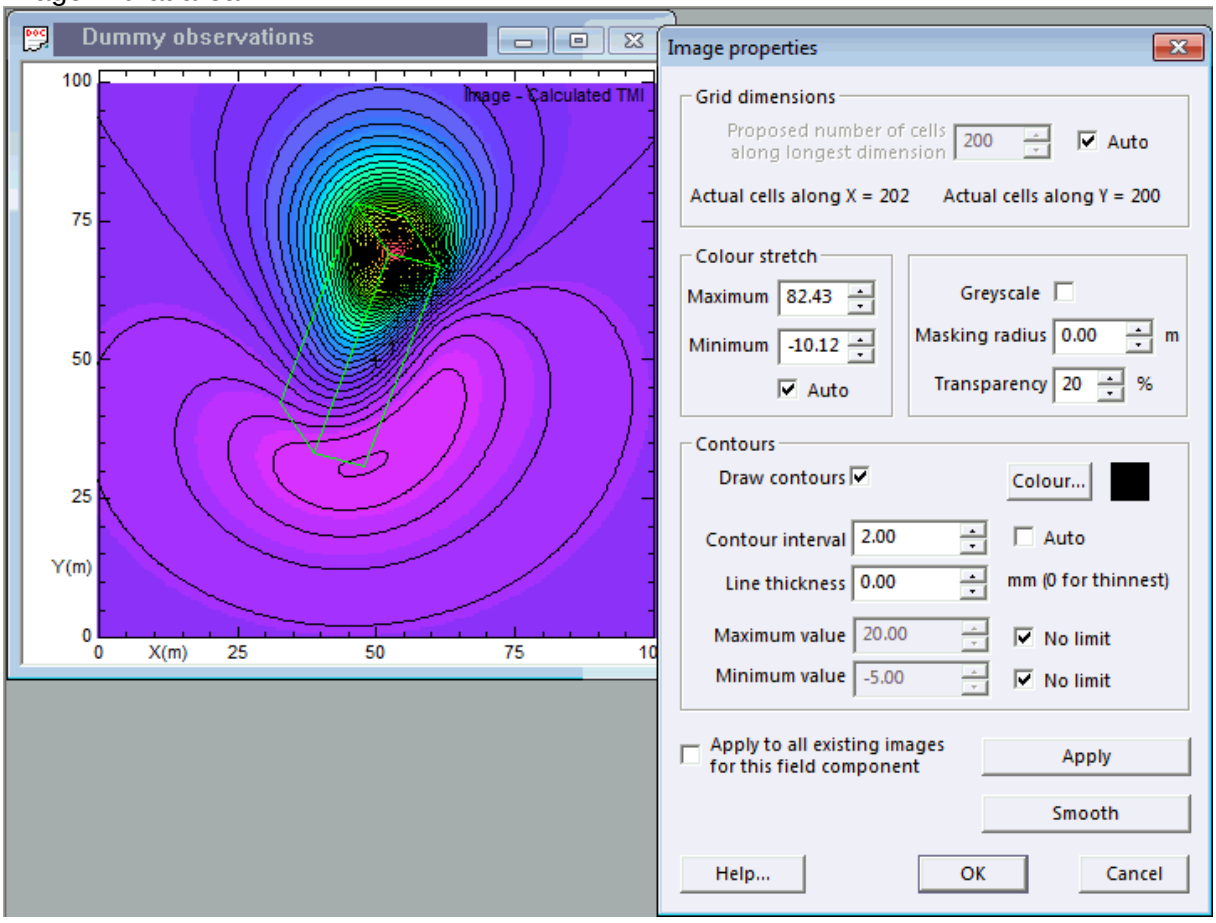
Greyscale – Select this to map the data values to levels of grey with black as low and white as high. When printing an image on a monochrome printer (File | Print command) this generally produces a better effect than using the colour mapping provided by your printer driver.

Masking radius - An image cell will be drawn with the background colour if it does not have any observations closer than the masking radius. Note that "zero" implies "infinity", and no masking is done.

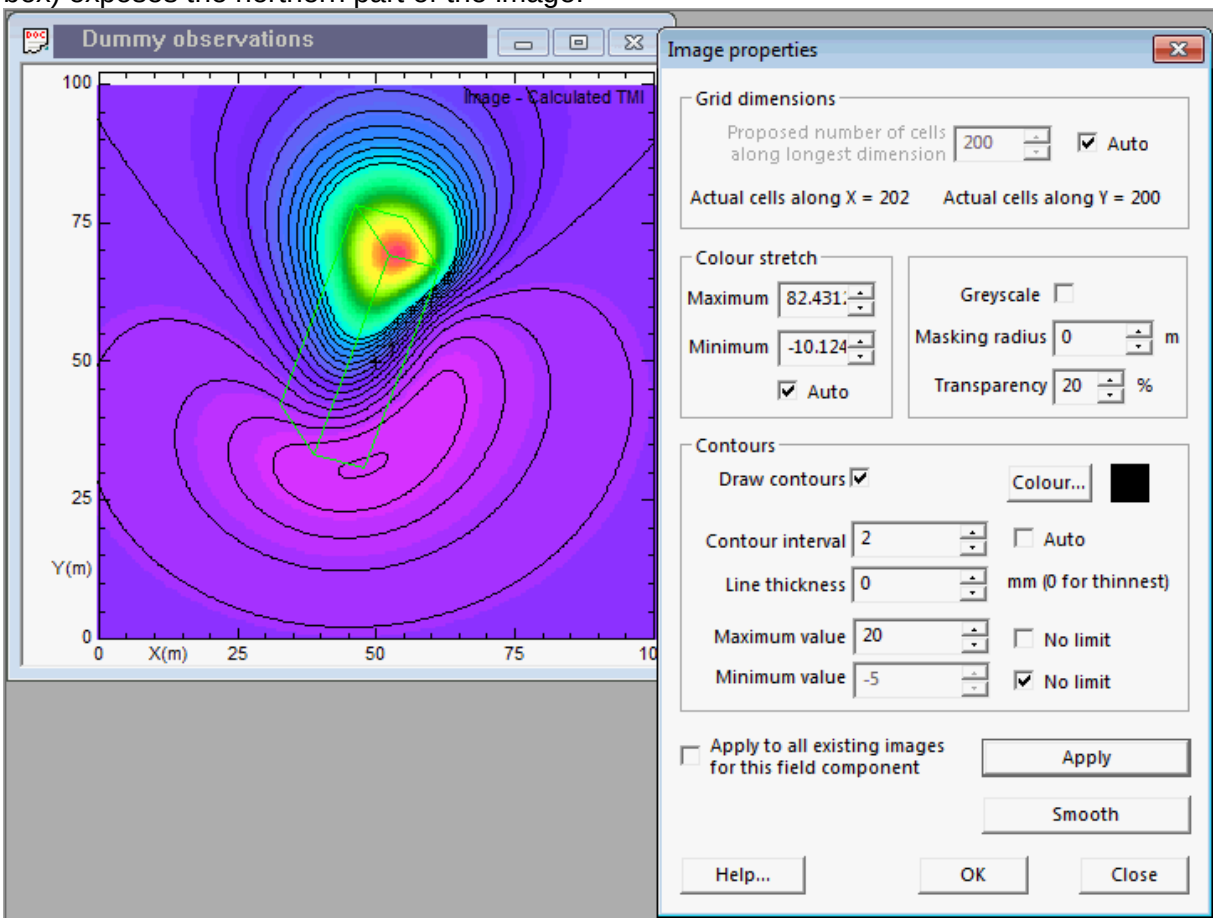
Transparency – Increase this number to lighten the image. This improves the visibility of the underlying model.

Contours – Use these settings to overlay contours on the image. Contours are not labelled – they are intended simply as a guide for the eye. If you wish to set the **Contour interval** manually you must also clear **Auto**, otherwise *Potent* will over-ride your setting.

The **Maximum value** and **Minimum value** settings allow you to specify limits outside which contours will not be drawn. In the following example the user wants to specify a contour interval of 2 in order to emphasise detail in the southern half of the image. This causes the overly dense contours in the north to obscure the image in that area.



Specifying a **Maximum value** of 20 (after unchecking the associated **No limit** box) exposes the northern part of the image.



Apply to all… – When this is selected the settings will be applied to all plan windows that display images of the calculated and/or residual fields for this dataset and field component.

Apply – Click this button to apply the settings without exiting the dialog box.

Smooth – This applies a simple 9 point moving average operator to the grid, which smoothes out any irregularities that are occasionally left by the gridding algorithm. Use with care, as repeated use will degrade the grid. (This feature was introduced for use when creating a body from a contour, for which smooth contours are essential.)

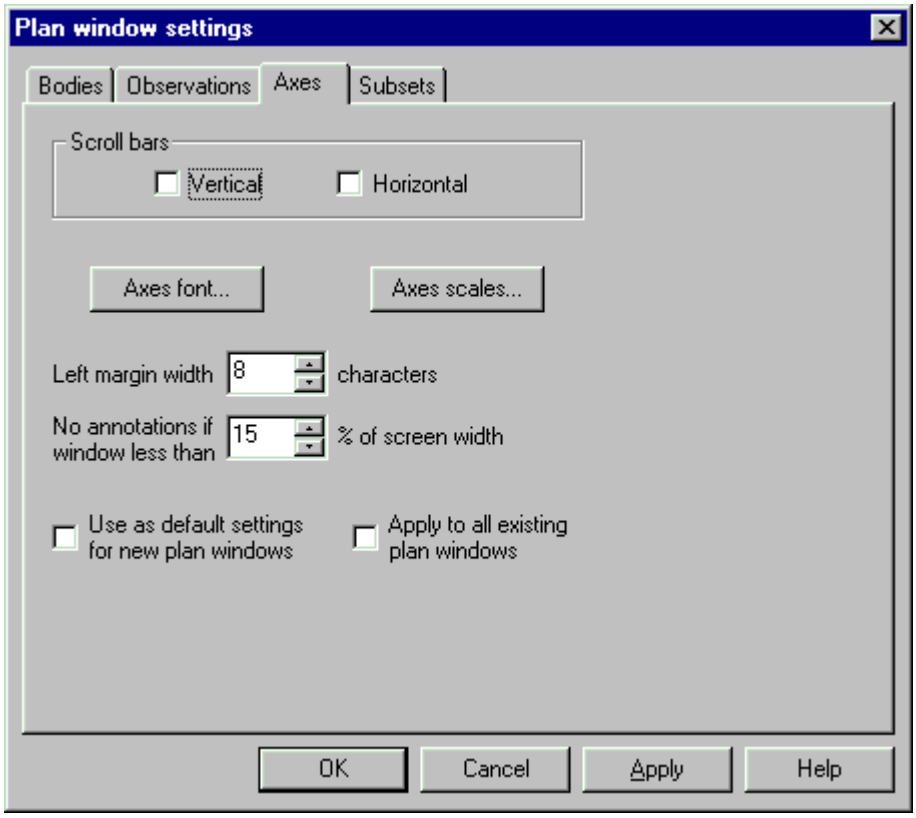
OK – Accept the settings and return to the Plan Window settings – Observations dialog box. Note that the settings will not take effect until you click **OK** or **Apply** in the parent dialog box.

See also – Features of Potent Dialog Boxes, and Plan windows, in the User's Guide.

--- Return to the Window menu.

Plan Window settings - Axes

This dialog box is a page of the **Plan window settings** dialog box. Use it to specify how axes are drawn in plan windows.



Scroll bars – Select a box to add a scroll bar to that axis.

Axes font – Click this button to set the font for both axes.

Axes scales – Click this button to invoke the Plan window scales dialog box.

Left margin width – Set the width of the area (to the left of the plotting area of the plan) that holds the labels for the Y-axis. The width is set in units of the average character width.

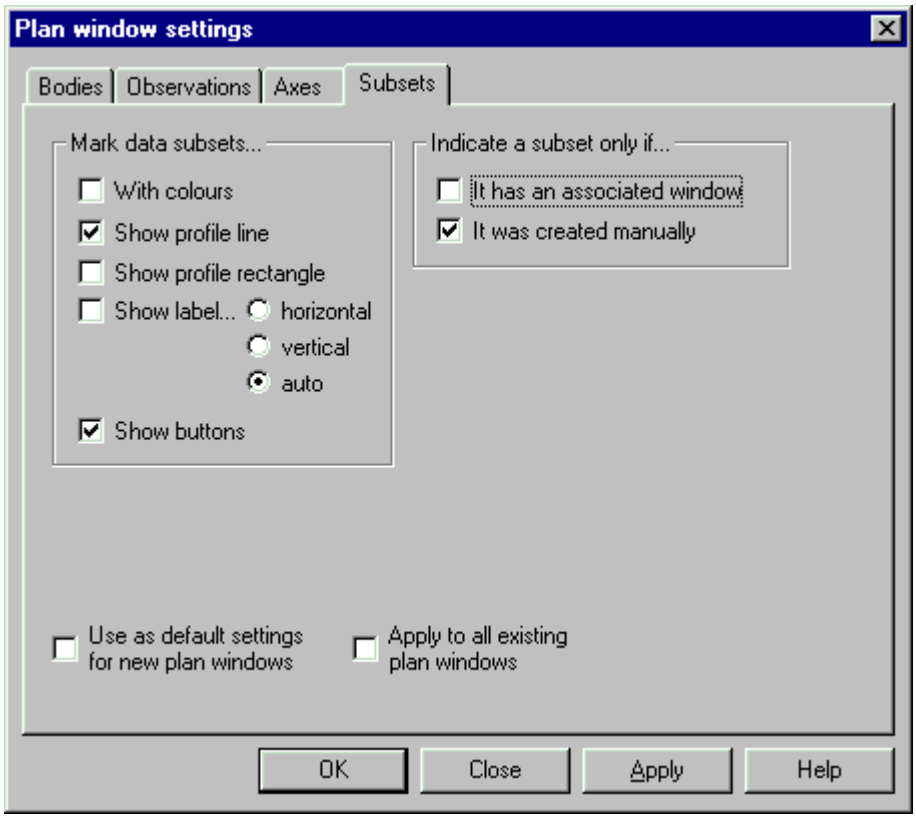
No annotations if… - When a window is small, the annotations take up a disproportional amount of the available plotting area, leaving little space for the actual data. When you set a non-zero value in this box, *Potent* will not draw annotations on the window if its width is less than that percentage of the physical screen width. Set the value to zero if you want *Potent* to always display annotations.

See also – Features of Potent Dialog Boxes, and Plan windows, in the User's Guide.

--- Return to the Window menu.

Plan Window settings - Subsets

This dialog box is a page of the Plan window settings dialog box. Use it to specify how subsets are drawn in plan windows.



Mark data subsets group

With colours – Draw the observations belonging to subsets, using their associated colours, subject to the "**Indicate a subset only if**" condition. You can change the colour by using the Observations | Subset management command.

Show profile line – Draw the centre-line of the subset, subject to the "**Indicate a subset only if**" condition. This is the P-axis that forms the horizontal axis in profile windows.

Show profile rectangle – Draw the rectangle (if any) that was used to create the subset with the subset tool, subject to the "**Indicate a subset only if**" condition.

Show label – Annotate the ends of profile lines with the labels belonging to subsets, subject to the "**Indicate a subset only if**" condition. (You can change the label by using the Observations | Subset management command.) Use the radio buttons to determine the direction in which the label text will be oriented. Select **Auto** to have *Potent* choose a direction based on the bearing of the centre-line of the subset.

Show buttons – Select this box to display the buttons that mark the ends of subsets.

Indicate a subset only if group

It has an associated window – Select this box to suppress the representation of subsets that do not have an associated window. A subset will be represented even if its window is minimised.

It was created manually – Select this box to suppress the representation of predefined subsets.

See also – Features of Potent Dialog Boxes, and Plan windows, in the User's Guide.

--- Return to the Window menu.

Profile window settings
This dialog box is displayed when you choose the Window | Options command when the active window is a profile window. The dialog box has three tabs:

Profile window settings - Cross-section

Profile window settings – Profile panes

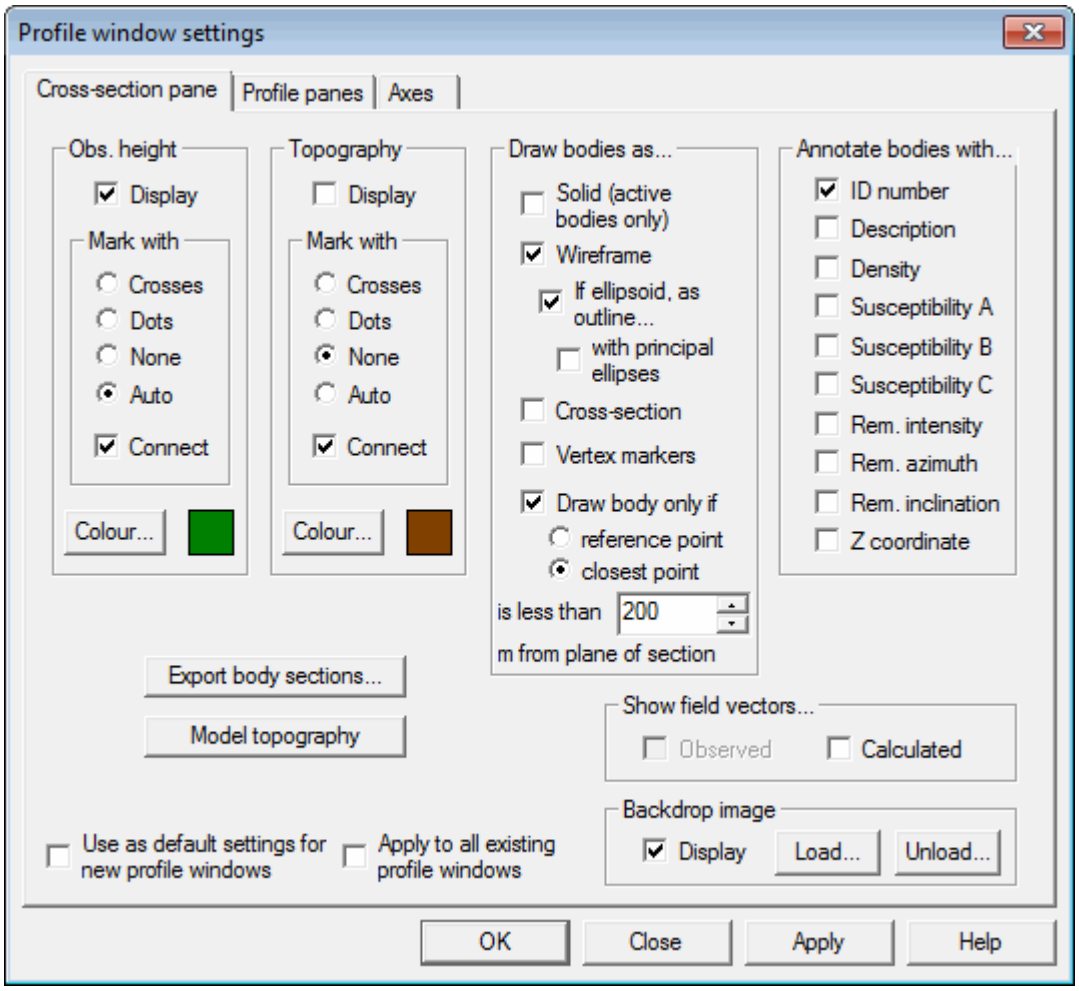
Profile window settings - Axes

This command is available from the popup menu.

--- Return to the Window menu.

Profile window settings - Cross-section

This dialog box is a page of the Profile window settings dialog box. Use it to specify how the cross-section pane is drawn in profile windows.



Observation height group

Display – Clear this box to suppress drawing of observation heights, while preserving the other options in the group.

Crosses – Mark the position of observations by small crosses.

Dots - Mark the position of observations by dots. A “dot” is a single pixel on the screen, but is drawn as a very small cross on hardcopy (where a single dot is usually almost invisible).

None – Don’t mark the observation points.

Auto – If the number of observations is greater than 500, use dots, otherwise use crosses.

Connect – Select this to draw lines between adjacent values.

Colour – Click this button to set the colour with which the observation point markers (dots or crosses) are drawn.

Topography group

This group controls drawing of a topographic profile, which is derived from the height and ground clearance fields that can be associated with an observation. Options are identical to those for the Observation height.

Draw bodies as group

Select **Solid** to cause active bodies to be drawn as solid objects, using OpenGL.

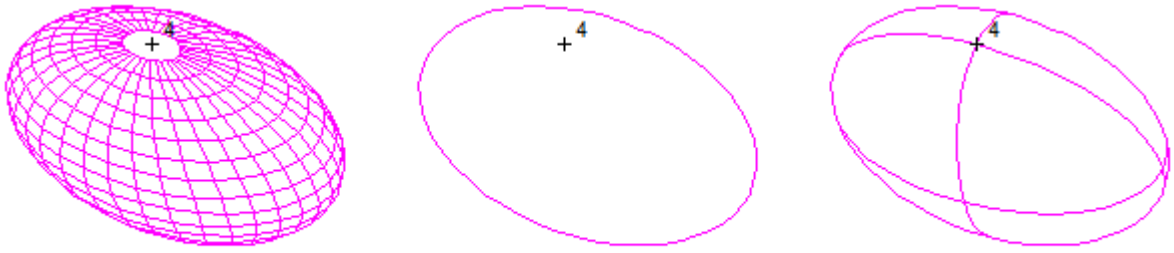
Select **Wireframe** to cause bodies to be drawn using a 3D wireframe representation. You can change the mesh size, as described in **Ellipsoid drawing** under File | Preferences.

Selecting both **Solid** and **Wireframe** has the effect of highlighting the edges of bodies. Clearing both options causes a body to be represented on plots only by its annotation (see below).

For ellipsoid or sphere body types, selecting **If ellipsoid, as outline…** causes only the outer edge of each body to be drawn on the section. For an ellipsoid, this is the ellipse that is the projection of the body onto the plane of the section. If you have many spheres or ellipsoids in your model then this option can create a clearer representation than is generated using the unqualified **Wireframe** option.

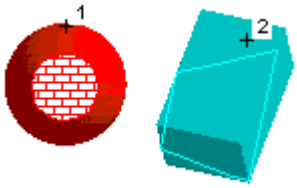
If the **outline** option is chosen then you may also select the **with principal axes** option. For an ellipsoid (not a sphere), as well as drawing the outline ellipse it also draws the visible sections of the three principal ellipses (those in the ellipsoid’s A, B and C planes). This adds a visual cue as to shape while, for a multi-body model, still producing a clearer result than might be obtained using the **Wireframe** option alone.

The following image shows the ways in which the three wireframe options represent an ellipsoid.



See also **Ellipsoid drawing** under File | Preferences.

Cross-section – If this is selected, *Potent* will draw the cross-sections of bodies in the plane of the profile.

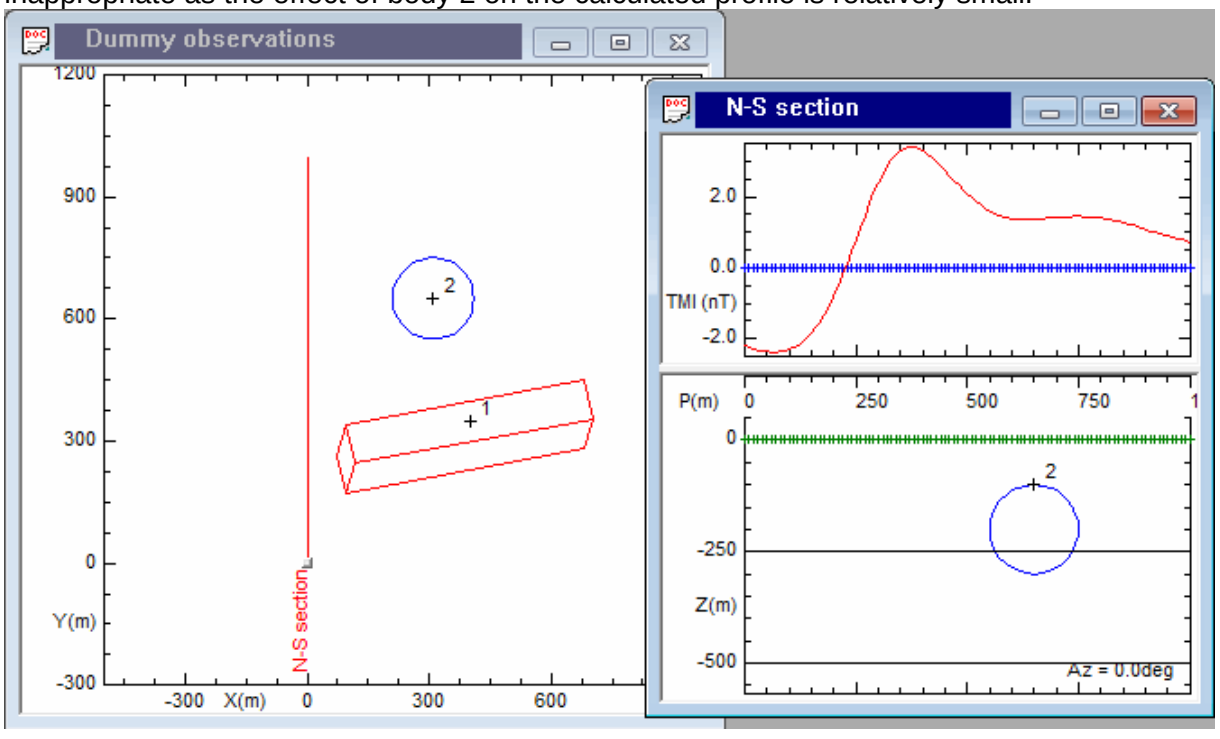


Use the **Cross-section pattern** button in the edit body dialog box (General tab) to specify how the cross-section is represented for particular bodies. The default is to draw the outline, without any fill, as in the case of the rectangular prism in the above example.

Vertex markers – Select this option to cause the vertices of polygonal bodies to be marked by small crosses. This can make them easier to pick up using the Body shape tool.

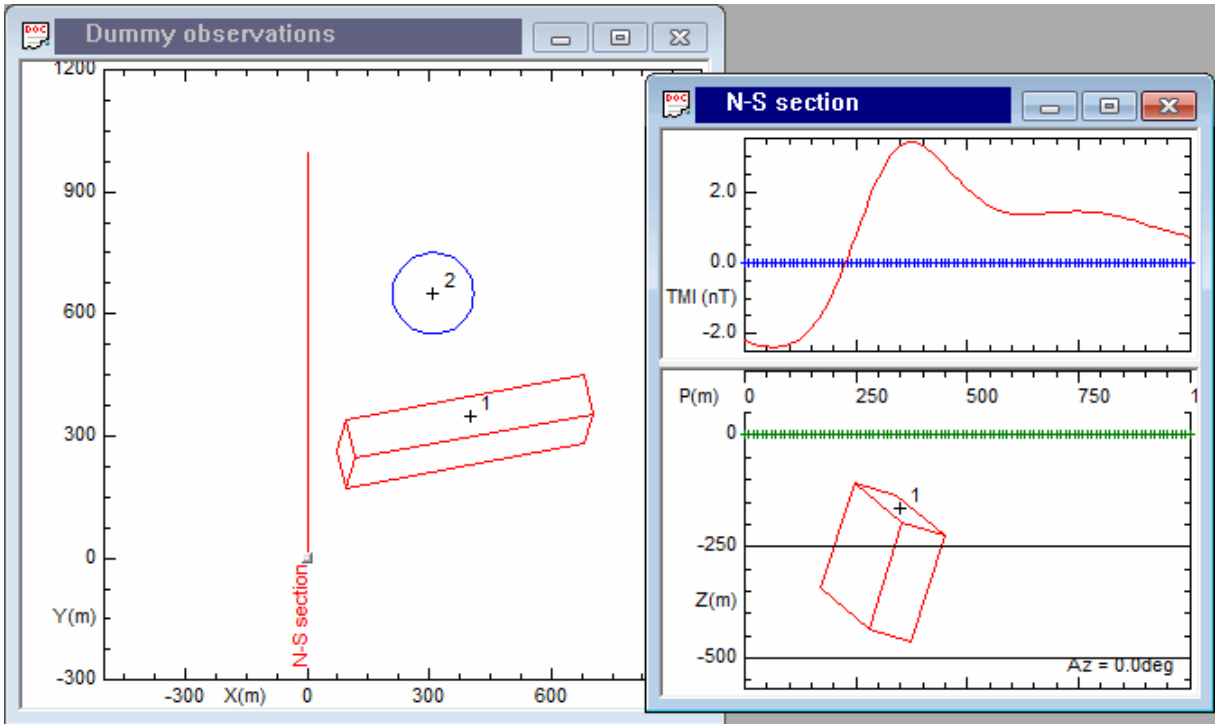
Draw body only if – If this is selected then you can suppress the drawing of 3D bodies on the basis of the distance of either their reference point or their closest point from the plane of the profile. This setting can be over-ridden for a particular body by checking the **Always show on profile plots** option in the edit body dialog box (**General** tab).

In this example the **Reference point** option was chosen with an **is less than** distance of 350m. (This reproduces the behaviour of versions of Potent before v4.15.) Body 2 (blue sphere) is drawn but not body 1 (red prism). This is clearly inappropriate as the effect of body 2 on the calculated profile is relatively small.



A better result is obtained in the next example. The **Closest point** option was

chosen with an **is less than** distance of 150m, which excludes body 2 (sphere). However body 1 (prism) *is* drawn as the distance of its closest (western) vertex is less than this distance.



(In these examples there is little point in excluding either body. However in complex models with many overlapping bodies spread over a large area, suppressing the drawing of bodies that are remote from the section can produce a much clearer plot.)

Annotate bodies with group

By default a body's position is marked by a cross, drawn at the reference point, and annotated with the body number.

You can add further annotations by checking other options in this group.
If you clear **all** annotation options, *Potent* also omits the cross.

The position of the annotation relative to the cross can be adjusted by using the **Annotation offsets** controls in the Edit Body – General dialog box.

Show field vectors group

Observed – This option is available only if the observations include the three magnetic or gravity components. Select this option to cause *Potent* to draw vectors of the observed field, radiating from each observation point.

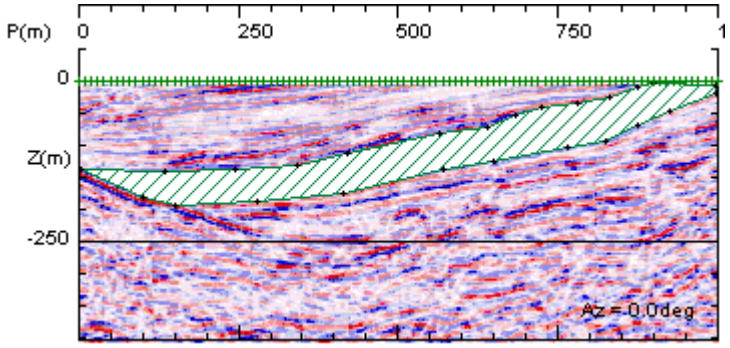
Calculated – Select this option to cause *Potent* to draw vectors of the calculated field. The vectors radiate from the observation points. If the observation components are all magnetic, then magnetic field vectors are drawn. Similarly, gravity vectors are drawn if the field components are all gravitational. If the components are a mixture of magnetic and gravitational, then magnetic vectors are drawn.

Backdrop image group

This feature is intended mainly to assist in stratigraphic modelling. It allows you to display an image from a bitmap file (must be .BMP format) as an aid in positioning your model.

Click the **Load** button and specify an appropriate bitmap file. Initially the image is sized to fill the entire cross-section pane. Choose the **Locate backdrop image** tool to specify the rectangular area that you want the image to occupy.

In this example a seismic section has been used as a guide to help draw a layer (using *Potent*'s **Body shape** tool).



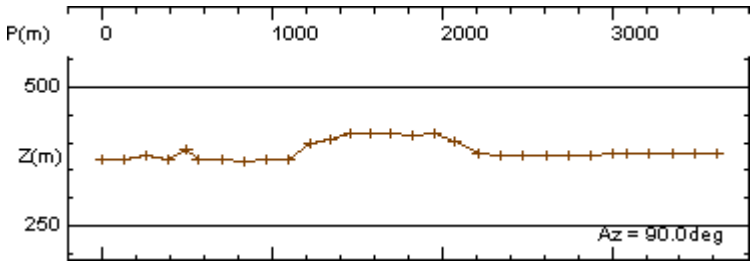
(Seismic image courtesy of GNS, New Zealand.)

See also - Stratigraphic (layered) models in the User's Guide.

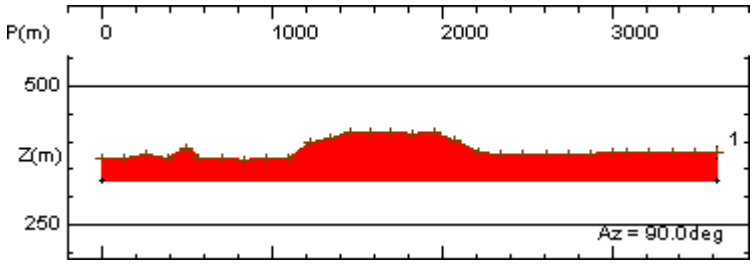
Model topography

Click this button to create a 2D polygonal prism that fits the topography. Note that topography is the difference between height and ground clearance, so the option is useful only if the observations have at least one of these defined.

This example shows a topographic section, vertically exaggerated.



When the **Model topography** button is clicked, *Potent* creates the following polygonal prism. In this example the body has been edited to make it active and to set its cross-section colour to red.



This feature is intended for gravity modelling. Note that the prism is created with vertical ends. You will probably need to drag these out either side to taper the body's calculated field.

Export body sections

Click this button to save the intersection points of polygonal cross-section bodies as a text file. Potent looks at the entire model, and writes an ASCII list of vertices for each polygonal body. An entry is also written as a placeholder for non-polygonal bodies.

The vertices in the list are the XYZ coordinates of the intersection of the body with the plane of the profile. Finite-length bodies are treated as having infinite strike for the purpose of determining the intersection.

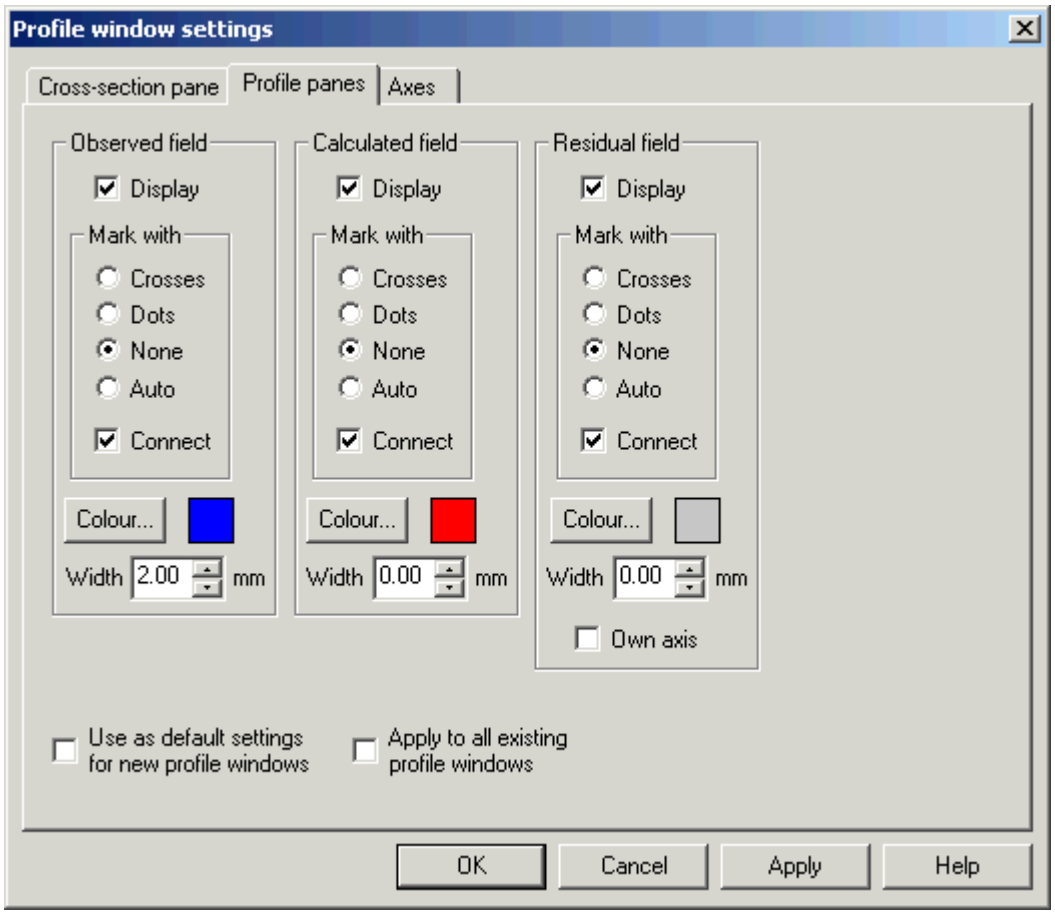
The format of the file has been kept very simple, so you can easily edit it with a text editor before using it for your own purposes.

See also – Features of Potent Dialog Boxes, and Profile windows, in the User's Guide.

--- Return to the Window menu.

Profile window settings - Profile panes

This dialog box is a page of the Profile window settings dialog box. Use it to specify how profile panes are drawn in profile windows.



Most of the controls in the three field groups function identically:

Display – Clear this box to suppress drawing of the field, while preserving the other options in the group.

Crosses – Mark the field values by small crosses.

Dots - Mark the field values by dots. A “dot” is a single pixel on the screen, but is drawn as a very small cross on hardcopy (where a single dot is usually almost invisible).

None – Don’t mark the field values.

Auto – If the number of values is greater than 500, use dots, otherwise use crosses.

Connect – Select this box to draw lines between adjacent values.

Colour – Click this button to set the colour with which the field value markers (dots or crosses) are drawn.

Width – Set the line thickness in mm for printed output. (The line thickness is increased on the screen by a reduced amount.)

The exception is **Own axis** in the **Residual field** group. If **Own axis** is selected the residual field profile is drawn on its own “zero-level” baseline axis, which is located a quarter of the way from the bottom of the profile window. If cleared, the residual field profile is drawn using the same baseline as the observed and calculated field.

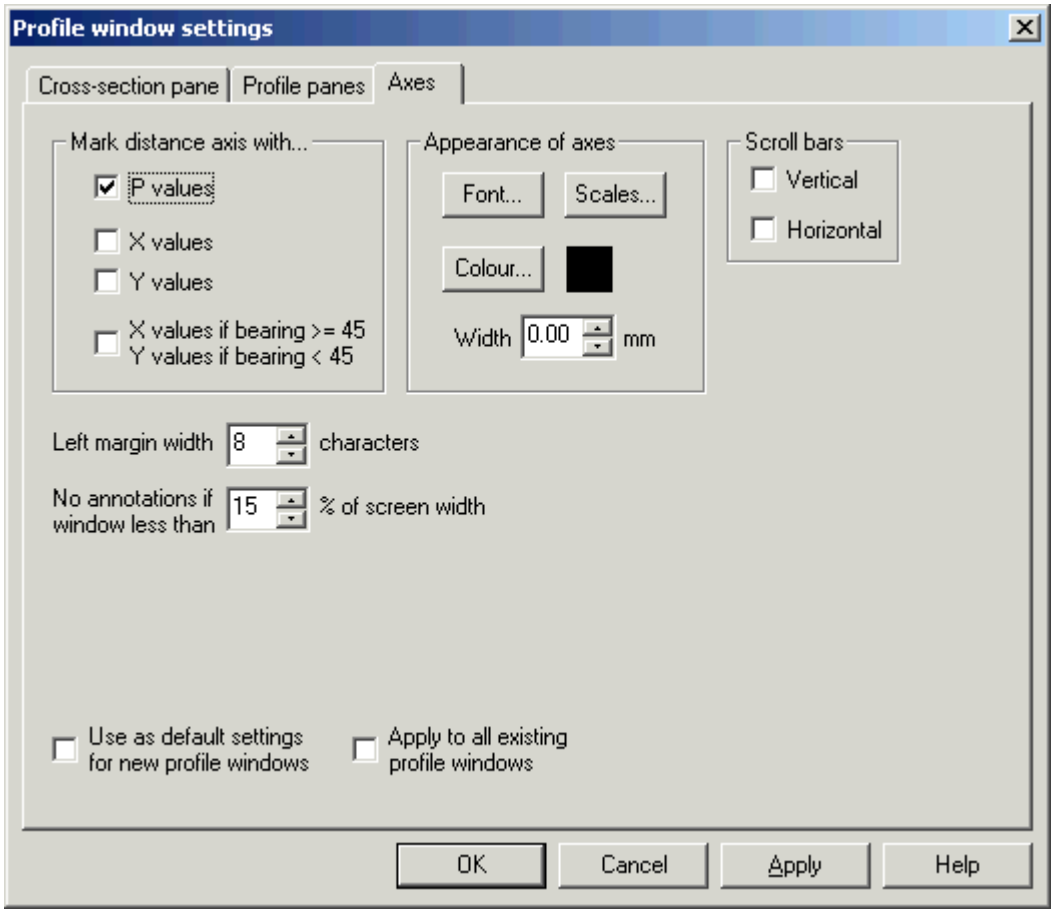
Select **own axis** if you want to be able to separate the the residual profile from the others to improve clarity.

See also – Features of Potent Dialog Boxes, and Profile windows, in the User’s Guide.

--- Return to the Window menu.

Profile window settings - Axes

This dialog box is a page of the Profile window settings dialog box. Use it to specify how axes are drawn in profile windows.



Mark distance axis with… group

The **P-axis** is the “natural” distance axis for plotting sections, as it represents true metres along the direction of the section. However, it often is convenient to display the corresponding X or Y values, particularly if the section is exactly parallel to one of these.

Select the appropriate box to add the required extra axis. Note that the numbers on the X-axis or Y-axis will not correspond to metres along the section unless the section is parallel to the axis.

Appearance of axes group

Font – Click this button to set the font for all axes.

Scales – Click this button to invoke the Profile window scales dialog box.

Colour – Set the colour of the axes.

Width – Set the line thickness in mm for printed output. (The line thickness is increased on the screen by a reduced amount.)

Scroll bars group

Select **Vertical** or **Horizontal** to add a scroll bar to that axis.

Left margin width – Set the width of the area that holds the labels for the vertical axes. The width is set in units of the average character width.

No annotations if… - When a window is small, the annotations take up a disproportional amount of the available plotting area, leaving little space for the actual data. When you set a non-zero value in this box, *Potent* will not draw annotations on the window if its width is less than that percentage of the physical screen width. Set the value to zero if you want *Potent* to always display annotations.

See also – Features of Potent Dialog Boxes, and Profile windows, in the User's Guide.

--- Return to the Window menu.

Down-hole window settings
Down-hole windows are controlled in the same way as profile windows, making allowance for the fact that the profile panes are arranged to the left of the cross-section pane. Refer to the Profile window settings and Profile window scales topics for details.

See also – Working with down-hole data in the User Guide.

--- Return to the Window menu.

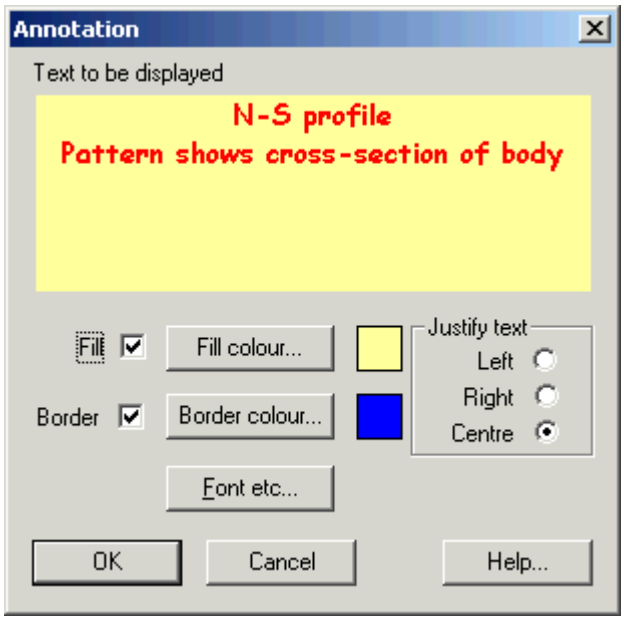
Window | Redraw

Use this command to refresh the drawing of the active window.

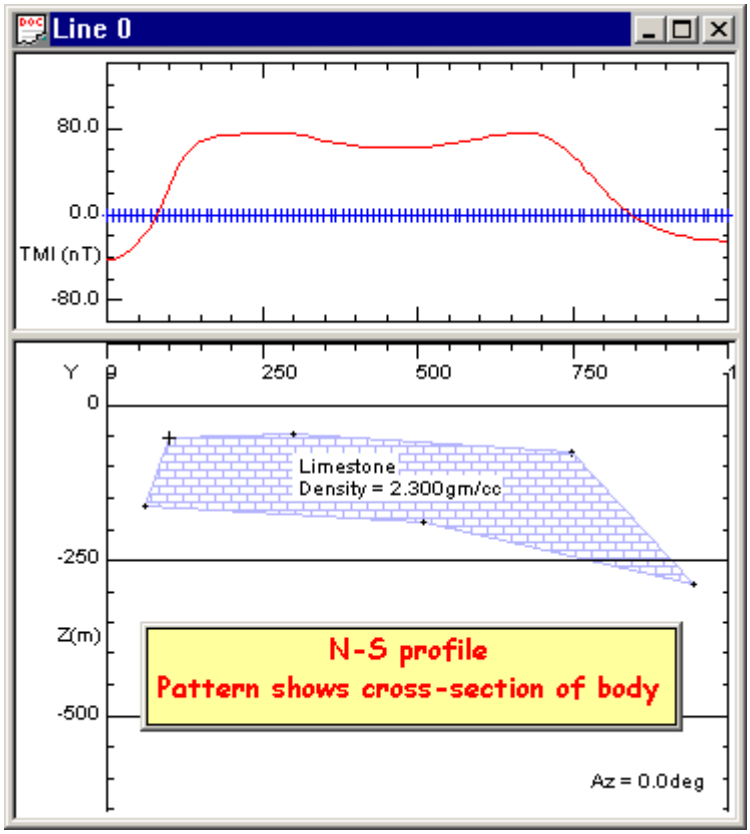
--- Return to the Window menu.

Window | Annotations

Use this command to add a new Annotation box to the active window. It may be invoked either from the **Windows** menu, or from the popup menu that appears when you right-click on a window. The **Annotation** dialog box is displayed:



Type your text into the text box (the Enter key inserts a new line), and set formatting as required. When you click **OK** an annotation box is added to the window.



Refer to Free-format annotation boxes in the User's Guide for more information.

--- Return to the Window menu.

Window | Auto-fit profile panes

When a profile or down-hole window is active, and there is more than one field component, this command causes the profile panes to be sized evenly to fill the space adjacent to the cross-section pane.

Help menu

Help | Help topics

This command displays the main Help index.

--- Return to the Help menu.

Help | About

This command displays the Welcome dialog box.

--- Return to the Help menu.

Help | Set user name

Type in a name that will be used to label various types of output, particularly hardcopy. This would typically be your company name.

--- Return to the Help menu.

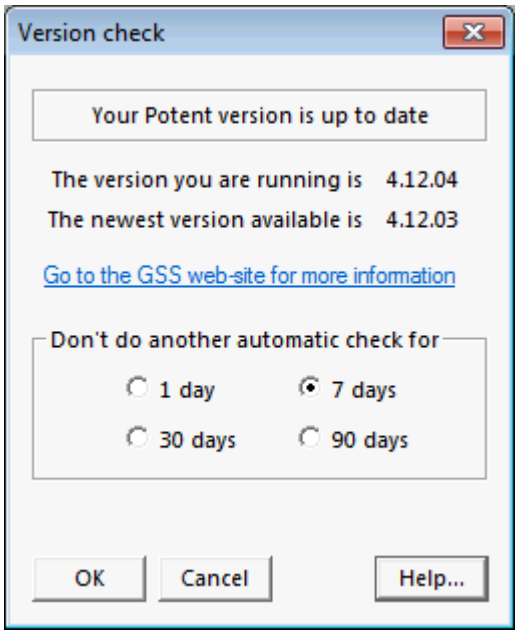
Help | View licence conditions

Choose this command to view the *Potent* **Conditions of use** dialog. Click **Yes** to accept the conditions, in which case *Potent* will proceed normally. If you choose **No**, *Potent* will continue for the current session, but when you run it again the **Conditions of use** dialog will be displayed immediately and *Potent* will continue only if you click **Yes**.

--- Return to the Help menu.

Help | Check for updates

This command causes Potent to check for an Internet connection and, if present, to compare the version of Potent you are running to the newest version available from the **Downloads** page of the GSS web-site. The **Version check** dialog box is displayed.



This check also is performed automatically when Potent starts up. If an Internet connection is found (Potent will wait for a maximum of one second for a connection to be established), and there is no newer version on the web-site, then the process is silent (i.e. no dialog box is displayed). The frequency of these automatic checks can be controlled by the radio buttons in the dialog box.

(Potent does not send any information to the GSS web-site – it merely reads the version number that is encoded in the web page.)

--- Return to the Help menu.