QC-TOOL

USER'S GUIDE

Eikon Technologies Ltd.

What is QCTool

QCTool provides you with an efficient means of data quality control prior to advanced processing and interpretation. It is not confined to a special type of data and can be used in any field where data analyses and handling are involved. It offers three modes of data display - tables, grids and plots, which, being interrelated, make the cumbersome job of data checking and editing as easy as a click of the mouse

QCTool also offers a full suite of gravity and magnetic data processing as well as advanced processing tools for a wide variety of data.

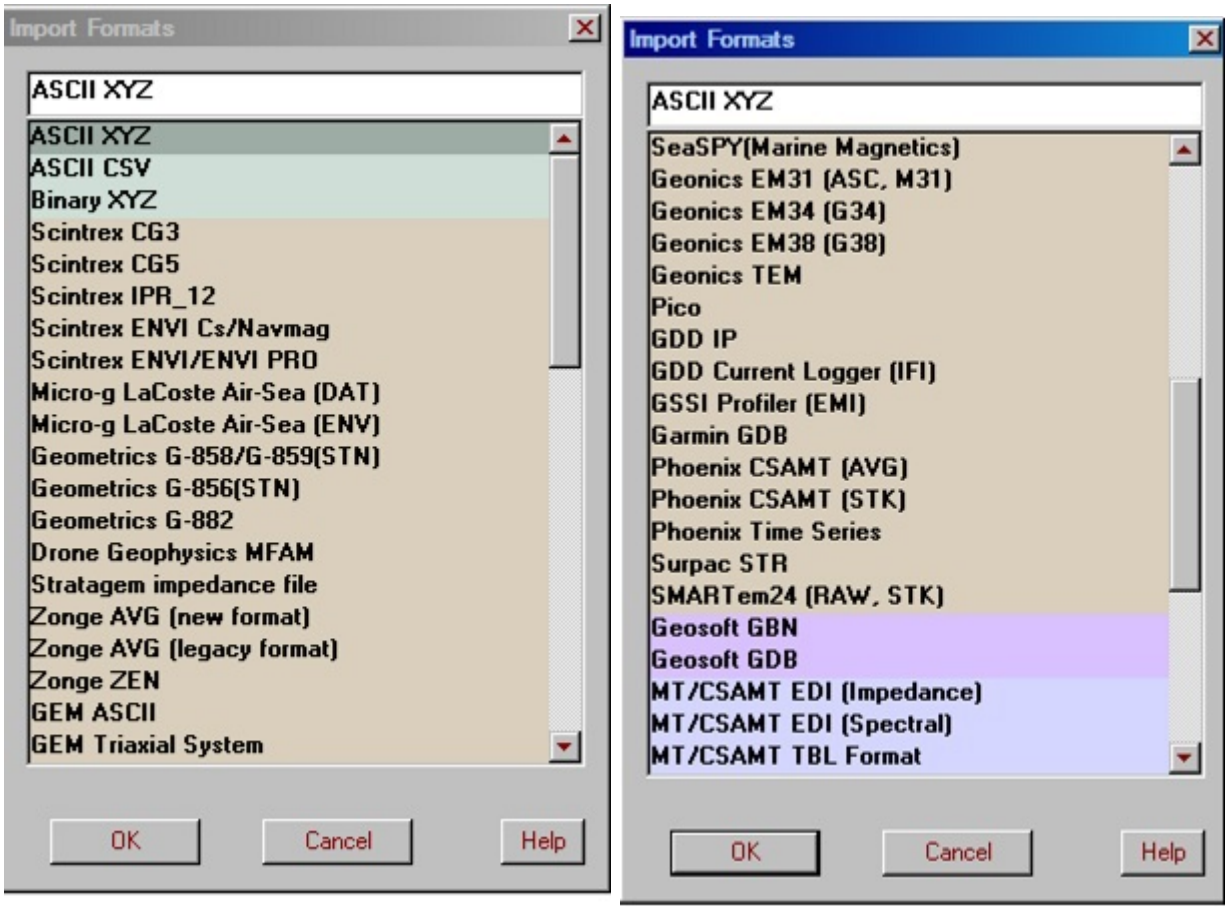
Terms (Notational Conventions)

Point	a row of a table (a single data point sample with N associated data channels)
Channel	data representing the same characteristic and forming a column of a table
Line	a data subset in a sequence of measurements shown in QCTool normally as a separate spreadsheet
Fiducial Channel	a channel containing reference information (time of measurement, etc.) arranged in a certain order; as a result each **point** has its own fiducial, an index to be utilized as a basis for merging different data sets (files)
Grid	a 2D array of data points
Interpolation	the process of estimating data at the vertex of a **grid** based on the data collected at data points which are possibly randomly distributed

Importing Data

How to Import Data to QCTool

1. Select **File/Import File**. The **Import Formats** window appears, containing many formats:



2. Select a required format and click **OK**. In the Windows-style **Import File** dialog to open, browse for the file to import.

QCTool permits import of the following data formats:

- ASCII XYZ(.txt, .xyz)
- ASCII CSV(.csv)
- ASCII GEM
- Binary XYZ
- Geosoft GBN
- Geosoft GDB
- Scintrex CG3 Dump
- Scintrex CG5 Dump(.dmp)
- Scintrex IPR_12(.dmp, .raw)
- Scintrex ENVI Cs/Navmag
- Scintrex ENVI/ENVI PRO
- SeaSPY(Marine Magnetics)(.mag)

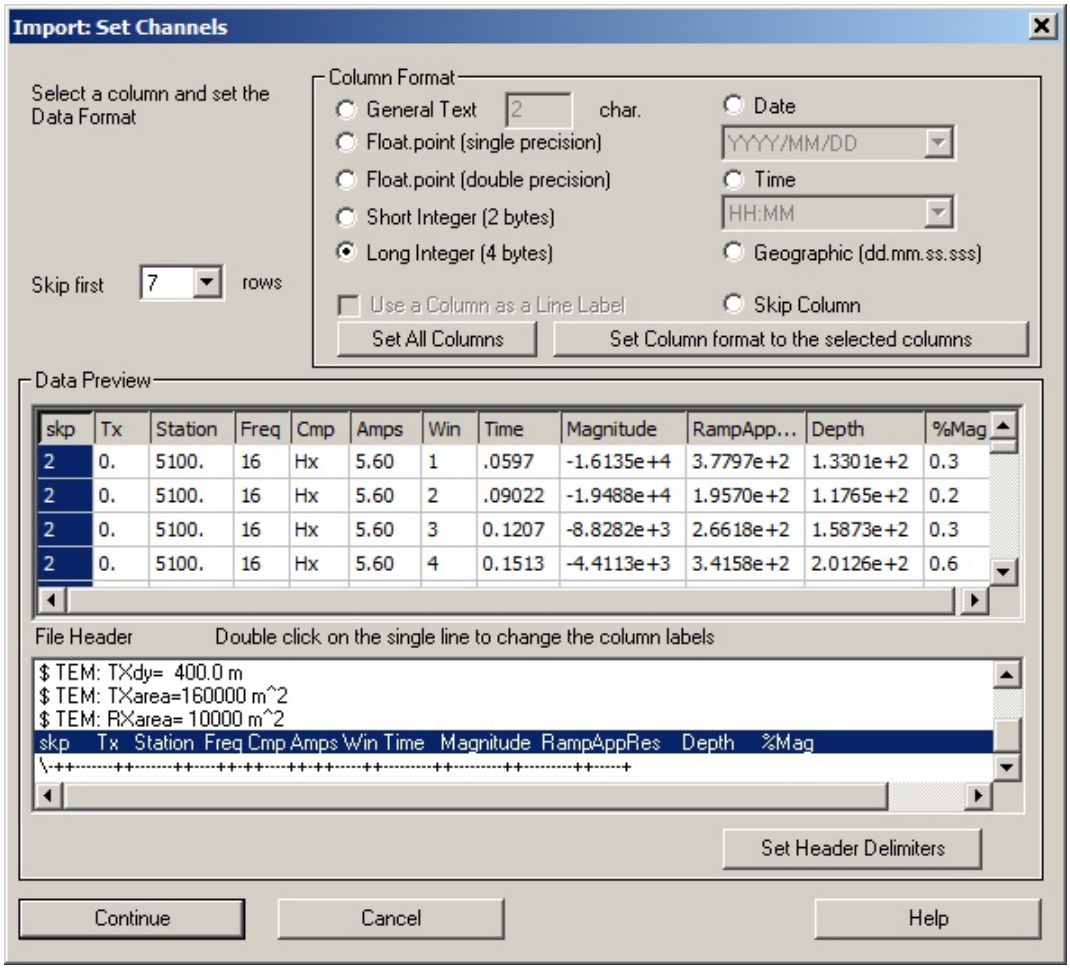
- Pico (.p*) files.
- Geometrics G-858/G-859/G-856/G-882(.stn)
- SEGY format
- Stratagem impedance file
- MT/CSAMT (impedance and spectral .EDI and .tbl formats)
- VLF (.tbl format)
- IRIS (VLF ASCII)
- Universal Sounding Format(.usf)
- Zonge AVG (legacy new and Zen formats)
- Geonics (.asc, .m31, .g34, .g38 and TEM formats)
- GEM triaxial bird
- Geophysical logs (.las)
- GPS Exchange Format(GPX)(.gpx)
- Grid Exchange file (.gxf)
- IAGA2002 Observatory Data
- Micro-g LaCoste Air-Sea(.dat,.env)
- GDD (IP, Current logger)
- GSSI Profiler (.emi and .dat)
- Phoenix (.avg, .strk and time series formats)
- SMARTem25 (.raw, .stk)
- EGR Grid Format
- ArcGIS (.flt format)
- Band Interleaved (.bil format)
- CDED DEM
- Geosoft grid file (.grd format)

3. Select the file to import and click **Open**. This brings up the **Save As** dialog, offering you to save your file as a .qct file. You can leave the old name of your file or, if desired, rename it.

4. Click **OK** to import.

ASCII Columnar Format

If you are importing the ASCII data, the **Import: Set Channels** dialog appears:



To specify the header line

If a file of data to be imported has a single-line header, the latter is automatically displayed as a header in the **Data Preview** table. If your header is not automatically displayed increase the number of skips until the header line shows in the **File Header** box.

If the initial header consists of two or more lines, each column in the **Data Preview** table is assigned a consequent number (Ch1, Ch2 and so on), whereas the **File Header** field below offers a number of choices.

- Select a header line in the **File Header** field.

The standard delimiters between columns are a space, a comma, and a tab. If this is the case, skip this paragraph and go to the next step. If you have a different delimiter in your initial file, click the **Set Header Delimiters** button to open the **Delimiters** dialog. Check the **Other** box and specify the kind of the delimiter used in the now active field to the right.

- Double-click the selected header line to incorporate it into the table. A message asks you to confirm the replacement.

- Click **OK** to confirm.

If the **File Header** field does not contain any items, it means that the first line of the file to import is not marked as a comment line (preceded by a slash "/"). In this case:

- If you have one first line, select 1 from the **Skip first rows** dropdown list above the **Data Preview** table. If you have two or more first lines, select the respective number. This removes the first line(s) from the table and places it (them) in the **File Header** field.

- In **File Header** field, double-click a required line to insert it as a header in the **Data Preview** table.

- Click **OK** in the confirmation dialog to appear.

To specify the format of data to import

You can specify the format of each column of your data:

- Select a column in the **Data Preview** table of the dialog. Note that the table contains only a portion of the data to be imported (about 20 rows).

  In the **Column Format** section:

- Select the **General Text** button if a column contains text.

  The box to the right shows an automatically estimated maximum number of characters per row in this column. Since the estimation is based on the portion of data displayed in the **Data Preview** table, you probably will want to edit it. For example, your data set contains 10,000 points with record numbers from 1 to 10000. The **Data Preview** table displays only 20 first rows with record numbers not exceeding 2 characters. As a result, the **General Text** box contains a 2-character estimate. To provide for your 5-character data, change 2 to 5.

- If your data contains date or time and you want to edit its format, click on the header of the respective column, turn the **Date** or **Time** button on and select the required format from the dropdown list.

- Select **Skip Column** to remove a column from further processing. This option is especially convenient when a column contains text and there is no point to subject it to processing.

- Select between the single- (4-byte) and double- (8-byte) precision and between a short (2-byte) and long (4-byte) integer.

- Repeat the same procedure for other columns.

- Check the **Use a Column as a Line Label** box to have a separate table for each data subset, or “line”.

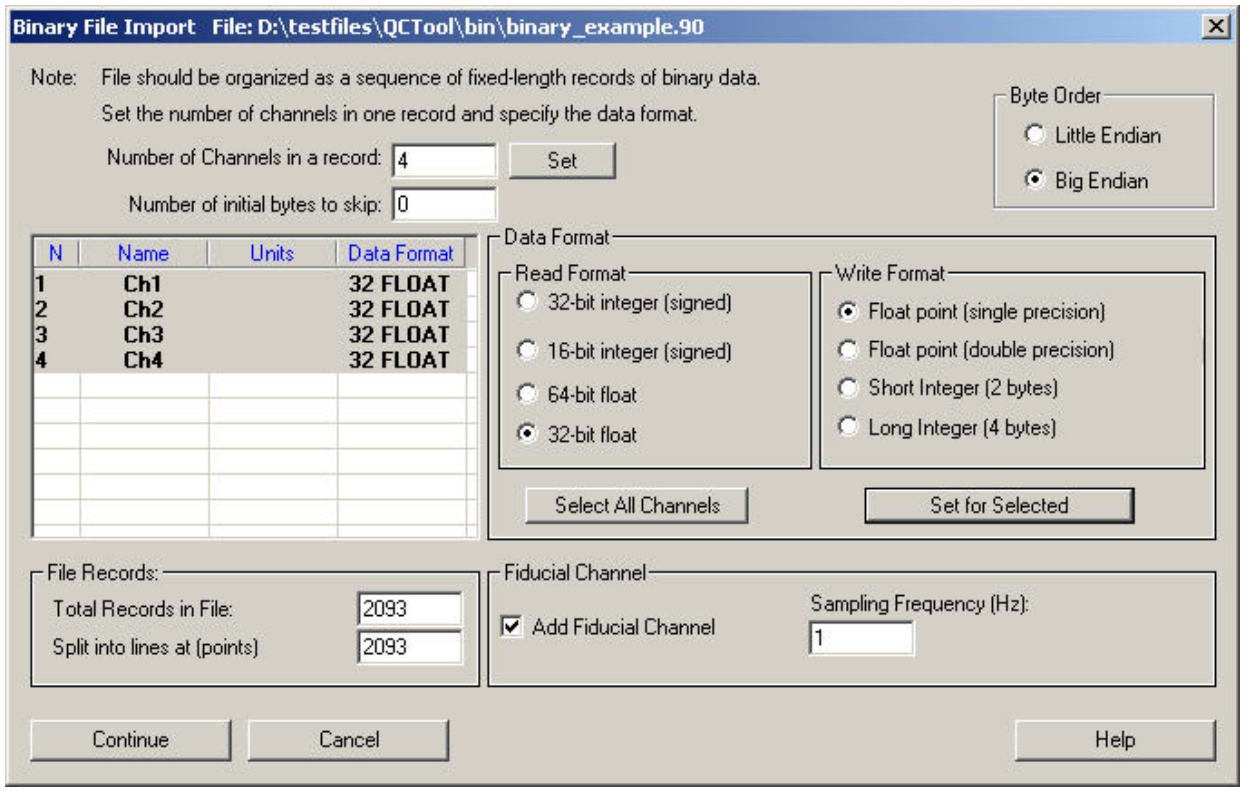
It applies if every row of your initial data contains a line number. In this case all line numbers are imported into a separate column. If you select **Skip Column** to remove it from the table, QC Tool will still use information from this column to perform the subdivision by lines. ***Note****. Currently, this subdivision is possible only if lines in your initial file are identified by the word “LINE”.*

All above steps performed, click **Continue** to complete data import.

Note*. The format adjustments will show only after import.*

Binary XYZ Format

If you are importing the binary XYZ format, the **Binary File Import** dialog appears:



- Specify the number of channels in a record in the box at the top of the dialog and click **Set**.
- If the file begins with data you do not want to import, enter a value in the edit box labelled **Number of initial bytes to skip**
- Selecting **Big Endian** in the **Byte Order** section indicates the file contains values where the first byte is the most significant byte. If the most significant byte is the last byte, choose **Little Endian**.

The table below and the **Data Format** section become active.

- Click on the cell in the **Name** column of the table to enter the name of the channel and on the cell in the **Units** column to enter the units of measurement.
- In the **Read Format** section, select between the four options, 16-bit integer, 32-bit integer, 32-bit float, and 64-bit float. The **Write Format** section changes accordingly.

- Click **Set for Selected** to insert the data format in the selected row of the table

OR

- Click **Select All Channels** to select the whole table and then **Set for Selected** to add the same data format to all rows.

All channels having been specified, the **File Records** section in the bottom left-hand corner of the dialog becomes active.

- In this section, check the total number of records in your file and specify, if required, an interval to be used to cut your data into lines.

- In the **Fiducial Channel** section, leave the **Fiducial Channel** box checked to add this channel to your table and insert the value of sampling frequency in the respective field. This sampling frequency could represent any quantity to be used to number or order the data. For example, if the sampling frequency were set to be 1 then the Fiducial would simply be the record number. As pointed out in the Terms section, an important purpose of this channel is to provide basis for merging different data sets.

- Click **Continue** to complete data import.

Geosoft GBN file import

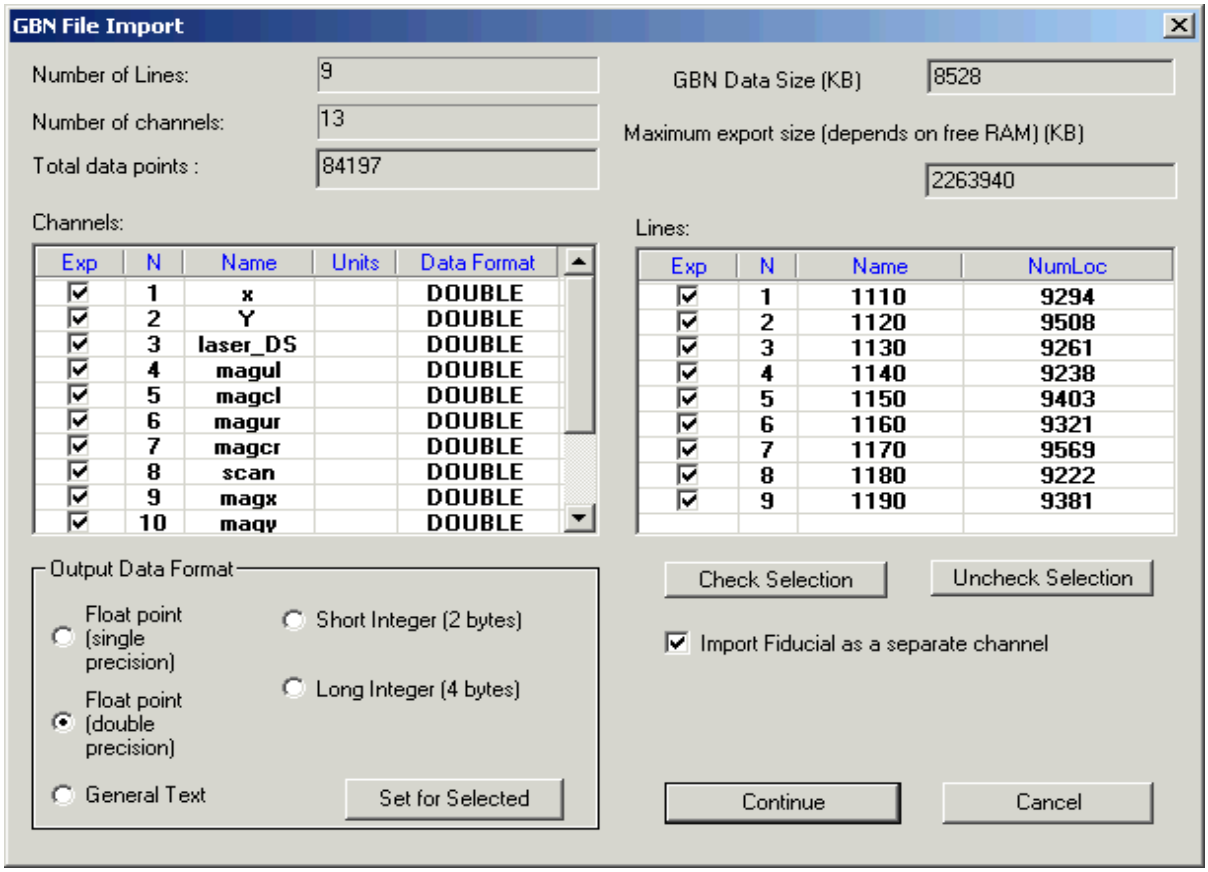
- Select **File/Import** on the QCTool menu. The Import Formats dialog appears.

- From this dialog, select **Geosoft GBN** and click **OK**. In the **Import file** dialog to appear, browse for a required *.gbn file and click **Open**.

  The standard Windows-style **Save As** dialog opens, offering you to save your *.gbn file in the *.qct format.

- Click **OK**.

  The **GBN File Import** dialog that appears contains general information on your file - the number of lines and the number of channels in a record. The left-hand table below shows all the channels available in your file; the right-hand table shows all the lines available in your file.



- Select or deselect the channels to import in the **Channels** table. You can do it manually by checking or clearing the boxes in the **Exp** column. Or, you can select one or more channels at a time and use the **Check Selection** and **Uncheck Selection** buttons in the right-hand part of the dialog.

- To specify the output data format for an individual channel:
  Click on a channel in the table, select one of the five available formats in the **Output Data Format** section and click **Set for Selected**.

- Select or deselect the lines to import in the **Lines** table. You can do it manually by checking or clearing the boxes in the **Exp** column. Or, you can select one or more channels at a time and use the **Check Selection** and **Uncheck Selection** buttons under the table.

- Check the **Import Fiducial as a separate channel** to create a channel for the Geosoft file's fiducial column usually displayed as the first column.

- The amount of data you have selected to import is displayed at the top of the window beside **GBN Data Size**. This value needs to be less than the **Maximum export size** in order to import your data.

- Click **Continue** to complete import into QCTool.

Geosoft GDB file import

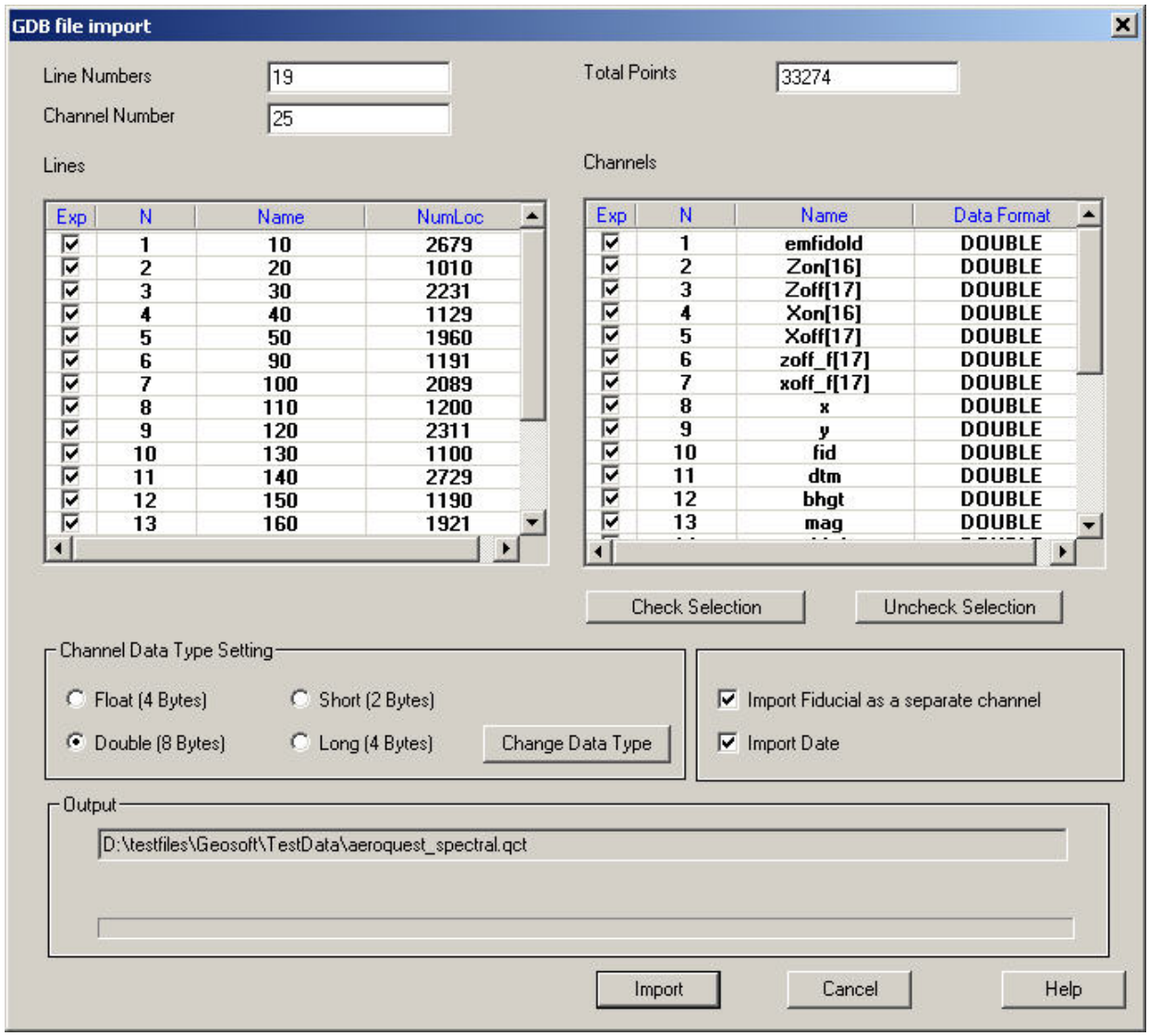
- Select **File/Import** on the QCTool menu. The Import Formats dialog appears.

- From this dialog, select **Geosoft GDB** and click **OK**. In the **Import file** dialog to appear, browse for a *.gdb file and click **Open**.

  The standard Windows-style **Save As** dialog opens, offering you to save your *.gdb file in the *.qct format.

- Click **OK**.

  The **GDB File Import** dialog that appears contains general information on your file - the number of lines, the number of channels and the total number of points. The left-hand table below shows all the lines available in your file; the right-hand table shows all the channels available in your file.



- Select or deselect the channels to import in the **Channels** table. You can do it manually by checking or clearing the boxes in the **Exp** column. Or, you can select one or more channels at a time and use the **Check Selection** and **Uncheck Selection** buttons in the right-hand part of the dialog.

- To specify the output data format for each channel:
  Click on a channel in the table, select one of the four available formats in the **Channel Data Type Setting** section and click **Change Data Type**.

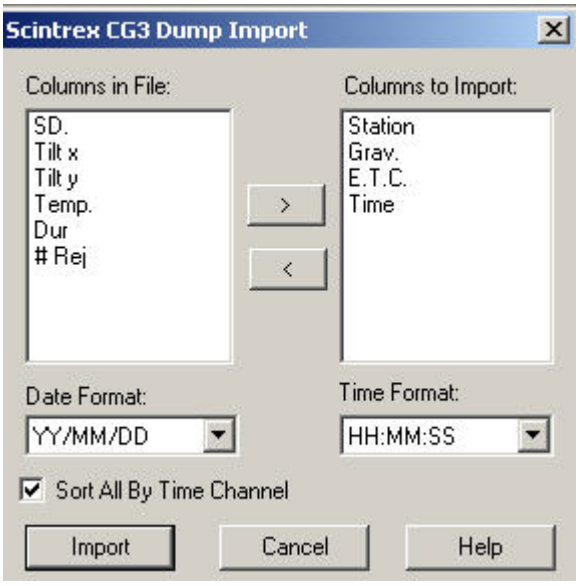
- Select or deselect the lines to import in the **Lines** table. You can do it manually by checking or clearing the boxes in the **Exp** column. Or, you can select one or more channels at a time and use the **Check Selection** and **Uncheck Selection** buttons under the table.

- Check the **Import Fiducial as a separate channel** to create a channel for the Geosoft file's fiducial column usually displayed as the first column.

- Click **Import** to complete import into QCTool.

Scintrex CG3 Dump Format

On selecting **Scintrex CG3 Dump** in the **Import Formats** dialog (see How to Import Data), the **Scintrex CG3 Dump Import** dialog appears:



- In the **Columns in File** field, select columns that you want to import and click the > button. The selected columns will appear in the **Columns to Import** field. To remove a column from the **Columns to Import** field, select this column and click the < button.

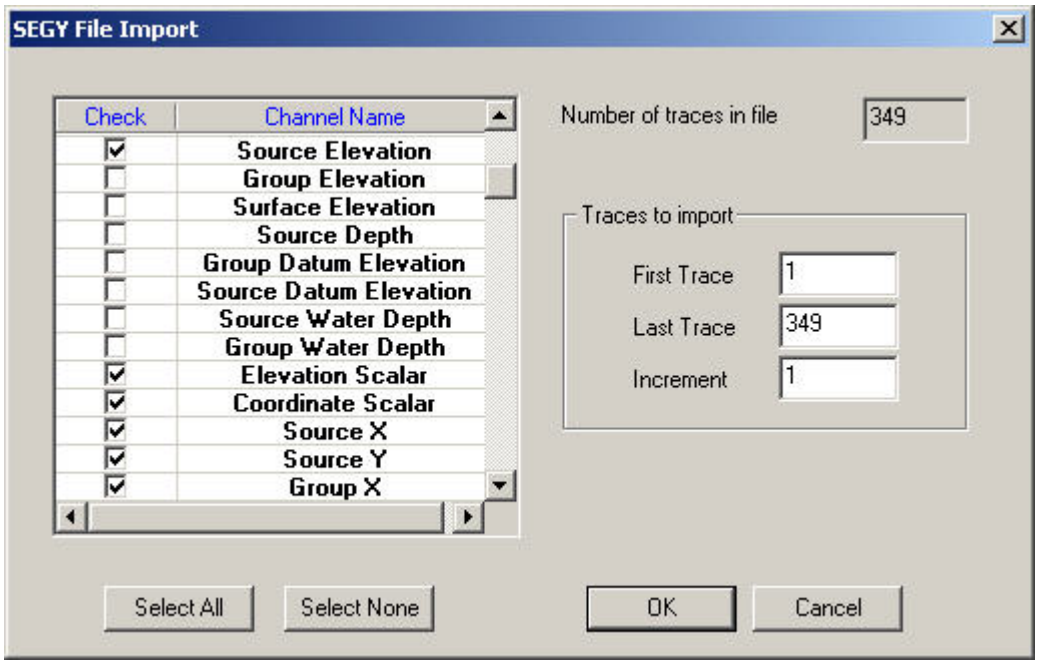
- Select a required date and time format from the respective dropdown lists below. This is how the date and time will appear in QCTool.

- Leave the **Sort All by Time Channel** box selected to sort your data by time in ascending order. If you de-select this box, you data will appear in the same order as in the raw file.

- Click **Import**.

SEGY Format

If you are importing using the SEGY format, after selecting the input and output files, the **SEGY File Import** window appears:



- Specify the channels you would like to import by selecting the box in the **Check** column.
- You may quickly **Select All** or **Select None** by clicking the respective button.
- In the **Traces to import** section, you may specify how many traces you would like to import from the file by entering a range from the **First Trace** to the **Last Trace**. It is also possible to enter an **Increment** so traces will be skipped. e.g. A value of 1 will import all traces. A value of 3 will import every third trace.

Working With Spreadsheets

Spreadsheets Overview

If a user imports a completed or an existing *.qct file opened, a spreadsheet-like “three-dimensional” table appears containing data in the format you specified (see How to Import Data to QCTool). This table has “depth” and can be divided into as many smaller tables as the number of data subsets, or “lines”, in the initial file. They can be further subdivided inside the application. You may also have a version that allows merging of these subsets or “lines”. The links between the tables differ from those used in similar applications, like Excel, since QCTool provides a very strict data structuring. The tables, numerous as they may be in a file, all have the same data structure and, therefore, are easy to shuffle through and to work within.

See Also

Change the Format of Data

Change the Colour and Font of your Spreadsheet

Copy and Paste Data in your Spreadsheet

Rename a Column Header

Insert a New Column

Delete/Restore a Column

Calculate/Recalculate a Channel, Create Row Indices

View the Channel Statistics

Delete a Single Value

Delete/Restore a Point (a Row)

Delete/Restore a Line (a Table)

Remove Outliers

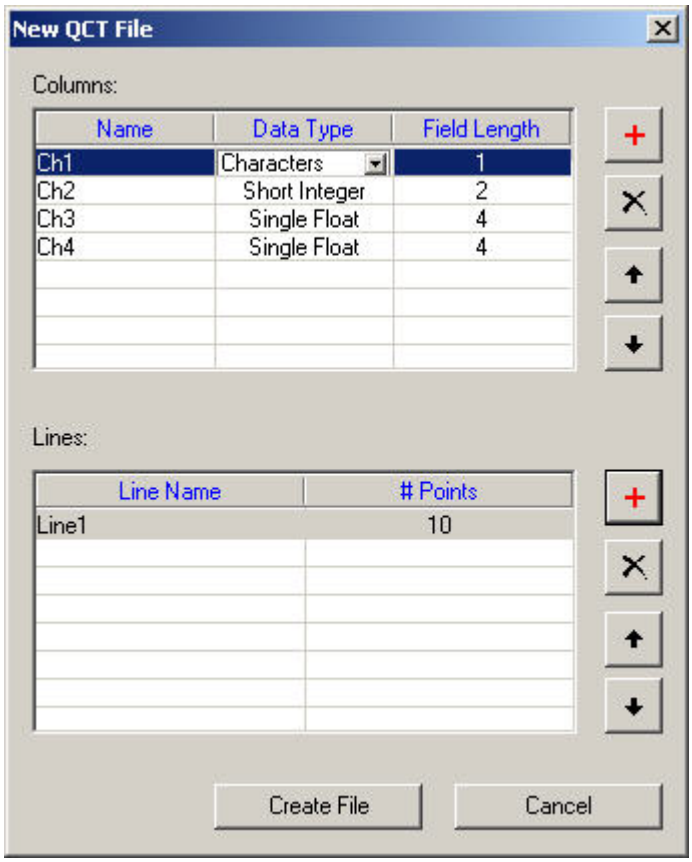
Cut a Line into Two or More Parts

New command (File menu)

Use this command to create a new qct-file to be displayed in a new spreadsheet window.

Toolbar: 

The following window will be launched. 4 new columns were added and the format of each channel was selected



- Add to the **Columns** list using the button with the red plus sign next to the list box.
- Add to the **Lines** list.
- Delete an entry using the appropriate  button.
- Scroll through list boxes using the appropriate  and  buttons.

- Click **Create File**, choose a filename and an empty spreadsheet will be displayed according to the parameters you entered.

- The resulting empty file can be edited by clicking on a cell and then pressing an arrow key or enter to proceed to the next cell.

To change the format of data

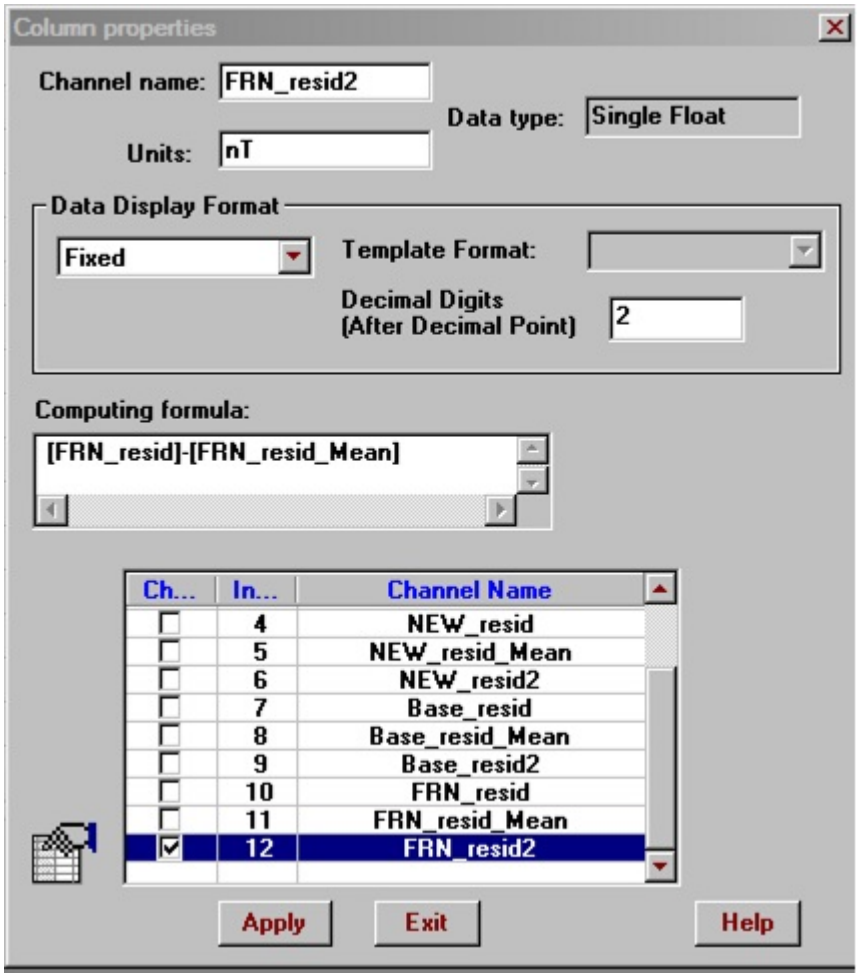
Although you specified the format of your data during import or when you created a new channel or imported a channel from another file, you may decide to change it during your further work.You can easily do this right from the table.

- Click on the header of the column, the format of which you want to change, and select the **Column Format** button  in the second row of the QC toolbar that is available when you work with spreadsheets

OR

- Right-click on the column header and select **Column Format** from the popup menu to appear.

  The **Channel properties** window appears:



In the **Data Display Format** section of the dialog:

- Select a required format from the dropdown list. For single- or double-precision data, you have the **Fixed** and **Scientific** options. If your data represent **Date** or **Time**, select one or the other from this list. This activates the **Template Format** menu which, depending on your selection, will offer you various formats of date and time.
- Specify the number of digits after the decimal point in the respective box.
- Select the columns this format should apply to with a checkmark.
- Click **Apply**.

Computing formula field contains the formula entered in the **Formula Calculator** function for the currently selected channel. It cannot be edited here. To change it, use the **Formula Calculator** interface.

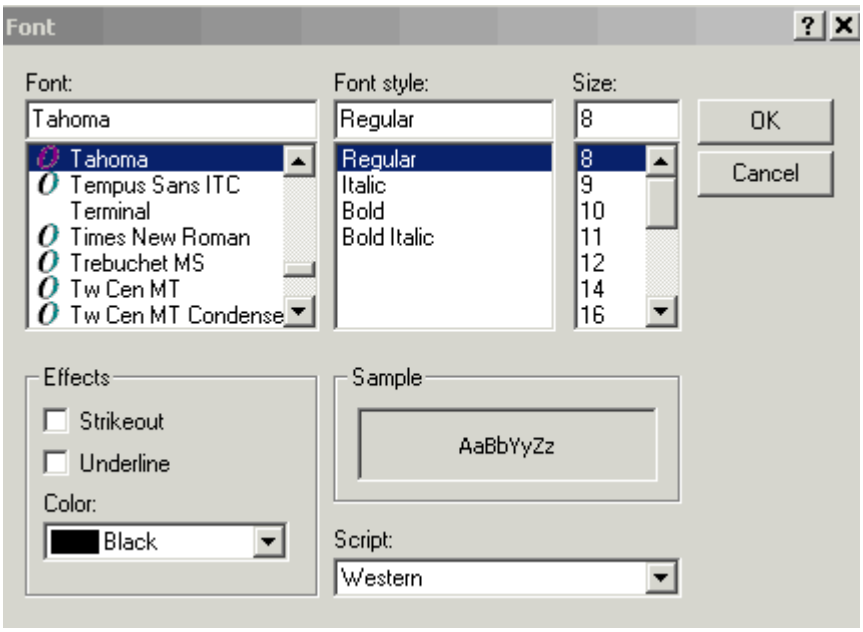
To change name and units of the current channel, edit respective fields in the top left corner of the window and click **Apply**.

During certain imports, some channels might be assigned an additional 'attribute' value. For example, Geonics gate channels have "Middle of the gate period" stored in imported .qct file. This string value can be seen and edited in the edit control to the right of **Computing formula** box. In the screenshoot on this page, "Window center" is attribute name; "106.9 µs" - its value.

All changes become visible when you click **Apply** button. After editing is completed, click **Exit**.

To change the colour and font of your spreadsheet

In the top DOS type menu bar, select **View/Spreadsheet Font** on the QCTool menu. The standard **Font** interface opens:



Make your adjustments as required and click **OK**. The changes will apply to the entire spreadsheet. Being saved in your computer, the settings will remain the same even if you switch to another line/data set or close QCTool and open it again.

To copy and paste data in your spreadsheet

The **Copy** and **Paste** functionalities are similar to those available in all like applications.

- Select the cells/rows/columns you want to copy and click the **Copy** button  on the QC toolbar or press Ctrl+C or select Copy on the right mouse menu.
- Place the cursor where you want your data to appear and click the **Paste** button  on the toolbar or press Ctrl+V or select Paste on the right mouse menu.

To rename a column header

- Right-click on the header to change and select **Column Properties** from the popup menu to appear.

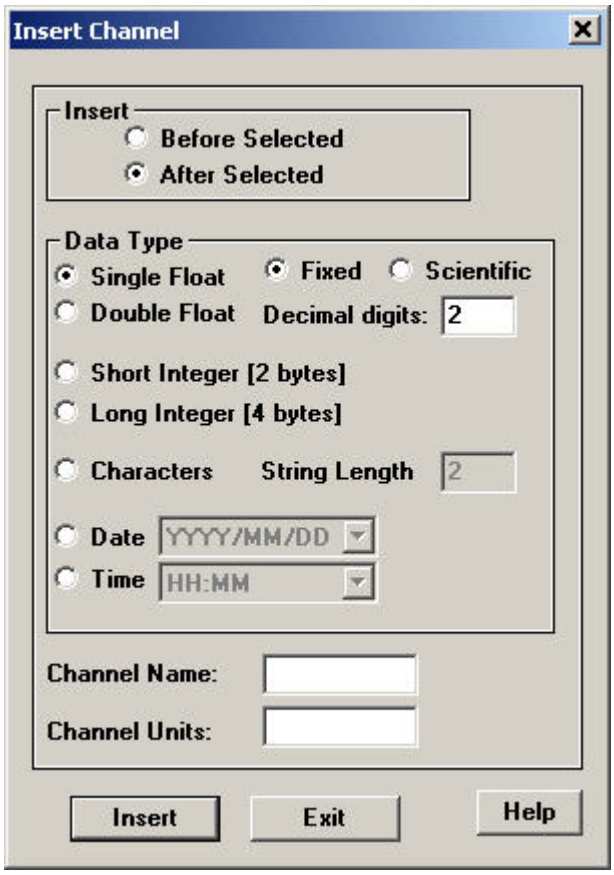
- Change the name and, if necessary, units in this dialog.

- Click **Apply**.

To insert a new column

- Right-click on the header of the column, next to which you want to create a new column.

- In the popup menu to appear, select **Insert Channel**. The **Insert Channel** dialog opens:



- In the **Insert** section, specify the location of a new column relative to the column selected.

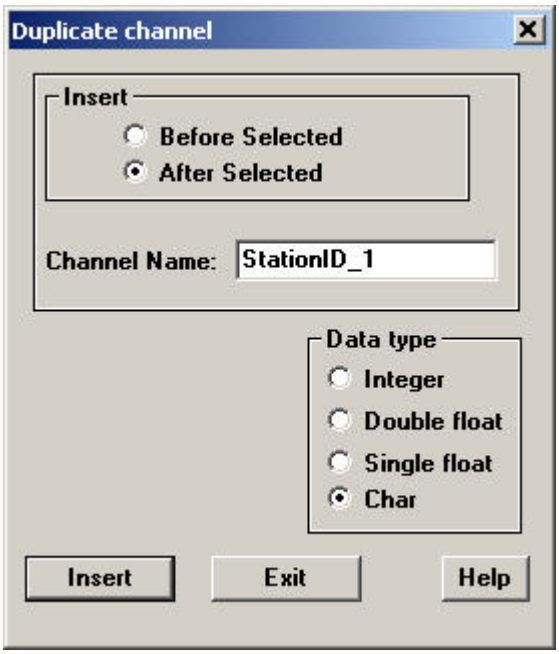
- In the **Data Type** section, specify the format of data this column will contain.

- Specify the header of the column in the **Channel Name** box and the units of measurement in the **Channel Units** box.

- Click **Insert**.

To duplicate currently selected column

- Right-click on the header of the column, next to which you want to create a new column.



- In the popup menu to appear, select **Duplicate Channel**. The **Duplicate Channel** dialog opens:

- In the "Result" section of the dialog, set desired type of newly created channel.

- Click **Insert**.

To delete/restore a column

- Right-click on the header of the column to delete.
- In the popup menu to appear, select **Delete Channel**. The column will get highlighted in dark blue, which means that it has been removed from processing.
- To restore an earlier deleted column, right-click on its header to bring up the popup menu and select **Restore Channel**.

To calculate/recalculate channels, create row indices (Formula Calculator)

The Formula Calculator is used in the following cases:

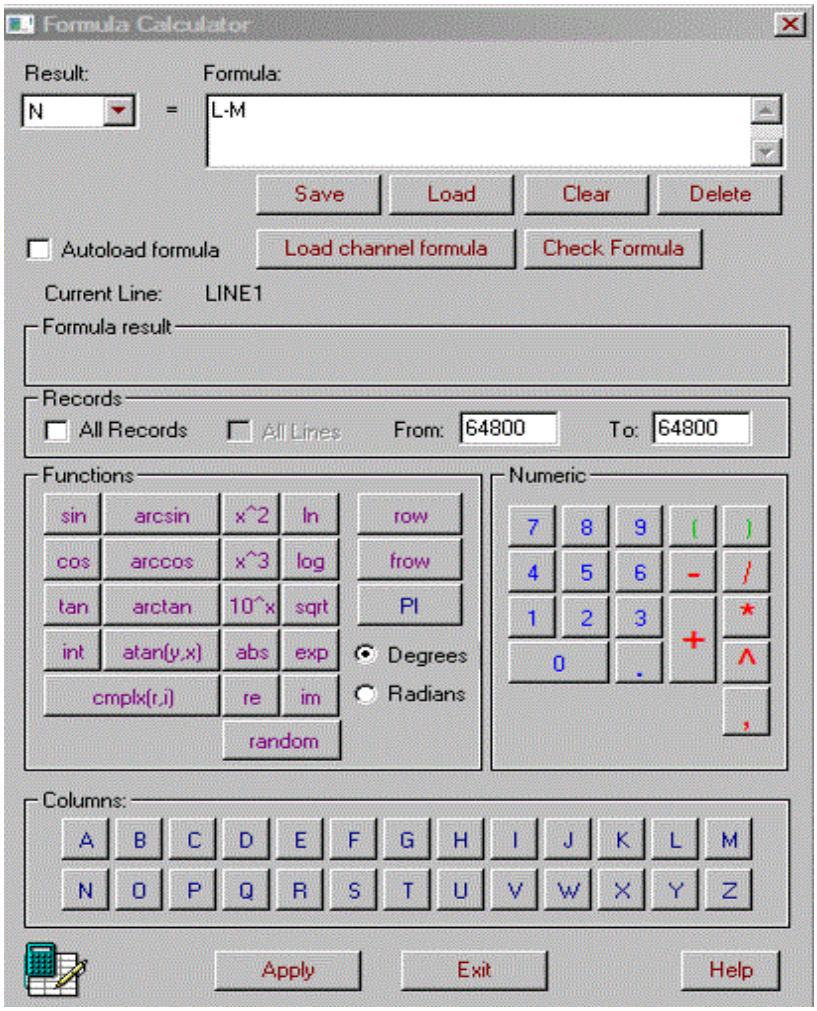
- When you need to recalculate values in a number of cells of a column
- When you need to recalculate an entire column in one spreadsheet or all spreadsheets
- When you need to calculate a new column; in this case, prior to the calculation, insert a new column as described in Insert a New Column
- When you need to assign each row an index.

To bring up the **Formula Calculator**:

- Select a required number of cells in the column or double-click on the header of this column to select all the cells in it and click the **Calculate Data** button  .

  OR

- Right-click on the selected cells/column/newly-added column and select **Calculate** from the popup menu to appear. The **Formula Calculator** window opens:



In this window:

- The **Result** dropdown list shows the index (a letter) of the column for the calculated value to appear.

- In the **Formula** section:

  - Work out your formula using the standard **Functions** and **Numeric** keyboards below. The **Columns** keyboard inserts the indices (letters) of the columns and is used if your formula is based on the values from other channels.
  - Use the standard **Delete** button to edit your formula.
  - Click the **Save** button to save your formula as a .fml file, so that in future you can use it again.
  - Click the **Load** button to load a .fml file and use an already available formula.

- Click the **Clear** button to remove or replace the formula.
- Formulas are stored to the .qct file and if the user has performed calculations previously, the formula may be loaded by clicking **Load channel formula**
- Click **Check Formula** to make sure that the formula contains no errors. In the **Formula Result** section below, you will see a sample of calculation for the first row or, if your formula is incorrect, an error message.

- In the **Rows** section of the window:
  - Specify the range of rows in a new column to contain the calculated parameter. If you have already selected a certain number of rows right in the table, the **From** and **To** fields will contain the first and the last number of these rows, respectively.
  - Check the **All Rows** box to select all rows. The **From** and **To** fields will become disabled.
  - Check both the **All Rows** and **All Lines** boxes to select the same column throughout all of your tables (lines)

    ***Note**. If you specify the first and the last row in the **From** and **To** fields and simultaneously check the **All Lines** box, the number of the rows to be selected in each table (line) will not exceed the range specified, no matter how many more rows the table contains.*

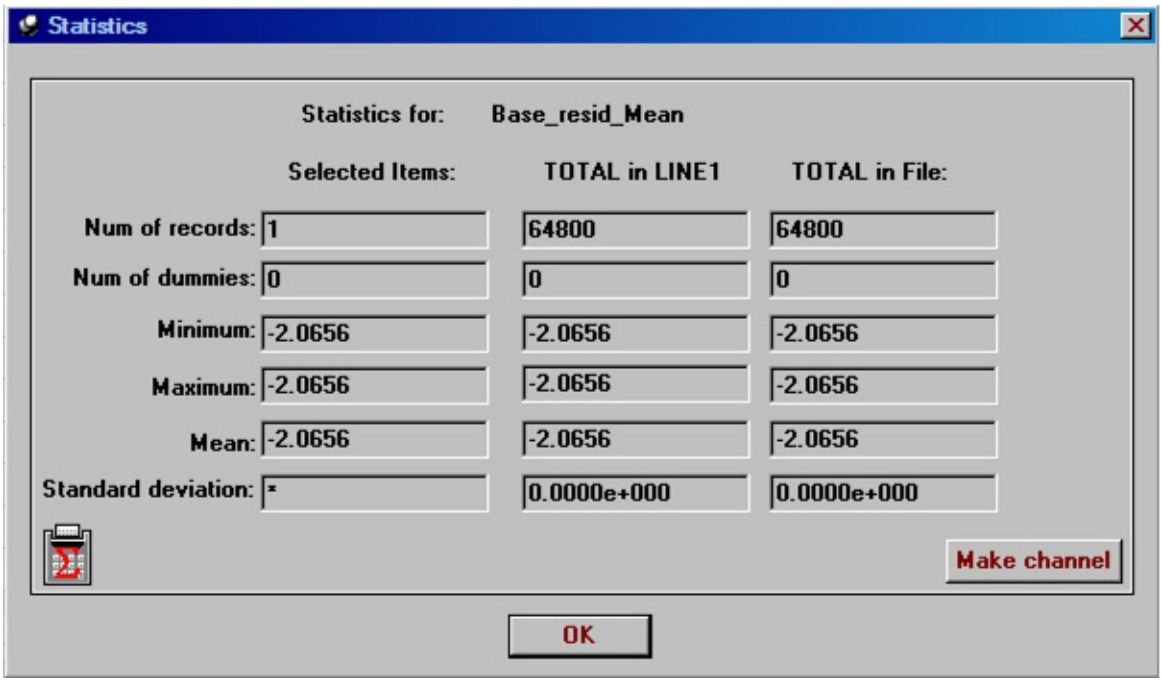
- To create indices for the rows specified in the **Rows** section of the calculator:
  - Click **row** in the **Functions** section of the calculator if you want the indices to reset with each new line.
  - Click **frow** if you want the indices to continue throughout all of the lines.
- Click **Apply** and **Exit** to close the dialog and see the results.
- The **random** button will generate a random number between 0 and 1.

To view the channel statistics

- Select any group of cells in a given column.
- Right-click on this group or on the column header and select **Statistics** from the popup menu to appear.

  OR

  Click the  button in second row of the QC toolbar.



To create a channel for a statistics value

- Click the **Make channel** button and select a statistics value, using the radio buttons that appear
- Click the **Save** button and select a name, format and insertion point for the new channel

To delete a single value

- Right-click on the cell containing a value to delete and select **Set Dummy Value** from the popup menu to appear.

  This will replace the value by an asterisk and automatically remove it from further processing.

To delete/restore a point (record, row)

- Select the point/row/record you want to remove.
- Click the **Delete Point(s)** button  on the QC toolbar

OR

- Right-click on the point number in the first column and select **Delete Point(s)** from the popup menu to appear.

  The whole row will get highlighted in pink, and the  sign will appear in the first column before the point number.

- To restore a point, right-click on this sign and select **Restore Point(s)** from the popup menu to appear.

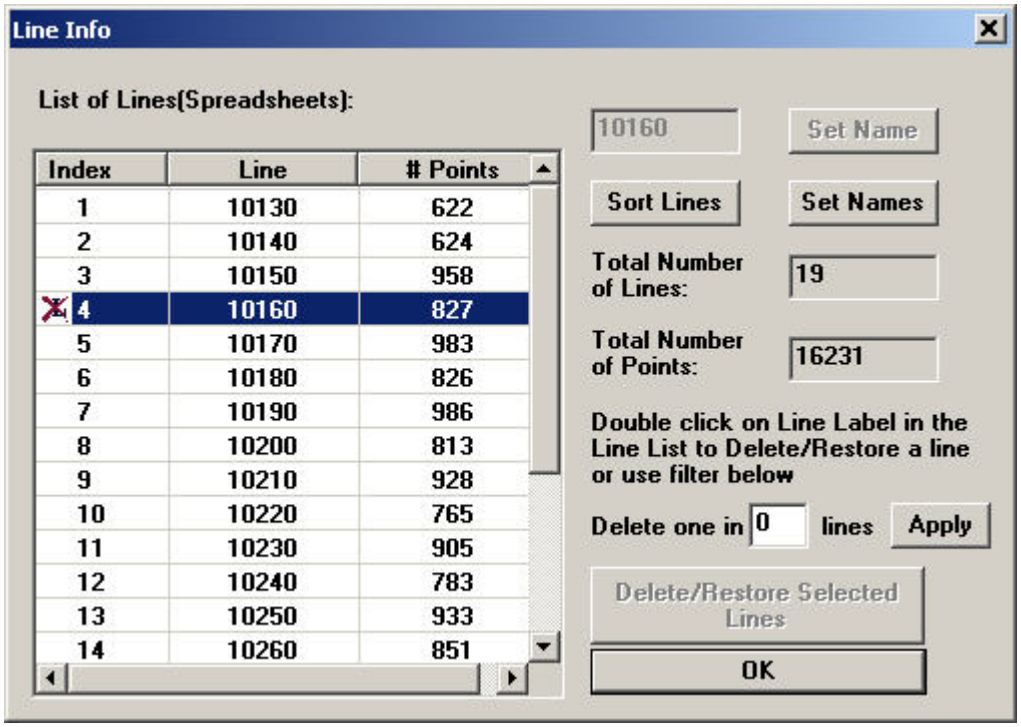
  ***Note**. You cannot restore a point after you have closed your file and then opened it again.*

To delete/restore a line (a table/spreadsheet)

Click the **Delete Line** button on the QC toolbar. The line you are currently in is removed from the dropdown **List of Lines** on the main QC toolbar, whereas your spreadsheet automatically switches to the next line.

However you can always restore the line you deleted:

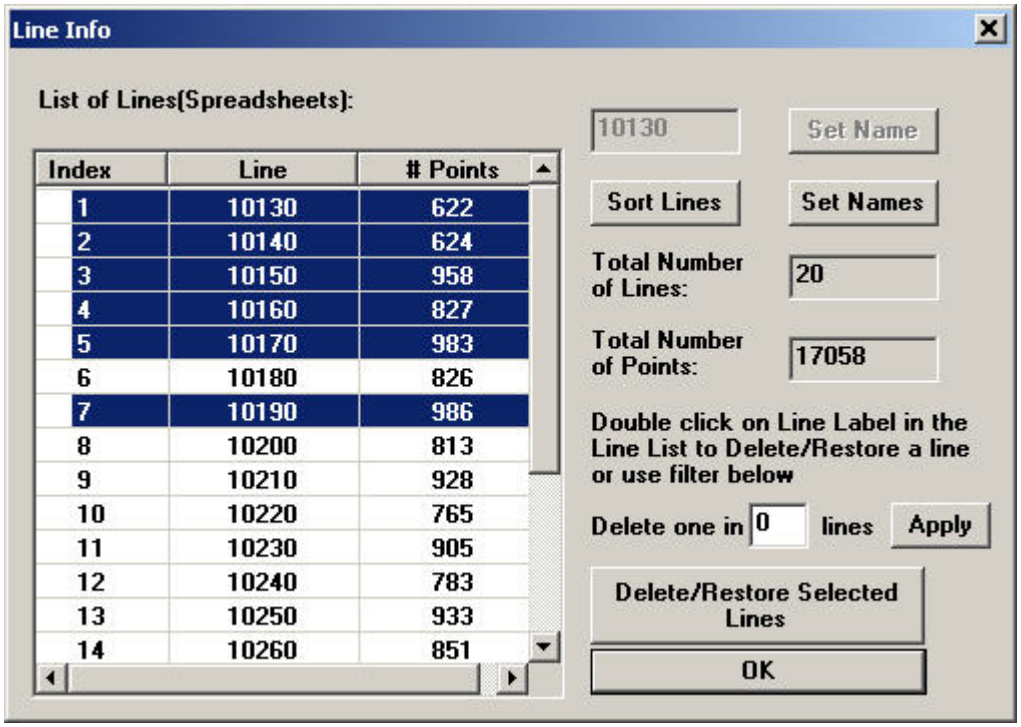
- Click the **Line Info** button on the QC toolbar to open the window below:

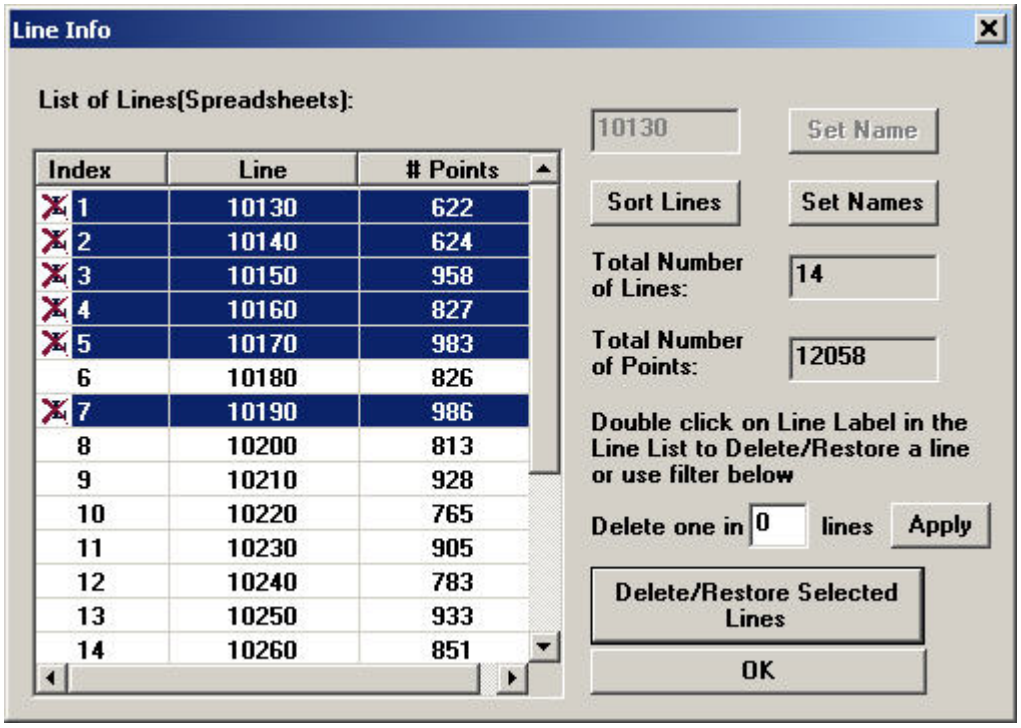


In the **List of Lines**, you can see the sign before the line you deleted.

- Double-click this sign to restore the line.

Use Shift or Ctrl key to select lines. Delete/Restore Selected Lines is enabled automatically. Click the button





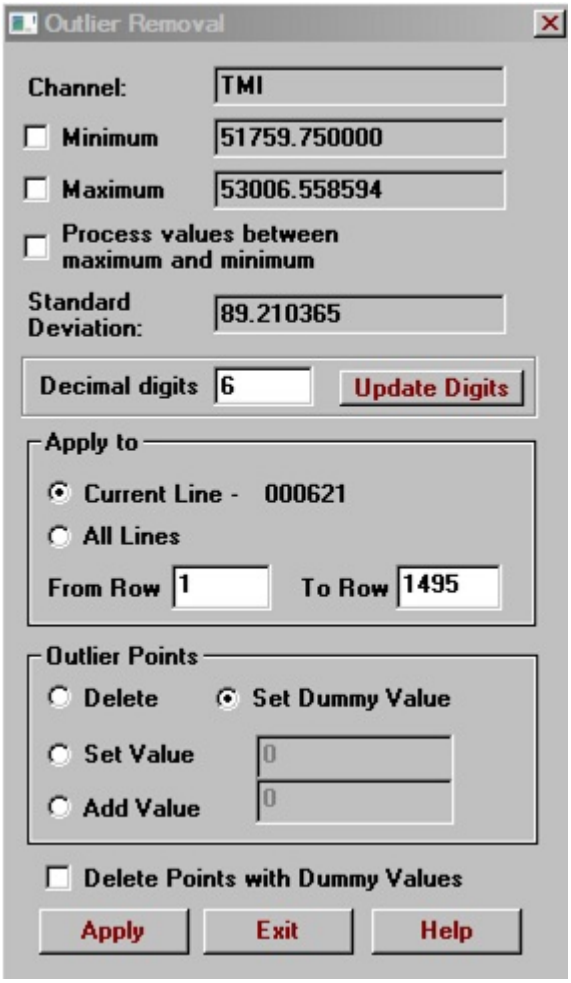
The values in the **Total Number of Lines** and **Total Number of Points** boxes in the right part of the dialog update accordingly to reflect the deletion or restoration of the line(s).

***Note**. You can also delete a line right in the **Line Info** dialog. Double-click on the line you want to delete, and the* *sign will appear to indicate that the line has been removed.*

Click **Set Names** to generate line names for each line by entering a **Name Root** and index preferences. For example, a Name Root of "L", starting index of 0 and increment of 100 would result in line names L0, L100, L200, etc.

To remove outliers

- Right-click on the header of the channel and select **Processing** then **Outlier Removal** from the popup menu to appear. The following window opens:



- The **Channel** field contains the name of the channel you selected.
- Check the **Minimum** and **Maximum** boxes and type in the limits. In this case, the processing will apply to values either above the maximum or below the minimum.
- You may specify the minimum and maximum as limits for a range of values to be deleted by selecting **Process values between maximum and minimum**
- **Decimal digits** displays the number significant digits that will be displayed after the decimal point for **Minimum**, **Maximum**, and **Standard Deviation**.

In the **Apply to** section, select **Current Line** to remove outliers only from the line you are currently working with and **All Lines** to remove outliers from all available lines. You may also specify a range of rows. The default values are for all rows.

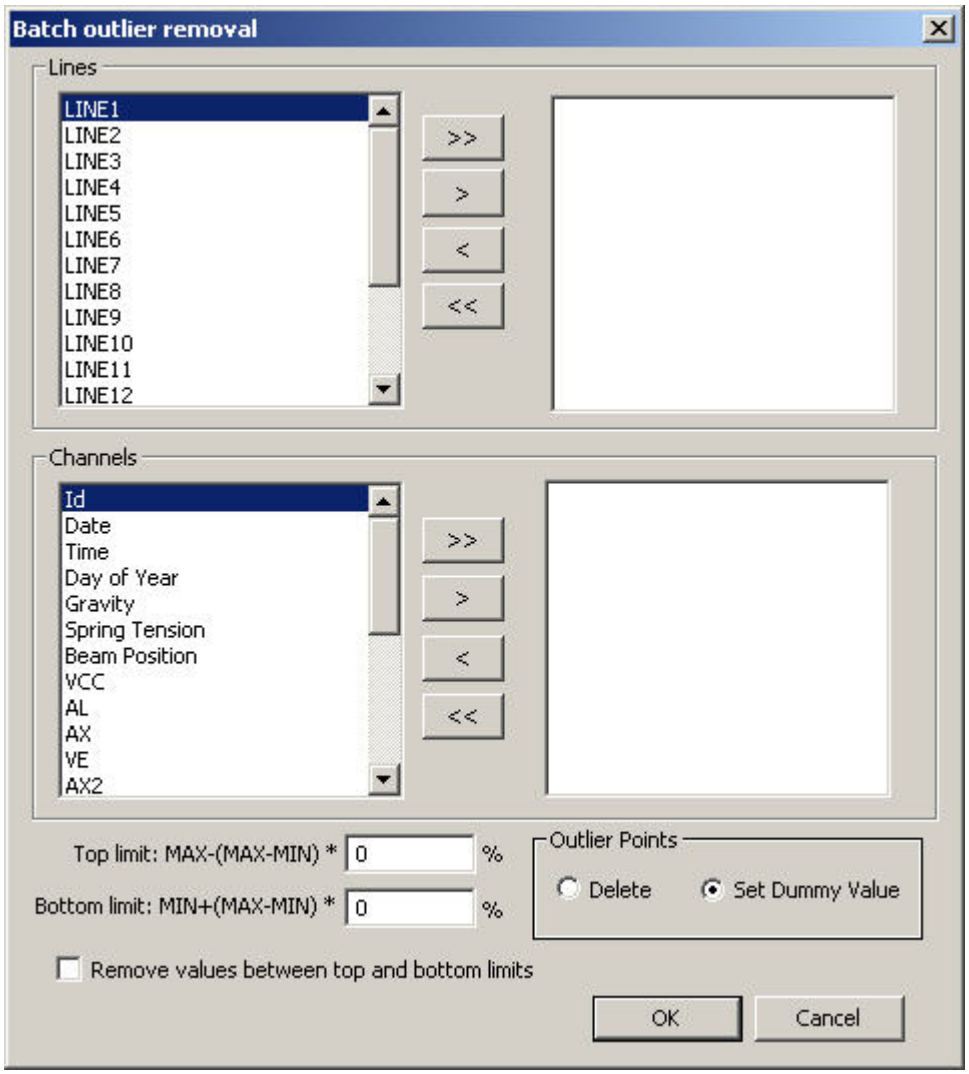
- In the **Outlier Points** section, select **Delete** to remove the whole rows containing outlier points. The rows highlighted in pink mean that the respective points have been deleted.
- Select **Set Dummy Value** to replace all the outliers in the column with asterisks and thus remove them from processing. If later you want to delete the whole rows containing the dummy values, check the **Delete Points with Dummy Values** box or from the main menu select **Processing/Delete Dummies**. The rows highlighted in pink mean that the respective points have been deleted.
- Select **Set Value** to replace all the outliers with a specified value.
- Select **Add Value** to add to the outliers

***Note**. To restore the points, select them in the record number column (**N**), right-click, and choose **Restore Point(s)** from the popup menu to appear.*

- Click **Apply**.

Batch outlier removal

Right-click on the header of a channel and select **Processing** then **Batch Outlier Removal** from the popup menu. The following window opens:



You may select any number of lines or channels from the respective boxes.

>> selects all lines or channels for processing

 selects the currently highlighted items for processing

 removes the currently highlighted items from processing

 removes all lines or channels from processing

- Specify the top and bottom limits according to percentage of range from minimum to maximum. The minimum and maximum values are calculated separately for each channel.
- Check the box labelled **Remove values between top and bottom limits** if you would like to define an outlier as between the top and bottom limits instead of outside the limits
- In the **Outlier Points** section, select **Delete** to remove the whole rows containing outlier points. The rows highlighted in mauve mean that the respective points have been deleted.
- Select **Set Dummy Value** to replace all the outliers in the channel with asterisks and thus remove them from processing. If later you want to delete the whole rows containing the dummy values, from the main menu select **Processing/Delete Dummies**. The rows highlighted in mauve mean that the respective points have been deleted.

***Note**. To restore the points, select them in the record number column (**N**), right-click, and choose **Restore Point(s)** from the popup menu.*

To cut a line into two or more parts

This operation may be performed from the spreadsheet, the plotter or the grid

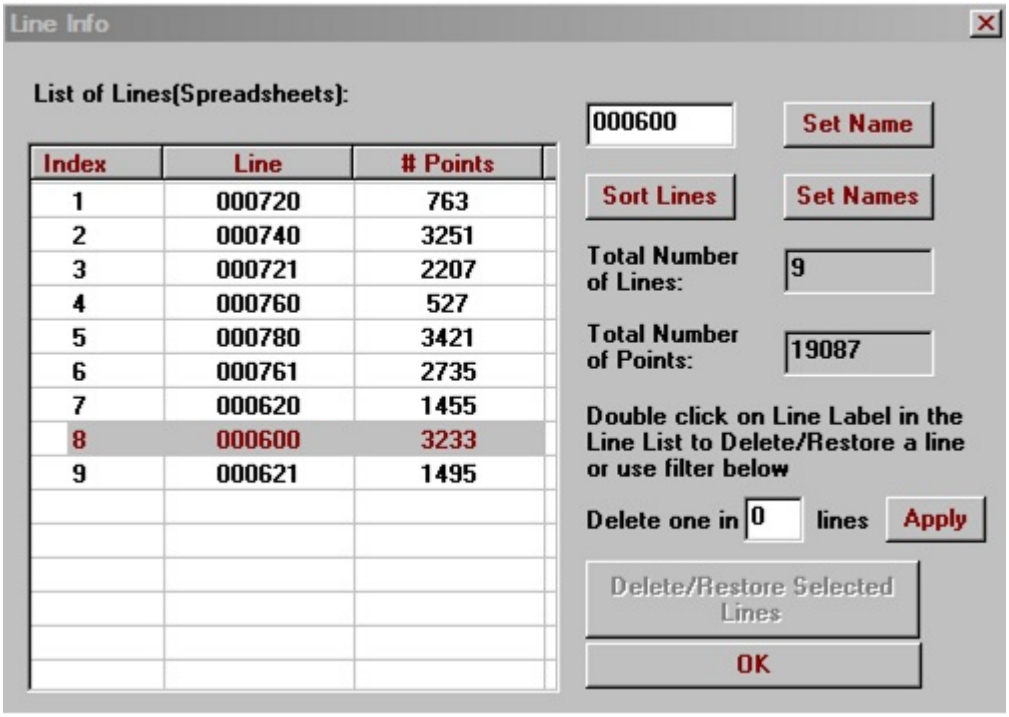
- Select the point where you want to cut your line into two portions.
- Click the **Cut Line** button  on the QC toolbar.

  In the dropdown **List of Lines**, both portions retain the original number of the line, with the second portion also having the extension “_1”. Normally you will rename the new lines as desired.

- Repeat these steps to make as many cuts as desired.

To rename a line

- Click the **Line Info** button  . The respective dialog appears:



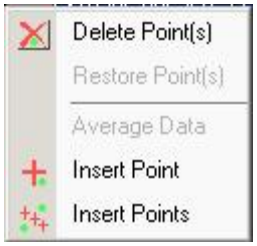
- In this dialog, select the line to rename from the **List of Lines**.

- Type a new name in the box to the right of the **List of Lines** and click the **Set Name** button.

How to Insert data points to QCTool

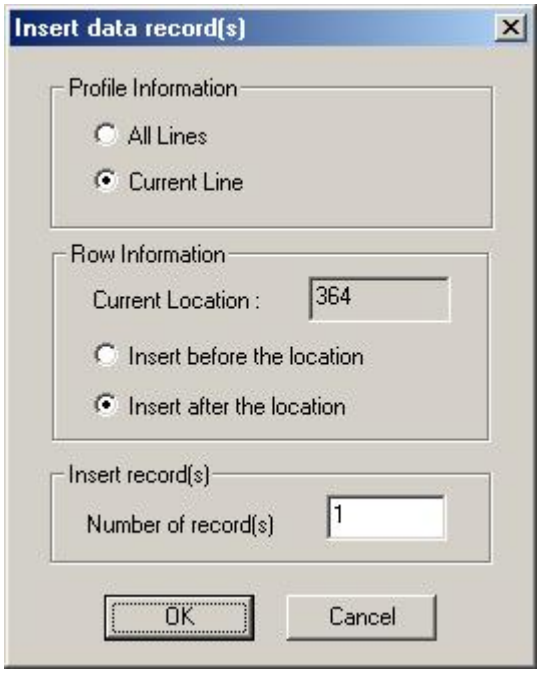
Insert a Point

Click the  button on the QCTool toolbar which will insert a new record/point at the point selected in the spreadsheet
or right click in the N column of the QC spreadsheet.



The menu provides means to add a single record or multiple records either above or below the selected location.

Click the  button on QCTool toolbar or right click in the N column of the QC spreadsheet and select the same icon.
The following interface appears:

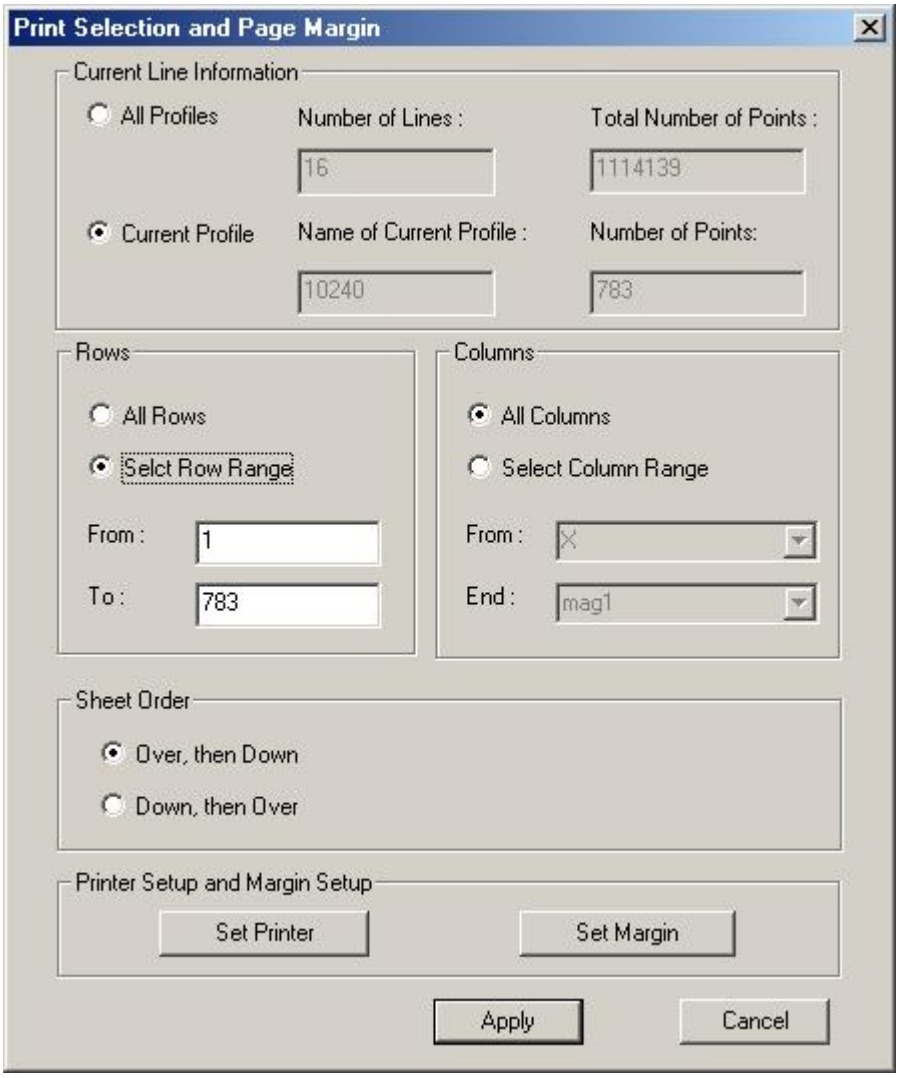


- Select all lines or current line
- Choose the insert position
- Input a number of insert records
- Click OK to insert new rows

Print Selected data

Use this command to print only a portion of your data.

Click File -> Print Selection Area. The following window appears.



- Specify in the **Current Line Information** whether the data for all profiles should be printed or only the data for the current profile.
- In the **Rows** and **Columns** sections, the range of rows and columns to be printed can be specified. All rows and columns are selected by default.

- In the **Sheet Order** section, select **Over, then Down** to have the next printed page to continue from the last cell in the current row. Select **Down, then Over** to have the next printed page continue from the last cell in the current column.

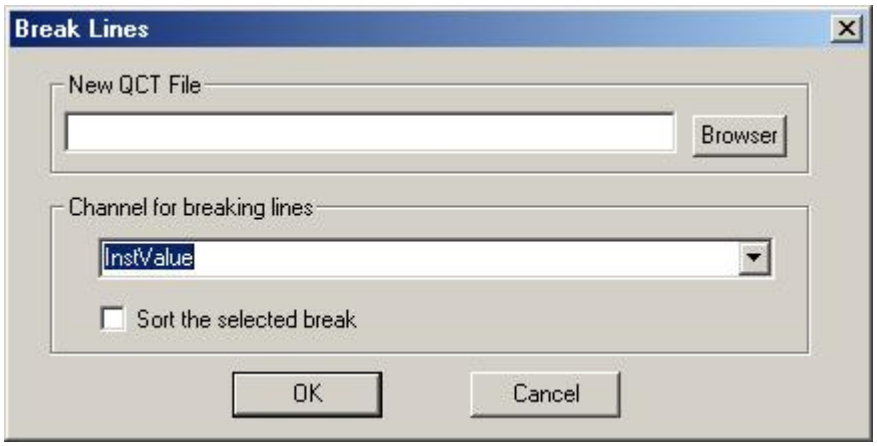
- The **Set Printer** button brings up the the printer properties window.

- Set the left, right, top and bottom margins with the **Set Margins** button.

- Click Apply then select File->Print Preview or File->Print to see the results of the settings you chose.

Break Lines

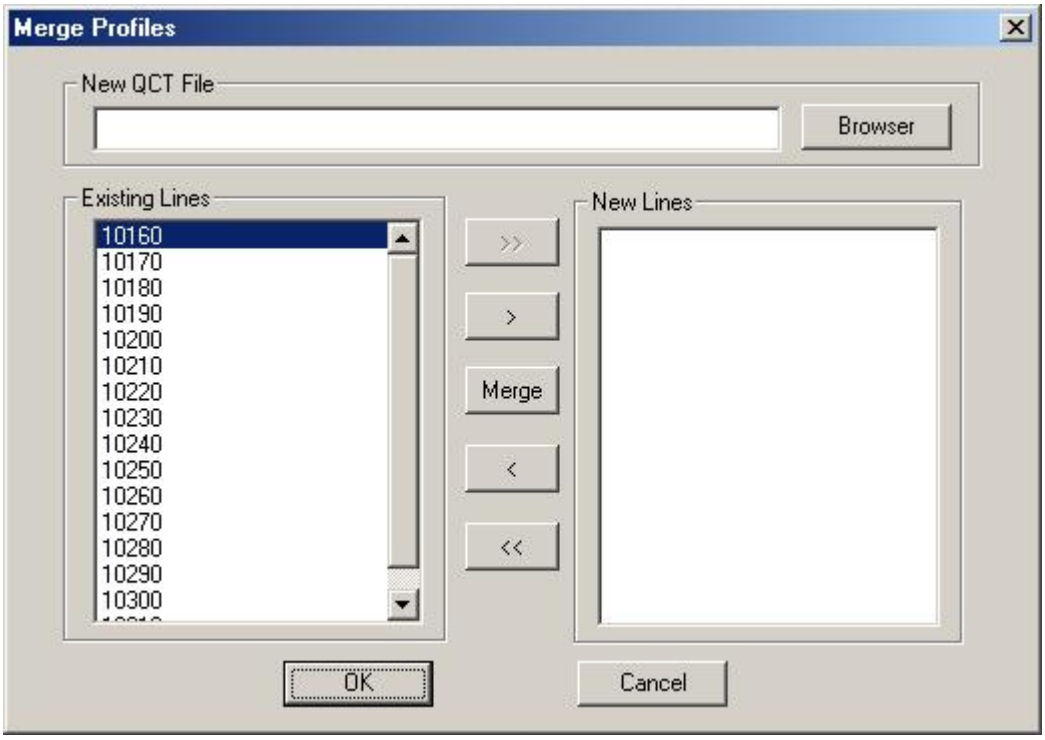
Click the Tools -> Lines -> Break Lines. The dialog appears.



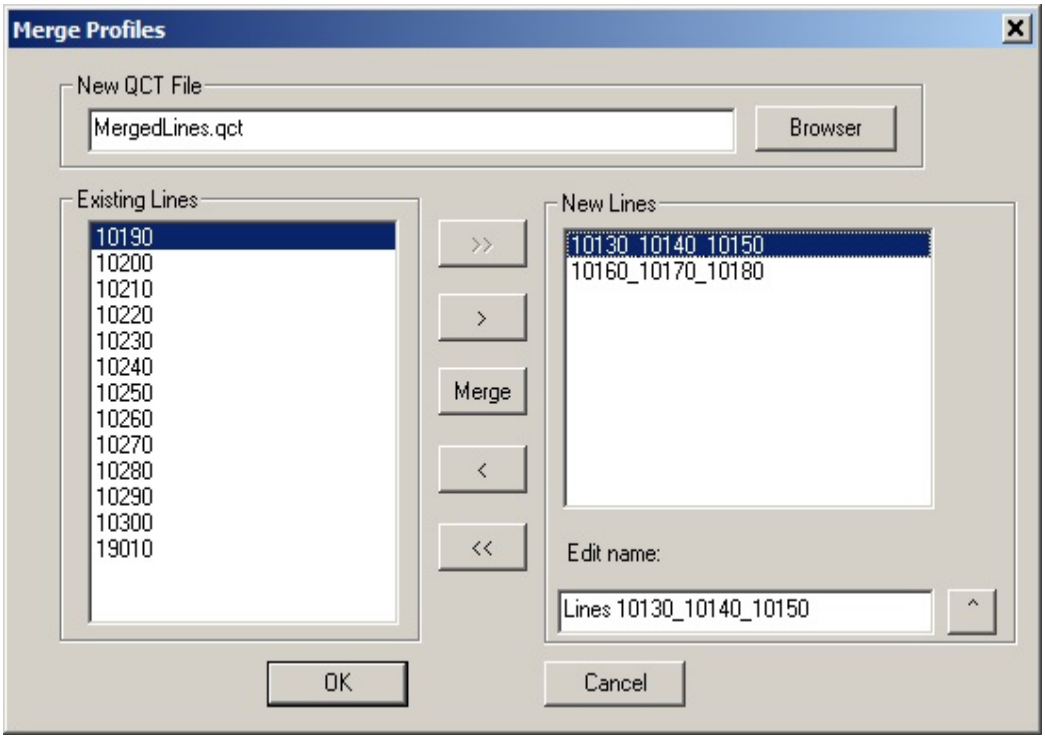
- Input a file name for saving the result
- Select a data channel for breaking lines
- The data will split according to the values in the chosen data channel
- Select the “Sort the selected break” check box if you want to sort the names of the lines

Merge Lines

Click the Tools -> Lines -> Merge Lines. The dialog appears.



- Input a file name for saving the result
- Use shift or ctrl key to select existing lines, click Merge



- Repeat if required
- Click OK

Line names in right listbox allow in-place editing.

Plots and Grids

The QC Plot and QC Grid provide a graphical representation of your imported or processed data. They facilitate preliminary estimation of your survey results and allow fast and efficient detection of erroneous data. For your convenience, you can arrange your window (this option is available from the **Window** menu) to simultaneously view your plot, grid, and the respective table. Selecting any point in the plot or grid will automatically highlight its value in the table.

See also

Working with Plots

Plot Data
Switch Between Curves on the Same Plot
Set the Same Scale for All Channels
Display Plots by Segments
Switch Between Lines
Set a Channel as the X Axis

Adjust the X Axis Settings

Edit Data from the Plot
Zoom In on a Plot Fragment
Customize the Plot Appearance
Remove a Plotted Channel
Remove a Trend

Working with Grids

Grid Data
Change to Another Channel
Edit Data from the Grid
Zoom In and Out
Zoom In on a Grid Fragment
Move Grid Elements

Measure the Distance Between Points

Interpolation
Grid Customization
Calibrate a Map to Use as a Grid Underlay

PRINTING GRIDS AND PLOTS

To plot data

- Click on the header of the channel to be plotted and engage the **Plot Data** button  on the main toolbar

OR

- Right-click on the header and select **Plot Channel** from the popup menu to appear.

The plot is generated automatically. Below the main toolbar, the QCPlot toolbar appears:

If your qctool files contains channels with associated time windows, you may right click on an existing plot and select **Plot Decay** to display a decay plot in a new window. See Set Time Channels to assign time windows to channels.

To switch between curves on the same plot

You can select as many channels as you like and they all will be displayed in different colors on the same plot. But you can view plots of only one line (table/spreadsheet) at a time.

To switch between the curves (data channels):

- Double-click on the curve you want to switch to

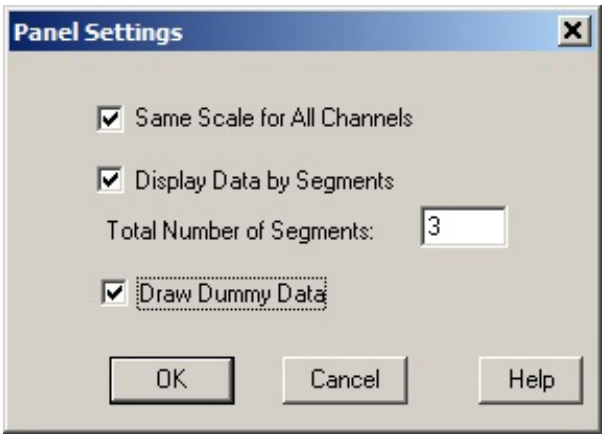
  OR

- Select the respective column in the table.

  The y-axis labels will change their color to that of the curve you switched to; the scale and units of measurements will also change accordingly.

To set the same scale for all channels

- Click the **Panel Settings** button  on the QC Plot toolbar. The respective dialog appears:



- Check **Same Scale for All Channels** and click **OK**. The Y-axis turns black, which means that it has become general for all the curves.

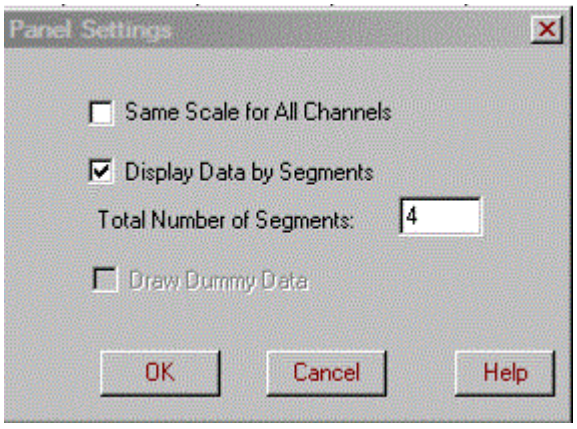
- Note: to speed up loading of graphics, the plot automatically splits in several segments if a large amount of data. If you wish to view all segments, turn off the multiple segment display.

- De-select this option in the **Panel Settings** dialog to return to the initial view.

To display plots by segments

If your line has too many data points, it is often more convenient to plot it by segments.

- Click the **Panel Settings** button  on the QC Plot toolbar. The respective dialog appears:

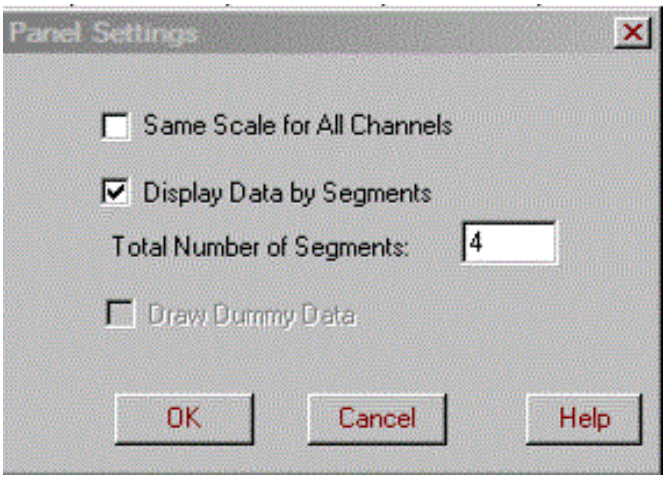


- In this dialog, check the **Display Data by Segments** box. The **Total Number of Segments** box becomes active.
- Specify the number of segments to divide your plot into and click **OK**. The plot will display first segment by default.
- To toggle through the segments, use the forward and back arrow buttons or select the required segment from the **Current Line Segment** dropdown list on the QC Plot toolbar:  .

To display dummy values

Dummy values indicated in the spreadsheet by a '*' can be displayed in a plot. Any dummy values will be set equal to the maximum value in the plot.

- Click the **Panel Settings** button  on the QC Plot toolbar. The following window appears:



- Check the **Display Dummy Values** box and click OK.
- De-select this option in the **Panel Settings** window to return to the initial view.

To switch between lines/tables/profiles/spreadsheets

- Select a required line in the dropdown **List of Lines** on the main QC toolbar: 

.
-
- The plot automatically switches to the line you selected.
- Or click left/right arrows

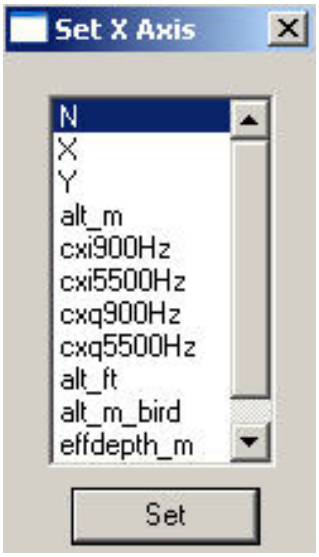
To set a channel as the X-axis

- Click the **Change X Axis Channel** button on the QC Plot toolbar

OR

- Double-click the X-axis label in the bottom right-hand corner of the plot.

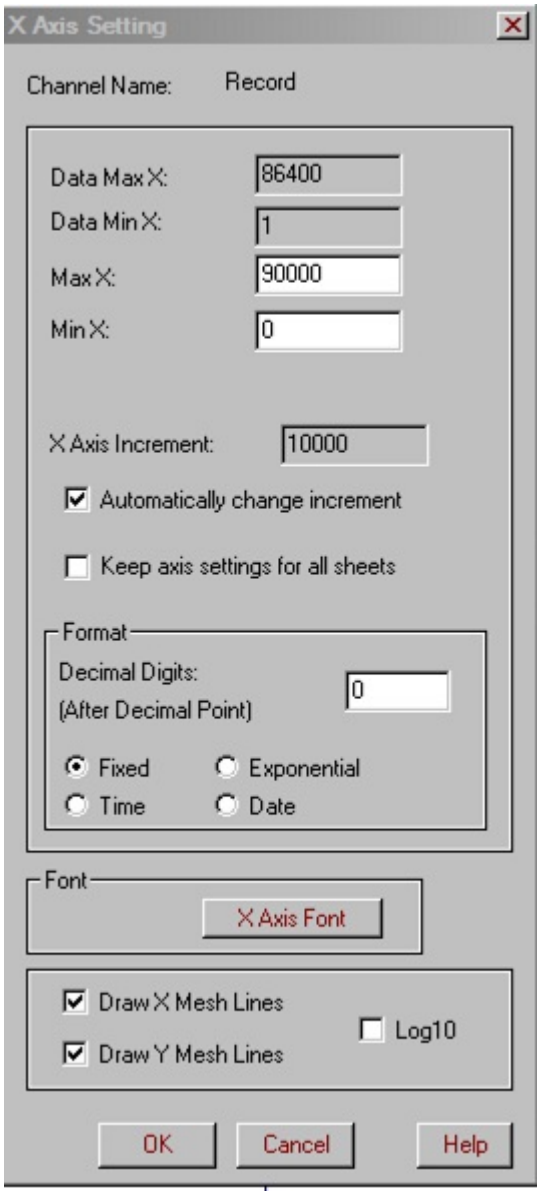
  The **Set X Axis** dialog opens:



- Select a required channel in this dialog and click **Set**. The X-axis will change to the scale of the selected channel. Its label will be replaced by the name of the selected channel.

To adjust the X-axis settings

Select the **Change X-axis Scaling** button on the QC Plot toolbar or double click anywhere on the X-axis. The **X Axis Settings** dialog opens:



- Edit the X-axis settings as needed in the active boxes of the dialog. These settings include the maximum and minimum X values and the X-axis increment.

  If you diminish **Max X**, the plot will go beyond the boundaries of the window and you will see it only up to the point specified as **Max**; in this case, use the scroll bar in the

bottom of the window to scroll forward and view the entire plot.

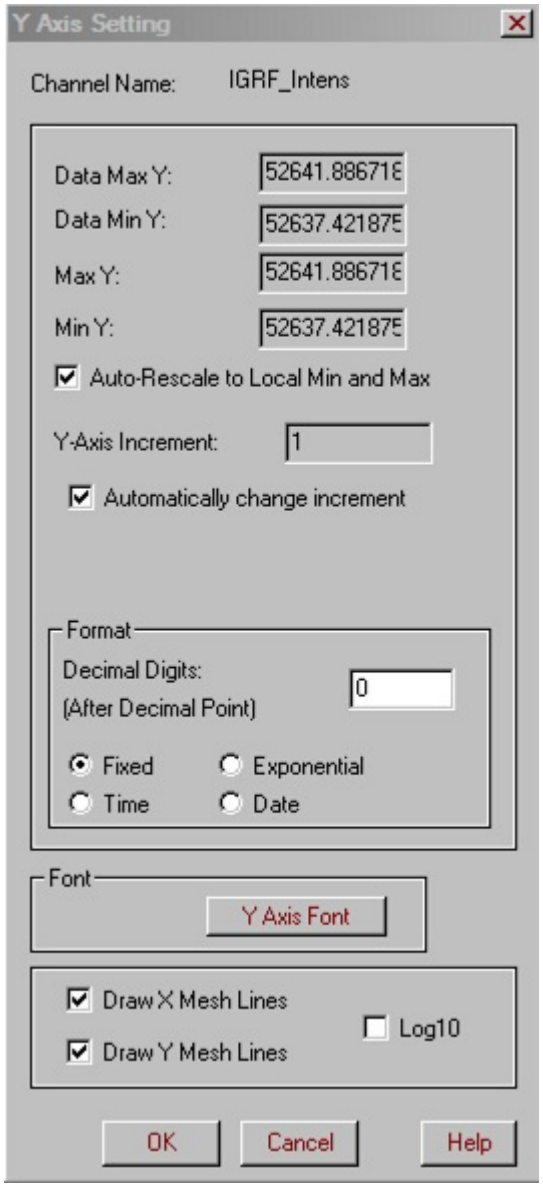
Check **Automatically change increment**, the value of X-Axis Increment will be calculated automatically and cannot be edited. Otherwise, the value of X-Axis Increment is fixed

Select **Keep axis settings for all profiles** and the settings will not change when a new profile is displayed

- Specify the format of the axis labels in the **Format** section.
- Click **X Axis Font** to select the font style and size of the X Axis labels.
- Select **Draw X Mesh Lines** to display vertical dotted lines for the x axis. A similar checkbox is available for the y axis.

To adjust the Y-axis settings

Select the **Change Y-axis Scaling** button  on the QC Plot toolbar or double click anywhere on the Y-axis. The **Y Axis Settings** dialog opens:



- Edit the Y-axis settings as needed in the active boxes of the dialog. These settings include the maximum and minimum Y values and the Y-axis increment.

  If you diminish **Max Y**, the plot will go beyond the boundaries of the window and you will see it only up to the point specified as **Max**; in this case, use the scroll bar in the

bottom of the window to scroll forward and view the entire plot.

Check **Automatically change increment**, the value of Y-Axis Increment will be calculated automatically and cannot be edited. Otherwise, the value of Y-Axis Increment is fixed

- Specify the format of the axis labels in the **Format** section.
- Click **Y Axis Font** to select the font style and size of the Y Axis labels.
- Select **Draw X Mesh Lines** to display vertical dotted lines for the x axis. A similar checkbox is available for the y axis.

To set automatic Y axis scaling

- Click the **Y Axis Rescale** button  on the QC Plot toolbar
- The y axis limits will be adjusted to match the visible minimum and maximum values of the current channel.

To edit data right in the plot

If any error, your plot is distorted.

- Click on the erroneous point.
- To delete this point, click the **Delete Point(s)** button  on the main QC toolbar. Repeat it until your plot takes a normal shape.
- To remove the whole line, click the **Delete Line** button  . Your display will automatically switch to the next line.
- To split a line into parts, click the **Cut Line** button  .

As a result, your line will be cut into 2 portions; with the first to be displayed by default. To display its second portion, select it from the dropdown **List of Lines** on the main QC toolbar: 

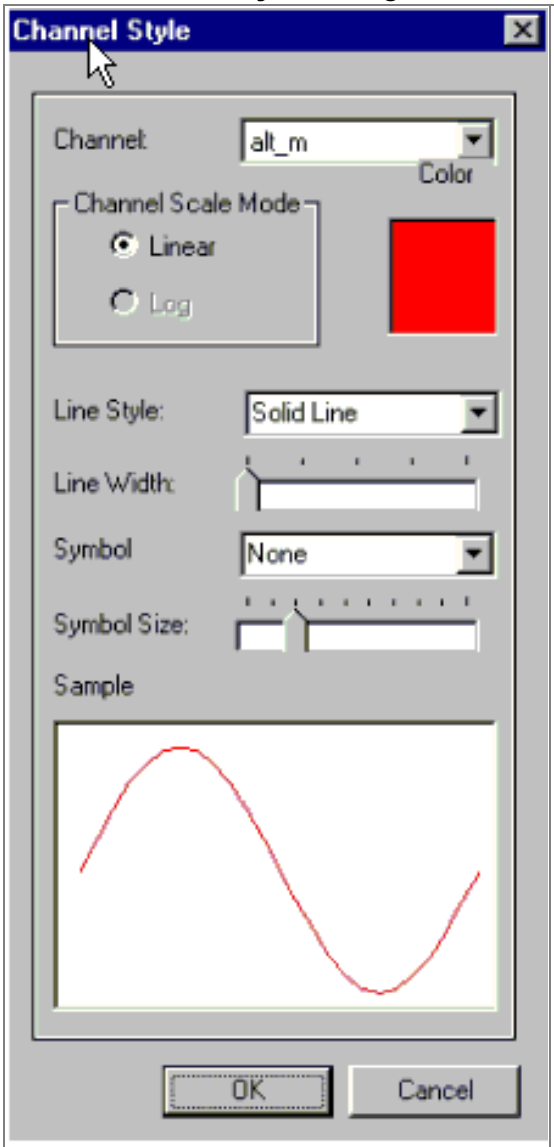
. It will have the same number as the first one and the extension “_1”..

To zoom in on a plot fragment

- Click the **Zooming** button  on the QC Plot toolbar.
- Click and drag to select the fragment you want to magnify.
- To return to the initial view, click the **Home View** button  on the QC Plot toolbar.

To customize the plot appearance

- Click the **Change Plot Style** button on the QC Plot toolbar to bring up the **Channel Style** dialog:



- Select the channel you want to change the appearance of from the **Channel** dropdown list.
- Select **Linear** or **Log** in the **Channel Scale Mode** section.
- Click the coloured square on the right to adjust the color using the standard Windows-style palette.
- Select the line style from the respective dropdown list.
- The **Line Width** option is applicable only to **Solid Line**. Other styles do not show on the plot unless the minimum line width is selected.
- Choose a required symbol from the respective list and change its size using the slider.

  This functionality is especially useful when you need to view the changes you are making right in the plot. For example, increasing the symbol size allows you to pinpoint deleted points.
- In the **Sample** field below, you can simultaneously view an example of your changes.

To remove a plotted channel

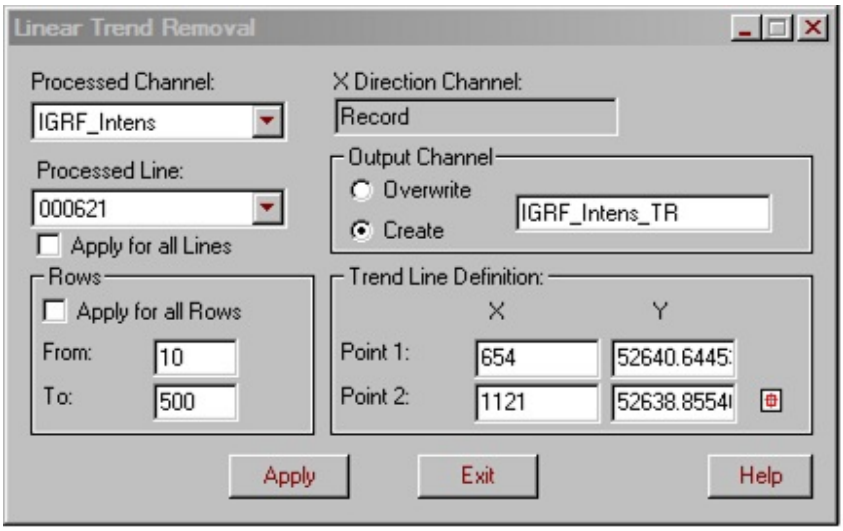
- In the table, right-click on the header of the channel you want to remove from your plot

- and select the respective command from the popup menu
- The channel will be removed from the plot.

To remove a trend

Your data may often be influenced by various factors and as a result contain trends distorting its actual appearance. If it is the case:

- Click the **Linear Trend Removal** button on the QC Plot toolbar to bring up the respective dialog:



  The **Processed Channel** and **Processed Line** lists show the channel and line you are currently working with. To switch to another channel and line, select them from these two lists.

- To remove a trend from all lines of data, check the **Apply to all Lines** box.

- To remove a trend from the whole line, check the **Apply to all Rows** box in the **Rows** section of the dialog.

- To remove a trend from a certain portion of the line, specify the first and the last point of this portion in the **From** and **To** boxes. You can either type in the values or do the following:

1. Click in the **From** box. The sign appears to the right, indicating that you are about to insert a new value in this box (you will always find this sign next to the boxes you are working with in the **Rows** or **Trend Line Definition** sections of the dialog).

2. Click on the point of your curve which you want to be the starting point of trend removal. The X coordinate of this point appears in the **From** box.

3. Click in the **To** box and then on the point of your curve which you want to be the ending point of trend removal. The X coordinate of this point appears in the **To** box.

- To specify trend direction, enter the X and Y coordinates of the starting and ending points of the trend in the **Trend Line Definition** section. You can either type in these coordinates or insert them automatically by clicking on the respective points on the plot.

- Since the result will appear not only in the plotter but also in the respective spreadsheet, select between the **Overwrite** and **Create** options in the **Output Channel** section of the dialog. If you select **Create**, a new column will be added to your spreadsheet containing the results of trend removal. If you choose the **Overwrite** option, the box to the right will turn into a dropdown list, offering you to select the channel to be replaced with a new one containing these results.

- Click **Apply**.

To save a plot

Click the **Save As** button on the QC Plot toolbar or right click on the plot and select **Raster to File** and follow the standard Save operations to create your file in a standard raster image format.

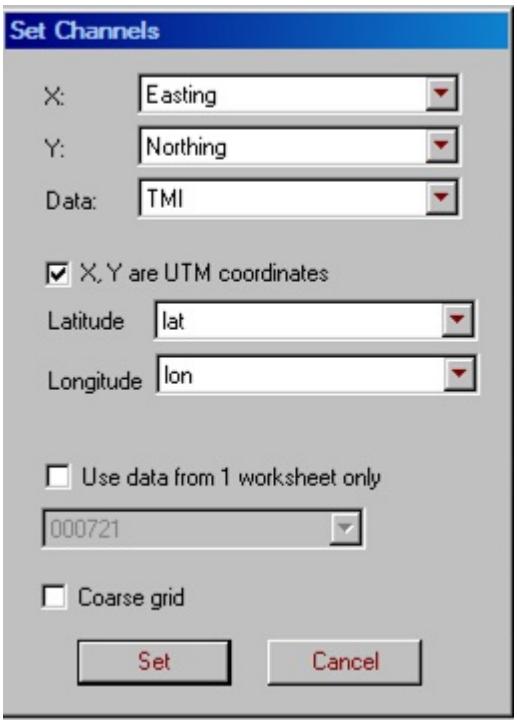
To grid or map data

Mapping data displays all data points with a colour associated with the values much like contours. This allows you to select a point and it will be shown on the GRID tool with relevant information and values.

- Click on the header of the channel to be plotted and select the QC Grid button  on the QC toolbar

OR

- Right-click on the header and select **Map Channel** from the popup menu. The **Set Channel** dialog appears:



- Select a channel to use as the X coordinate from the **X** dropdown list.

- Select a channel to use as the Y coordinate from the **Y** dropdown list.

- If required, reselect a channel to grid from the **Data** dropdown list. The channel selected by default is the one you specified in the table.

- Check the **X and Y - UTM coordinates** box if you would like to display longitude and latitude lines over your grid.

- If the data set is very large then displaying every data point can be time consuming and so if you select “Coarse grid” , a compressed version of the data will be displayed.

- Click **Set**.

Your grid is generated. Each data point in the resulting stack of lines is assigned its own color. To see the value in a given point, click on the point and look its value up in the respective table. To move across the grid, you can also use the arrow keys.

To change to another channel

The generated grid uses the channels you specified in the **Set Channels** dialog (see To Grid Data). To change to another channel:

- Select the  button on the QCGrid toolbar

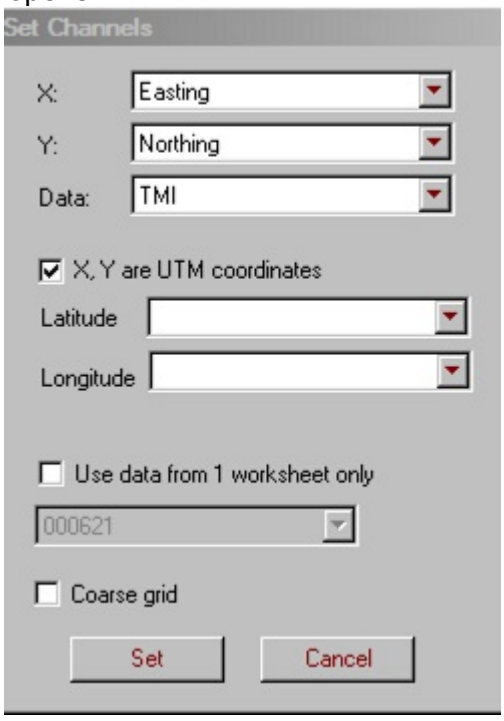
  OR

- Click the button  on the QCGrid toolbar and click the **Change Channel** button 

  OR

- Right click the mouse on the new channel and select **Map channel**

The **Set Channels** window opens:



- Reselect the channels in the **X** and **Y** dropdown lists.

- Reselect the channel to grid in the **Data** dropdown list.

- Click **Set**.

No matter which display you had – prior to or after interpolation, - your new display is not interpolated.

To edit data right in the grid

There may be positional errors or data errors. You may desired to delete some points or divide lines into two or more parts

- Click on the erroneous point and use the **Delete Point(s)** button  to remove it from the grid. Repeat this operation until your grid display becomes normal.

- Use the **Delete Line** button  on the main QC toolbar to remove a whole line.

- Click the **Cut Line** button  to cut a line into 2 or more portions.

  In this case, your change does not show on the grid unless you zoom in on the respective fragment (see To zoom in on a grid fragment). You can also check it up in the **List of Lines** on the main QC toolbar, in the respective table or plot, or in the **Line Info** dialog that opens by clicking the **Line Info** button  .

To zoom in and out

- Use the **Zoom In**  and **Zoom Out**  buttons on the QC Grid toolbar under the main QC toolbar.

To zoom in on a grid fragment

- Click the **Zoom Selected**  button on the toolbar. The cursor will change into a magnifying glass.
- Click and drag to select a fragment to be zoomed.
- To return to the initial scale, select **Home View**  from the same toolbar.

***Note**. To change back to the arrow manipulator, select it from the toolbar*  *.*

To move grid elements

- Select **Move** from the toolbar. The arrow cursor changes to the hand manipulator  .

- Click and drag labels, symbols and other objects to the place you need.

  This functionality is especially useful when you need to customize the appearance of your grid (see the Grid Customization section).

***Note**. To change back to the arrow manipulator, select it from the toolbar*  .

To measure the distance between different points

- Select **Distance** from the toolbar. Your cursor changes to the ruler  .
- Click on the starting point and drag to the ending point.
- Check the result in the Status Bar at the bottom of the window.

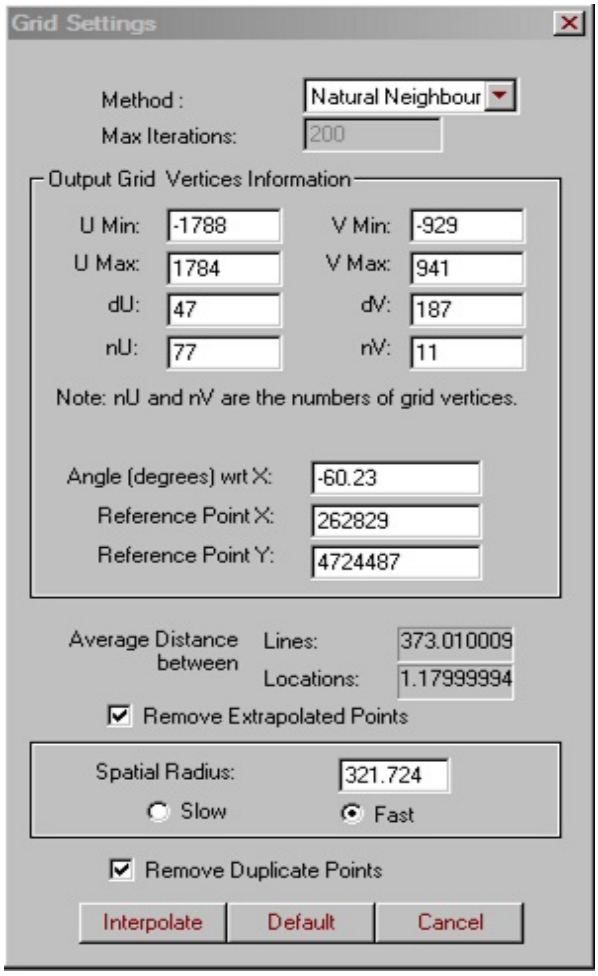
***Note**. After measuring a distance, the cursor automatically changes back to the arrow manipulator.*

To perform interpolation

During the interpolation process, QCTool interpolates data from data points onto a regular (rectangular) grid. You can interpolate the whole grid or its portion. In the latter case, before interpolation, zoom in on a required fragment as described in To zoom in on a grid fragment.

- Click the **Interpolate** button on the QC Grid toolbar or double click on the grid and the **Tools** interface will open. Select **Contour** and then **Interpolation**

  The **Grid Settings** dialog opens:



- Choose the method of interpolation in the **Method** dropdown list.

  By default, the **Natural Neighbour** option is selected. If desired, you can change it to **Minimum Curvature**. This enables the **Max Iterations** box below, for you to type the required number of iterations.

- The software attempts to find the best center for interpolation and a default angle based upon the direction of the survey lines. U,V are the grid vectors within the rotated grid space

- Edit the **U Min**, **U Max**, **V Min** and **V Max** settings as needed.

- Increase or decrease **dU** and **dV** (length of the grid cell side) or **nU** and **nV** (number of points) to provide higher or lower resolution.

- Change the **dU** and **dV** values and click in the **nU** and **nV** fields: the latter is recalculated automatically. You can reach the same effect by changing the **nU** and **nV** values and clicking in the **dU** and **dV** fields.

- Edit the grid angle to examine your display from a different perspective.

  The grid has its own coordinate system (u,v) which is rotated from the coordinate system (x,y) chosen for the 2D display. In the example here, the grid is aligned along the x,y direction (angle = 0) and ‘u’ is parallel to ‘y’.

- Edit, if necessary, the **X** and **Y** reference points in the respective boxes.

- If you are not satisfied with your changes, click the **Default** button to change back to the initial settings.

- Check the **Remove Extrapolated Points** box and set a required spatial radius to restrict the area of interpolation. For example, a spatial radius of 55 means that if there are no data in the radius of 55 m around a given point – a grid cell center, – this cell will be removed from interpolation.

  Note. *The interpolation process interpolates data onto the entire regular rectangular grid. However parts of this grid, which are far away from the data, may be overly extrapolated and thus should be removed for careful viewing.*

- Select between the slow and fast algorithms (slow is more accurate, but fast is almost always sufficient).

- Check the **Remove Duplicate Points** box to provide the respective operation.

- Click **Interpolate**.

The interpolation process can take some time depending on such grid settings as the number of initial points, grid dimensions or the interpolation method.

If you interpolated only a grid fragment:

- After interpolation, select **Home View** from the QC Grid toolbar. You will see the interpolated fragment stand out against the primary grid background.

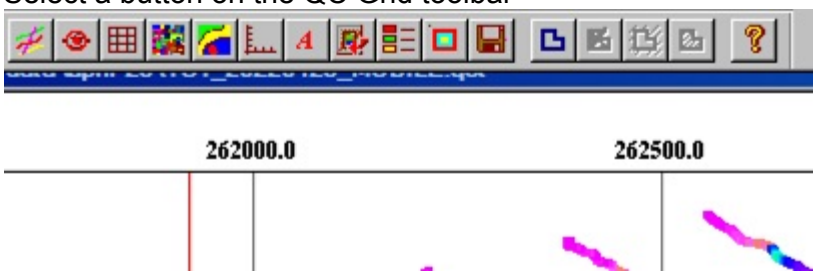
Customization Dialogs and Menus

You can change the grid appearance using two methods:

- Select the **Show Plot Components** dropdown menu  on the QC Grid toolbar and toggle on or off various features

OR

- Select a button on the QC Grid toolbar



- or right click in the grid window and select an item from the popup menu for the advanced customization of your grid appearance.

Related topics:

Display and Customize Lines

Change the Draw Mode

Draw and Customize the Mesh Grid

Draw and Customize Contours

Display and Customize the Coordinate Grid

Display Longitudes and Latitudes over a UTM-Based Grid

Display a Grid Proportionally

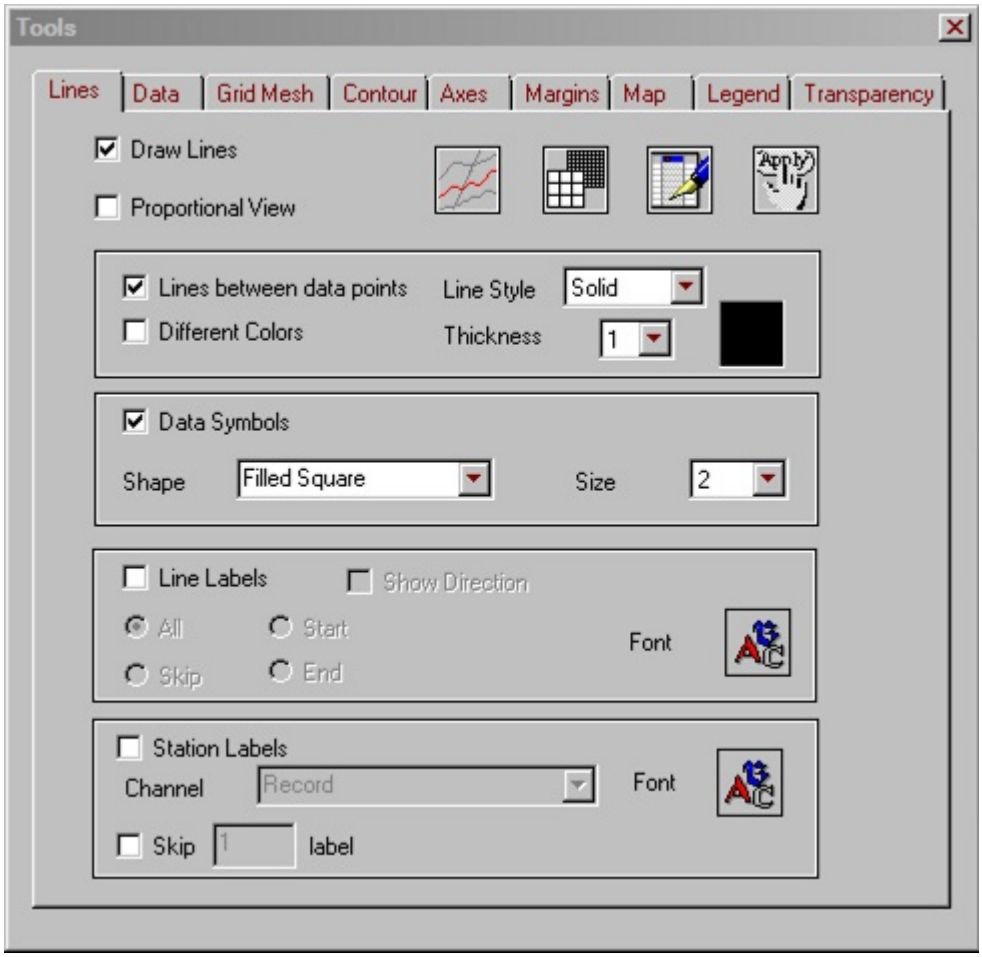
Use a Map Underlay

Customize the Legend and Scale

Customize the Grid Layout

To display and customize lines

Click the  button on the QC Grid toolbar or right click in the grid window and select **Lines**. The following interface appears:



- Check the **Draw** button to switch lines on and off (or select **Show Lines** in the **Show Layers** dropdown menu on the QC Grid toolbar).

- To assign the same colour to all of the lines, leave the **Different Colors** box in the **Lines** section unchecked. Select a required thickness of the lines from the respective dropdown list and click on the color square to the right to bring up the standard color palette.

- To assign different colors to the lines, check the **Different Colors** box. Select a required thickness from the respective dropdown list.

- To change the shape and size of symbols (points) on your grid, check the **Data Symbols** box. The **Data Symbols** section becomes active. Select the required shape and size from the respective dropdown lists.

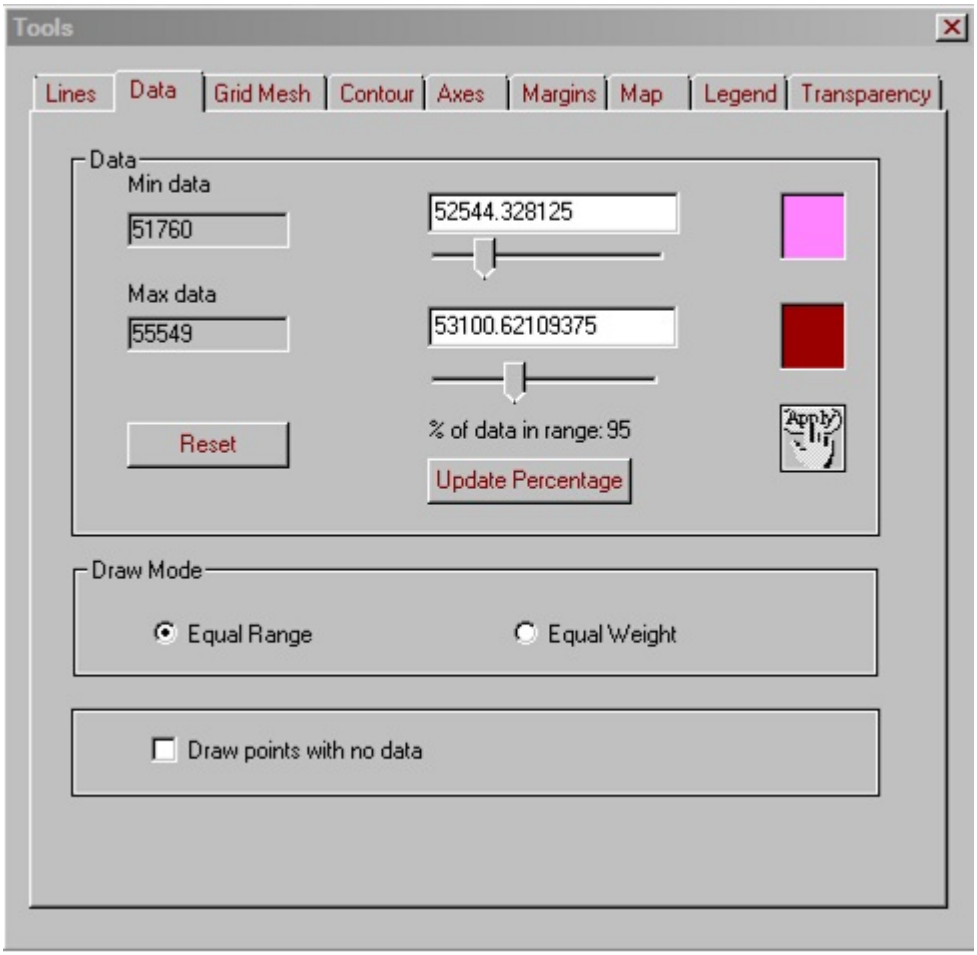
- To display the labels (line numbers), check the **Line Labels** box. Select between the four options below. The **All** button displays all labels, the **Skip** button – every second label, the **Start** and **End** options – the labels at the beginning and at the end of lines.

- To edit the font and style of the labels, click the **Font** button  in the bottom right-hand corner of the dialog.

- To display station labels, select the **Station Labels** box. Select the channel used to obtain a label for each station.

To change the draw mode of your grid

Click the  button on the QC Grid toolbar or right click in the grid window and select **Data**. The following interface appears:



- Select **Equal Range** to assign different colors to equal ranges independently of the number of points in each range or select **Equal Weight** to assign different colors to different ranges covering the same number of points.

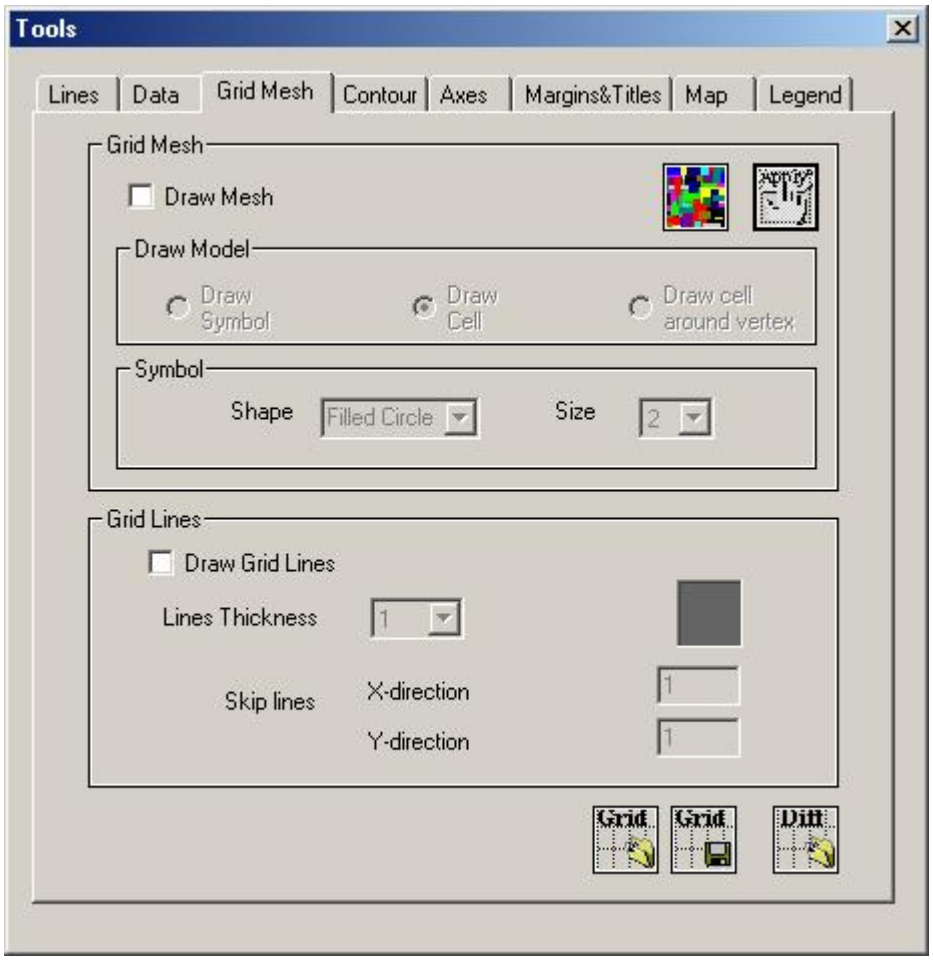
- Specify the range of data to be displayed (75% of data are displayed by default) using the sliders to the right of the **Min Data** and **Max Data** fields that show the absolute minima and maxima of your data. Or, type in these values manually. Click **Apply**.

- Click on the colored squares to the right to open the standard Windows-style palette and to specify colors for the start and end of the data range.

- Select **Draw points with no data** to see points with dummy values

To draw and customize the mesh grid

Click the button  on QC Grid toolbar The following interface appears:



- Check **Draw Mesh** to see your grid cells filled with color (or **Show Interpolated Grid** in the **Show Layers** dropdown menu on the QC Grid toolbar). This functionality is accessible only if your data have been interpolated; if not, a message prompts you to carry out interpolation. Click the **Interpolate** button  and follow the steps as described in To perform interpolation.

  The **Draw Grid Symbols** option displays each grid cell as a set of four data (vertices) and assigns a certain color to each data dependending on its value.

  The **Draw Grid Cells** option calculates the average of the data located in the vertices of a grid cell. The cell is filled with the color assigned to this average value.

The **Draw Cells around Grid Vertices** option displays your grid as a set of cells drawn around each grid vertex and filled with a certain color assigned to the data value in the vertex.

- If you selected the **Draw Symbol** button, select the shape and size of the symbols to be used from the respective dropdown lists in the **Symbol** section of the dialog.

- Select the way for the color to fill the cells (**Draw Symbol**, **Draw Cells**, **Draw Cell Around Vertex**).

- Check the **Grid lines** box to draw the mesh grid. This functionality is available only if your data have been interpolated; if not, a message prompts you to carry out interpolation. Click the **Interpolate** button  and follow the steps as described in To perform interpolation.

- Click the **Load Grid from File** button below to load an already available mesh grid.

- Select the grid line thickness from the respective dropdown list and click on the colored square to the right to specify the color.

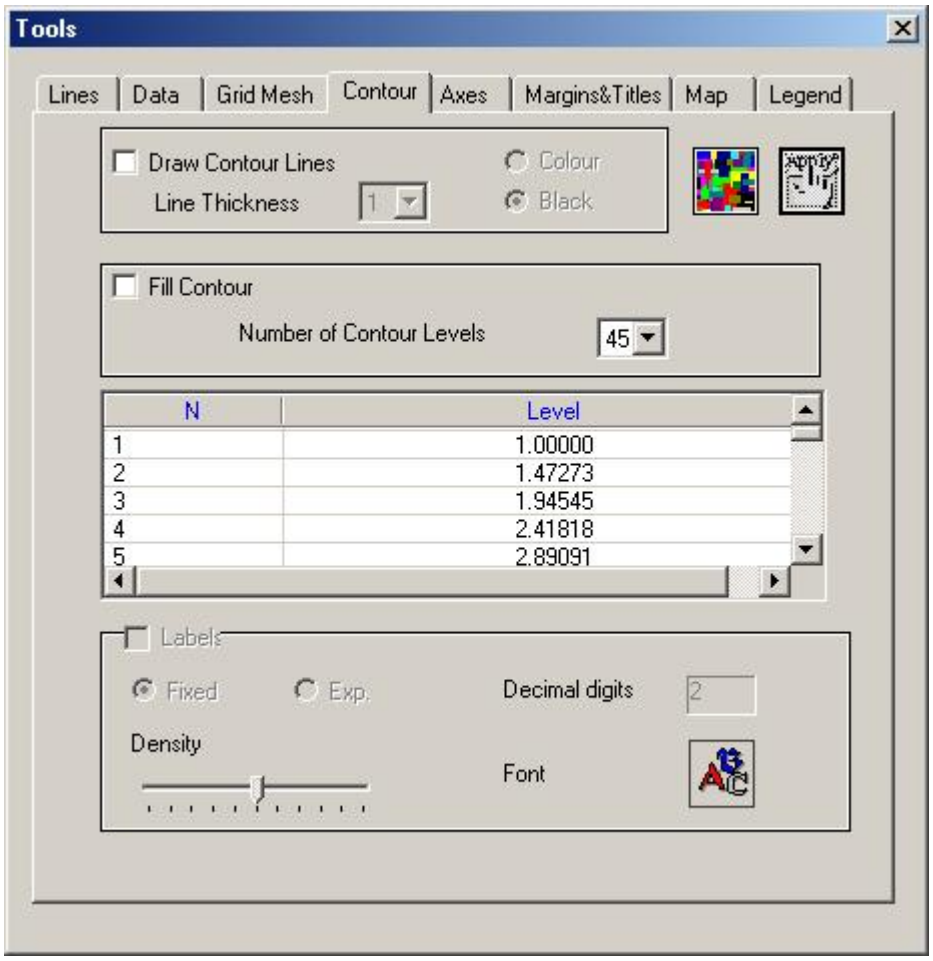
- Change the density of your grid as needed both in the X- and Y-directions in the **Skip Lines** section.

- Save your grid as an egr file to use it later with other data. You may also plot the difference between the currently displayed grid and a grid that you load.

- Resulting interpolated grids my be saved or loaded here. Also, you may create the difference between grids here as well

To draw and customize a contour display

Click the button  on the QC Grid toolbar or the **Contour** tab on the **Tools** menu. The following interface appears:



- Check the **Draw Contour Lines** box to draw contours (=**Show Contour** in the **Show Layers** dropdown list on the QC Grid toolbar). This functionality is accessible only if your data have been interpolated; if not, a message prompts you to carry out interpolation. Click the **Interpolate** button  and follow the steps as described in To perform interpolation.

- Check the **Fill Contour** box to display filled contours.

- Select between the color and black representation of the contour lines and labels.

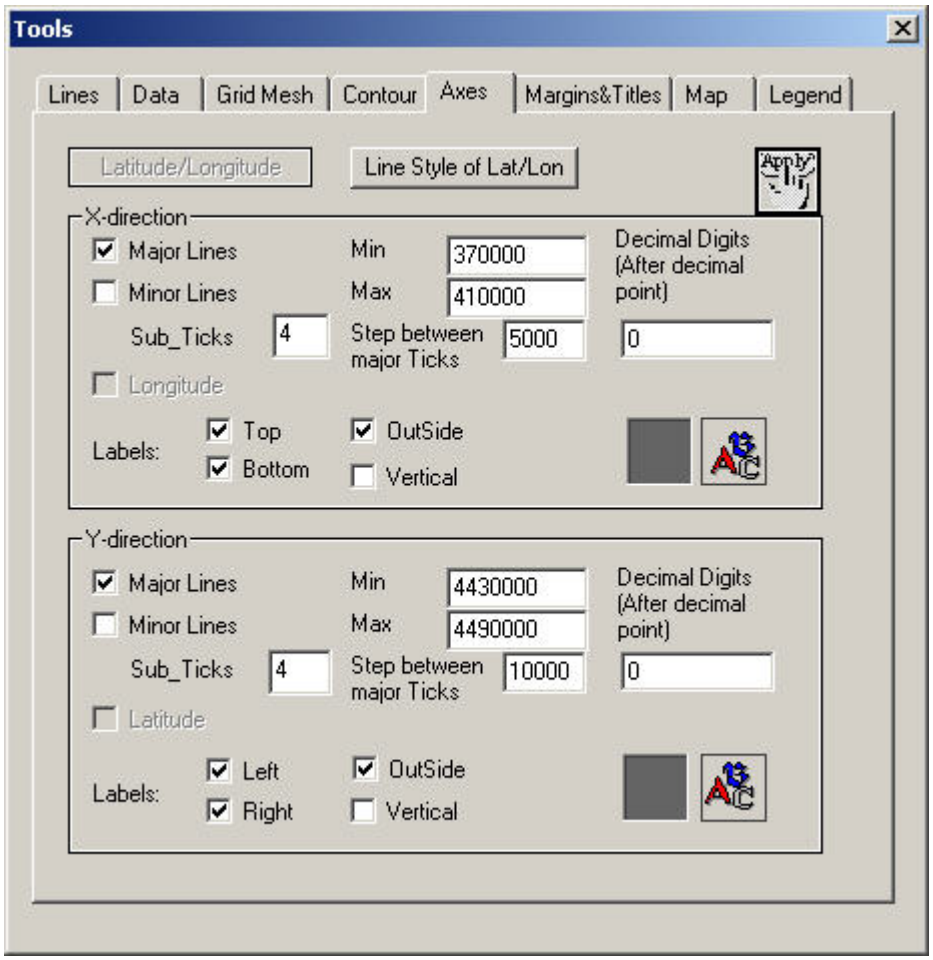
- Choose the number of contour levels and line thickness from the respective dropdown lists.

- Check the **Labels** box to display the contour values and specify their format in the respective section (fixed or exponential, the number of decimal digits).

- Use the **Density** slider to increase or decrease the amount of labels to be displayed and click the  button to adjust their font and style.

To display and customize the coordinate grid

Click the button  on the QCGrid toolbar or the **Axes** tab on the **Tools** gui. The dialog appears.



- Check the **Major Lines** and **Minor Lines** boxes in both the **X-direction** and **Y-direction** sections to show the coordinate grid.
- Adjust the minimum and maximum coordinates in the respective boxes to display the portion of your grid you are primarily interested in.
- Change the step between major ticks in the respective boxes to increase or decrease the density of your coordinate grid.
- Click the color square in the bottom right-hand corner of both sections to edit the color using the standard Windows-style palette.

- Select between the **Top** and **Bottom** positions of the labels in the **X-direction** section and between the **Left** and **Right** positions in the **Y-direction** section.

- Select the mode of outside and vertical positions of the labels in the X-direction section and the Y-direction section

- Input the number of Sub_Ticks of Minor lines in the X-direction section and the Y-direction section

- Click the  button to specify the font and style of the labels in the standard Windows-style **Font** dialog.

- Specify the number of decimal digits to be displayed in the respective box.

- Click the **Line Style of Lat/Lon** to specify colour, font, line style and width for the latitude and longitude display.

Note*.To switch the coordinate grid off, de-select* ***Major*** *and* ***Minor Lines*** *or de-select* ***Show Coordinate Grid*** *from the* ***Change Layer*** *dropdown list on the QC Grid toolbar.*

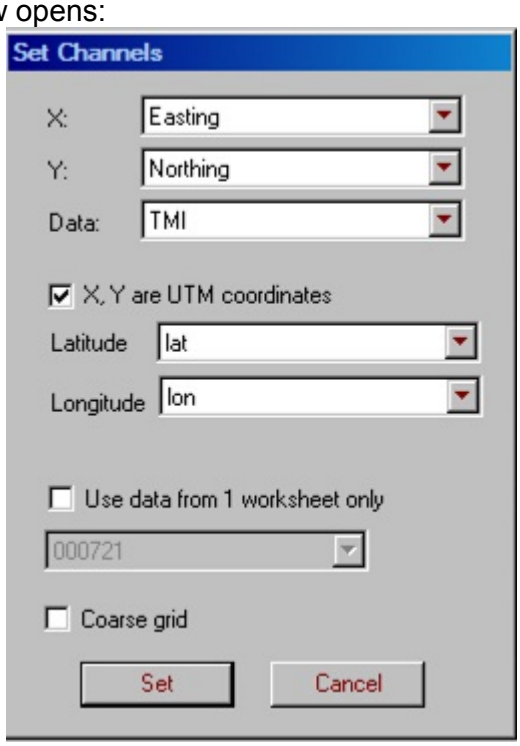
Display latitudes and longitudes over a UTM grid

- Select the  button on the QCGrid toolbar

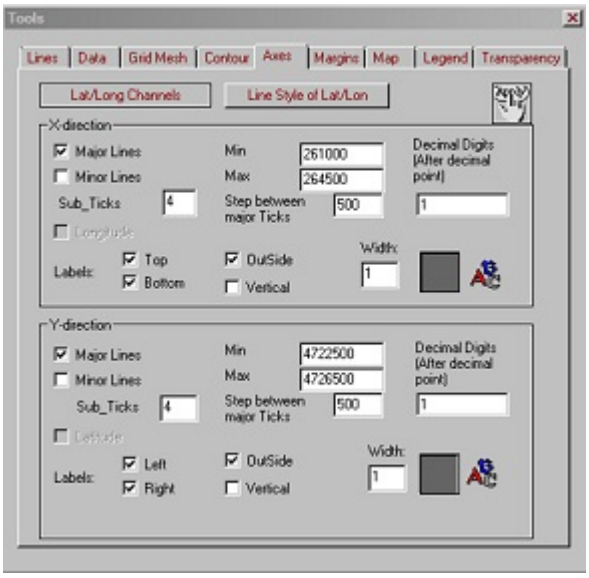
  OR

- Click the button  on the QCGrid toolbar and click the **Change Channel** button 

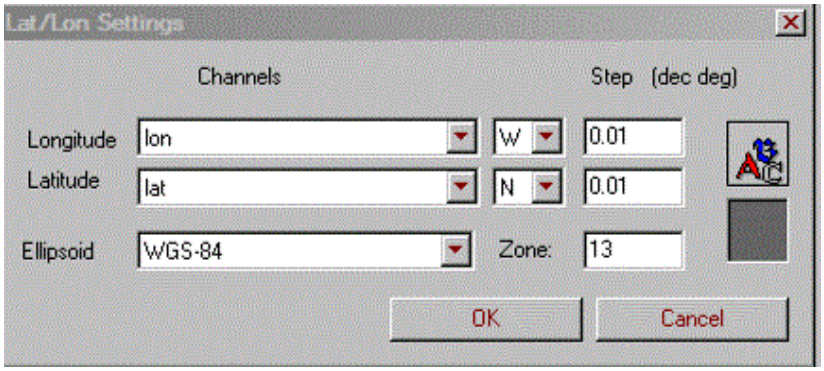
The **Set Channels** window opens:



- Check the **X and Y - UTM Coordinates** box and click **Set**.
- Go to the **Axes** tab of the **Tools** dialog



- and click the **Lat/Long** button. The **Lat/Lon Settings** dialog opens:



In this dialog:

- Select between the W and E, N and S directions.
- Specify the step in the respective boxes to the right.
- Select a required ellipsoid from the respective list in the bottom of the dialog.
- Click **OK**.

Back in the **Axes** tab, check the **Longitude** box in the **X-direction** section and the **Latitude** box in the **Y-direction** section. Longitudes and latitudes will appear over your grid.

To display your grid proportionally

- Go to the **Lines** tab of the **Tools** dialog.
- Check the **Proportional View** button. The sides of your grid will become of equal length.

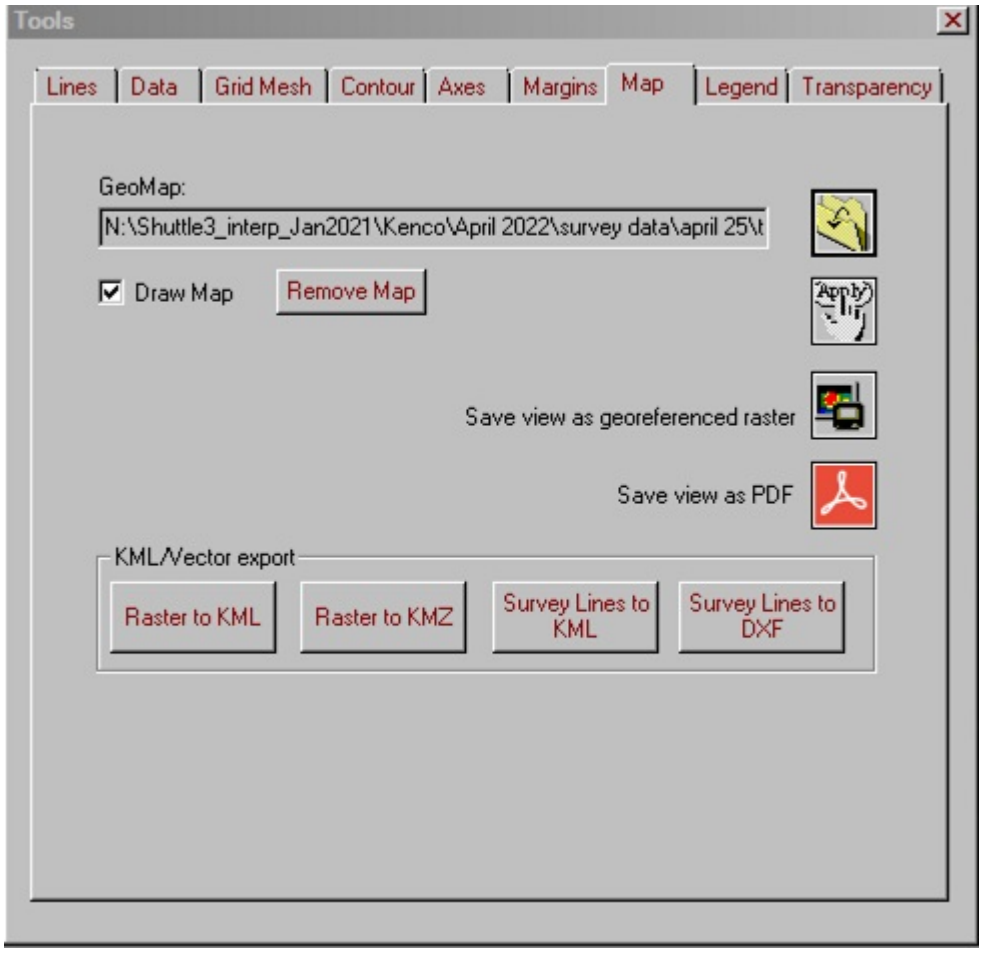
  OR

- Click the proportional button 

Grids will be displayed proportionally by default if there are multiple profiles.

To underlay a map

Click the  button on the QC Grid toolbar The following interface appears:

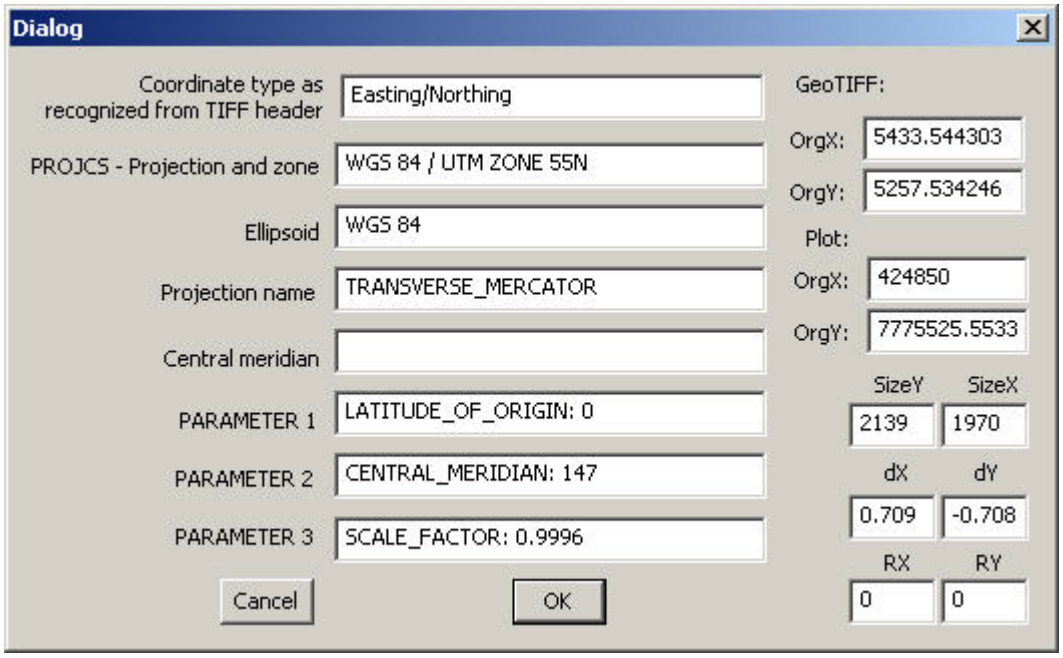


- Click the  button to display the **Open Georeferenced File** window. Select and load a map.

There are a number of supported map file formats. They are PEGeoMap(.map), GeoTIFF, AutoCAD(.dxf), ArcView World, MapInfo and Google Earth (.kml)

> ***Note***. *See* **Calibrate a map.** *for instructions on how to create a .map file in PEGeoMap*

Loading a GeoTIFF file launches the following window that displays the geographic information extracted from the file:



You may adjust the **OrgX** and **OrgY** GeoTIFF values to shift the map's location

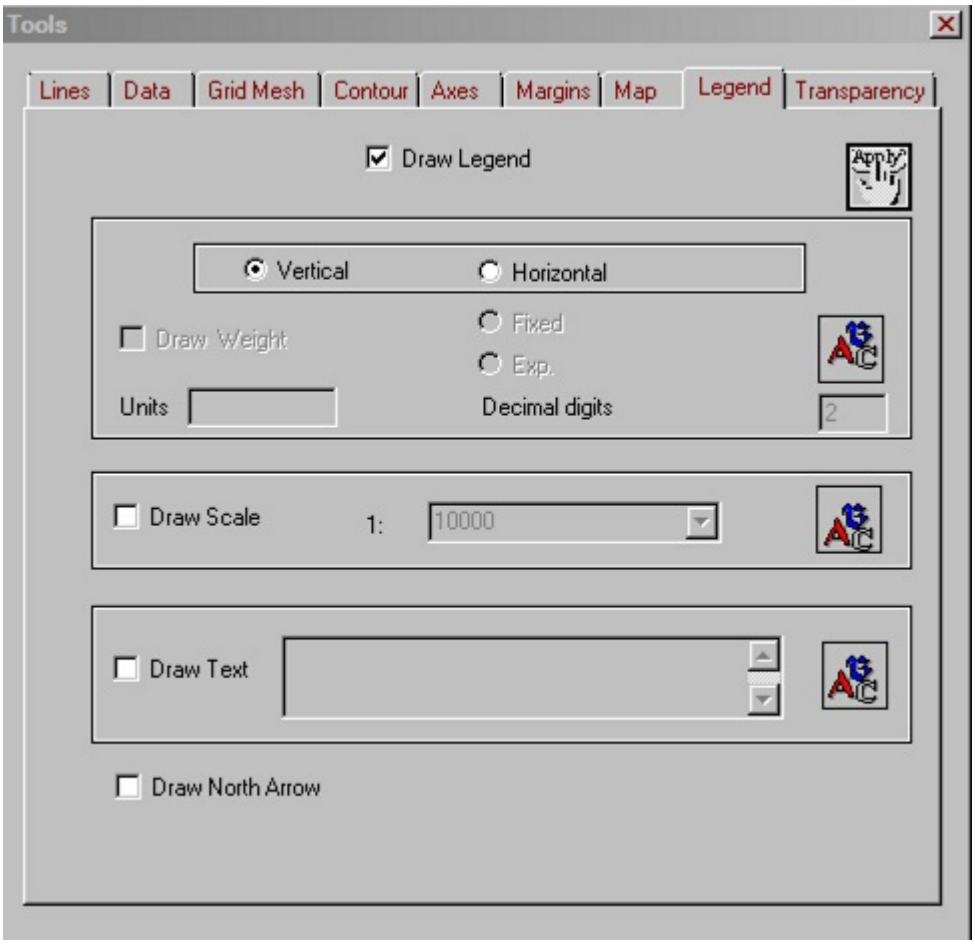
Loading a DXF file enables edit boxes to specify the line width and resolution of the DXF vector graphic

When loading a KML file you need to specify the type of coordinate and the current datum as when exporting to KML

- De-select the **Draw Map** box to remove the map.

To customize the legend and scale

Click the  button on the QC Grid toolbar or right click in the grid window and select **Legend**. The following interface appears:



- Check the **Draw Legend** box to display the legend (or select **Legend** in the **Show Layers** dropdown menu on the QC Grid toolbar).
- Specify the mode (vertical or horizontal)
- Specify the format (fixed or exponential), the number of decimal digits, and font  for the values in the legend.
- To move the legend, click on the legend and move it in a desired direction with the hand manipulator to appear.

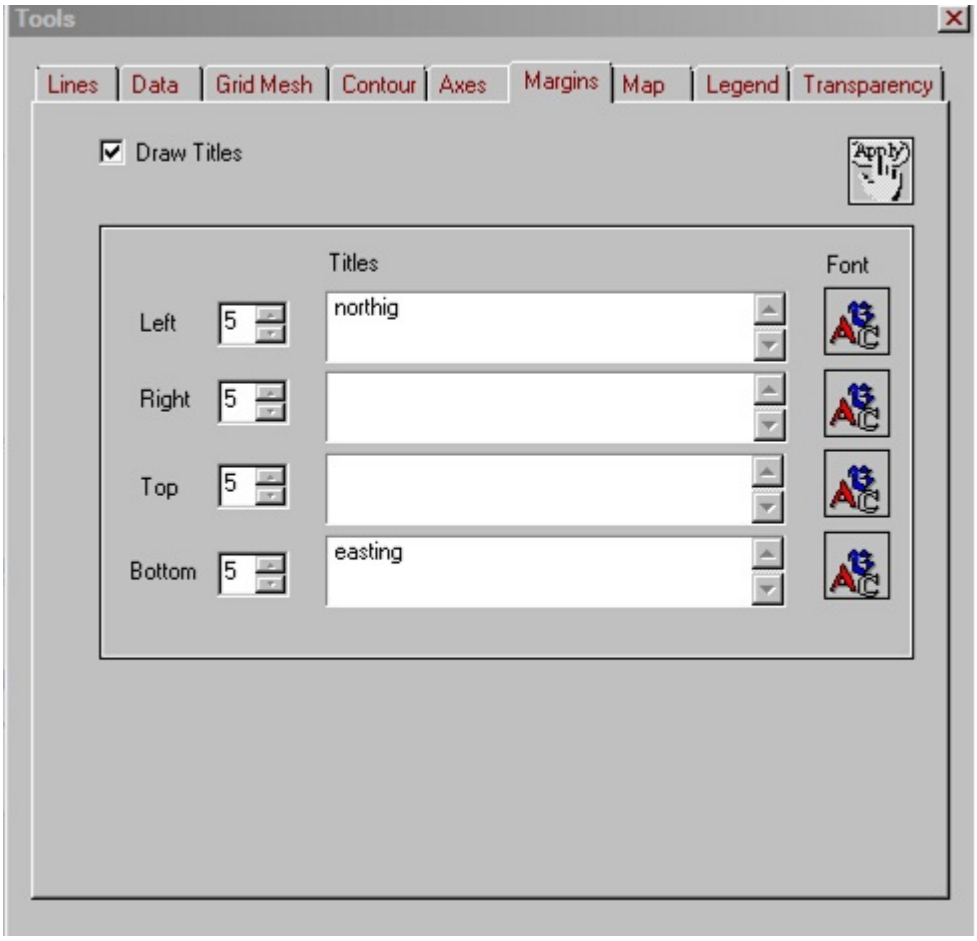
- Check the **Scale** box to authorize scale changes and bring up the scale rule.

- Choose a required scale from the respective dropdown list and click **Apply** in the upper right-hand corner of the dialog (or click anywhere in your plot) to view the result. The scale rule will change accordingly.

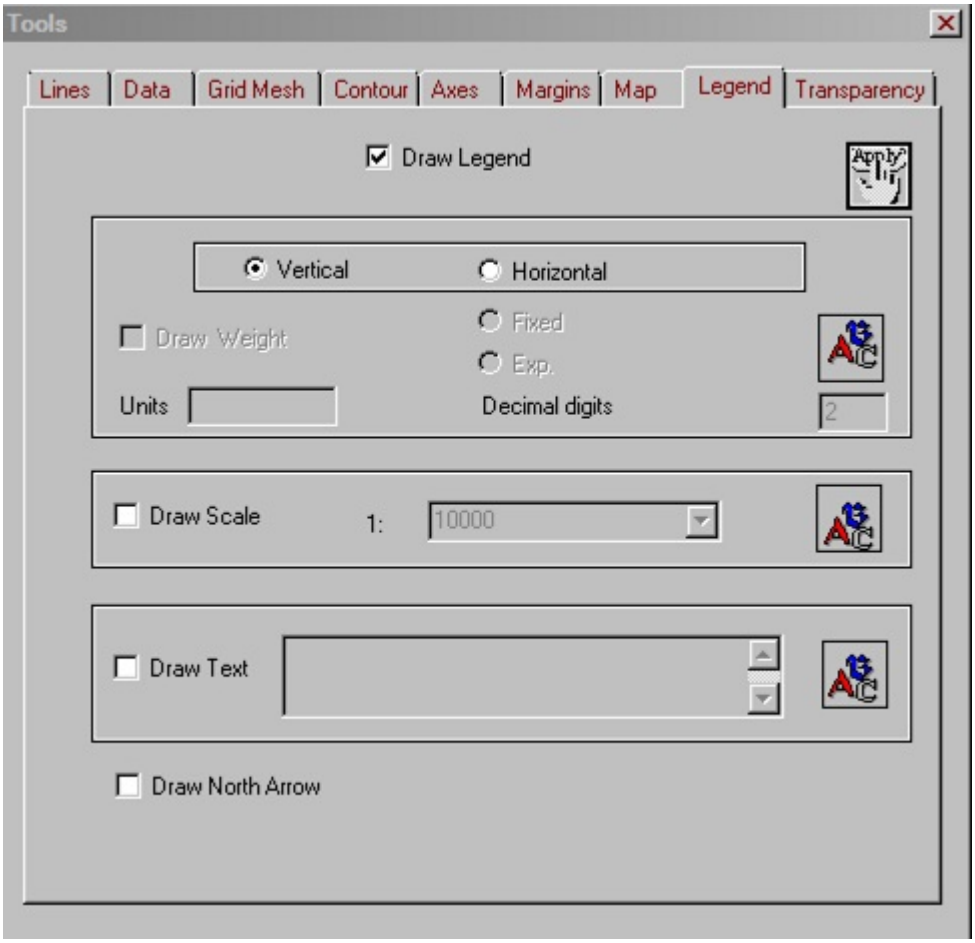
- Check the **Draw North Arrow** box in the bottom of the dialog to display an arrow pointing north.

To customize the grid layout

Click the  button on the QC Grid toolbar The following interface appears:



- Specify the margins around your grid display using the **Left**, **Right**, **Top**, and **Bottom** scroll lists
- Type in the titles, click the  button on the right to specify their font and style, and click **Apply** in the upper right-hand corner of the tab.
- Go the **Legend** tab:



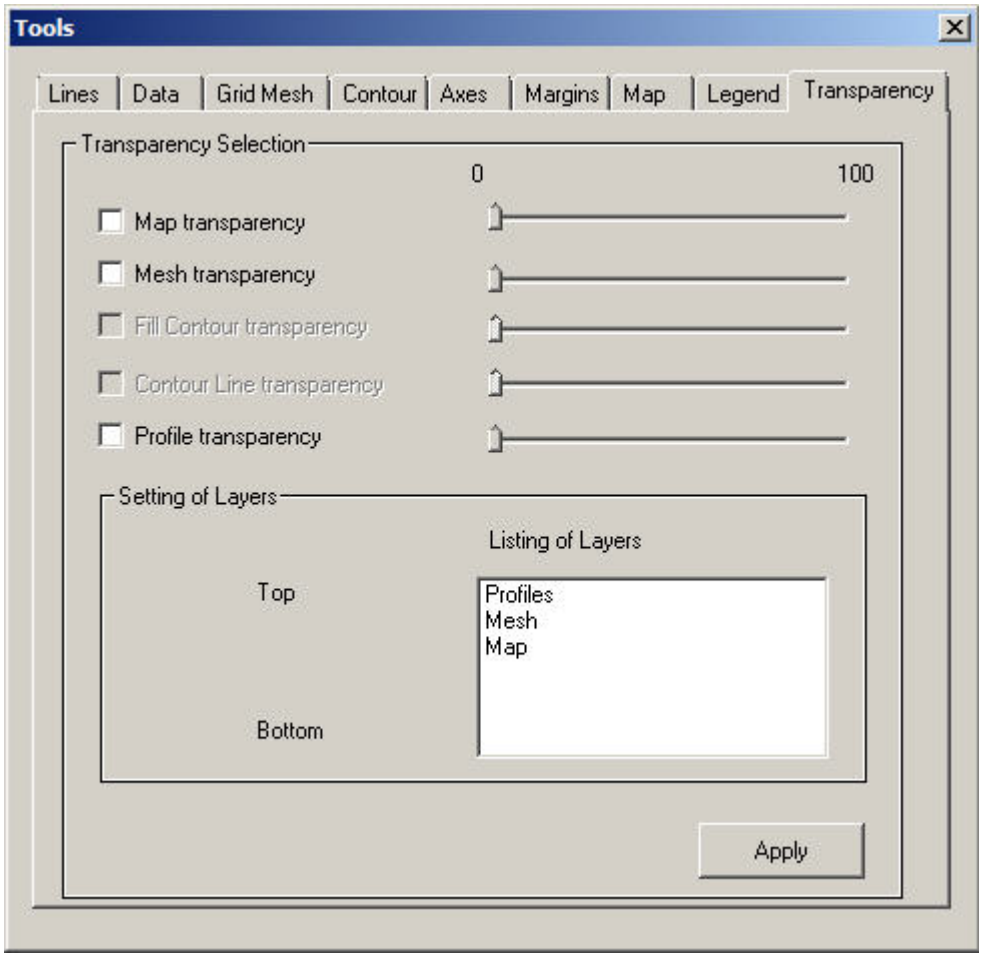
- Check the **Draw Text** box to add comments to your display.

  Type in the field below, press **Ctrl+Enter** to insert a carriage return, click the  button to adjust the font and style of your labels and texts in the standard Windows-style **Font** dialog.

To change the transparency of grid layers

First, a map will be loaded through the **Map** tab

Click the  button on the QC Grid toolbar or the **Transparency** tab on the main **Tools** gui.
The following interface appears:



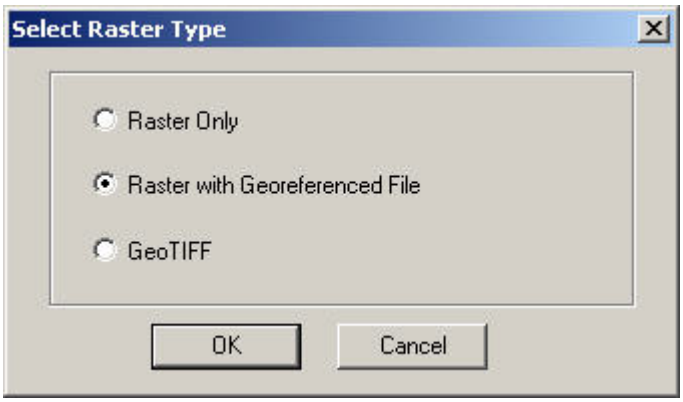
- Select the checkbox related to the layer you would like to add transparency
- Adjust the appropriate slider to set the level of transparency. 0 will have no transparency. 100 will render the layer invisible.

- In the **Listing of Layers** box you may drag the entries to put the layers in a different order
- Click **Apply** to see the effect of your selections

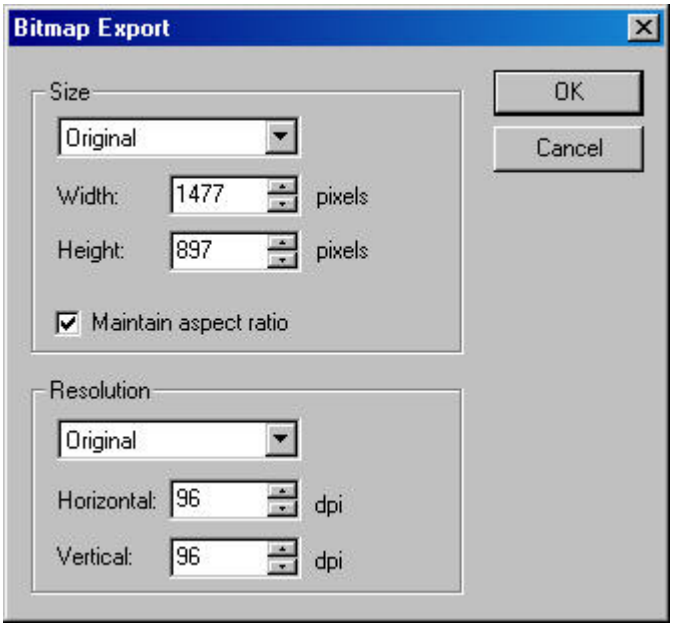
To save a grid in a map format

Click the **Save View As** button  on the QC Grid toolbar and follow the standard saving operations. Or double click on map to open up the **Tools** gui and then select **Map** then **Save view as georeferenced raster**

Select Raster Type

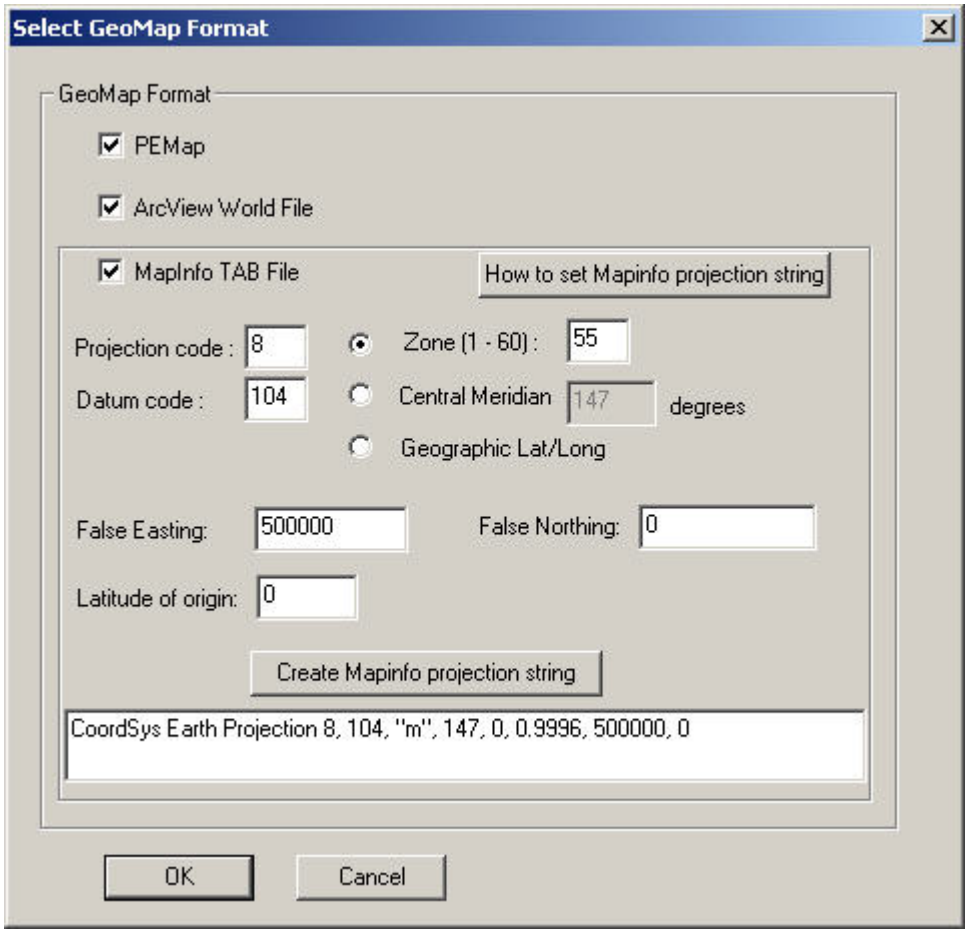


Select Raster Resolution

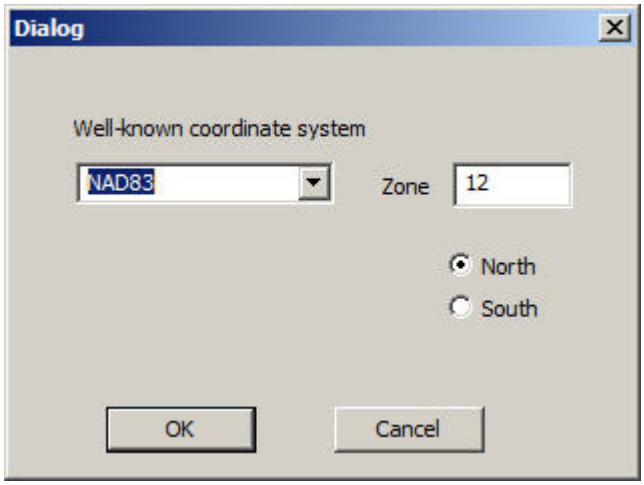


If the **Raster with Georeferenced File** option was chosen, three georeferenced file formats are available to work with in PEGeoMap, ArcView and MapInfo respectively. Additional positioning information must be specified for the MapInfo file. A MapInfo projection string is required. One can be generated from the settings you have selected by clicking the **Create**

MapInfo projection string button.
MapInfo Projection String Details



When saving to GeoTIFF format, you will need to select a well-known coordinate system and zone in this window:



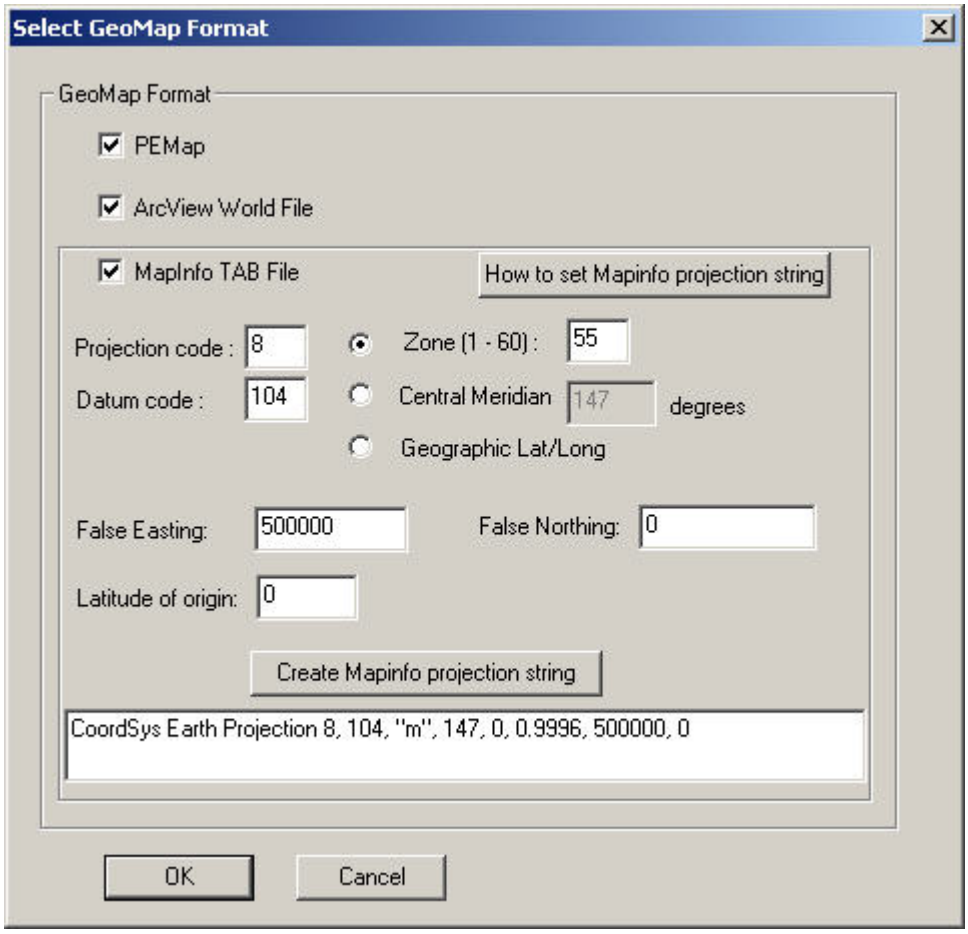
If you are not using one of the systems listed, you should convert the coordinates first before plotting and exporting to GeoTIFF

Other export options are available on the map tab.

Also, you can save legend (colour value scale) separately, and then insert it into printed page if needed (see PEGeoMap PageLayout description). For this, right button click over the legend, popup menu string "Save legend bar as raster" will appear.

NB:
Note that when you save a grid which is not drawn as "proportional" (i.e., width and height ratio does not reflect real topographic sizes), saved raster also will not be proportional. However, obtained georeferenced image will be geographically correct. When read into mapping software (PEGeoMap, ArcMap, GlobalMapper, etc.), it will be positioned accordingly to its coordinates.

MapInfo Projection String



You can construct the string by entering the necessary values on the interface and clicking the **Create MapInfo projection string** button. Otherwise, find the string you need from another source and enter it manually.

A complete table of projections and datum codes for MapInfo is in the \BIN\MAPINFO\MAPINFO_CODES.TXT file supplied with your installation. In the text file, you can find some strings by EPSG number. The number is prefixed by '\p'. The header of the text file describes how to create a proper CoordSys Earth Projection string using the appropriate #MAPINFO line.

The **Create MapInfo projection string** button makes strings only for UTM and geographic lat/long projection codes. For other projections, you must type in the string manually.

When saving KML as .TAB, the string is created automatically as WGS84 lat/long. If source map was a .TAB, the default string comes from that source map.

Example MapInfo projection strings:

- (UTM): CoordSys Earth Projection 8, 116, "m", 141, 0, 0.9996, 500000, 10000000

where: 8 - UTM projection; 116 - GDA94 datum; m - units; 141 - central meridian, 0 - latitude of origin;
0.9996 - UTM stretch coefficient; 500000- false easting; 10000000- false northing (for Southern hemisphere; 0 for Northern).

- (geographic): CoordSys Earth Projection 1, 104

where: 1 - geographic latitude/longitude; 104 - WGS84 datum.

- (Lambert): CoordSys Earth Projection 3, 104, "m", 3, 46.5, 49, 44, 700000, 6600000

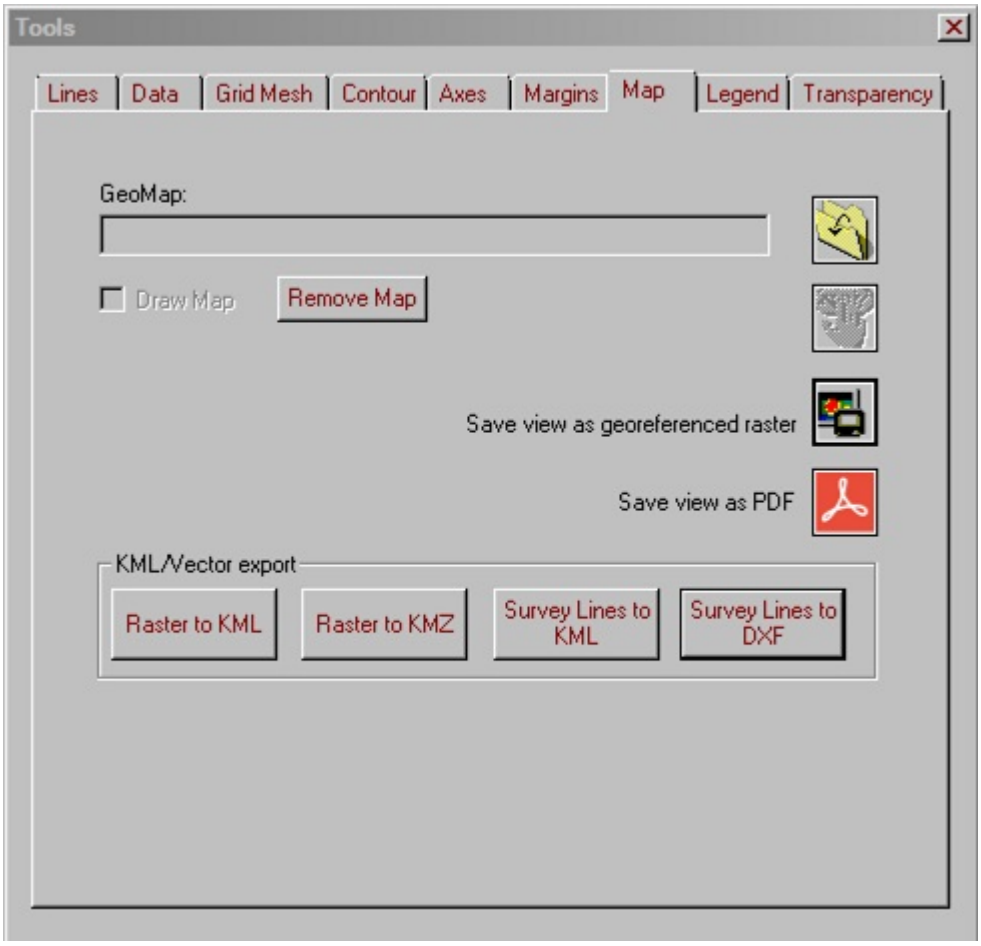
where: 3 - Lambert projection; 104 - WGS84 datum; m - units; 3 - origin longitude; 46.5 - origin latitude;
49 and 44 - standard paralels; 700000- false easting; 6600000- false northing.

-------- FREQUENTLY USED PROJECTION CODES: --------
1 - geographic lat/long; 8 - UTM; 10 - Global Mercator; 3 - Lambert conformal conic.

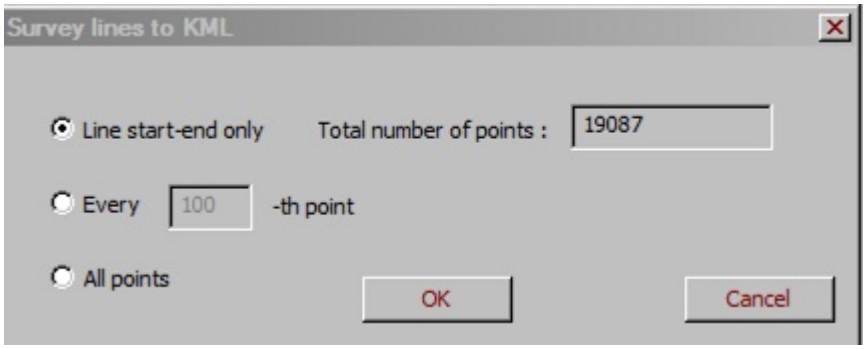
-------- FREQUENTLY USED DATUM CODES: --------
104 - WGS84; 74 - NAD83; 116 - GDA94; 12 - AGD66.

To save to a georeferenced or vector format

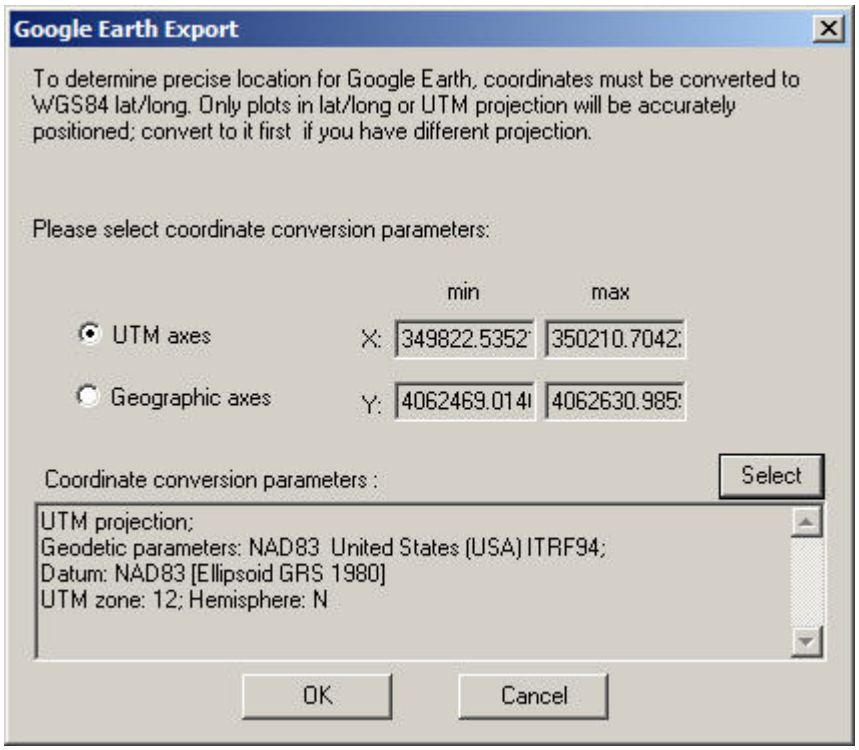
Click the  button on the QC Grid toolbar to see the following interface



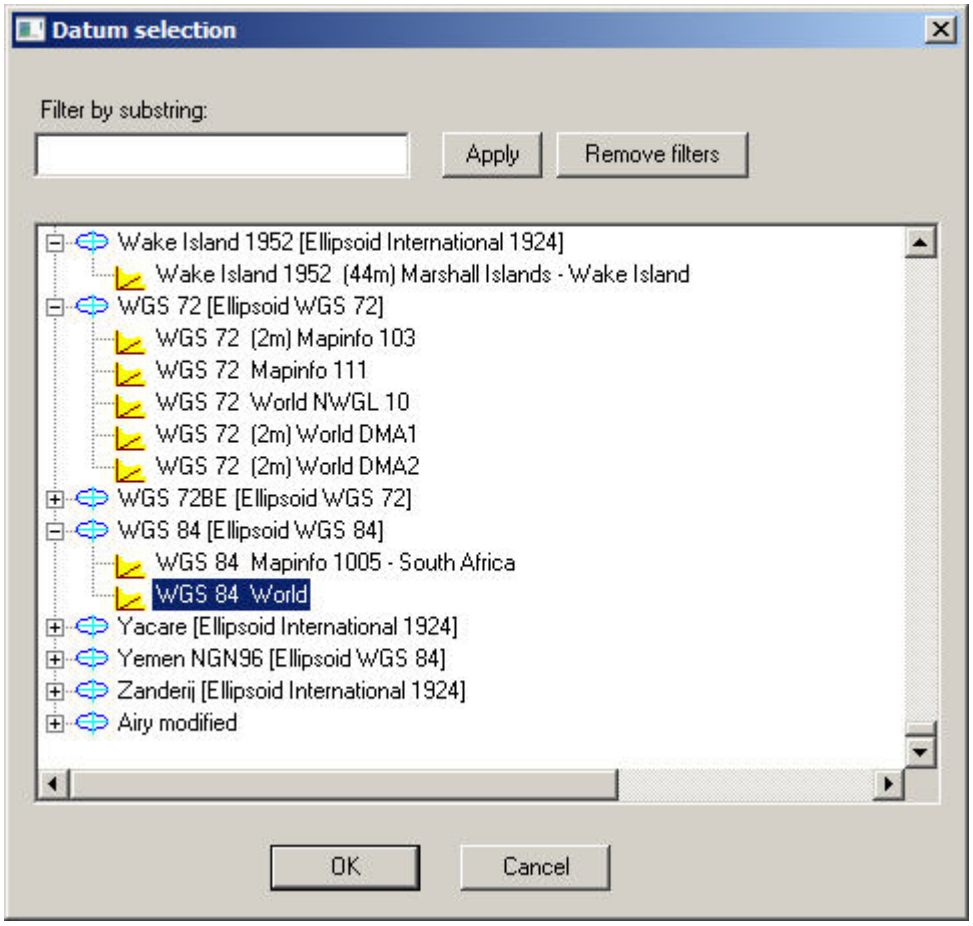
The grid image can be saved to Google Earth KML format by clicking the **Raster to KML** button
Survey lines can saved to either Google Earth KML format or AutoCAD DXF format by clicking the appropriate "**Survey Lines**" button. In addition, you can choose whether only the line endpoints will be saved or at a specified interval:



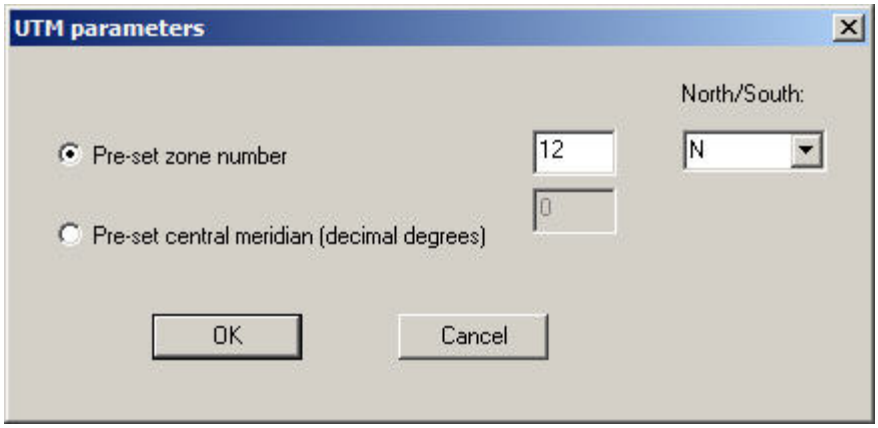
When saving to KML format, this window will appear in order to convert the coordinates to the WGS84 datum:



First specify whether your coordinates are UTM or geographic latitude/longitude. Next click **Select** and you will have to choose the datum used for your current coordinates in the following window:



You may display only the ellipsoids containing the text in the **Filter by substring** box by clicking **Apply**. Display all ellipsoids by clicking **Remove filters**.
If converting from UTM coordinates, you will need to specify either the zone or central meridian in this window:



Click **OK** on the main kml export window and the kml file will be created after you select a file name, folder and image size.

Note that if you make 2 kml exports of the same data: raster overlay from QCGrid and vector survey lines, and then open them both in GoogleEarth, they appear to be slightly shifted against each other, especially when they cover comparatively large area (>10 km). This is due to the fact that raster overlay has precisely positioned only its 4 corners, while in vector survey lines each point is positioned in its exact geographic place.

Other export options are available by clicking the  button.

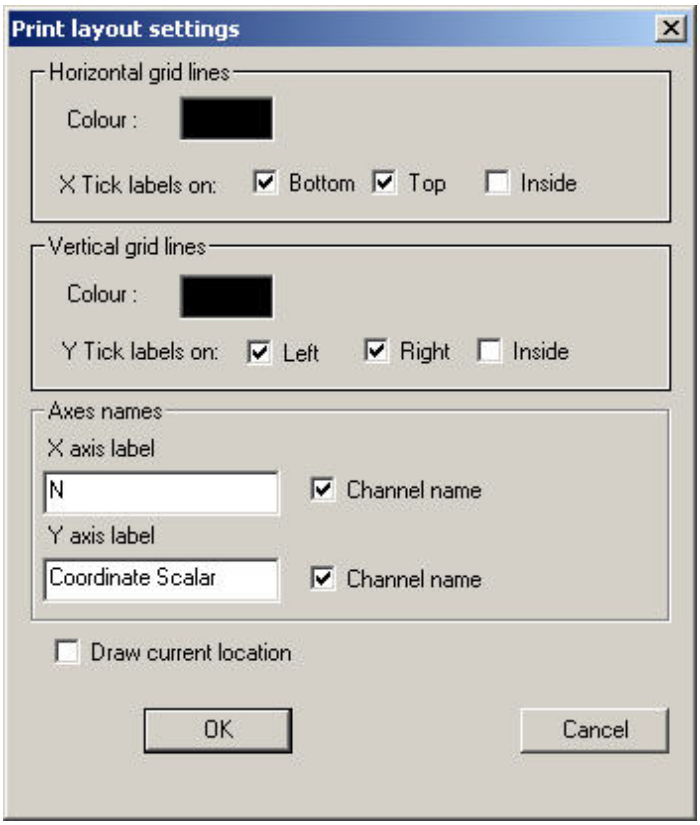
Printing Grids and Plots

To preview a grid/plot before printing:

- Select **Print Preview** from the **File** menu.
- Use the **Next Page** and **Previous Page** buttons on the **Print Preview** toolbar to toggle through available pages.
- Click the **Two Pages** button to preview two pages at a time, click it again to switch back to the **One Page** mode.
- Click the **Zoom In** button to take a closer look at your plot and the **Zoom Out** button to move it away.
- To close the **Preview** mode, click **Close**.

To modify a plot before printing:

- Right-click on the plot and select **Additional Formatting**(This is not available for a grid). The **Print layout settings** window appears:

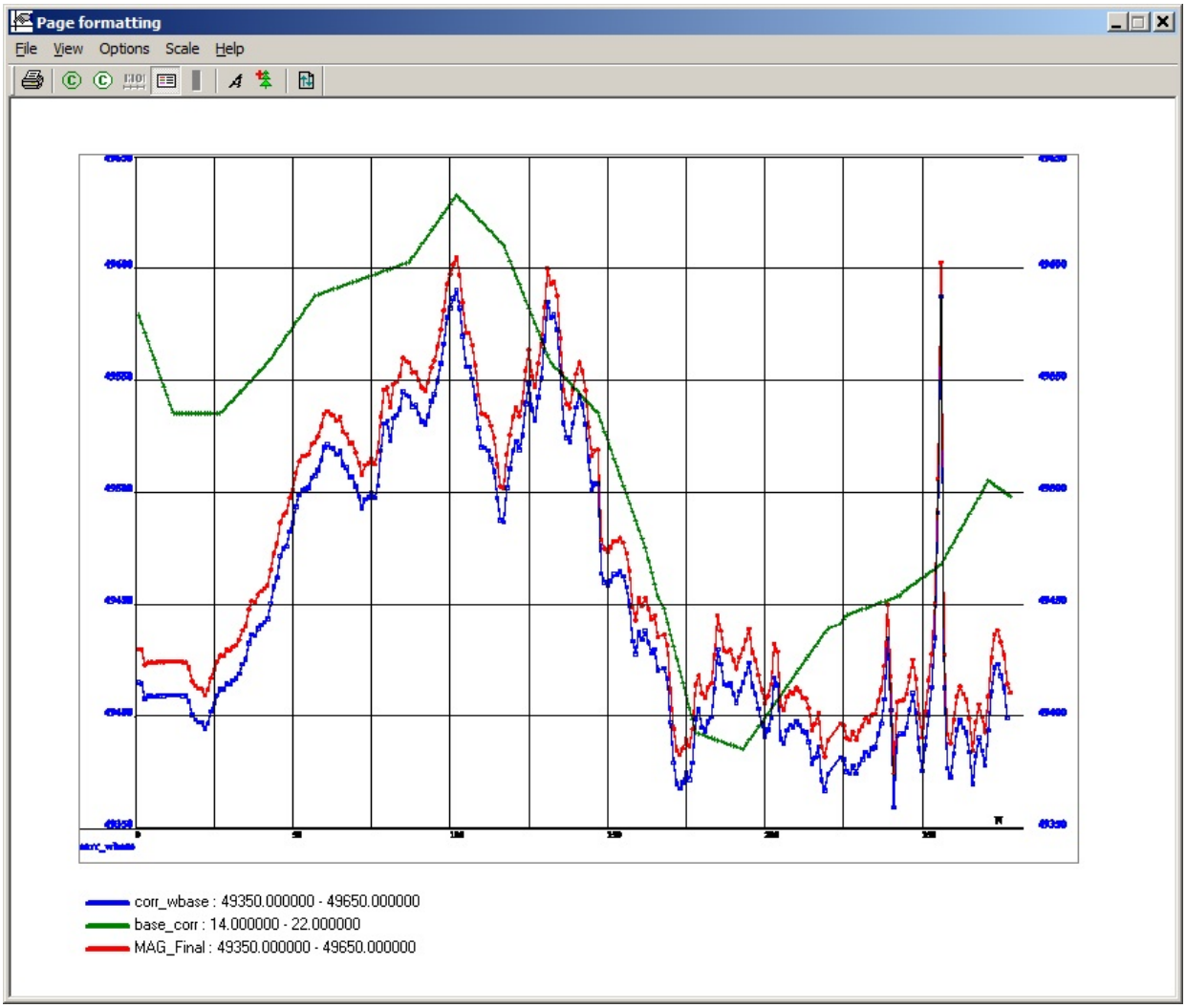


- In the **Horizontal/Vertical grid lines** section, you may select a new colour for the respective grid line.
- You may specify whether you would like **X Tick labels** at the **Top**, **Bottom** or both. As well, **Y Tick Labels** can be displayed on the **Left**, **Right** or both. Tick labels can also be displayed **Inside** the axis by selecting the respective checkbox.
- You may change the **X axis label** or the **Y axis label**. Select the **Channel name** checkbox to change back to the original channel name.
- Select the **Draw current location** to display the cursor indicating the current location on your printout.

To print a grid/plot:

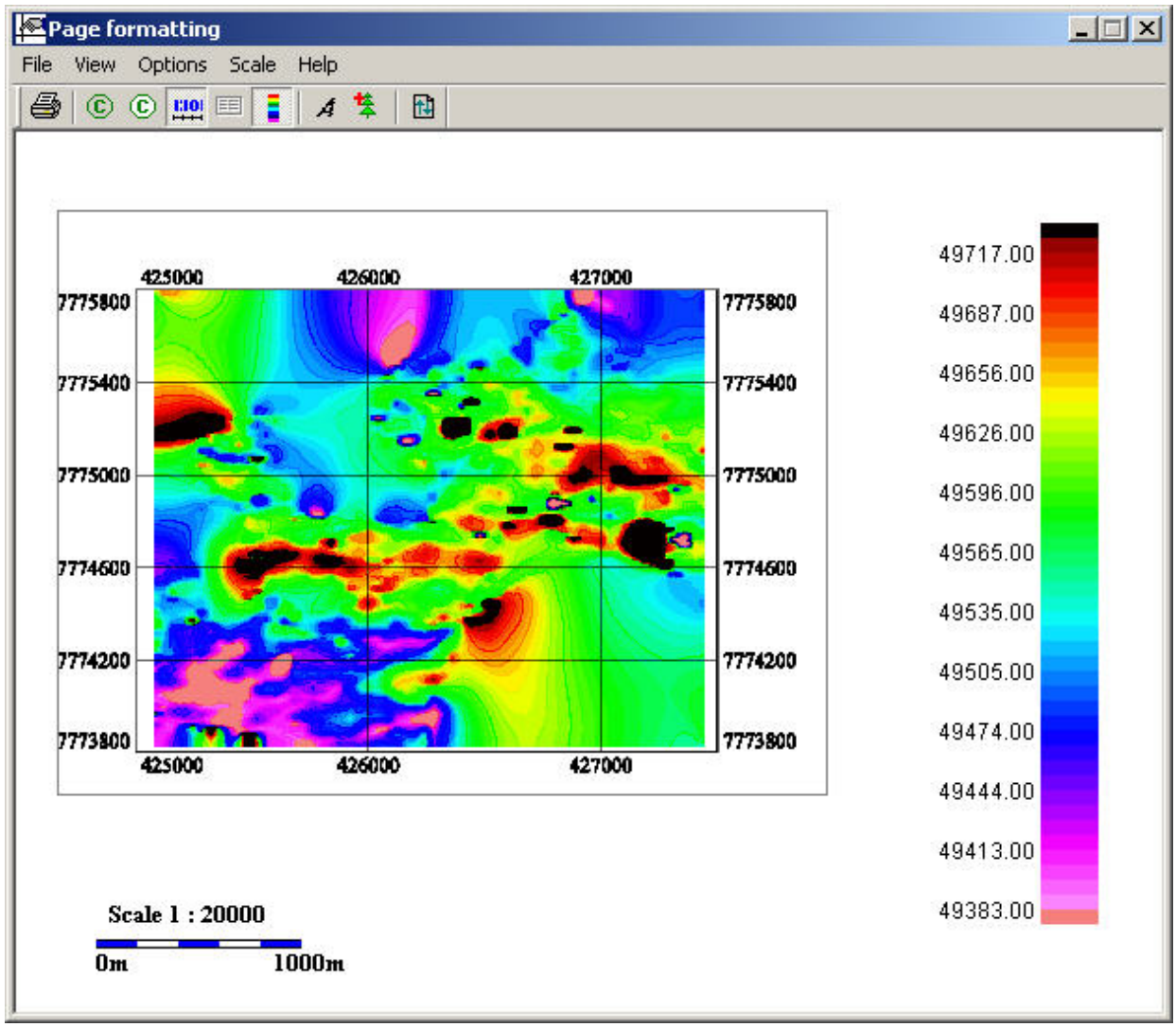
- Click **Print** in the preview mode to print the grid/plot right away.
- You may also select **Print** from the **File** menu. In the standard **Print** dialog to open, specify printing properties and click **OK** to start printing.

- If you have selected items from the Print layout settings window, you need to right-click on your plot and select **Print Layout** to see the effect of your selections. See Print Layout for more information.



Print Layout

Right click on a grid/plot, select **Print Layout** and the following window is displayed:



Moving and resizing page components can be done by use of trackers. To activate the appropriate tracker, left-click on the component.

To repaint the entire page and clear trackers, click the button or right-click anywhere in the window.

The application window size is calculated to match the aspect ratio of the current printer page size. Changing the width of the application window forces the window's aspect ratio to match that of the selected paper size.

Scale If you are displaying a grid proportionally, you may choose to have a scale on your hard copy. The scale is recalculated if the grid changes size. To select a specific scale value, select Scale/Scale to Value from the menu and enter the new value for the scale you would like. Toggle scale visibility with the  button or by selecting View/Scale in the menu. To change font of the scale, right click on it and select **Font**.

Raster images
You may add pictures by clicking the  button or selecting **Options/Add raster image** from the menu. Navigate to the location of the raster image you would like to add then click on the location where you would like the image to be. To retain aspect ratio when resizing a raster image component, hold the Ctrl key while resizing or right click and select **Resize proportionally to width** Delete the image by right-clicking and selecting **Delete**

A logo is a special type of raster image. Its path is saved between QCTool sessions. Toggle its visibility by clicking the  button or selecting **View/Logo** from the menu. Select a logo by clicking the  button or selection **Options/Change Logo Image** from the menu.

Text Boxes
You may add a text box by clicking the  button or selecting **Options/Add Text Block** from the menu. Double click on the location where you would like the text box to be and start entering your text. Single click the text box to edit its contents later. To move/resize a text block, right-click on it and select **Move/Resize**. To change font of a text box, select your text, right click on it and select **Font**.

You may add a special text box by selecting **View/Title Box** from the menu. A title box is a text box with a border and divided into sections.

Right click and select **Edit** to change its contents.

Delete both types of text boxes by right-clicking and selecting **Delete**.

Legend

For a plot click the  button to toggle the visibility of a legend or select **View/Legend** from the menu.
For a grid click the  button to toggle the visibility of a legend or select **View/Palette** from the menu.

What is PEGeoMap

PEGeoMap

PEGeoMap is a compact mapping software. It allows creation of georeferenced maps from appropriate raster images; draw various objects on top of it; create layered maps with transparent layers; perform conversions between commonly used image and mapping formats.

Georeference types supported by PEGeomap:

- 1. MAPINFO .tab
- 2. ESRI world files (.jgw, .bpw, .pgw, .tfw)
- 3. Petros mapping format .map
- 4. GoogleEarth ground overlay .kml
- 5. Geotiff - tiff format with georeference written inside image file.

Types 2, 3, and 4 are two item formats: raster + text georeference file.
Georeferenced image can be then imported as underlying map into QCTool grid or EMIGMA Survey Editor and GridPresentation.
Also, it can be used in any kind of mapping software accepting these formtas, such as ARCMap, GlobalMapper, etc.

PEGeoMap

Using PEGeoMap

Pegeomap is a tool for working with georeferenced raster images. It can either use existent georeferenced files compatible with various other mapping software, or create a new one by the "Generate a Map" procedure [see below]. A georeferenced image can be then imported as an underlying map into QCTool grid or EMIGMA GridPresentation and SurveyEditor.

PEGeoMap can be launched from the QCTool group on the windows Start bar, from QCTool Grid toolbar, or started from Windows Explorer.

Georeference types used in PEGeomap: MAPINFO tab, ESRI world file, map, kml ground overlay - 2 items formats: raster + text georeference file;
geotiff - tiff format with georeference written inside image file.

To open a georeferenced map:
Select in menu File/Open geomap/ select georeference file: jgw, bpw, pgw, tfw, map, tab, kml or geotiff.
Or, it can be any kind of an image without georeference: bmp, jpg, png, etc.; in this case, use File/Open Raster menu item.
Also, you can drag and drop any such file from Windows Explorer into the application window.

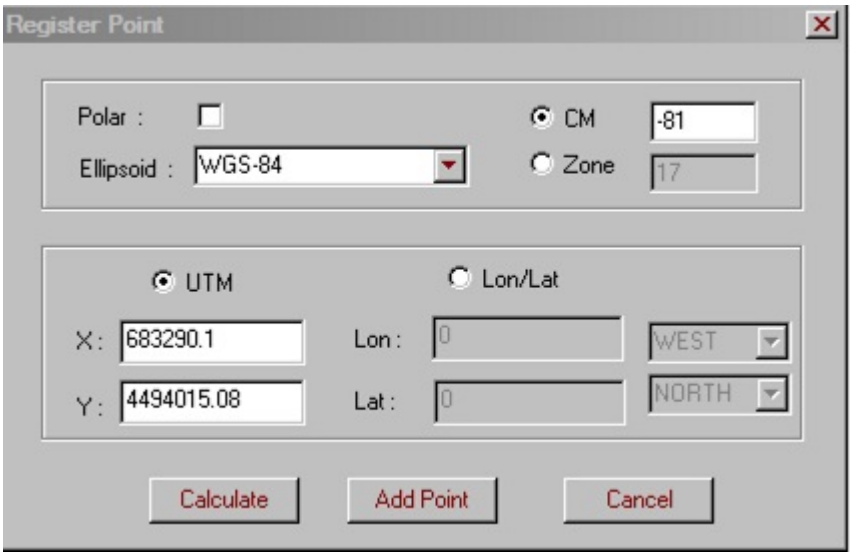
If georeference was successfully discovered, status bar displays valid X,Y coordinates; otherwise they are 0.

To Generate a Map (create georeference for raster image):

- A raster format file (gif, jpg, etc.) needs to be loaded. If a file is not already loaded you will be given the opportunity to load the

last used file. If you would like a different one, click **No** and a window will appear offering you to select a file. You can also load a raster file from the toolbar using the  button. You can load a map file using the  button.

- A raster file needs to be calibrated with at least three points.
- Double-click on the first point in the image. The following window appears:



- Check the **Polar** box if your map comes from a polar region
- Select the required ellipsoid datum from the respective dropdown list
- Specify the central meridian(CM) and zone in the respective boxes
- Select between **UTM** and **Lon/Lat**. The appropriate boxes below will become active.
- Type in the coordinates of the point to add and click **Add Point**. Repeat this operation for the rest of the points.
- To convert your UTM into Lat/Lon, select **UTM**, click **Calculate** and the conversion appears in the Lat/Lon area. Select **Lat/Lon** and **Calculate** to convert the other way. Lat/Lon will be

recalculated to UTM since raster transformation is based on UTM coordinates.

- Once at least three points have been entered, you may save your map(see below) and the Map Tools will be enabled.

Managing the registered points:

- Click the  button. The **Edit Registered Points** window will open. Select the cell you want to edit and type in a new value.
- Click the  button to remove all registered points.
- Use the  button to apply a shift to the x and y values of the map. The effect of the shift can be seen by looking at the X and Y values displayed on the status bar located at the bottom of the PEGeoMap window.

Customizing the appearance of the map:

- To toggle the display of grid lines on the image display, click the  button.
- To toggle between color and grayscale, click the  button
- To add a line to your map, click the  button and draw the line with the mouse. With the line selected:
  - Click the  button to change the line's width.
  - Click the  button to change the line's style.
  - Click the  button to change the line's colour.
  - Drag the boxes at the ends of the line to modify the start and end positions.
  - Use the Map Tools to add a line at specific coordinates.
- To add a symbol to your map, click the  button and drag the mouse to specify the desired dimensions. With the symbol selected:
  - Click the  button to change the symbol's style.
  - Click the  button to change the symbol's colour.

    - Drag one of the resize controls located around the symbol to change its dimensions.
    - Use the Map Tools to add a symbol at specific coordinates.
- To add an annotation to your map, click the  button. In the **Annotations** window to open, write your text. Use the Font section at the right end of the toolbar to change the appearance of the text. Click on the map at point where you want the annotation to appear and drag the mouse to specify its dimensions.
- To cancel any of the previous operations, click the **Undo** button 
- Any items that have been added to the map can be repositioned by dragging with the mouse.
- A popup menu is also available by right clicking on the selected item :
    - **Delete** - Deletes the item.
    - **Edit** - Displays the **Annotation** editing window.
    - **Paste** - Creates a duplicate of the selected item.
    - **Properties** - Display the window to edit the item's properties.

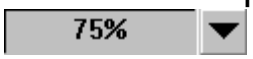
Image rotation

If original image was obtained by scanner or drawn in skewed local grid, for conversion to any of commonly used coordinates it might need to be rotated.

To rotate an image, select Draw tools/Rotate and set required angle. All point registering and drawings must be done after that.

Viewing the map:

- To zoom in on your map, click the **Zoom In**  button on the toolbar. Then, click on the image to the desired zoom level.

- To zoom out, click the **Zoom Out** button. Then, click on the image until the desired zoom level is reached.
- To zoom to a percentage of the actual size, click the down arrow 

and select a new zoom value.
- To zoom to 100% of the actual image size, click the button.
- To move a different viewing area, click the button and drag the image with the mouse.
- To measure a distance, click the button and drag the mouse between the two end points of the distance you would like measure. The value of the distance will appear on the right end of the status bar at the bottom of the PEGeoMap window. Three points need to be registered in order to use this feature.

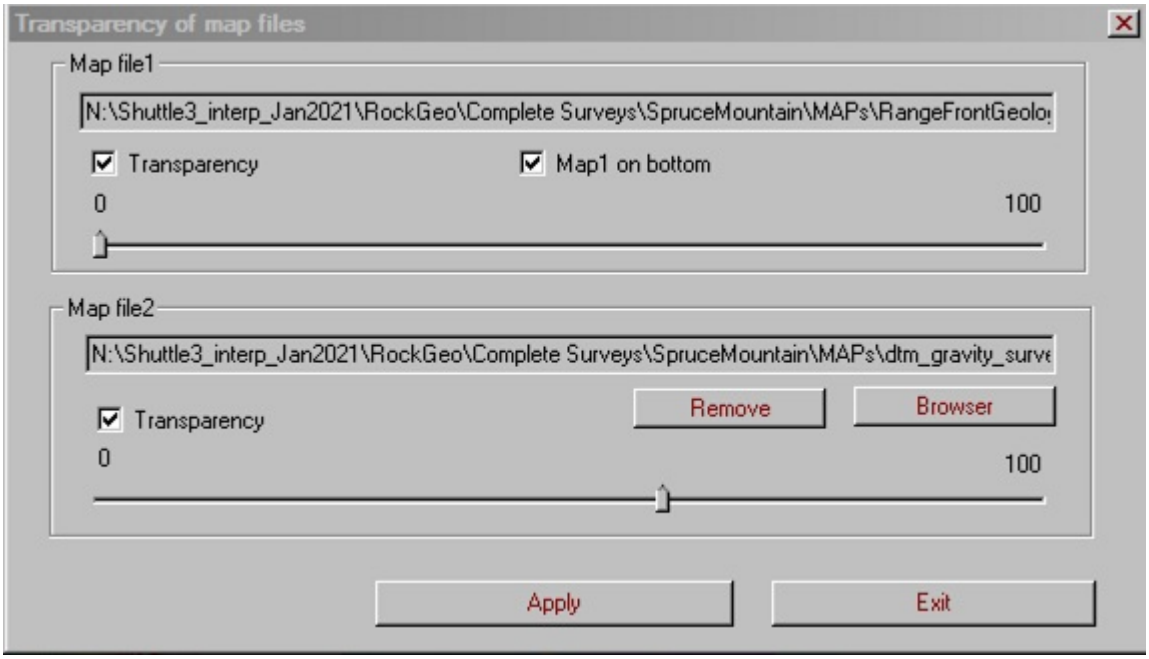
Saving your work:

- Save using the button. You may choose to save an image file and/or a *.map file. The *.map file will be available to other EMIGMA tools such as Grid Presentation and Survey Editor which can use the created raster image as an underlay.
- Click the button to quickly save an image file only. Various file types are supported.
- To save a section of the image to a file, click the button. Click and drag the mouse on the image to create a selection box containing the area you would like to save. Then, click the button to save the file.
- To customize the layout of a printed hardcopy, click the button. The Print layout window will appear and then you will be able to edit the appearance of your printout.

PEGeomap image layering:

PEGeoMap can display 2 layers with selected transparency.

- After loading your first map layer, select menu item **"Map Tools/Transparency".** The **Transparency of map files** window appears.
  Select a 2nd layer map by clicking the **"Map file2 - Browser"** button.
  Both layers must be georeferenced in the same coordinate system.



  Use the sliders to set the transparency as desired. Click **Apply** to update the appearance of the image.

  For successful saving of the layered image:

- Both layered images must have same raster type: bmp, jpg, etc. (and correspondingly, bpw, jgw... world files if chosen format is ESRI).

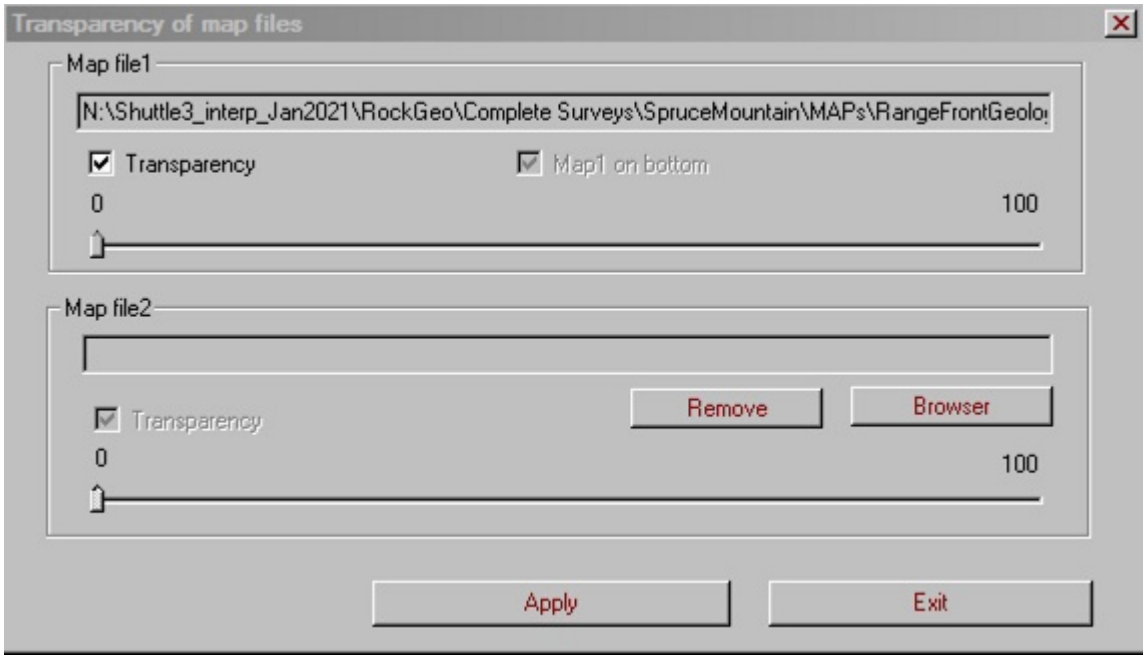
- The resolution of 1st layer will be used for the saved image. Generally, select an image with higher resolution for the first layer.

Map Tools

These functions are available only when current image is georeferenced: either originally, or by registering points.

PEGeomap image layering:
PEGeoMap can display two map layers with selected transparency.

After loading your first map layer, select menu item **"MapTools/Transparency" or click the**  **toolbar button .** The **Transparency of map files** window appears.
Select the 2nd layer map by clicking the **"Map file2 - Browser"** button.
Both layers must be georeferenced in the same coordinate system.



Use the sliders to set the transparency as desired. Click **Apply** to update the appearance of the image.

For successful saving of the layered image:

- Both layered images must have same raster type: bmp, jpg, etc. (and correspondingly, bpw, jgw... world files if chosen format is

ESRI).

- The resolution of 1st layer will be used for the saved image.

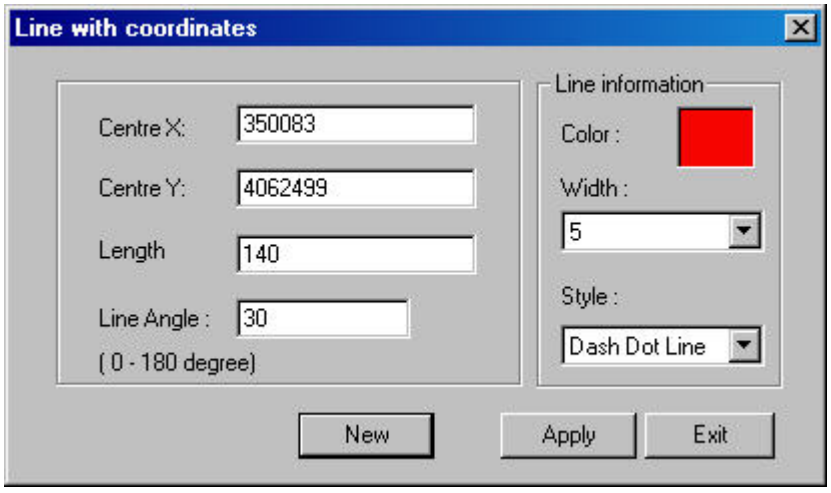
Generally, select an image with higher resolution for the first layer.

To see geographic coordinates:
Select from the menu Map Tools/Calculate Lat/Long. Select datum and zone number. Then, the latitude and longitude of the current cursor position will be displayed in the status bar.

Annotation Tools

Drawing positioned elements:

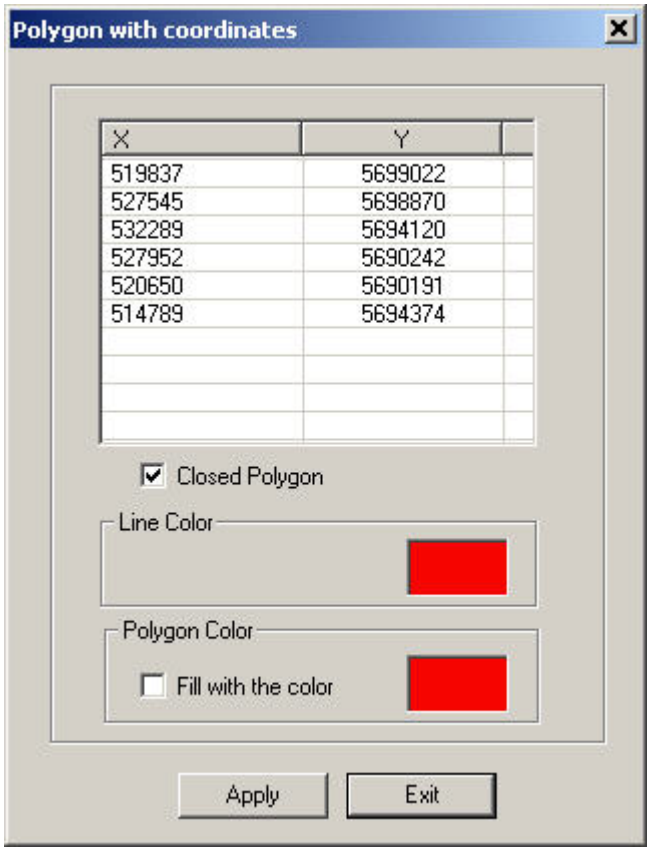
Click the  button to add a line to the map file. The following window appears:



- Use this window to specify the line dimensions, color and style.
- Colors may be changed by clicking on the box labelled **Color**.
- Click **New** add a new line to the map. Click **Apply** to modify an existing selected line.

A similar window is used when adding a symbol to the map file using the  button.

Click the  button to show the coordinates of an existing polygon. The following window appears:

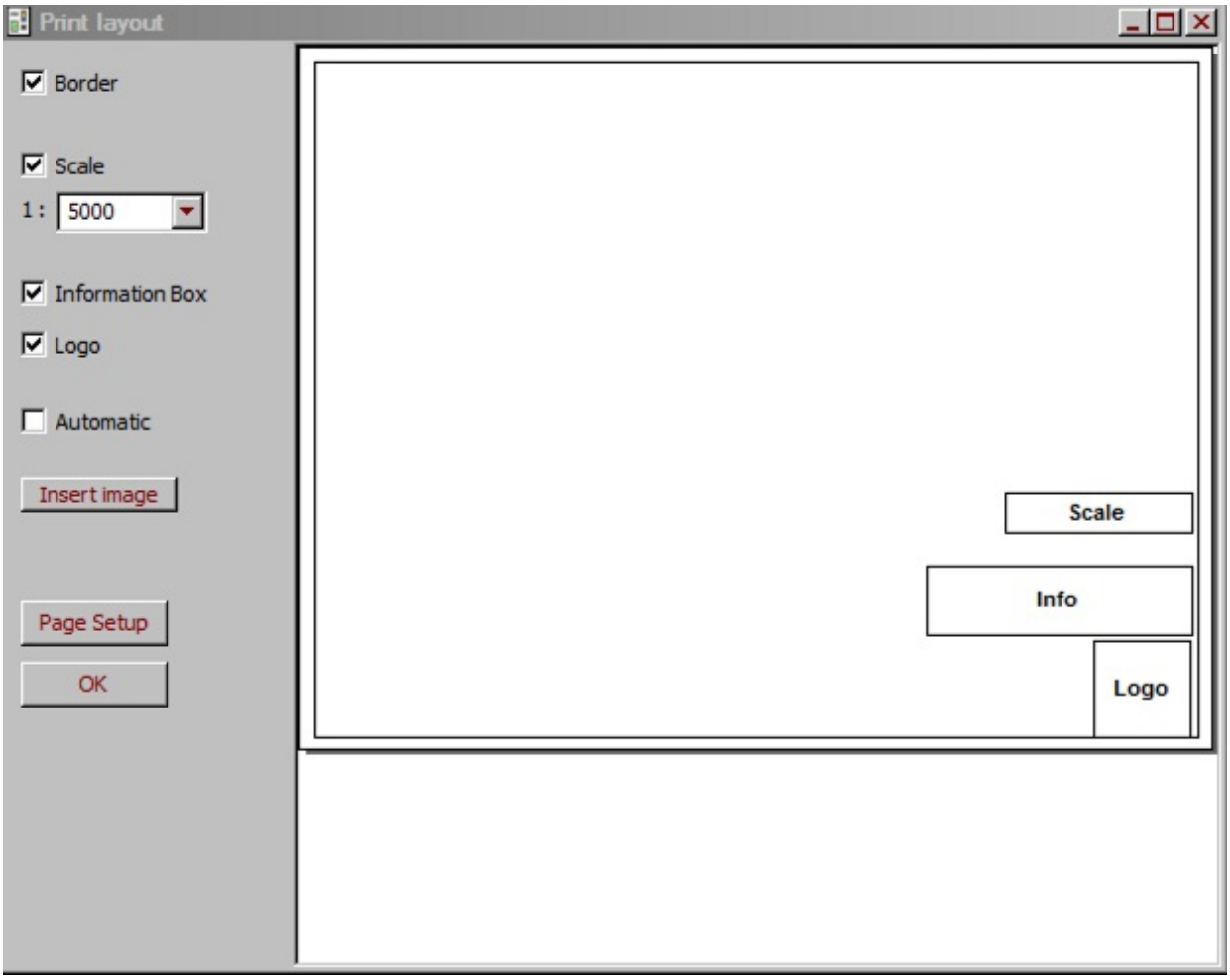


- Use this window to display the coordinates for a polygon as well as specify the line and fill color.
- Select **Fill with the color** to specify a filled polygon.
- Colors may be changed by clicking on a colored box.
- Select **Close Polygon** to draw a line connecting the first and last coordinates.

- Select a pair of coordinates and press delete on the keyboard to delete a point.
- Click **Apply** to modify an existing selected polygon.
  A polygon, symbol or line can also be modified by right clicking on the object and selecting **Properties**.

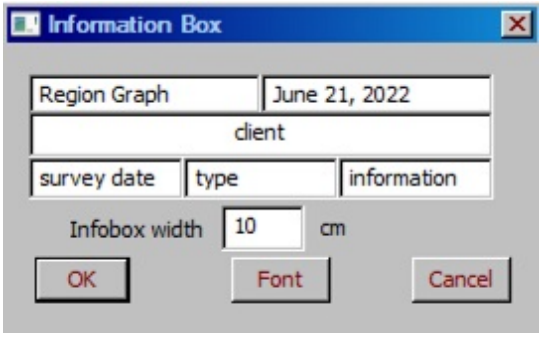
Print Layout

To customize the layout of a printed hardcopy, click the  button and the **Print layout** window will appear:

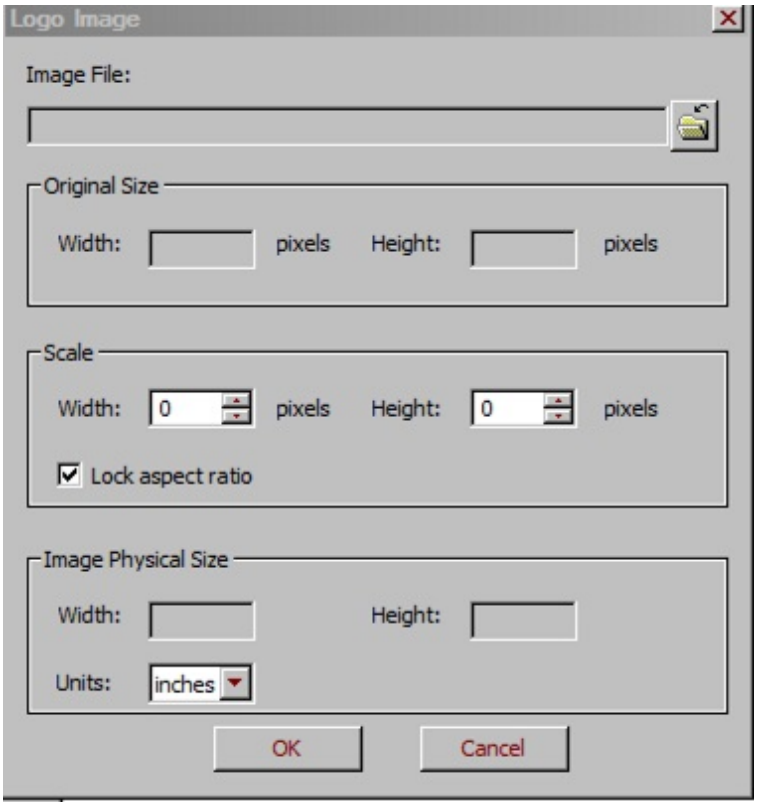


- The default setting is Automatic which will print the map image as displayed on the screen.
- Deselect the Automatic checkbox to add additional items to your printout. You can add a border, scale, information box, and logo.
- Any of these items may be moved by clicking and dragging the item on the Print layout window.
- Edit the information box content by double clicking the box labelled Info. Also, in this dialog you can select font and define

box metric size - its width in centimeters.



Edit the logo by double clicking the box labelled **Logo**. The **Logo Image** window will appear:



- Click the button to browse for the logo image file.
- You may change the size of the logo in the **Scale** section. Select **Lock aspect ratio** to have one dimension update its value when the other is modified so the image's aspect ratio is maintained.
- The measurements of the logo on the printed page are shown in the **Image Physical Size** section. The measurements can be displayed in inches, centimetres or millimetres.

- Click **OK** when finished.

You can insert multiple raster images into your page by clicking "Insert image" button. After selecting a file, click page where you want to place your image. Then, you can move images around the page, and, by double click, change their sizes or delete. You can insert colour scale legend bars saved in QCTool (described in "Save a grid") or EMIGMA's GridPresentation.

To save a file

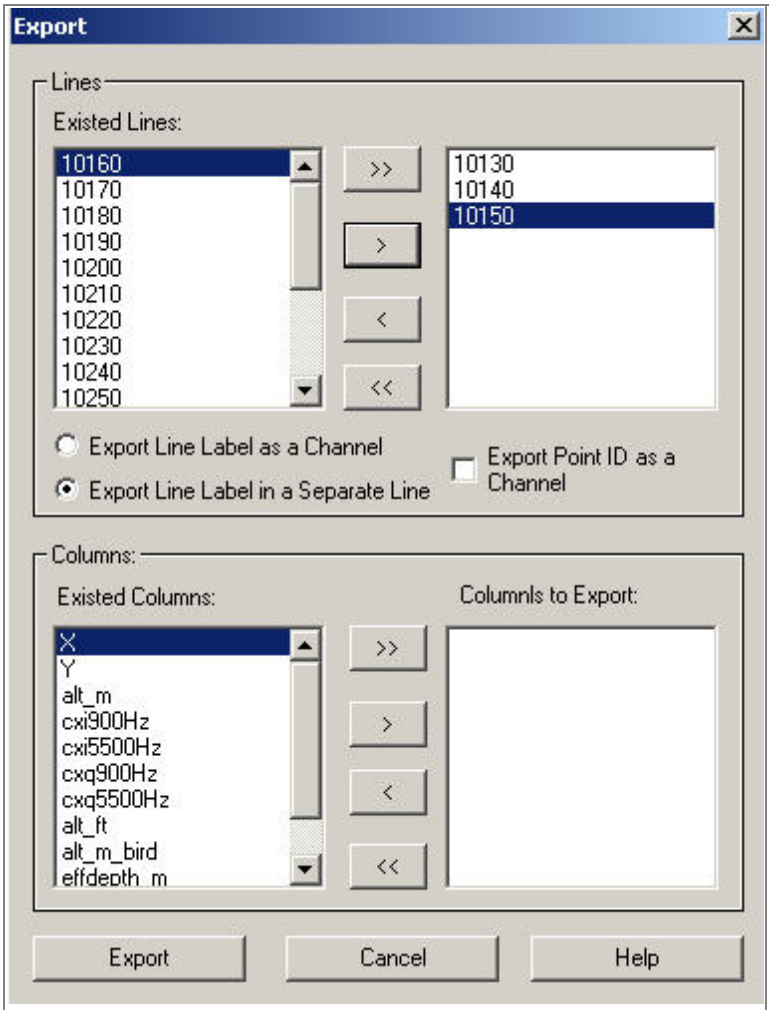
- Select **Save File** from the **File** menu (you can also use the standard **Save** button on the QC toolbar) to overwrite your old version.

- Select **Save As** to create a new filename.

To save data in other formats

You can save your file in several formats: .qct (QCTool), .xyz (XYZ ASCII), .xls (Microsoft Excel), .csv (comma separated values) and .gbn (Geosoft GBN):

- Select **File/Save As**. The **Save As** dialog appears.
- Select a required format from the **Save as Type** dropdown list and click **Save**. The **Export** dialog opens.



- In the **Lines** section of the dialog:
  - Select a line you want to export and click the > button. The line is moved to the field on the right.
  - To export all lines, click the >> button.
  - To cancel export, click the < and << buttons.
  - Select between the two options offered for the line label export: as a channel or in a separate line.
  - Check the **Export Point ID as a Channel** box to

	assign a special number (ID) to each row of your data and to export such numbers as a separate column. • In the **Channels** section, select the columns to export in the **Existing Columns** field and move them to the field on the right using the same method as described for the **Lines** section of the dialog. • Click **Export.**

Note*. If not indicated otherwise, the exported file will be saved with the same name, but a different extension.*

Managing Files

In certain cases, you may need to work with various files simultaneously. To facilitate this task, the QC Tool offers its own **File Manager**.

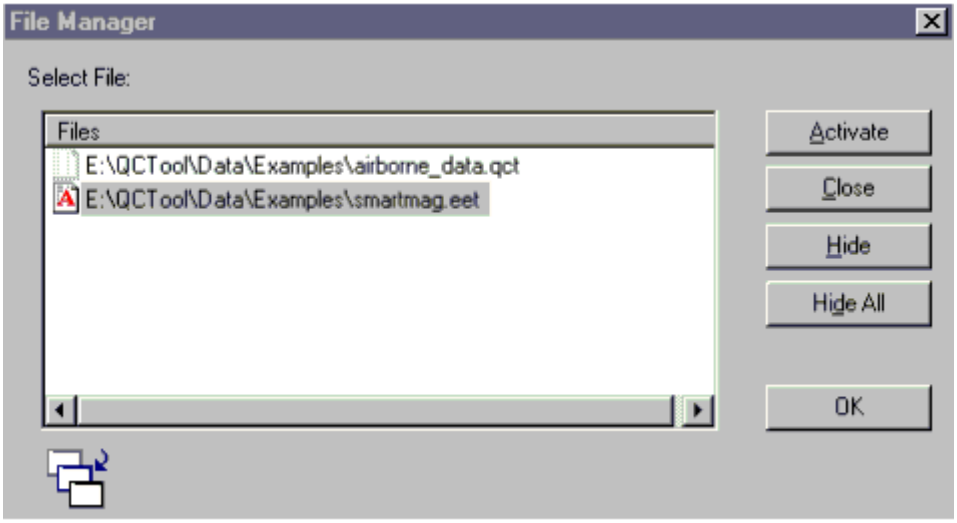
If you work with one file, have to switch to another, but want the first one to be at hand:

- Close all the files one by one.

  In each case, a message will prompt you to click the **Yes** button if you want to close your file and the **No** button if you want to hide it, i.e. “to put it aside for the time being”.

- Click **No** and continue your work with other files.
- When you want to open the file you hid, click the **File Manager** button  on the main QC toolbar.

The respective dialog appears:



- Select the file from the **Files** list and click **Activate**. The file will open.

You can also use the **File Manager** dialog to hide and close files:

- To hide a file, select it from the **Files** list and click **Hide**.

- To hide all the files, click **Hide All**.

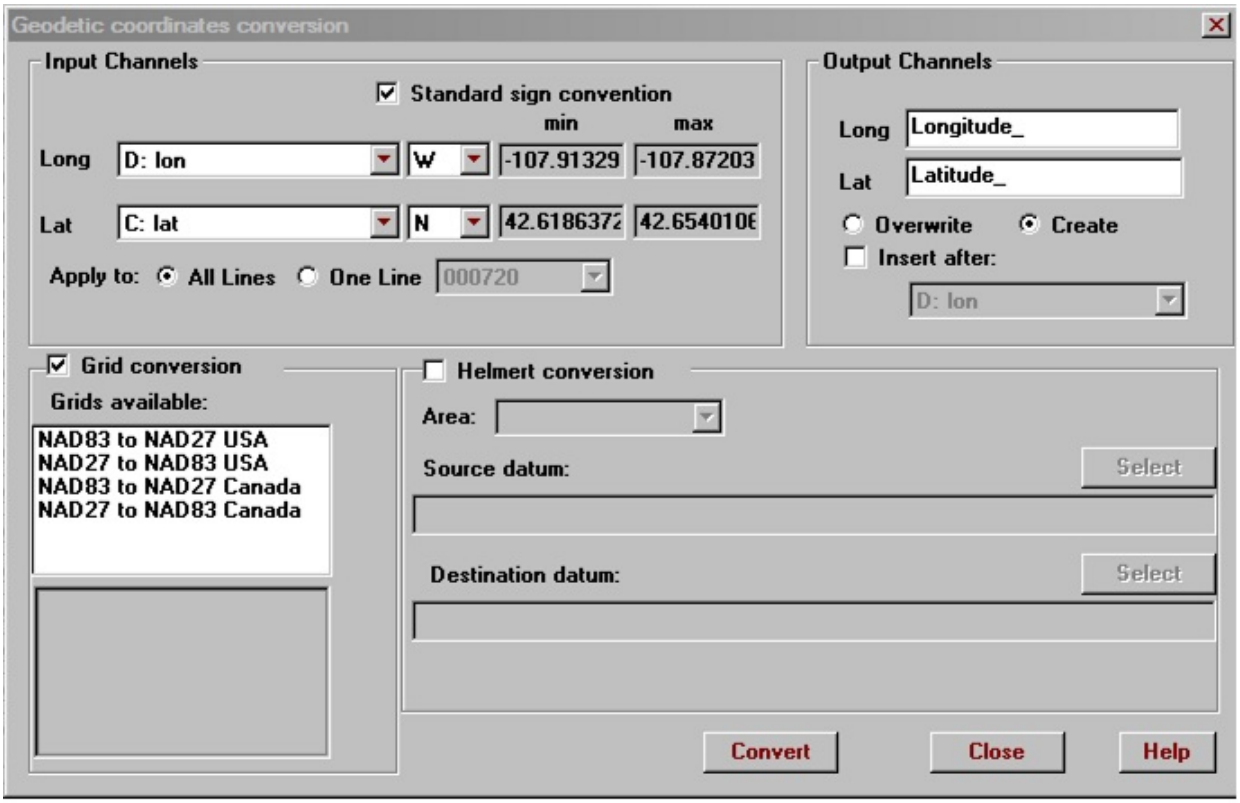
- To close a file, select it from the **Files** list and click **Close**.

***Note**. When you have a blank screen, click the*  ,  *and*  *buttons on the main QC toolbar. This will automatically restore the tables, plots and grids you hid last.*

Coordinate Tools

Geographic Datum Conversion

To convert coordinates between two different datums, select **Tools/Coordinates/Geographic Datum Conversion** to open the following window:



In the **Input Channels** section:

- Select the channels you want to convert from the dropdown lists. The Lat and Long channels are recognized automatically.
- Select the axis direction between W and E, N and S in accordance with your data.

If longitude or latitude is negative, W and S are selected automatically. Otherwise, you have to select the appropriate direction yourself.

Standard sign convention is deselected to indicate that negative values are opposite to the selected direction. e.g. -106W is the same as 106E

In the **Output Channels** section:

- You can leave the default header names offered by QCTool or change them as desired. Select **Insert before source columns** for the new columns to be inserted in that position otherwise they will be inserted as the last columns.

To convert from UTM to Latitude/Longitude:

- Click on the respective option in the upper left-hand section of the dialog.

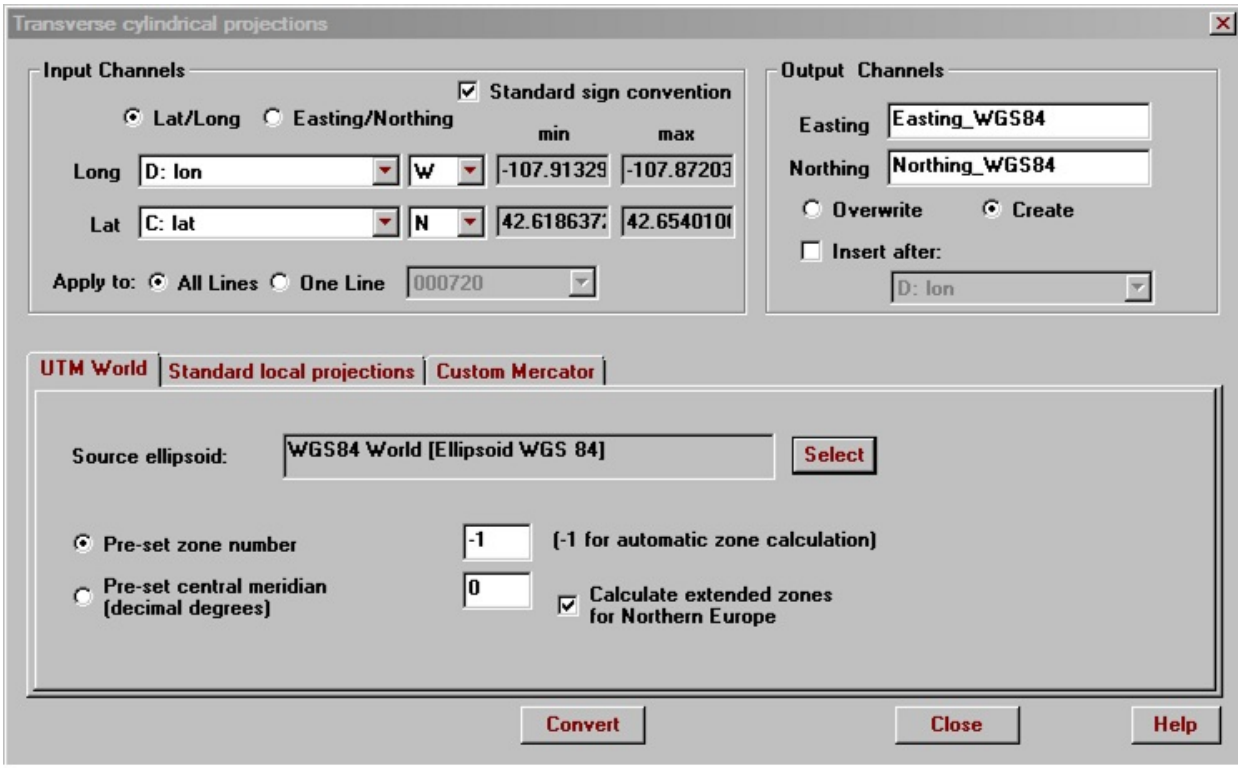
If grids are available, for more accurate results, you may make a selection in the **Grids available** box and the checkbox for **Grid conversion** will be selected.

Otherwise you may select the checkbox for **Helmert conversion** and then select the **Area**, **Source datum** and **Destination datum**.

Click **Convert** to create the new channels

Transverse cylindrical projections

To convert between Latitude/Longitude and UTM using cylindrical projections, select **Tools/Coordinates/Geographic<->Projected coordinates/Transverse cylindrical projections** and the following window opens:



In the **Input Channels** section:

- Select the channels you want to convert from the dropdown lists. The Lat and Long channels are recognized automatically.
- Select the axis direction between W and E, N and S in accordance with your data.

If longitude or latitude is negative, W and S are selected automatically. Otherwise, you have to select the appropriate direction yourself.

Standard sign convention is deselected to indicate that negative values are opposite to the selected direction. e.g. -106W is the same as 106E

In the **Output Channels** section:

- You can leave the default header names offered by QCTool or change them as desired. Select **Insert before source columns** for the new columns to be inserted in that position otherwise they will be inserted as the last columns.

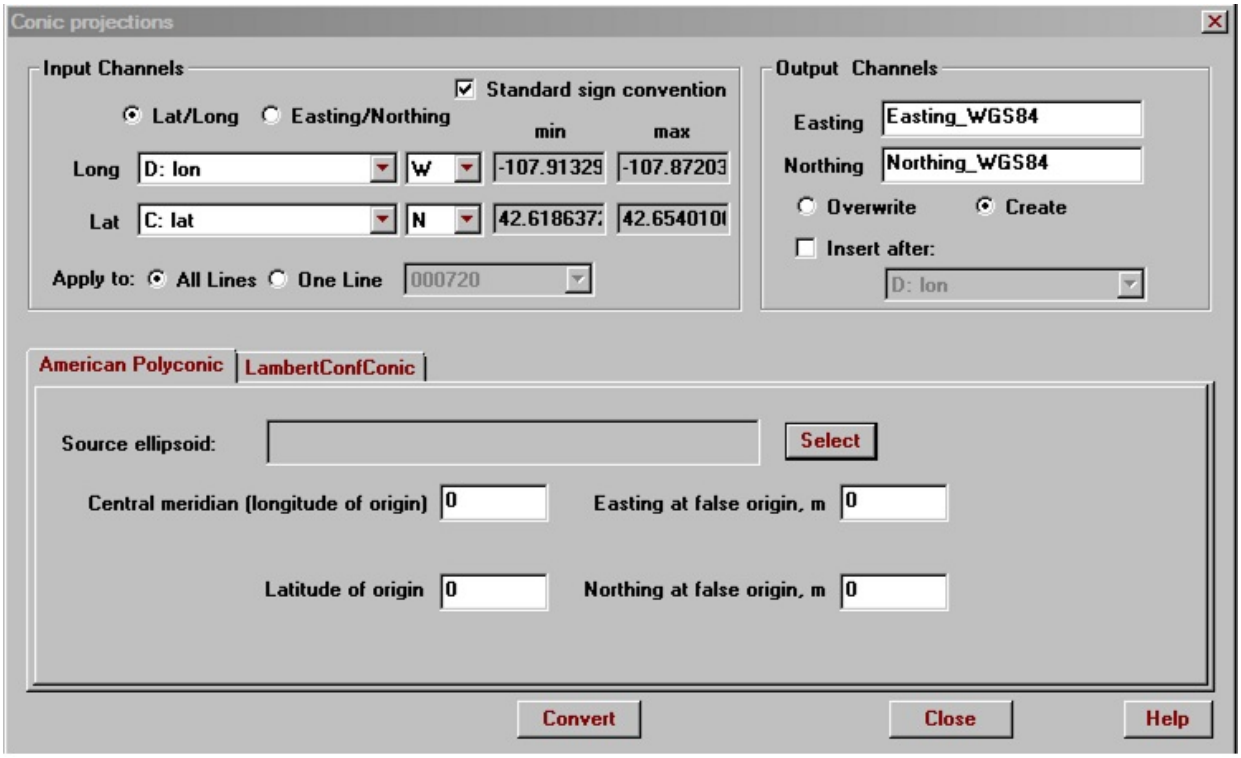
Select the **UTM World** tab to use the cylindrical projection formula defined in the document DMATM 8358.2 by the US Defense Mapping Agency

Standard local projections contains projections for specific locations in Europe and Africa and therefore will yield better results. Most European projections in the list are Gauss-Kruger projections. Otherwise, they are Mercator projections. Click the **See info** button for more information and map for each entry. You might be required to select the **Zone** for the projection you have selected.

Custom Mercator allows you to select the location of the **natural origin** and **false origin** as well as the **Scale factor** at the natural origin. Generic Mercator USGS formulas are used to calculate the result.

Conic Projections

To convert between Latitude/Longitude and UTM using conic projections, select **Tools/Coordinates/Geographic<->Projected coordinates/Conic projections** and the following window opens:



In the **Input Channels** section:

- Select the channels you want to convert from the dropdown lists. The Lat and Long channels are recognized automatically.
- Select the axis direction between W and E, N and S in accordance with your data.

If longitude or latitude is negative, W and S are selected automatically. Otherwise, you have to select the appropriate direction yourself.

Standard sign convention is deselected to indicate that negative values are opposite to the selected direction. e.g. -106W is the same as 106E

In the **Output Channels** section:

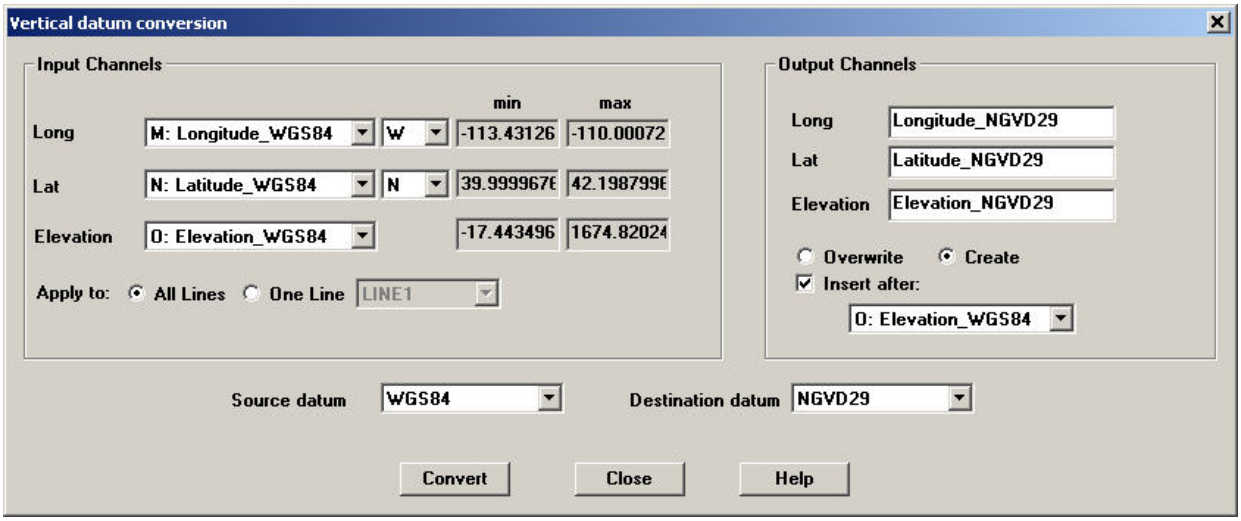
- You can leave the default header names offered by QCTool or change them as desired.
- Select **Insert before source columns** for the new columns to be inserted in that position otherwise they will be inserted as the last columns.

Select between **American polyconic** or **Lambert conformal conic** projections

Specify ellipsoid and the location of the origin and false origin on the tab you have selected. You may also specify a scale factor as well as standard parallel for the Lambert Conformal Conic option.

Vertical datum conversion

Select **Vertical datum conversion** from the **Tools/Coordinates** menu. The following window appears:



Input Channels Select the longitude channel in the combo box labelled **Long**. West or East will be selected by default depending on the sign of the longitude values.
Select the latitude channel in combo box labelled **Lat**. North or South will be selected by default depending on the sign of the latitude values.
Select the elevation channel in combo box labelled **Elevation** You may apply the conversion to **All Lines** or only **One Line**

Output Channels
The results can be saved to new channels by clicking the button labelled **Create**. The names of the new channels will be the those specified in the boxes above the **Create** button. New channels are inserted after the elevation channel selected for input. To insert the channels in a different location, select the **Insert after** checkbox and the channel that the new channels should be inserted after.
You may also overwrite existing channels by selecting the button labelled **Overwrite**. The channels that will be be overwritten can be

selected using the combo boxes above the **Overwrite** button.

Datums that are supported are NGVD29, NAVD88, NAD83 and WGS84. A conversion can be made between any two of these datums. Select the **Source datum** and **Destination datum** in the respective combo boxes.

Click **Convert** to process the data.

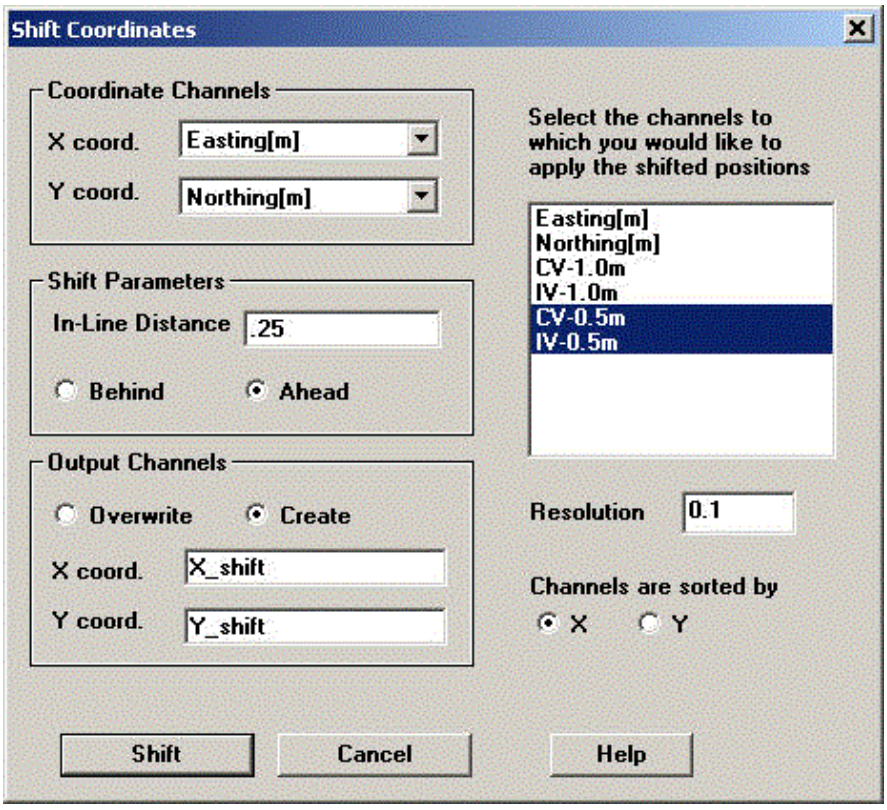
Shift Coordinates [Lag]

In some cases, you need to shift your coordinates by the distance between the receiver and the GPS station if, during the survey, they were located in different places.

The algorithm determines the horizontal tangent to the instantaneous position and adjusts forward or back according to the tangent vector.

Values in other channels can be updated to use the new shifted positions.

Select **Shift Coordinates** from the **Tools/Coordinates** menu. The **Shift Coordinates** window appears.



Coordinate Channels
Select the input channels in this section.

Shift Parameters

Specify the **In-Line Distance** in the related box. Select either the **Behind** or **Ahead** options depending on the receiver position relative to the GPS station.

Output Channels
Select the names of the new output channels containing the shifted coordinates when **Create** is selected. Select the channels to be overwritten when **Overwrite** is selected. Channels selected elsewhere in this tool cannot be selected to be overwritten.

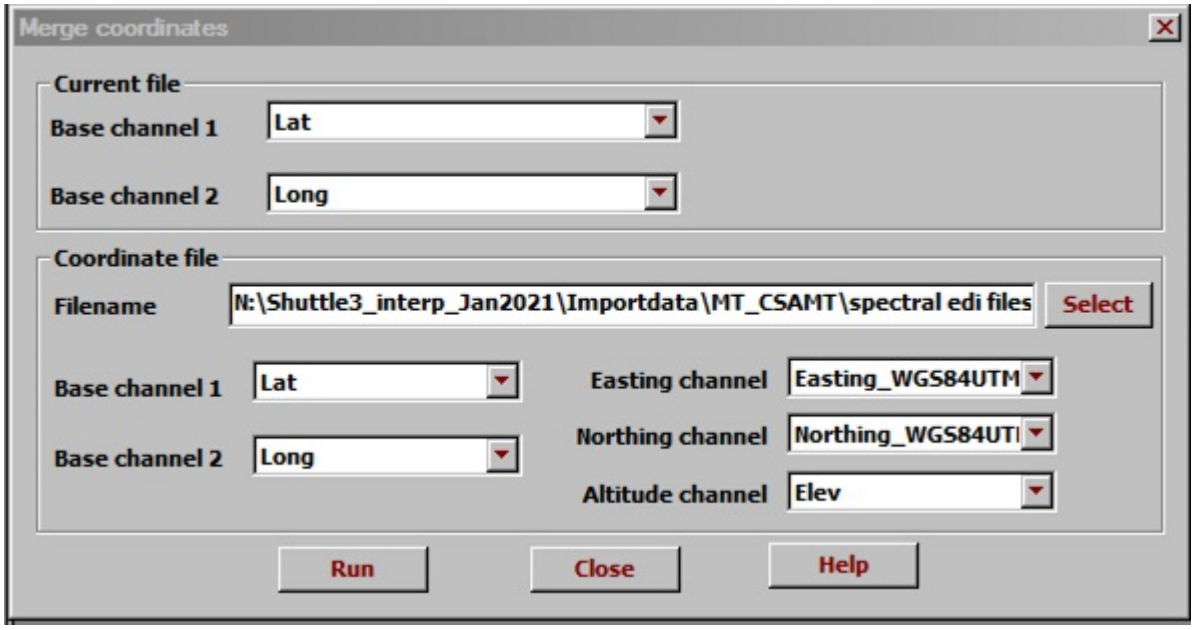
Channel Update
Select channels from the list box to the right if you would like them to be associated with the shifted coordinates instead of the original coordinates. This process requires that one of the original coordinate channels is sorted. Specify which channel by selecting **X** or **Y** under the label **Channels are sorted by**. This channel selection is enabled when a channel in the list box is selected. If the distance from the shifted coordinates to the original coordinates is less than the **Resolution** value, the updated value for the selected channel will be placed on the same row of the original coordinates. Otherwise, a new row will be created. New channels are created for the updated values with names based on the original channel name and the shift distance.

Click the **Shift** button to shift the coordinates and to add the new data to the spreadsheet.

Integrate Coordinate data into QCT file from another file

The purpose of this operation is to integrate GPS measurements into QCT file.

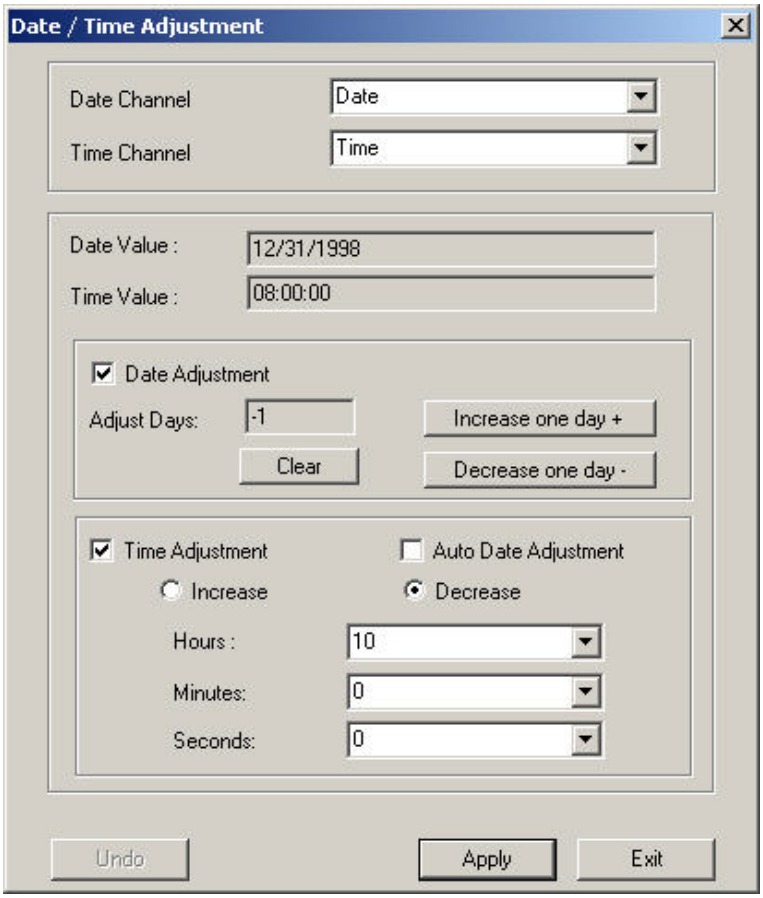
- Select menu item "Tools\Coordinates\Merge coordinates". The dialog appears.



- Press "Select" button and choose file with GPS data.
- Each point position is identified by "Line" and "Station" channels. Select these channels for current .QCT file and selected Elevation file.
- Select "Northing", "Easting" "Altitude" channels from GPS elevation file.
- Click "OK" to execute..

Adjust Date/Time

Select **Tools/Coordinates/Adjust Date / Time** and the following window opens allowing an shift in Time and/or Date



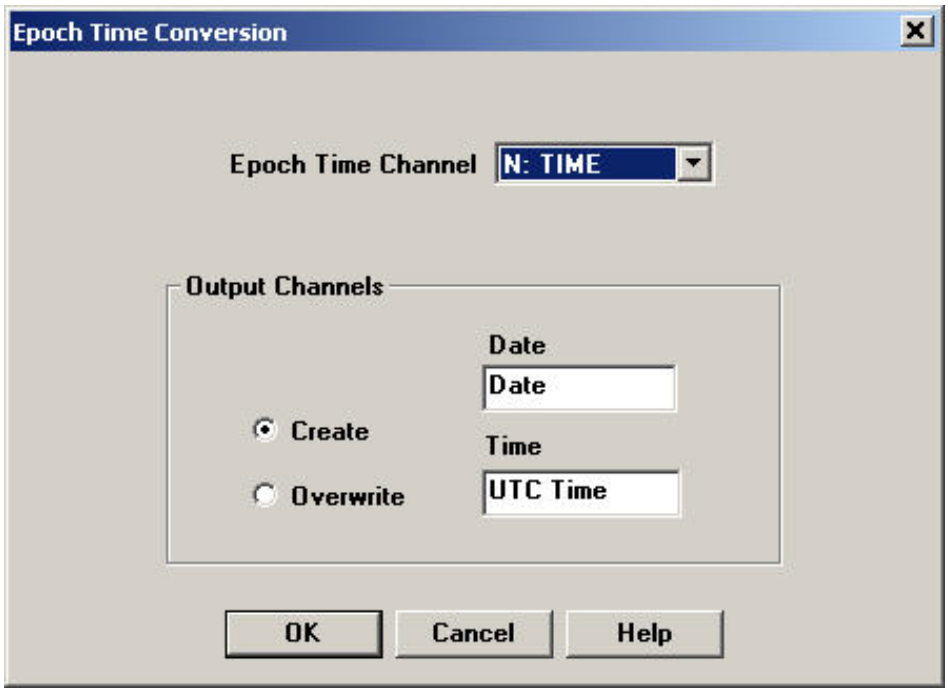
- Select the channels for date and time at the top of the window and example values from the first row appear in the boxes labelled **Date Value** and **Time Value**
- Select the **Date Adjustment** checkbox if you would like to shift the date, Select the **Time Adjustment** checkbox if you would like to shift the time. Select the **Auto Date Adjustment** to change both date and time channels according to the changes in time. This means if the time value goes past midnight, the related date value will be adjusted accordingly.
- Specify how much you would like to add to or subtract from the date and/or time then click **Apply**. The **Undo** button will then be enabled in case

you don't like the results.

Epoch Time Conversion

Epoch time is the number of seconds that have elapsed since January 1, 1970 at midnight UTC time. This tool will convert epoch time to a standard date and UTC time.

Select **Coordinates/Convert Epoch Time** from the **Tools** menu. . The following window appears:

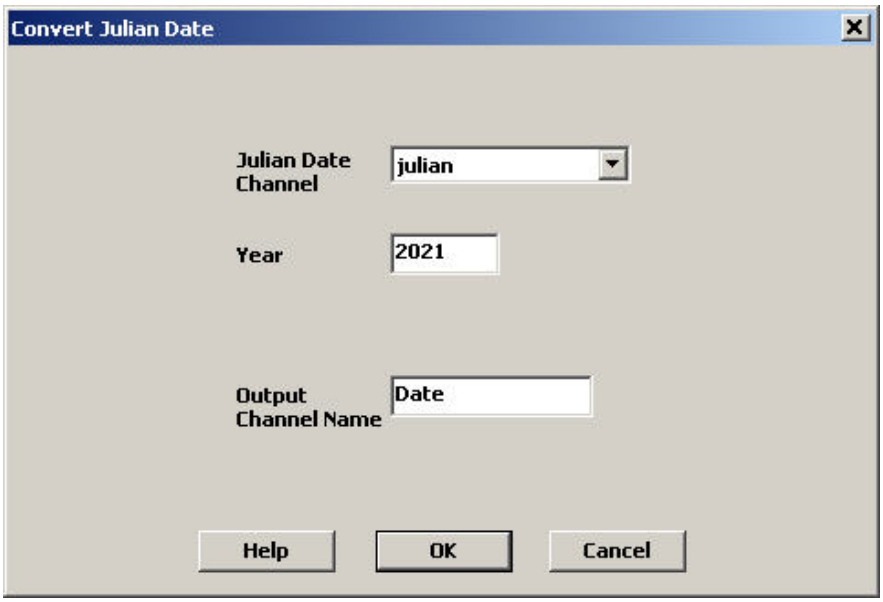


- Select the **Epoch Time Channel** in the drop down list at the top of the window.
- Select **Create** to place the output data in a new channels using the entered names.
- Select **Overwrite** to overwrite existing columns that can be selected in the drop down lists.
- Click the **OK** button when all the settings are correct.

Convert Julian Date

This tool will convert a Julian value for a year to a date in the format YYYY/MM/DD. The Julian values can be within the range between 1 and 366 inclusive. Any values that cannot be converted to a date will be displayed as an asterisk.

Select **Tools/Coordinate/Convert Julian Date** from the menu. The **Convert Julian Date** window appears:



Select the channel whose values you would like to convert in the selection box labelled **Julian Date Channel**

Specify the year that these Julian dates apply to in the **Year** box.

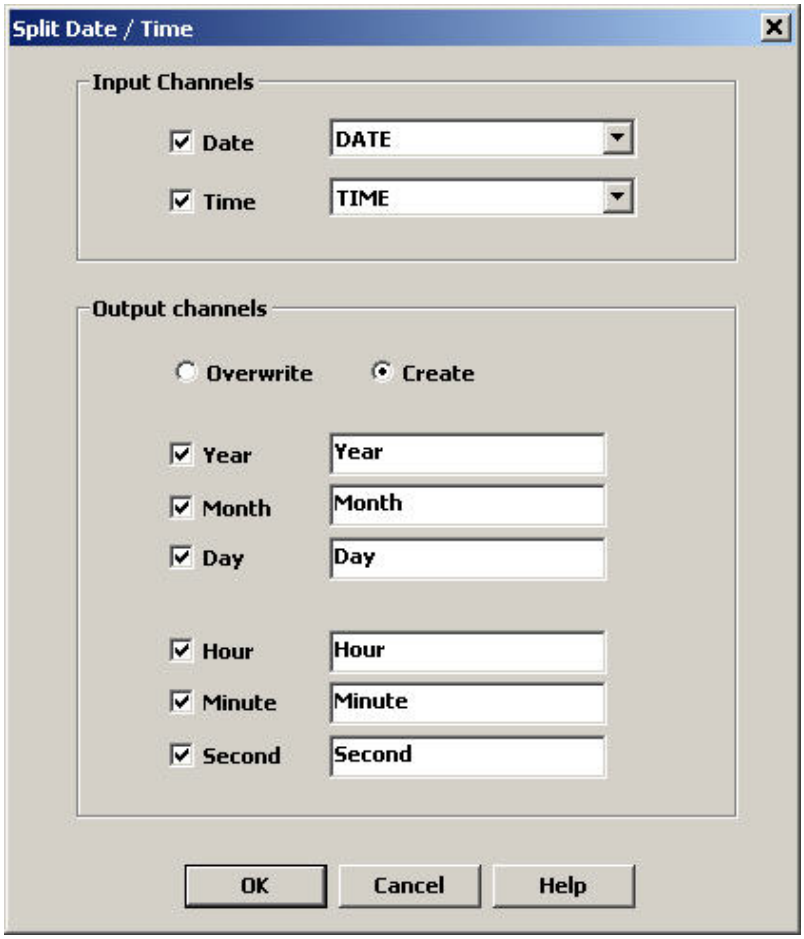
Select name for the new channel containing the converted date in the **Output Channel Name** box.

Click OK to create the new channel

Split Date/Time

This tool will split a date channel into year, month and day channels. For a time channel it will be split into hour, minute and second channels.

Select **Tools|Coordinates|Split Date/Time** from the menu. The **Split Date/Time** window appears:



Input Channels Select channels for **Date** and **Time**. Deselect the appropriate checkbox if you only want to split only one channel. At least one channel needs to be selected.

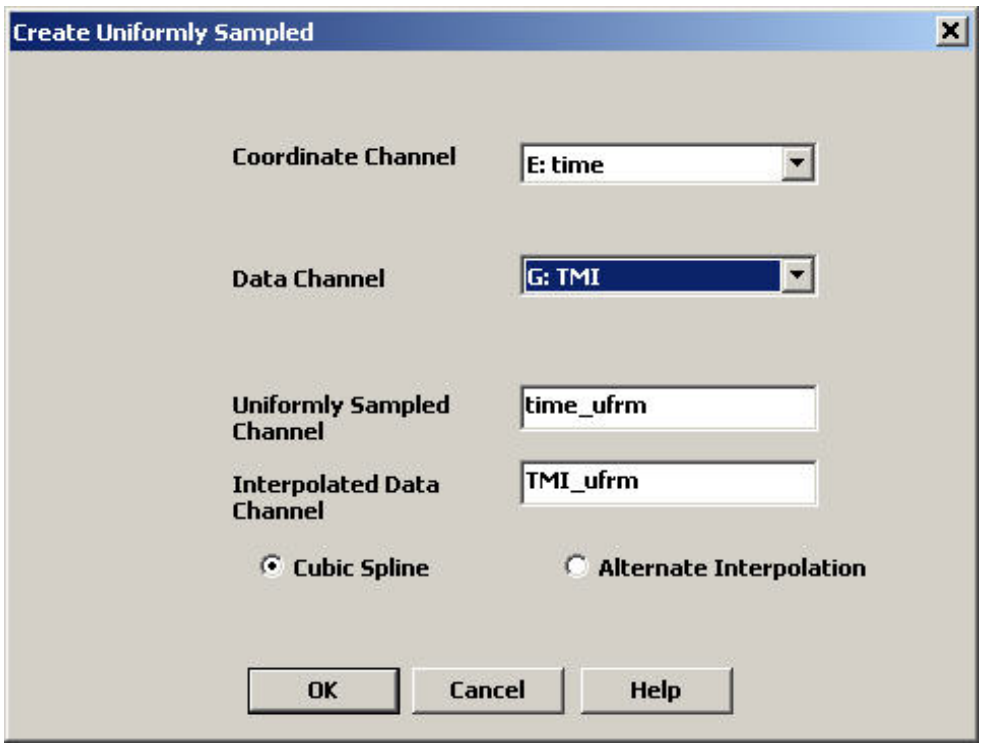
Output Channels
Using the checkboxes, select the channels that you would like to output.

Select **Overwrite** to put the output in existing channels. Select **Create** to create new channels for the output.

Create Uniformly Sampled

Data is often not collected uniformly sampled either spatially or in time. For some purposes, filtering is required to be used with uniformly sampled data. An example is DFT filtering or spectral analyses.

Select **Coordinates/Create Uniformly Sampled** from the **Tools** menu. . The following window appears:



- Select two channels for the **Coordinate Channel** and **Data Channel** Which will be modified so that the sampling rate of the values will be constant. The name of the resulting channel will be the name entered for **Uniformly Sampled Channel**.

- A cubic spline algorithm is used to interpolate new values at the uniformly sampled coordinates based on the values in the **Data**

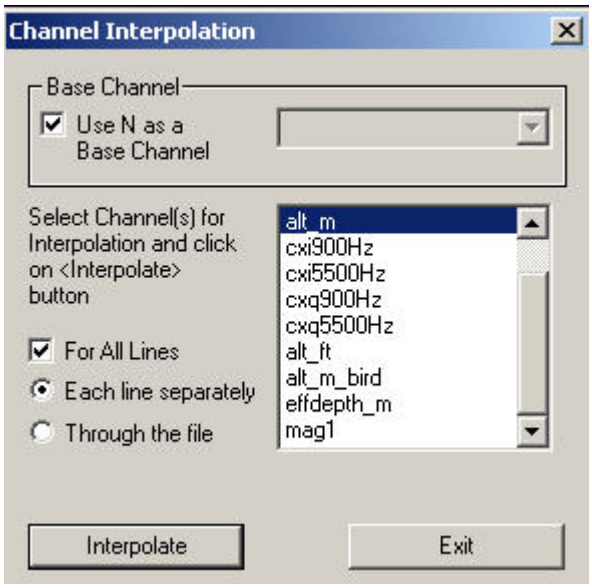
Channel. The result is saved in a column using the name entered for **Interpolated Data Channel**.

- Select **Alternate Interpolation** to use a different mathematical version of a cubic spline interpolation.
- Click the **OK** button to create the new channels.

Interpolating Data

This functionality is accessible only if your data contains a base channel (e.g. Fiducial) which may be both in an ascending and descending order. Interpolation is especially useful when you merge two or more files and there are a lot of missing data as a result. You can also use it to replace existing data that you consider doubtful by interpolated. In this case, select these data, right-click on them, and choose **Set Dummy Values** from the popup menu to appear. This will substitute your data with asterisks.

- Select **Tools/Data Channels/Interpolate Channels**. The **Channel Interpolation** dialog appears:

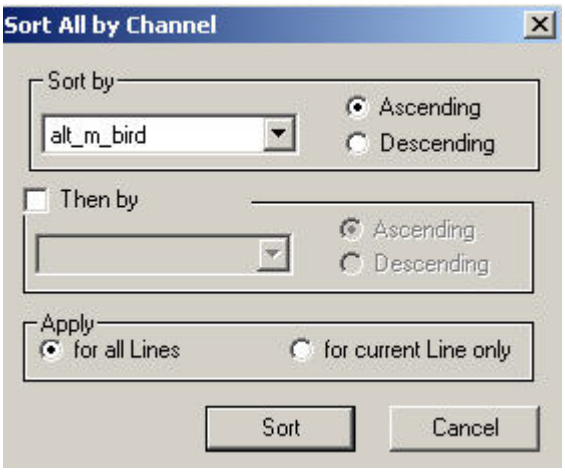


- Choose the base channel from the dropdown list in the upper section. If your data contain the Fiducial channel, it is selected by default. To use the record number **N** as a base channel, check the respective box.
- Select the channel(s) for interpolation from the list below.
- Check the **For All Lines** box to interpolate data in all lines. This box unchecked, your data are interpolated only within the line you are currently working with.
- If you checked the **For All Lines** box, select **Each line separately** if you want interpolation to be line-specific and **Through the file** if you want it to be continuous, throughout all of the lines.
- Click **Interpolate**.

Sorting by Channel

To sort your data by channel:

- Select **Tools/Data Channels/Sort All by Channel**. In the dialog to appear:



- Select the channel to sort your data by from the **Sort by** dropdown list and specify the **Ascending** or **Descending** order on the right.

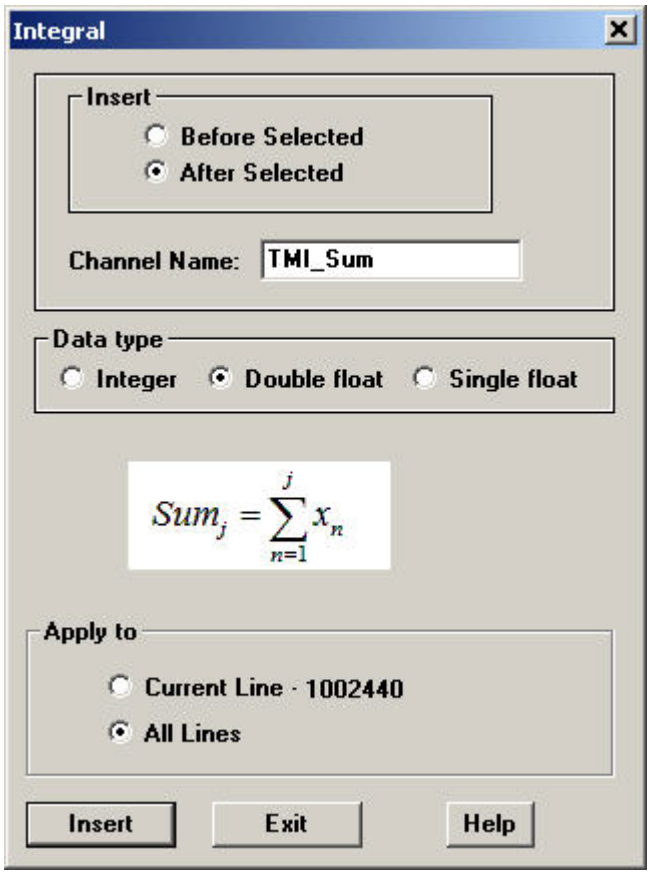
- To specify the secondary sort order of your data, check the **Then by** box. The section below is activated. Select the channel to sort by from the dropdown list and specify the **Ascending** or **Descending** order on the right.

- In the bottom **Apply** section, select **For all Lines** to sort your data by the specified channel throughout all of the lines and **For the current Line only** to sort data within the line you are currently working with.

- Click **Sort**.

Integral

Select the channel you would like to work with by clicking the channel name on the spreadsheet. Select **Tools/Data Channels/Integrals and Derivatives** then **Integral** and the following window opens:



Specify the new channel name and whether you would like the new channel to appear before or after the current channel.

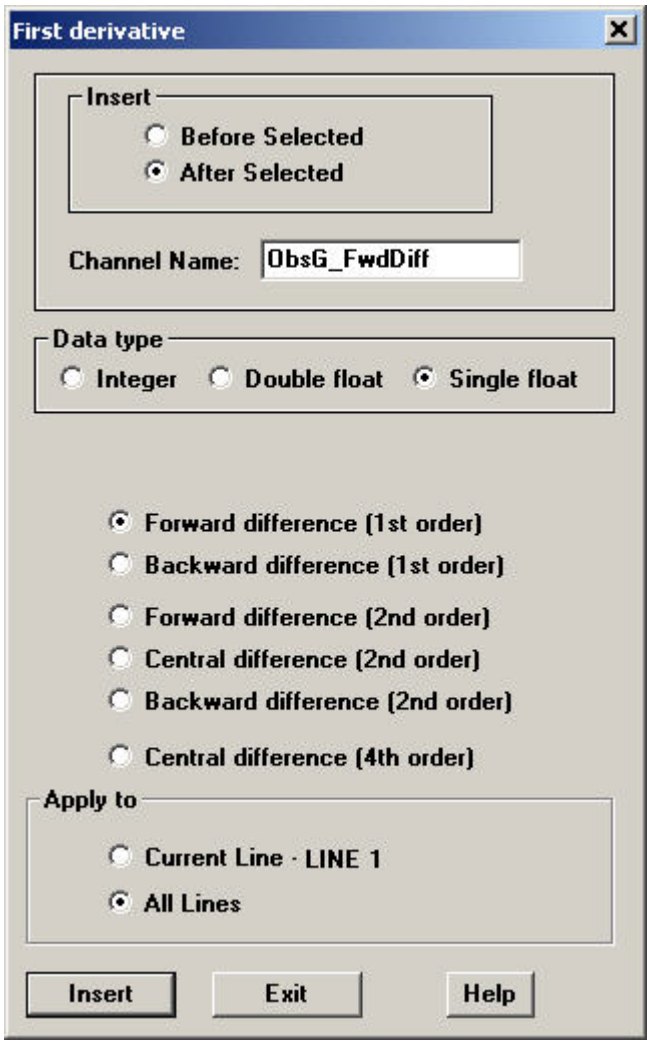
Select the whether you would like the new channel to be integer, double or float in the **Data type** section

Select whether you would like to calculate integrals for only the current line or all lines in the **Apply to** section.

Click the **Insert** button and a new channel will be created.
The value in row n in the new channel will be the sum of all the values from the selected channel in rows 1 to n

Derivative

Select the channel you would like to work with by clicking the channel name on the spreadsheet. Select **Tools/Data Channels/Integrals and Derivatives** then **First Derivative** and the following window opens:



- A **Second Derivative** option is also available but without any first order formulas
- Specify the new channel name and whether you would like the new channel to appear before or after the current channel.

- Select the whether you would like the new channel to be integer, double or float in the **Data type** section
- Select whether you would like to calculate integrals for only the current line or all lines in the **Apply to** section.
- Click the **Insert** button and a new channel will be created.

Below are the formulas used to calculate the derivatives depending the selection made. Values in initial or ending rows may not be calculated.

First order forward difference
f'(x): f(x+1)-f(x)

First order backward difference
f'(x): f(x)-f(x-1)

Second order forward difference
f'(x): {-3f(x)+4f(x+1)-f(x+2)}/2
f''(x): 2f(x)-5f(x+1)+4f(x+2)-f(x+3)

Second order central difference
f'(x): {f(x+1)-f(x-1)}/2
f''(x): f(x+1)-2f(x)+f(x-1)

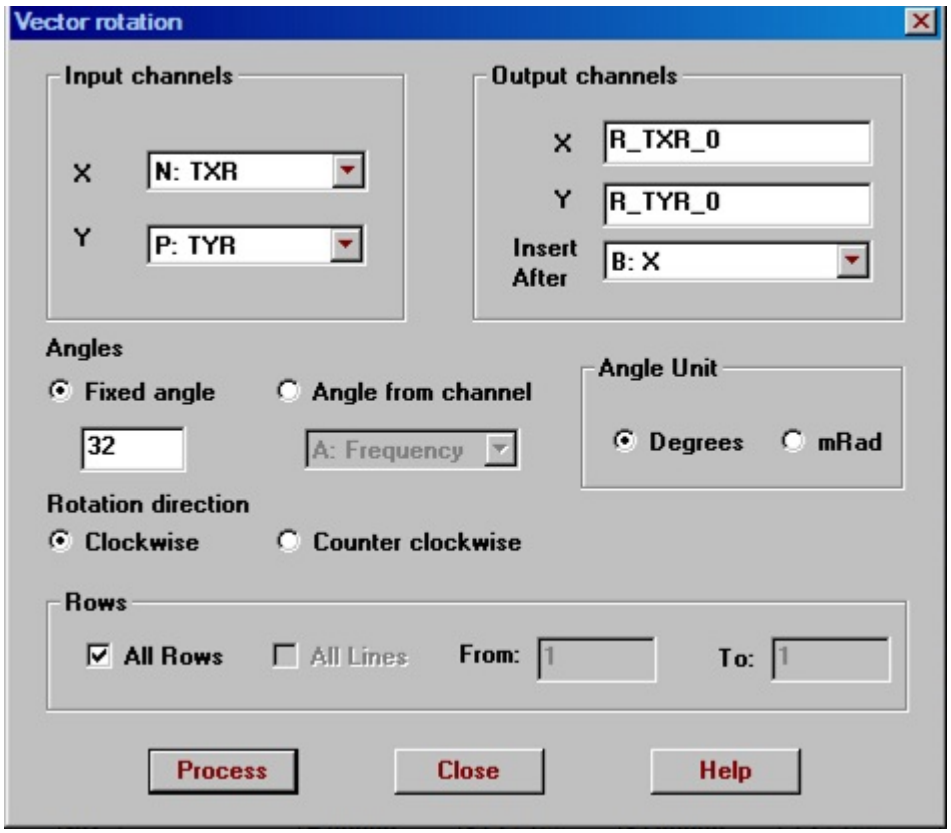
Second order backward difference
f'(x): {3f(x)-4f(x-1)+f(x-2)}/2
f''(x): 2f(x)-5f(x-1)+4f(x-2)-f(x-3)

Fourth order central difference
f'(x): {-f(x+2)+8f(x+1)-8f(x-1)+f(x-2)}/12
f''(x): {-f(x+2)+16f(x+1)-30f(x)+16f(x-1)-f(x-2)}/12

Vector Rotation

Rotate a 2D vector in the plane

Select the channel you would like to work with by clicking the channel name on the spreadsheet. Select **Data Channels/Rotate Vector** from the **Tools** menu. . The following window opens:



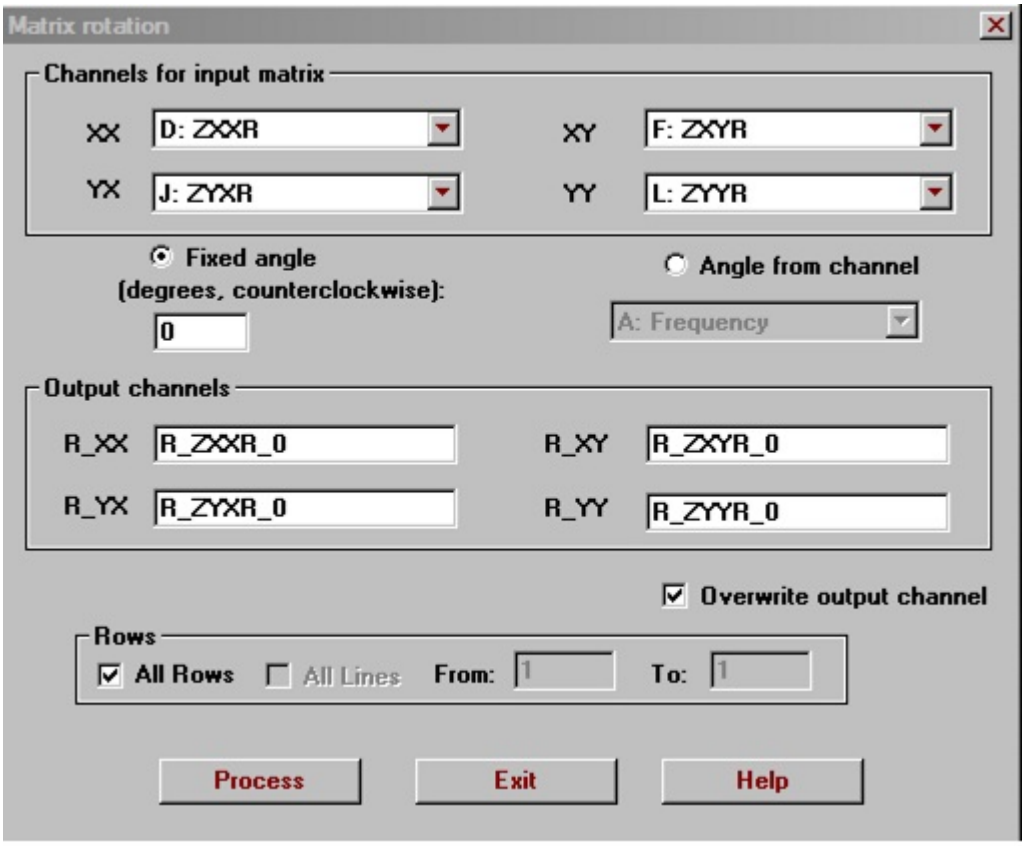
- Select the channels for the x and y components of the vectors to be rotated in the **Input channels** section.
- The **Output channels** section enables the names of the result columns to be selected as well as their insert point on the spreadsheet.
- The rotation angle can either be a **Fixed angle** or multiple angles in the channel selected below **Angle from channel**.
- Units for the rotation can either be degress or millradians, selected in the **Angle Unit** box.

- The direction of the rotation can be selected with the controls labelled **Rotation direction.**
- The **Rows** section of the interface is used to specify which rows and lines should be processed. A range of rows can be selected. Either the current line or all lines can be selected as well.
- Click the **Process** button to rotate the vectors and new channels will be created.

Matrix Rotation

Rotate a 2x2 matrix

Select the channel you would like to work with by clicking the channel name on the spreadsheet. Select **Data Channels/Rotate Matrix** from the **Tools** menu. . The following window appears:



- Select the channels for the elements of the matrix in the **Channels for input matrix** section.
- The names of the result columns can be selected in the **Output channels** section.
- The rotation angle can either be a **Fixed angle** or multiple angles in the channel selected below **Angle from channel**. The units must be in degrees.
- If the names selected for the output channels already exist, the existing channels can be overwritten by selecting the **Overwrite**

output channel checkbox. If the channels do not exist or the checkbox is not selected, new channels will be created.

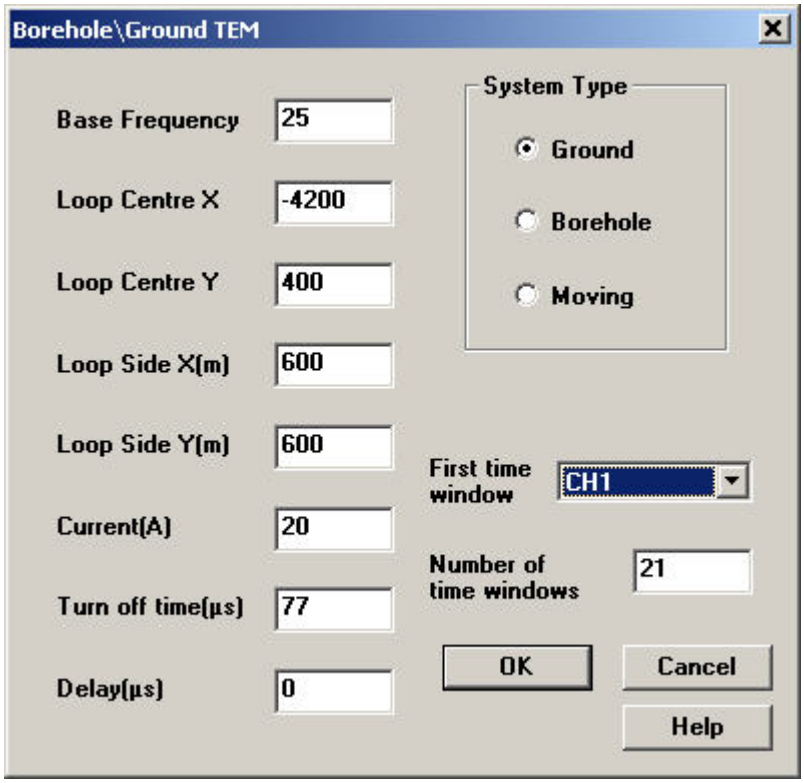
- The **Rows** section of the interface is used to specify which rows and lines should be processed. A range of rows can be selected. Either the current line or all lines can be selected as well.
- Click the **Process** button to rotate the matrix.

Borehole\Ground TEM

This tool creates a qct file that can imported as Geonics data into EMIGMA

Start with a file that has TEM data arranged in columns according to time window. All time window channels need to be contiguous on the spreadsheet. Coordinate channels are also required in the qct file when importing to EMIGMA.

Select **Data Channels|Borehole\Ground TEM** from the **Tools** menu. . The following window appears:



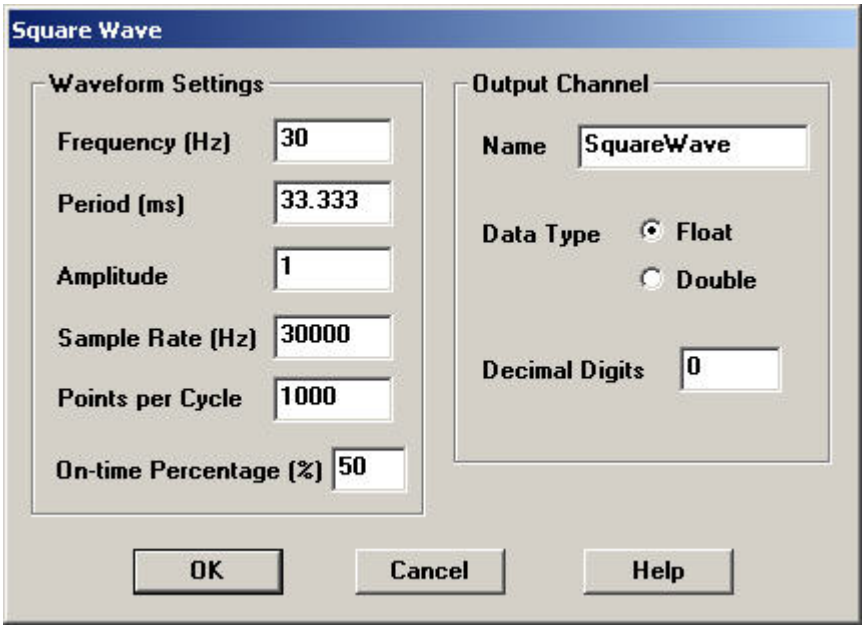
- Select the channel for the **First time window** and enter the number of time window channels in the file in the box labelled **Number of time windows**.
- The possible system types are fixed(ground), moving and borehole.

- Enter the values for the system parameters then click **OK** to add the channels necessary to create a Geonics TEM file.
- Time window start and end times depend on the base frequency that was entered so the base frequency value needs to be one that corresponds to a Geonics TEM instrument.

Square Wave

This tool creates a square waveform

Select **Data Channels/Square Wave** from the **Tools** menu. . The following window appears:

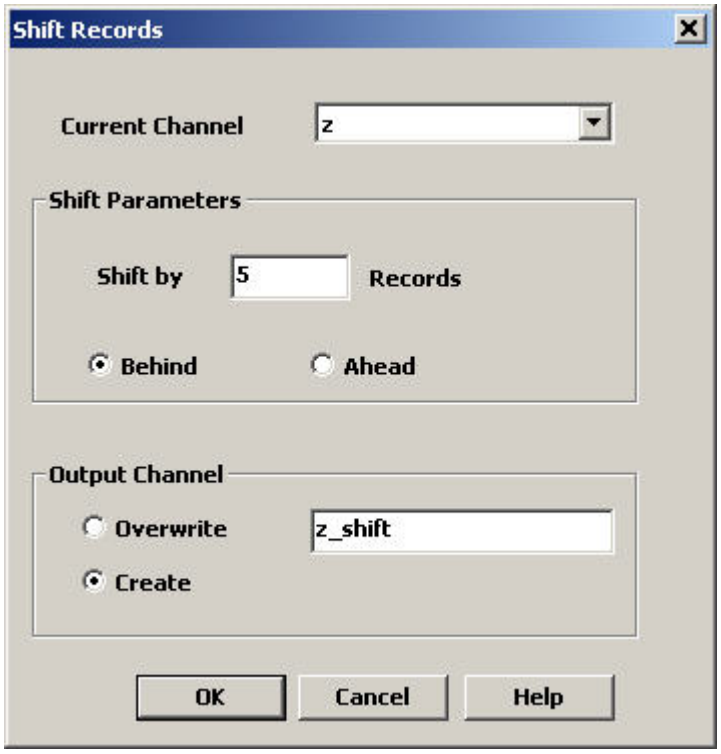


- The Frequency, Period, Sample Rate and Points per Cycle can be specified in the **Waveform Settings** section. Changing one of these settings will update the others. You may also specify the Amplitude and On-time Percentage in this section.
- The name of the result column can be selected in the **Output Channels** section. Also, the data type and number of decimal digits can be specified here.
- Click **OK** to create the square wave.

Shift Records

Values in a channel can be shifted by a number of records.

Select **Tools/Data Channels/Shift Records** from the menu. The **Shift Records** window appears:



Select the channel whose values you would like to shift in the selection box labelled **Current Channel**

Specify the number of records you would like to shift the values in the **Shift Parameters**. The values can be shift either up or down by selecting **Behind** or **Ahead** respectively. If the values are shifted down, dummy data will inserted in the new cells that appear at the top. Dummy data will be inserted at the bottom if the values are shifted up.

Output Channels Select **Overwrite** to put the shifted values in an existing channel. Select **Create** to create a new channel with the

specified name for the result.

Fill Character Channel

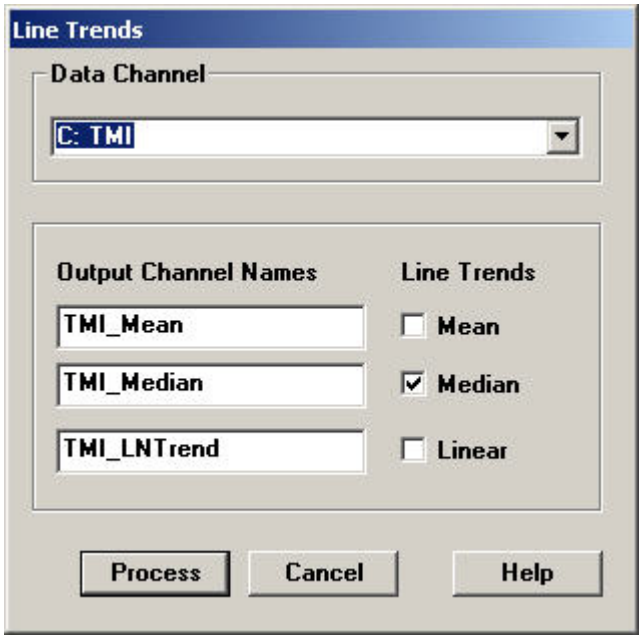
Select the channel you would like to work with by clicking the channel name on the spreadsheet. Select **Tools/Data Channels/Fill Character Channel** and the following window opens:

- Enter the text that you would like to save to the character channel in the box labelled **New Text**
- In the **Apply to** section, selecting **All Lines** insert the new text in all lines of the channel or else only the current line will be affected. The new text will be inserted in the range specified by **From Row** and **To Row** unless **All Records** is selected and then every row will be affected.
- You may change the current channel by clicking a new channel on the spreadsheet.
- You may change the current line by selecting a new line on the toolbar.
- Click the **Apply** button and a new channel will be created.

Line Trends

This tool will calculate certain trends in the data on each line or profile. Included are the Mean and Median for each line for the selected channel. Also, a Linear Trend or linear fit to the data for each line can be calculated.

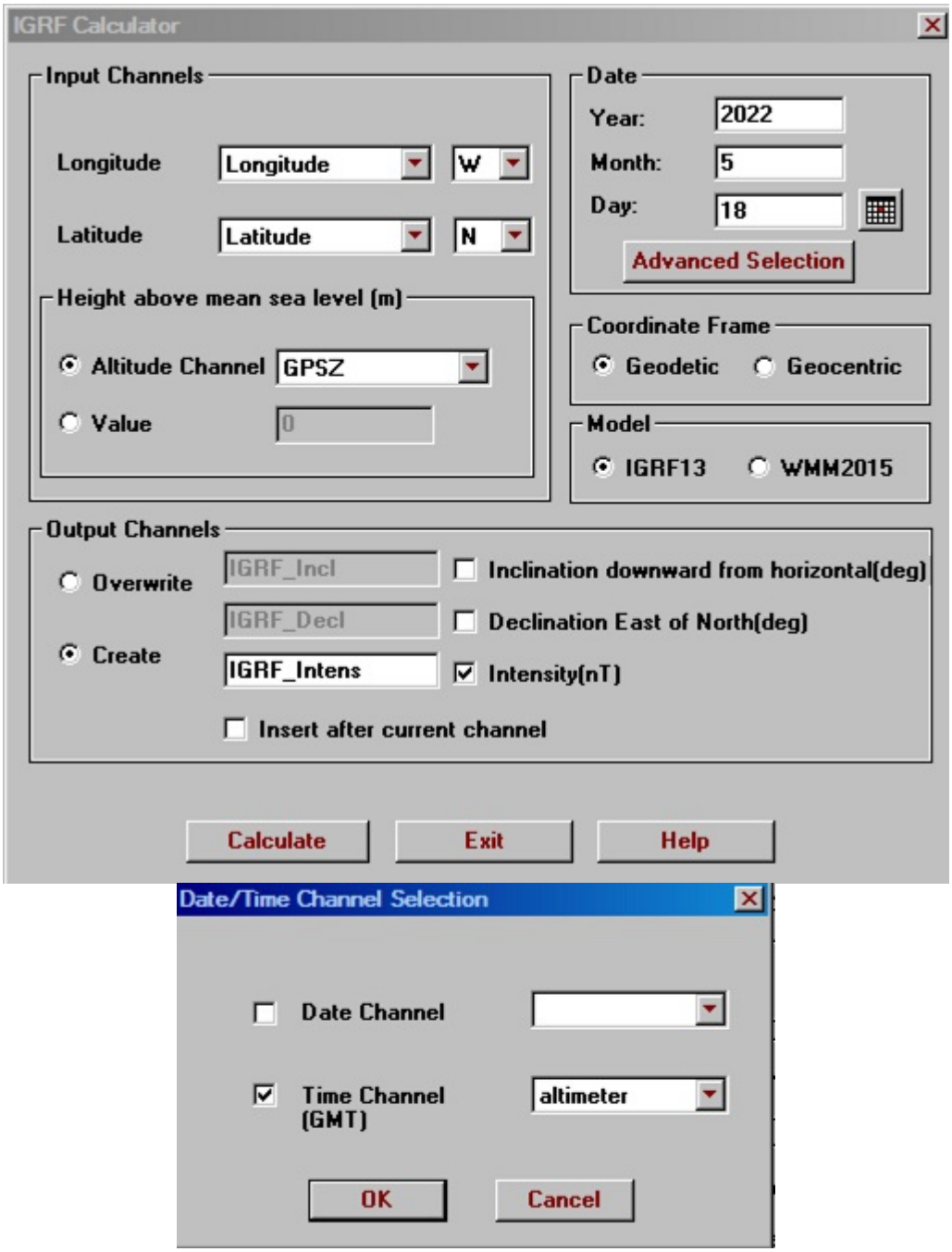
Select **Tools/Data Channels/Line Trends** from the main menu to display the following interface:



- You may choose to calculate **Mean**, **Median** and/or the **Linear** trend by clicking the appropriate checkbox(es).
- The names of the newly created channels containing the processed data can be edited in the boxes below **Output Channel Names.**
- Click **Process** when you are ready to create the new channels.

IGRF Calculator

Select **Processing/Magnetic Data Processing/IGRF Calculator** to open the following window:



To create IGRF channels, in the **Input Channels** section:

- Select the channels you want to convert from the dropdown lists. The Latitude and Longitude channels should be recognized automatically.

- Select between N and S in accordance with your latitude

- The world standard for longitude is that positive longitude is east from Greenwich (Central Meridian) and thus a negative is west. The default setting is thus (E)ast. However, it is quite common particularly in the United States to use a positive longitude to represent west from the Central Meridian. If your longitudes are positive and mean west, then select (W)est.

Specify the **Height of sea level** by selecting the channel which provides this information from the respective dropdown list. You can also select **Value** and enter a height in the respective box if you do not have an elevation channel.

- Make sure that the date and coordinate frame information is correct.

- In the **Model** section, you may select between the World Magnetic Model (WMM2015) or the International Geomagnetic Reference Field model (IGRF12). WMM2015 is only available for years between 2015 and 2019 inclusive. IGRF12 is available for years between 1940 and 2019 inclusive

In the **Output Channels** section:

- Select **Overwrite** if you want to replace existing columns in your table. The inclination, declination, and intensity boxes turn into dropdown lists. Choose the columns to overwrite from these lists.

- Select **Create** if you want the results of your conversion to appear in new columns. You can leave the default header names offered by QCTool or change them as desired.

- Select **Insert after current channel** if you want your created channels to appear after the currently selected channel. The new channels will appear after the last channel otherwise.

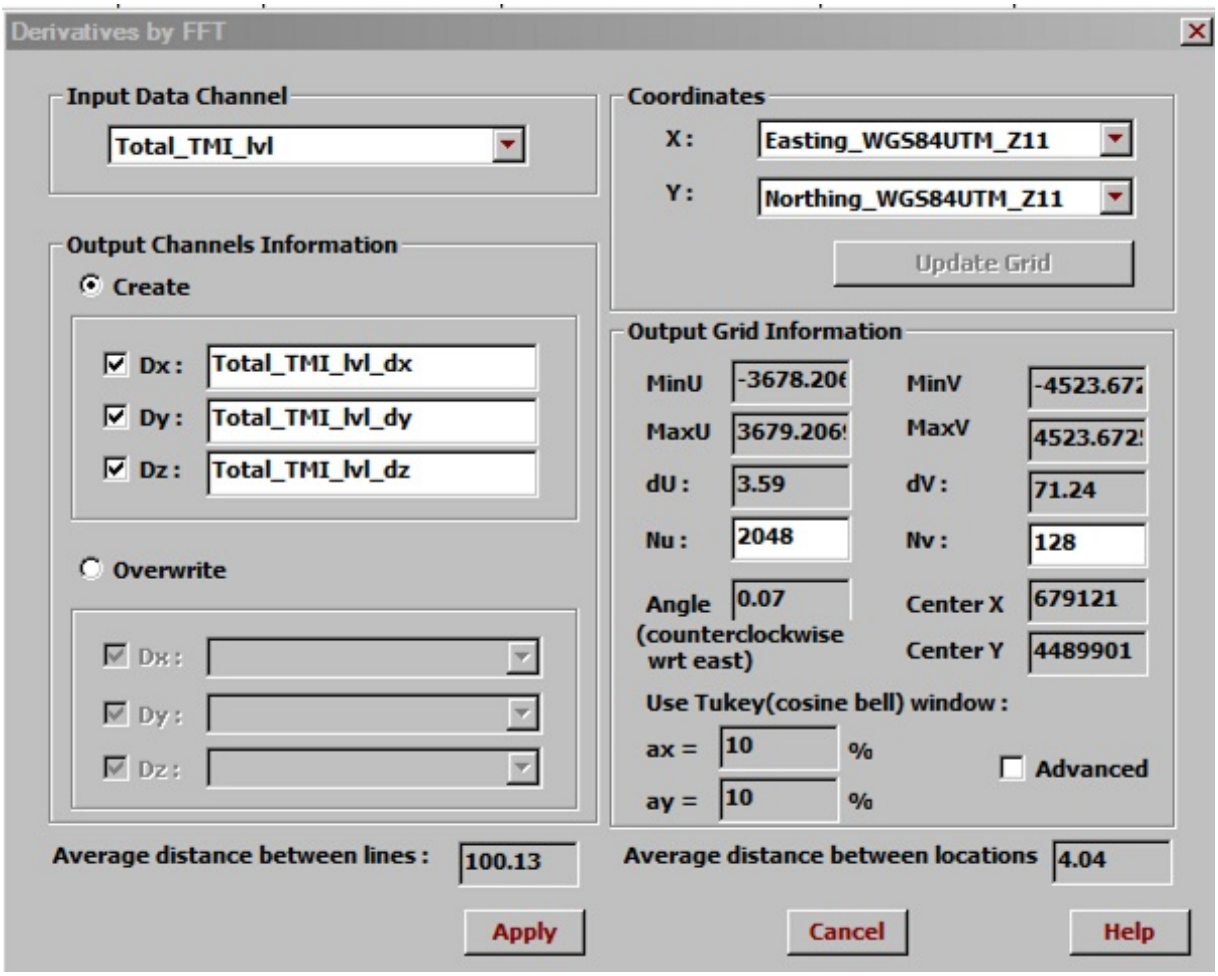
- Click **Calculate**.

Derivatives By FFT

This facility produces spatial derivatives of data via FFT procedures. It can be utilized with any type of data but theoretically it is intended for either magnetic or gravity data. The technique requires interpolation of the Input Data channel onto a grid which 2^{Nu} x 2^{Nv} in size.

Note: Our gridding techniques do not require square grid cells and thus the inline resolution (in this example, 2.42m) and the crossline resolution (200.16 in this example) do not need to be compromised.

Click the Processing -> Gravity/Magnetic data Processing -> Derivatives by FFT. The dialog appears.

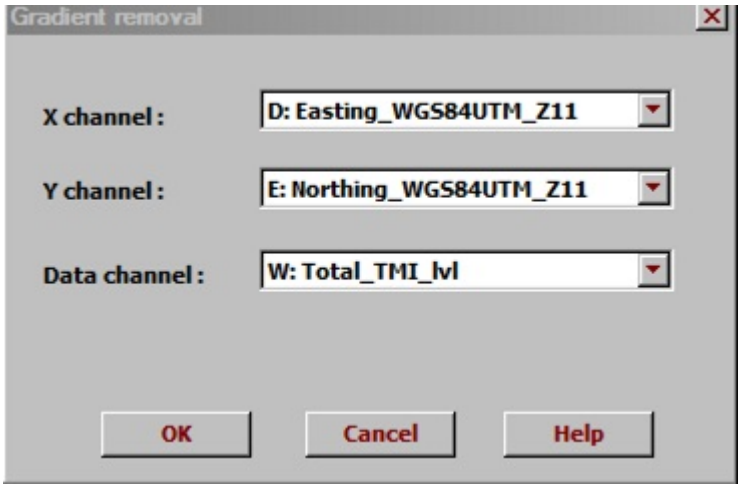


- Select a data channel in the **Input Data Channel** section.
- Select data channels for coordinates
- Click the **ADV** button to enable the **Output Grid Information** section.
- Click Apply to calculate

Gradient Removal

This tool removes a linear trend from your data. It is primarily intended for magnetic data to remove strong linear effects in the regional response.

Select **Gradient removal** from the **Processing/Magnetic Data Processing** menu. The following window appears:

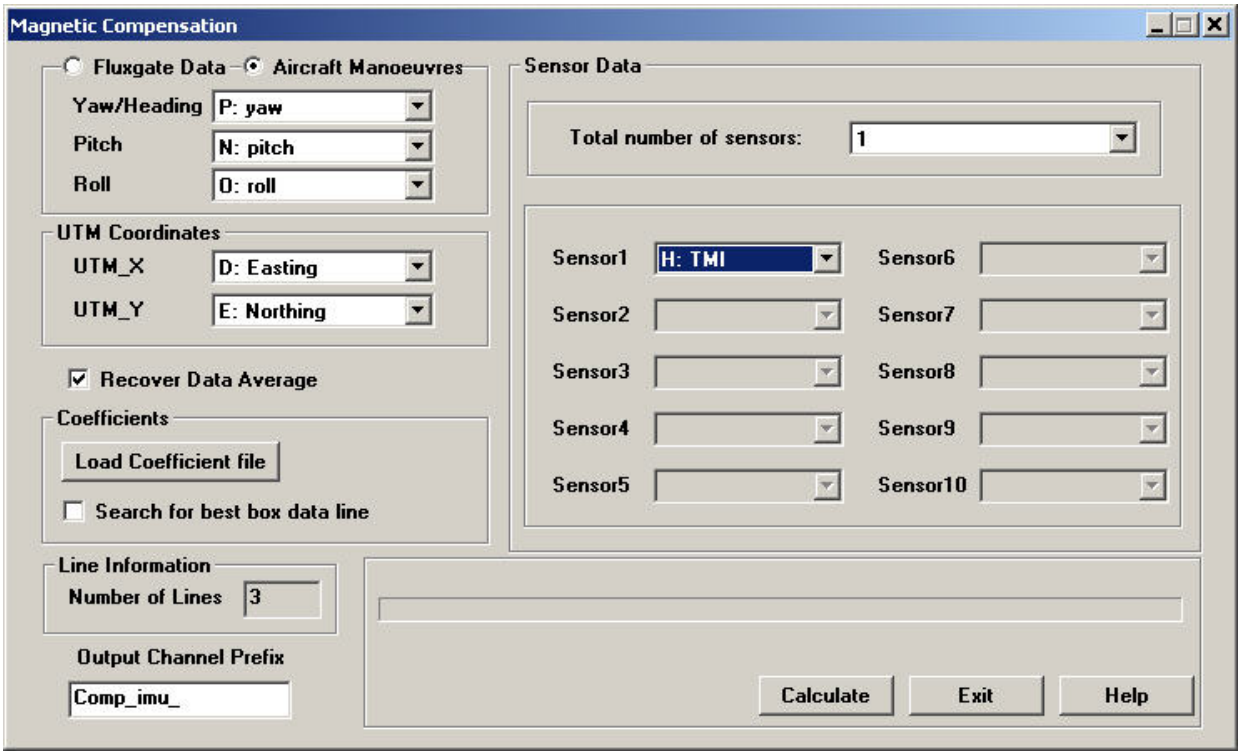


Select the easting channel in the combo box labelled **X channel**. Select the northing channel in combo box labelled **Y channel**. Select the channel with the data that has the linear trend you want removed in the combo box labelled **Data channel**.

The results of the gradient removal will be saved in new channel using the name of the original **Data channel**. A suffix is added to the name of the original channel to obtain the name of the new channel. E.g. if the original channel was named "TMI", the new channel would be named "TMI_NoGradient". The new channel is inserted directly after the original channel.

Magnetic Compensation

Select **Magnetic Compensation** from the **Processing/Magnetic Data Processing** menu. The **Magnetic Compensation** window appears.



If the coefficients are calculated utilizing the fluxgate data then select **Fluxgate Data** or otherwise select channels for yaw, pitch and roll by selecting **Aircraft Manoeuvres**. Select channels for the x and y coordinates in the **UTM Coordinates** section.

Sensor Data Specify the number of sensors in the box labeled **Total number of sensors**. You may select the channels for up to 10 sensors directly below.

Recover Data Average
Select the checkbox to correct the DC level to that of the original data line.

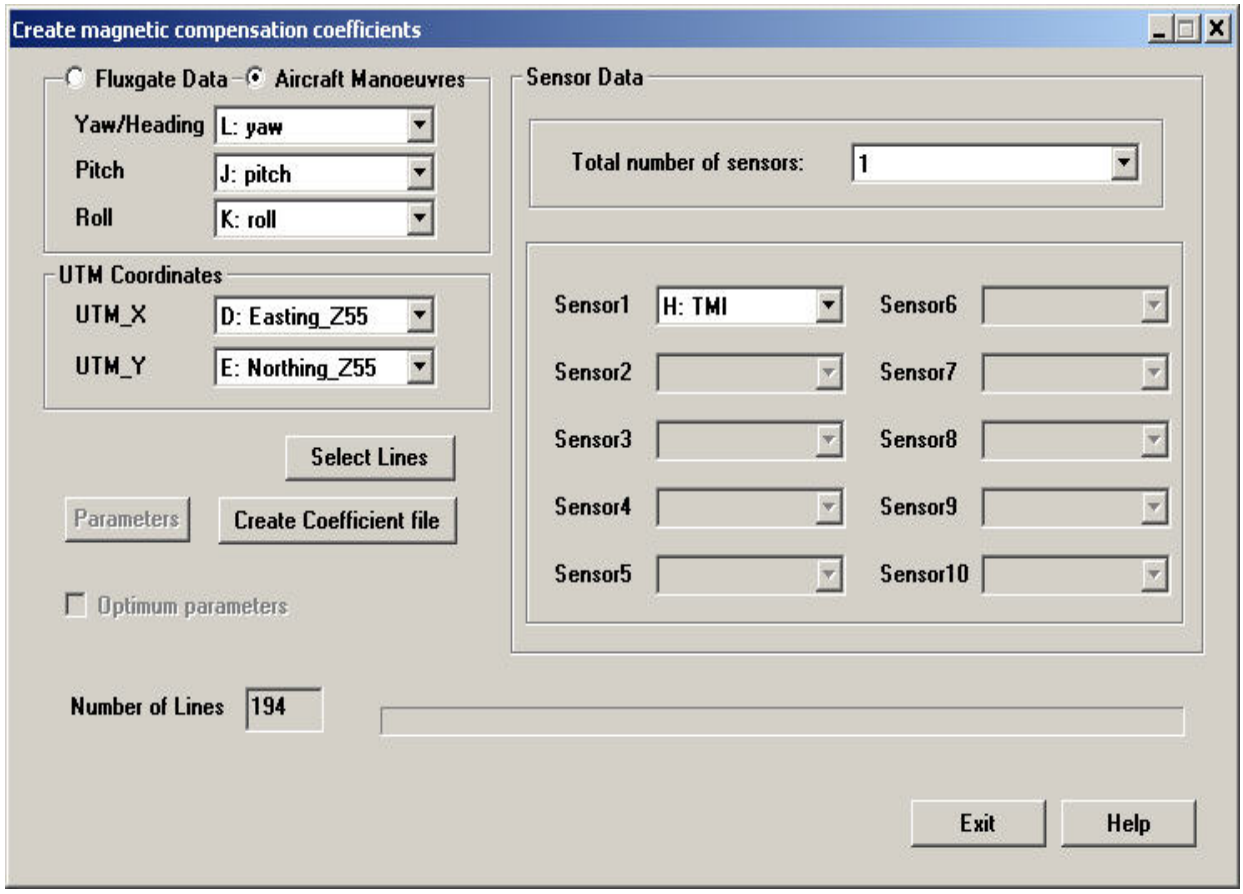
Coefficients
Click the **Load Coefficient File** button to specify the path of your coefficient file. The file can be created by using the Compensation Coefficient tool. The coefficients used for compensation has the angle closest to the line being compensated. The compensated data with the lowest figure of merit can be output instead by selecting the checkbox labelled **Search for the best box data line**.

Output Channel
The compensation results will be saved in a channel with the name entered here. If more than one sensor was specified the name of the sensor channel will be appended to the name entered here.

Click the **Calculate** button to perform the magnetic compensation.

Create Magnetic Compensation Coefficients

Select **Create Magnetic Compensation Coefficients** from the **Processing/Magnetic Data Processing** menu. The **Compensation Coefficient** window appears.



The coefficients can be derived in the traditional manner by using 3 fluxgate sensors or they may be calculated by any set of data that provides orientation. You may select channels for the components of the fluxgate data by selecting **Fluxgate Data** or you may select channels for yaw, pitch and roll by selecting **Aircraft Manoeuvres**. Select channels for the x and y coordinates in the **UTM Coordinates** section.

Sensor Data Specify the number of sensors in the box labeled **Total number of sensors**. You may select the channels for up to 10 sensors directly below.

Parameters
The **Parameters** button is enabled when using fluxgate data. Click it to select the number of terms to use when performing singular value decomposition. The number of terms can be determined automatically by the software instead if you select the **Optimum parameters** checkbox.

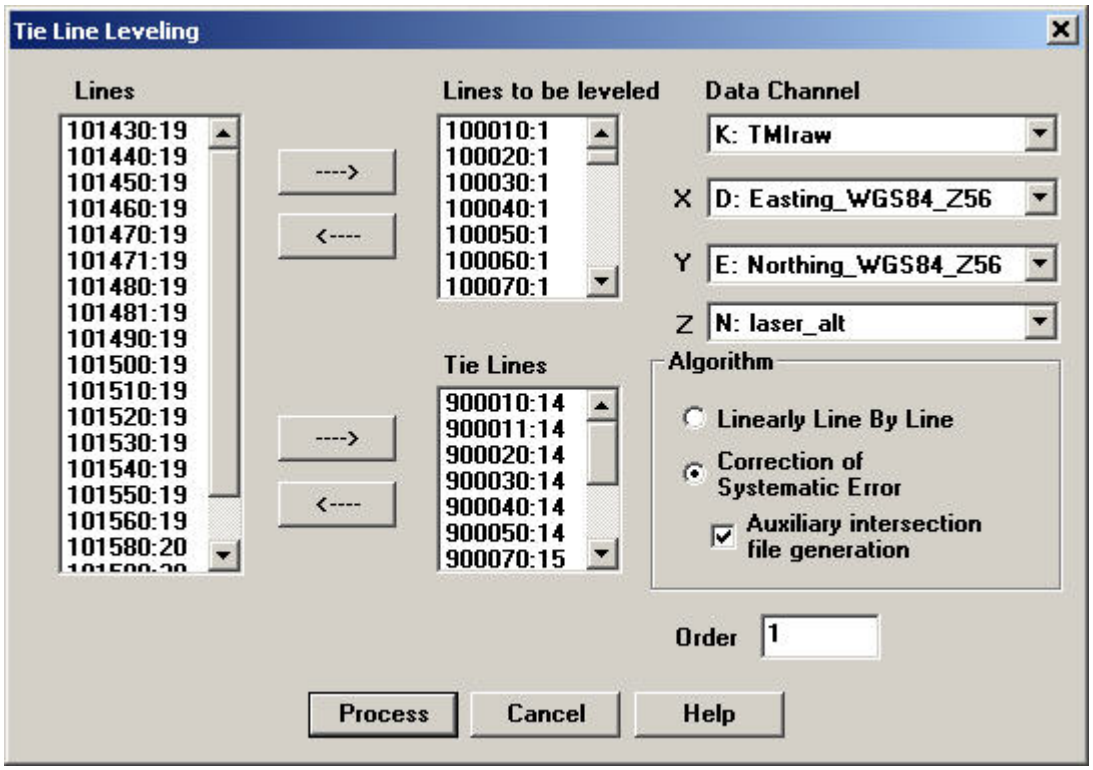
Click the **Select Lines** button if you do not want to calculate coefficients for all the lines.

Click the **Create Coefficient File** button to create and save the coefficient file that will be used when performing magnetic compensation.

Tie Line Leveling

When you have tie lines in your survey, you may use tie line leveling. This algorithm attempts to adjust the data so that data points which are at an intersection have a small a difference between them as possible. This is done by applying corrections between the intersecting points.

Select **Processing/Magnetic Data Processing/Tie line leveling** from the menu. The **Tie line leveling** window appears:



Channel Selection Select channels for **Data**, **X (easting)** and **Y(northing)**. The **Z** (altimeter or GPSZ) channel only needs to be selected if the **Auxiliary intersection file generation** checkbox is selected.

Line Selection
Select **Lines to be leveled** by selecting entries in the **Lines** box and clicking the button with an arrow pointing at the **Lines to be leveled** box. Select **Tie Lines** by clicking the button with an arrow pointing at the **Tie Lines** box. Lines can be removed from the **Lines to be leveled** or **Tie Lines** boxes by

clicking on the respective button with an arrow pointing at the **Lines** box. The resulting output will correct on the Lines and not the Tie Lines.

Linearly Line By Line
This algorithm splits the lines at each intersection and uses linear interpolation. For this algorithm, a line should have at least two tie lines that intersect with it for it to be processed. If a line has less than two intersections, its existing data will be simply copied to the output channel.

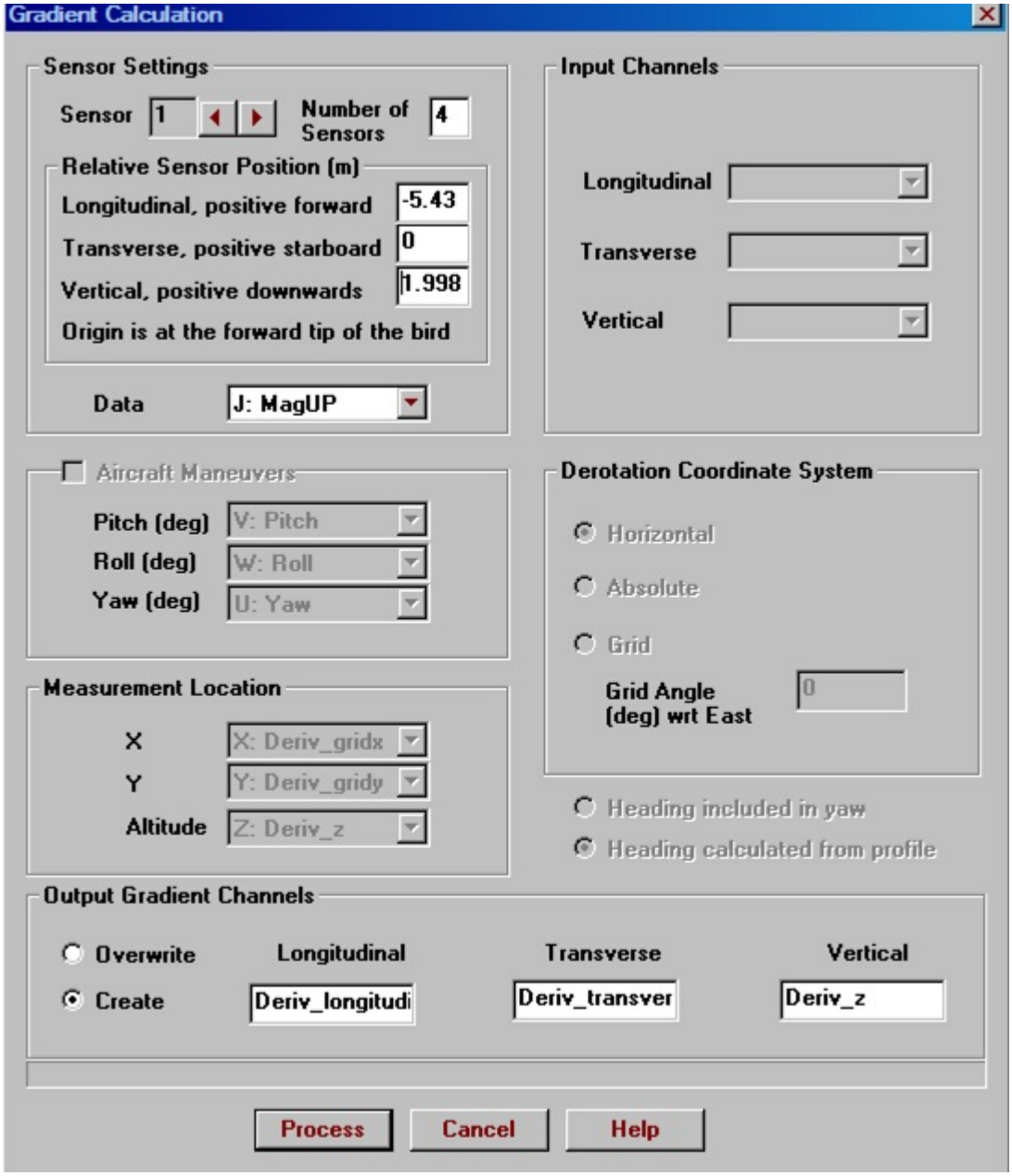
Correction of Systematic Error
This algorithm makes the data adjustment simultaneously utilizing all the lines and tie lines to minimize the data errors at the intersection points. The data errors are represented at the intersection points with a polynomial of a particular order. The **Order** edit box displays the order of the polynomial. It is only enabled for this algorithm. The order value needs to satisfy the following to be accepted:
(order+1)*(# of lines + # of tie lines) <= # of intersection points

Select **Auxiliary intersection file generation** to create a qct file that lists the locations of the intersections as well as the data values of the two lines at those intersections.

Gradient Calculation

The purpose of this function is to calculate the gradients between multiple sensors. For example, the derivatives or gradients between multiple TMI or vector magnetic sensors. These gradients are calculated in the frame of the moving system. For example, if it is a 3-sensor bird to be towed by a helicopter, then this calculates the gradients in the bird reference frame. Later, these gradients may be de-rotated into grid coordinates or geographic coordinates.

Select **Gradient Calculation** from the **Processing/Magnetic Data Processing** menu. The following window appears.



Number of Sensors Total number of magnetic sensors in the system. Default is 4.

Relative Sensor Position
Coordinates are relative to the aircraft's natural axes for the selected sensor. The origin is at the forward tip of the bird. The index of the selected sensor is displayed beside the label **Sensor**.

Data

Channel containing the total field data for the selected sensor

The parameters for each sensor must be entered individually. Click the left or right arrow button at the top of the **Sensor Settings** section to display and edit the settings for a different sensor. If the sensors are determined to have coordinates that are coplanar the **Aircraft Maneuvers** and **Measurement Location** sections will be enabled so a third gradient can be calculated.

Aircraft Maneuvers
Channels may be selected for **Pitch**, **Roll** and **Yaw**. Select the checkbox in top left corner of this section to do this. Units are in degrees.

Heading
Select **Heading calculated from profile** to indicate that the heading will be computed from the profile then added to the yaw.
Select **Heading included in yaw** when assuming the heading is included in yaw and additional calculations to determine the heading are not needed.

Measurement Location
Select the channels indicating the x, y, z coordinates where the data was measured.

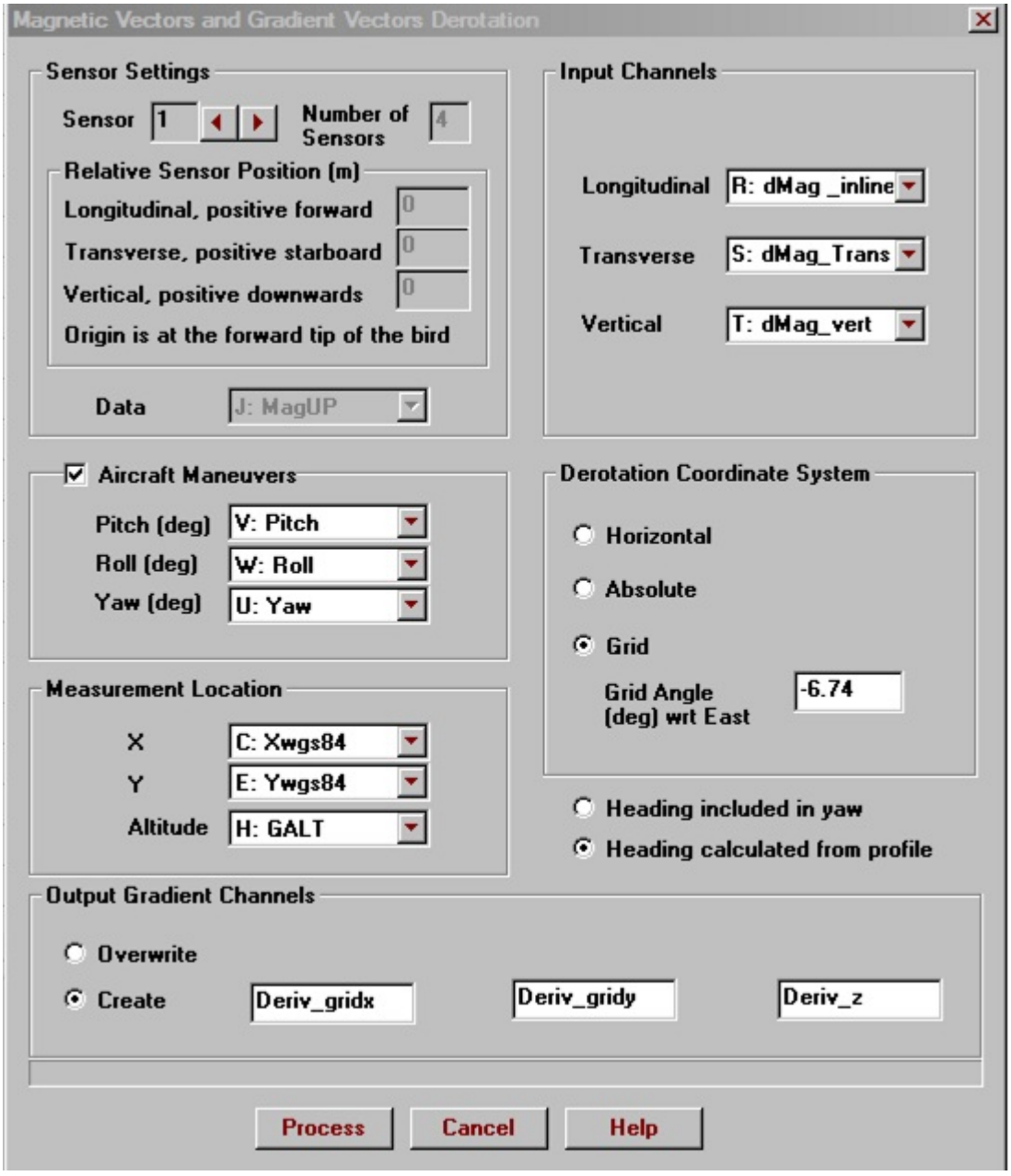
Output Gradient Channels
Three components of the total field will be saved to the qct file. Select **Create** to save the results in new channels with the selected names. Select **Overwrite** to save the results to existing channels.

Click the **Process** button to calculate the gradients. The results can be further processed by selecting **Vector Derotation** in the main menu.

Vector Derotation

This de-rotation function projects data onto a specified coordinate system. This assumes the vector data are with regard to a rigid frame. Thus, this can be applied to gradients of multiple total field sensors or components of the magnetic field (e.g. 3 component fluxgates).

Select **Vector Derotation** from the **Processing/Magnetic Data Processing** menu. The following window appears:



Input Channels Select the channels for the longitudinal, transverse and vertical components of the vector that will be derotated. These channels can be calculated by selecting **Gradient Calculation** from the main menu.

Aircraft Maneuvers

Channels may be selected for **Pitch**, **Roll** and **Yaw**. Select the checkbox in top left corner of this section to do this. Units are in degrees.

Derotation Coordinate System
Select **Grid** to project the data on a grid specified by a **Grid Angle** in degrees measured counter clockwise from east.
The data can also be projected on either an **Absolute** or **Horizontal** coordinate system. The x-axis will be along the profile if **Horizontal** is selected.

Heading
Select **Heading calculated from profile** to indicate that the heading will be computed from the profile then added to the yaw.
Select **Heading included in yaw** when assuming the heading is included in yaw and additional calculations to determine the heading are not needed.

Measurement Location
Select the channels indicating the x, y, z coordinates where the data was measured.

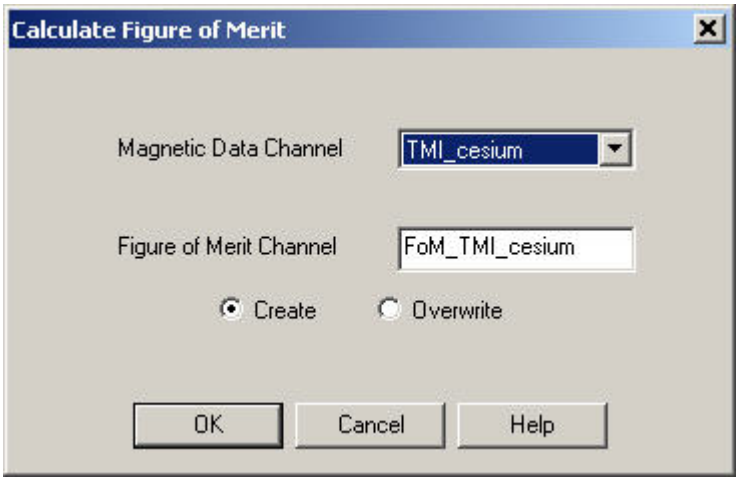
Output Gradient Channels
Three components of the total field will be saved to the qct file.
Select **Create** to save the results in new channels with the selected names. Select **Overwrite** to save the results to existing channels

Click the **Process** button to perform the derotation.

Calculate Figure of Merit

This function is a means to evaluate magnetic compensation results

Select **Figure of Merit** from the **Processing/Magnetic Data Processing** menu. The **Figure of Merit** window appears. This is normally done before and after magnetic compensation.



Select the input channel with the box beside the label **Magnetic Data Channel**. Normally this is done once with the original data and once with the compensated data.

Edit the name of the output channel in the box beside the label **Figure of Merit Channel** if you would like to save the results in a new channel. Otherwise, select **Overwrite** and select the channel you would like to overwrite. The FOM is calculated for each line of data.

Click the **OK** button to calculate the figure of merit.

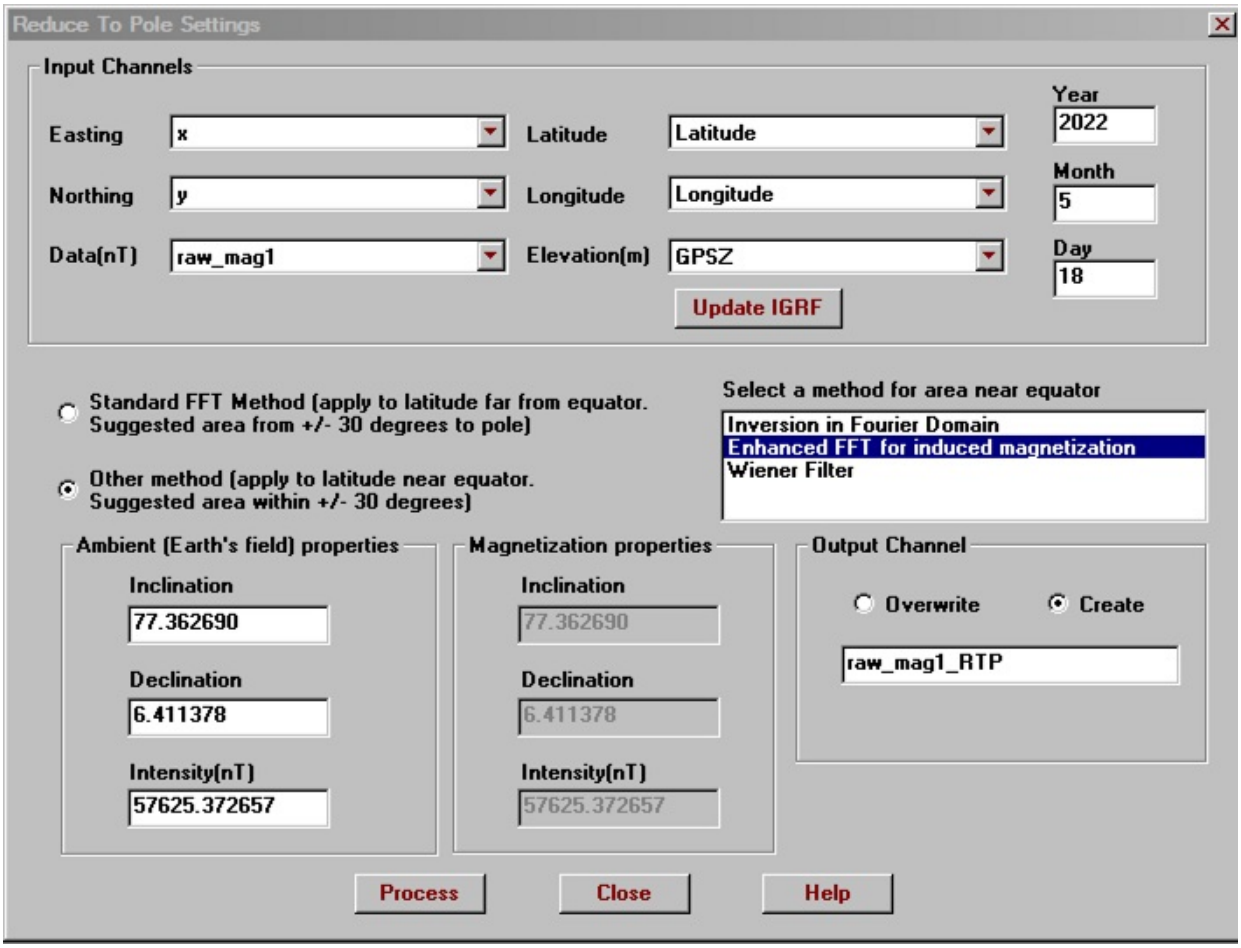
If you would like to have the ratio of FOM from correction to original, then

process the FOM separately for both original and corrected.
Create a new channel and then use the calculator to produce the ratio.

Reduce To Pole

You may convert the magnetic data you have to the magnetic field that is at the north pole. What this really means is that the magnetic data is modified to a source field which is perfectly downward as would be the case at the magnetic pole.

Select **Processing/Magnetic Data Processing/Reduce To Pole** from the menu. The **Reduce To Pole** window appears:

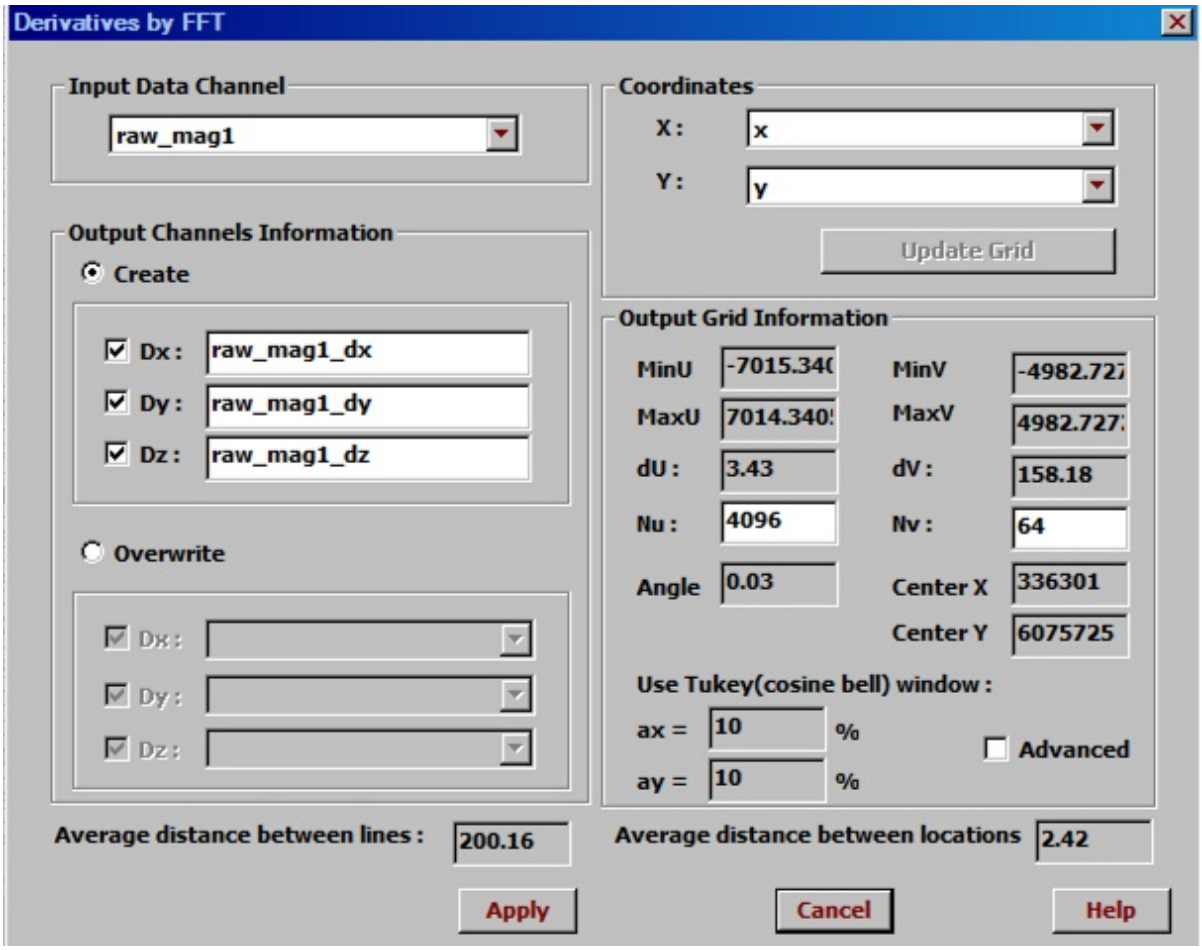


Input Channels Select channels for **Easting**, **Northing** and **Data**. **Latitude**, **Longitude** and **Elevation** channels as well as the date are only required to calculate the default **Inclination**, **Declination** and **Intensity** values. Click **Update IGRF** to update the values after you

selected a new channel or changed the date. Elevation is the height above sea level in metres.

Output Channel
Select **Overwrite** to put the output data in an existing channel. Select **Create** to create a new channel using the specified name.

Click **Process** to start the calculation. The calculation is performed using a regular grid that must be specified using the Derivatives by FFT tool that is automatically displayed.



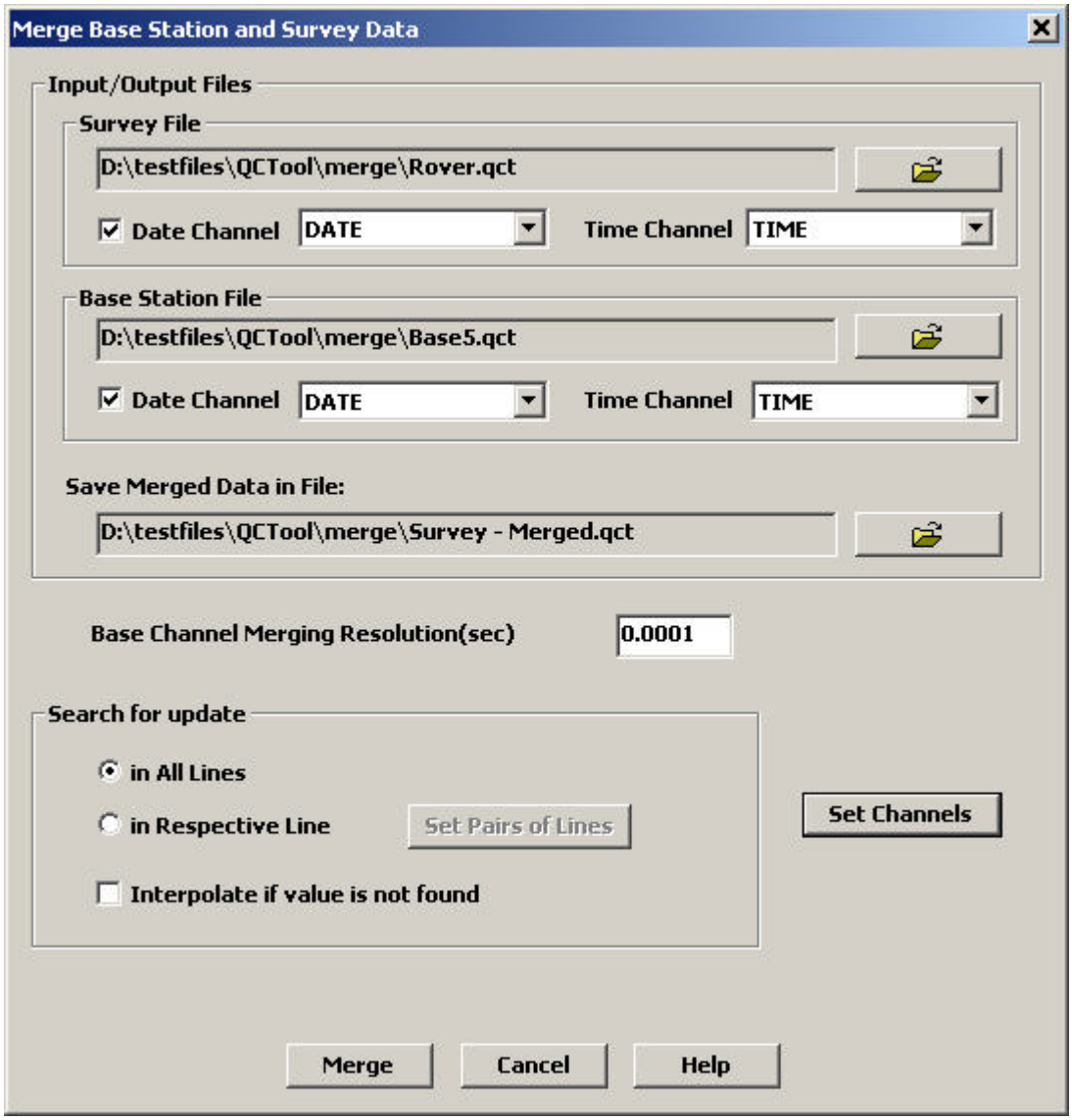
Once complete, the gridded data is interpolated to the original locations and saved in the specified channel. You have the option to save the gridded data as egr files with suffixes added to the file names as follows:
dat - magnetic data dx - dx derivative of magnetic data dy - dy

derivative of magnetic data dx - dz derivative of magnetic data rtp - magnetic data reduced to the pole

Merge Base Station Data

You may merge columns from one file containing base station data to another file.

Select **Processing/Magnetic Data Processing/Merge Base Station Data** from the menu. The following window appears:



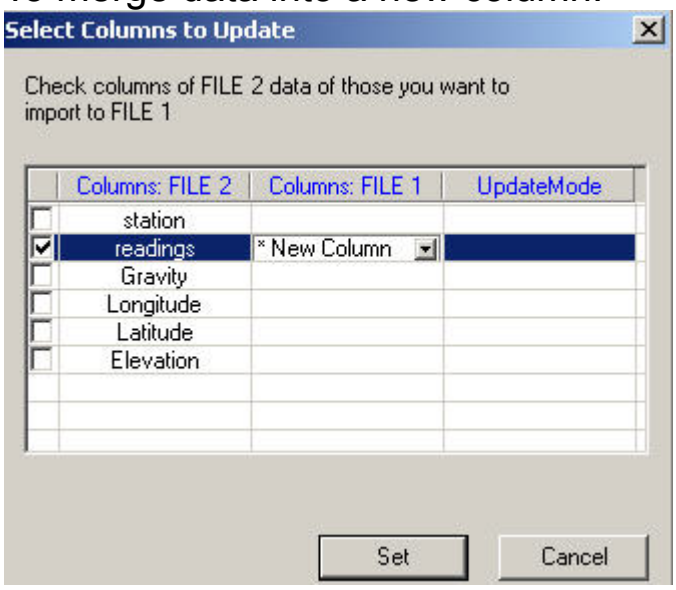
In the **Survey File** section, Click the button to the right to browse for the file to update. Select the time channel from the list below. You

may also select a channel for date if data in the file was collected on different days. Repeat for the **Base Station File** section. The **Save Merged Data in File** field in the bottom of the **Input/Output Files** section contains the location of the result file

Select the number of seconds for the **Base Channel Merging Resolution**. Times that have a difference less than this value will be considered the same.

Click the **Set Channels** button. The **Select Colums to Update** window appears, with the first column containing the data channels from the base station file. There are two ways of merging. The first merges data into a new column of the survey file and the second method updates an existing column.

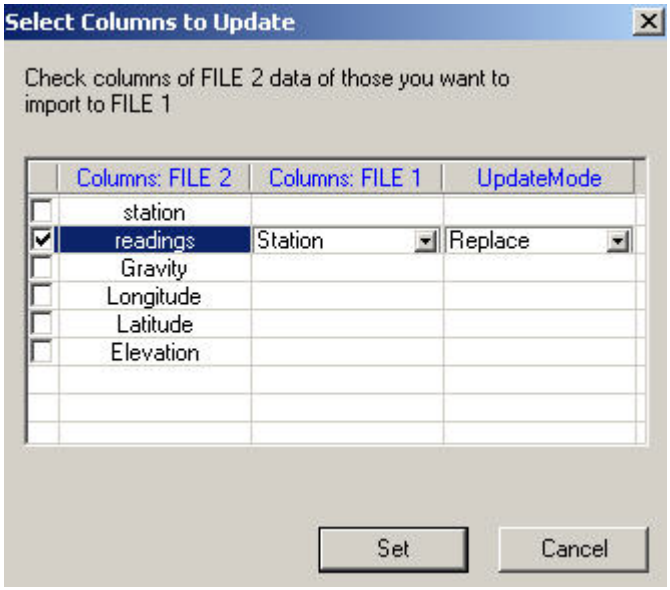
To merge data into a new column:



- Click in the checkbox next to the column to merge and leave **New Column** selected in the dropdown list that becomes active in the **Columns: File 3** column.

- Click **Set**

To update an existing column:



- Click in the checkbox next to the column to import and select the channel to update from the dropdown list in the **Columns: File 1** column.

- In the dropdown list that appears in the **Update Mode** column upon your selection, choose between three update modes: **Replace** (overwrites an existing column), **Average** (overwrites an existing column with the average calculated from the respective values in File 1 and File 2), and **Replace No Data** (overwrites only dummy values).

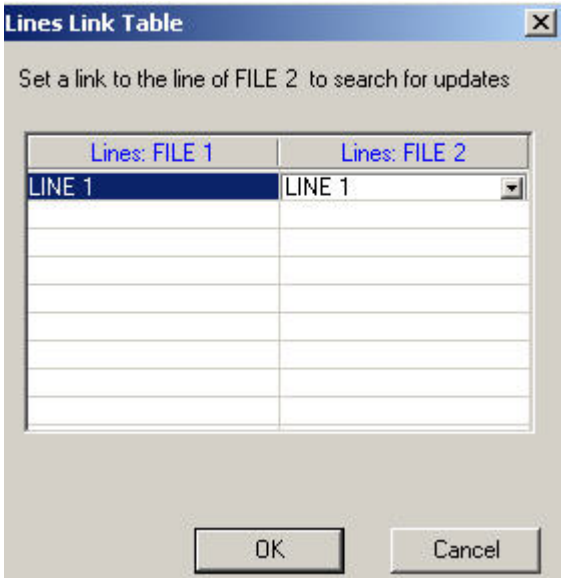
- Click **Set**.

In the **Search for Update** section of the **Merge Files** window:

- Select between two methods of searching for the update: **In All Lines** (this option searches data in all available lines and sorts them in order) and **In Respective Line** (this option searches and sorts data within each separate line). If you select the second option, you can also specify the pairs of lines from the survey and base station files.

To do this:

- Click the **Set Pairs of Lines** button. In the **Lines Link Table** window that appears:



- Select the line to update in the first (File 1) column. The second column (File 2) now contains a dropdown list with all the lines available in File 2.

- Select the desired line and click **OK**.

Back in the **Search for Update** section of the **Merge Files** window:

- Check the box next to **Interpolate if Value is not Found** if the column to import lacks a base channel vaue requested by the file to update.

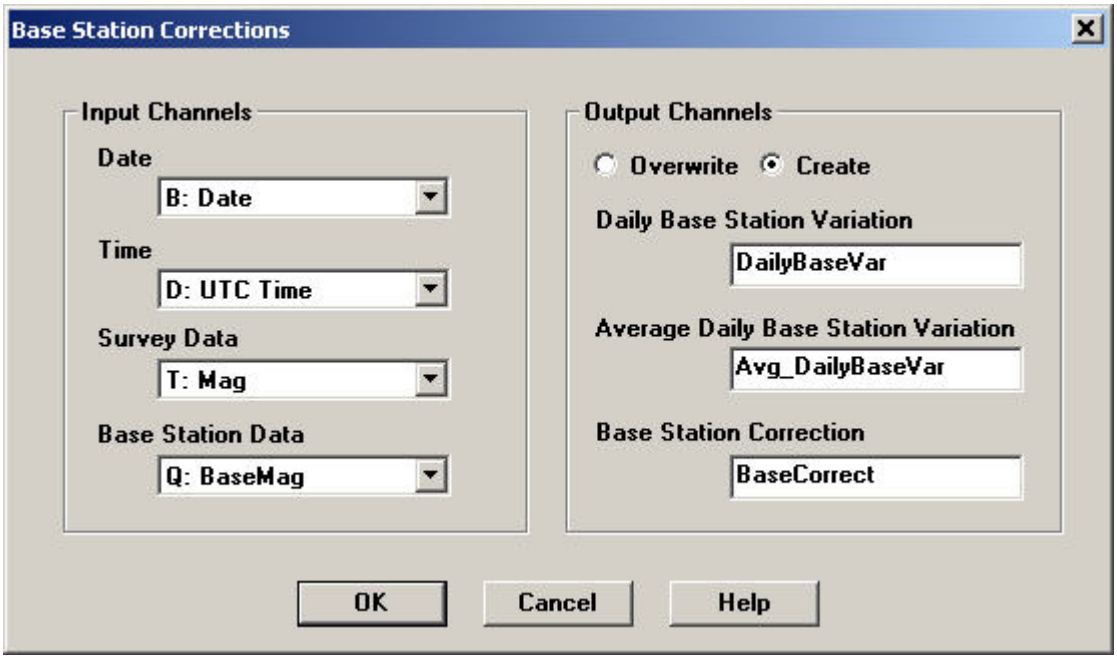
- Click **Merge** in the bottom of the **Merge Files** window to complete the merge.

BASE Station Corrections

This tool can be used to apply corrections for the temporal variations in magnetic data when base station data is available.
The tool may work with a single day or multiple days of surveying.

The tool is designed for use with a single base station data file. For temporal magnetic variations with 2 base stations please refer to the manual
QCTOOL_MAG_BASE_Processing.pdf found in your installation

Select **BASE Station Corrections** from the **Processing/Magnetic Data Processing** menu. The following window appears:



The tool assumes that the base station data has been merged with the survey data according to DATE and TIME
Normally UTC Time would be used but any time information can be used as long as the survey data and the base station data are linked by the same time coordinate.

Normally each sheet in the .qct file includes a single profile or flight line obtained on a single day. Although this is not critical only one DATE is allowed per sheet for this tool
If your data is organized such that there are multiple dates on any one sheet, the data must be reorganized.
Thus, if your data covers multiple days but is input as a single sheet, sort the sheet by date and then break into sheets according to DATE

Input Channels
Select the channels for the **Date**, **Time**, **Survey Data**, and **Base Station Data**. The values in the **Time** channel should be in a standard time format such as HH:MM:SS.SSS
If the base station data contains high frequency variations not in the survey data due to the distance of the base station from the survey data, first filter the base station data.
We suggestion the Gaussian filter for this process. Then use the resulting filtered data in the input "Base Station Data"

Output Channels
Results can be saved in new channels by selecting **Create** and entering the desired channel names. Results may instead be saved in existing channels by selecting **Overwrite** and specifying which channels should be replaced.

Daily Base Station Variation
A search is performed to locate the record with the earliest time for each day. Note: only the first date value on a sheet is used so there cannot be multiple days on a single sheet. The base station value at the earliest time is subtracted from all the base station values for that day and saved to the **Daily Base Station Variation** channel.

Average Daily Base Station Variation
The average of all the **Daily Base Station Variation** values is calculated for each day and saved to the **Average Daily Base Station Variation** channel.

Base Station Correction

The **Daily Base Station Variation** values are subtracted from the corresponding **Survey Data** values and saved to the **Base Station Correction** channel.

This process thus computes the variation from the beginning of the data and subtracts this variation from the data. If you wish to reduce the data relative to the mean of the base station data for each day, simply add **AvgDailyBaseVar** to **DailyBaseVar** and then subtract the resulting data from the survey data.

Gravity Data Corrections

Drift Correction

When you calculate Drift Correction, you want to eliminate the instrument error. Also you want to correct data for the tide effect. This may involve three steps. First, applying Meter Calibration correction, then, calculating and applying the tide correction, and finally, applying the Instrument Drift Correction.

Meter Calibration Correction

Tide Correction

Instrument Drift Correction

Meter Calibration Correction

For many older meters the manufacturers provided a single number for meter calibration. The number was the simple scale factor that converted the meter reading into milliGals by multiplying the reading times the factor. The scale factor was sometimes engraved into the meter nameplate. To apply this correction simply use QCTool calculator and apply instrument scale factor to the readings, which already corrected for tide effect.

Modern meters use a more complete description of the relationship between meter reading (counter units) and milliGals. Over the extended range of these meters the conversion from counter reading to milliGals is not perfectly linear; hence, a table rather than a constant is used. The table for the translation of instrument reading to milliGals is supplied by the manufacturer.

- Import this table to the QCTool to have a qct-file like this:

- To apply Meter Calibration table use the **Merge Files** utility in **Tools** and then **Update File Channels**.

- Select your gravity survey file as File 1. For the Base Station channel select the Readings channel.

- Select Instrument Calibration file as File 2. For the Base Station channel select the Readings channel.

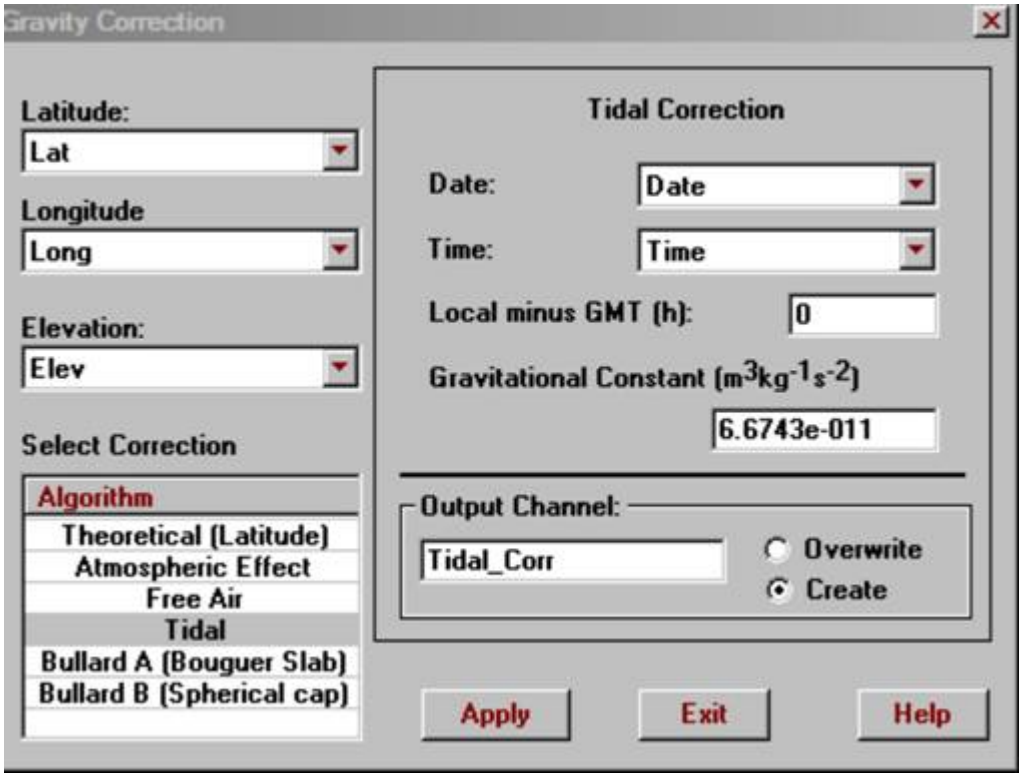
- In the **Select Columns to Update** window, select channel with translation channel. Leave New Column mode.

- IMPORTANT. Check the **Interpolate if Value is not found** check box. The calibrated reading data will be calculated automatically.

Tide Correction

Open your gravity survey file and correct data for the tide effect.

Select **Latitude, Free Air, Tide, Bouguer and Atmospheric Effect** from the **Processing/Gravity Data Processing/Corrections** menu. The following window appears:



- Specify Latitude, Longitude, Elevation channels in the respective drop down menus. Select **Tidal** algorithm in the correction algorithm list box. Specify Date and Time channels and GMT difference. Set the name of output channel. The default value of G is shown but you may change. Click on Apply button.
- The new channel with Tide correction will be created in your work file.
- Apply this correction to the channel with instrument readings by using the calculator. (Insert channel, apply formula *new channel = instrument reading - tide correction* (e.g. M = B-L) for all lines, all rows).

Instrument Drift Correction

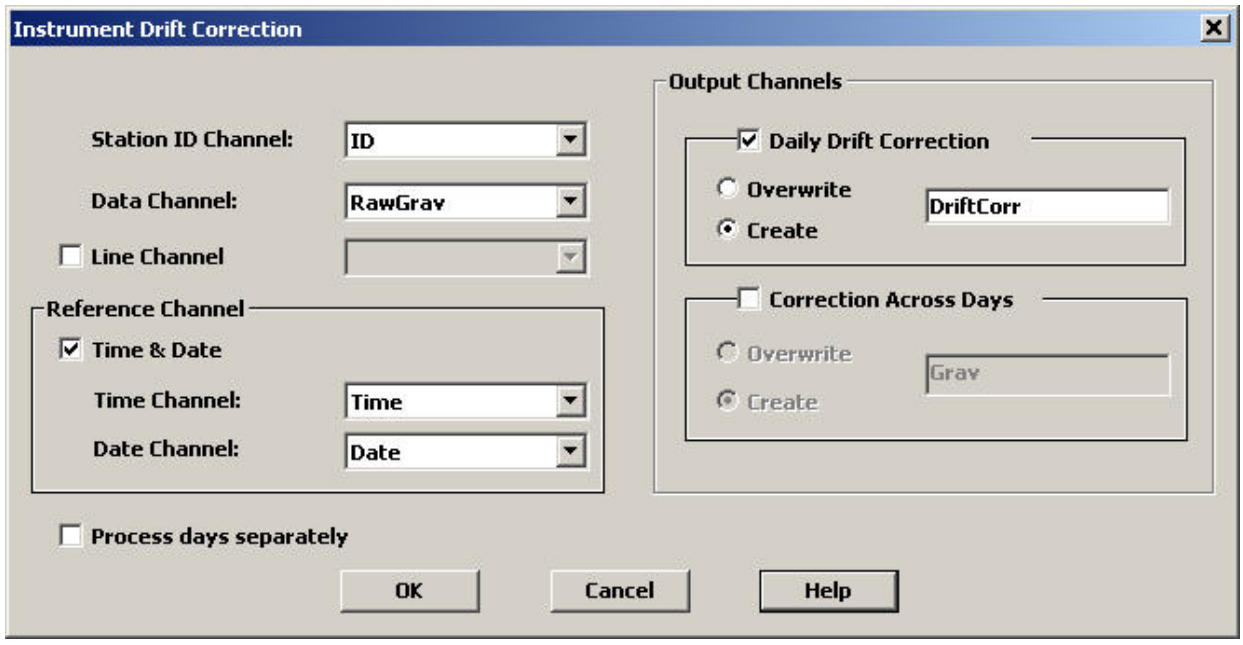
Characterizing the Drift

Meter drift is often assumed to be a linear phenomenon. If it were truly linear, the drift rate could be determined from only two readings. You could take a base station reading at the beginning of the survey and another at the same station at the end of the survey. This is called a loop tie. From this, you could solve for the constant drift rate and use this to compute the drift to remove from any observation as a function of time.

Rather than making a single loop tie, you could make several loop ties as the survey progresses. Within each loop tie, you might assume a linear drift but because you have many different drift segments, you can reasonably approximate a more complex drift. Once the drift curve is established, you can compute the drift to remove from any observation as a function of time.

The file needs to be sorted by time for each day before you begin.

Select **Processing/Drift Correction** from the menu. The **Instrument Drift Correction** window appears:



Ensure that you have selected the **Station ID Channel** (channel with a station label) as well as the **Data Channel**. Set the **Reference Channels** (Time and Date).

Line Channel - this optional channel is only needed if the same station label is used at different locations

Select **Process days separately** if a sheet has multiple days and the drift correction should be calculated using the first reading on each day. Each sheet will be processed separately regardless of whether this checkbox is selected or not.

Specify the **Output Channel(s)**. Select **Correction Across Days** to apply a least squares approximation so that the difference between base station data values for each day is reduced.

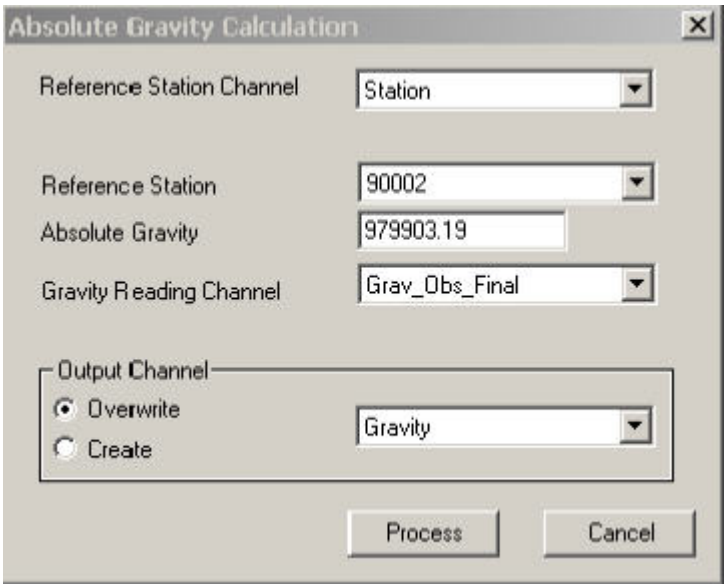
If **Correction Across Days** was not selected, apply the drift correction to the data channel by using the calculator adding function.

N: Grav_Obs	O: DriftCorr	P: Grav_Ob...
4834.865522	0.000000	4834.865522
4833.965497	-0.000352	4833.965145
4830.036361	-0.000785	4830.035576
4836.409874	-0.004162	4836.405712
4834.682339	-0.004332	4834.678007
4837.377858	-0.004524	4837.373334
4838.154712	-0.004751	4838.149961
4833.529221	-0.004908	4833.524313
4835.854897	-0.005167	4835.849730
4832.816510	-0.005298	4832.811212
4834.872262	-0.006740	4834.865522

Absolute Gravity Calculation

The absolute gravity can be calculated for all stations in the survey file assuming some stations have been measured at absolute gravity stations.

Select **Absolute Gravity Calculation** from the **Processing/Gravity Data Processing/Corrections** menu. The **Absolute Gravity Calculation** interface appears.



Select channels from the respective drop down menus.

- **Reference Station Channel** - channel with station label
- **Reference Station** - reference station for which absolute gravity data is known
- **Gravity Reading Channel** - your channel resulting from applying Drift Correction.

Set the **Absolute Gravity** for the specified reference station.

Specify the **Output Channel**.

Click the **Process** button to calculate the absolute gravity.

Note: If you have more than one raw file with gravimeter readings, import all of them into QCTool first. Then merge all files into one with the Append Files utility.

Terrain Correction

Regional and Local Topography Grid

This application has been designed to create a regional topography grid from data obtained from various sources.

For example, a regional topography grid can be created from DEM files in BIL, GTOPO30 or CDED format. The resulting grid can be used in the QCTool terrain correction algorithm.

GTOPO30 files can be downloaded from the USGS website, CDED from Resources Canada, and INEGI provides topographic data in .bil format.

NASA satellite DEM data at high resolution is now available almost everywhere in the world. One source is EarthExplorer.usgs.gov. The .bil format is most convenient for importing to QCTool to prepare the grids.

These sources also provide high resolution data and thus can be used for the more accurate local corrections.

Select **Processing/Gravity Data Processing/Corrections** from the main menu. Then, select **Create Grid from DEM data**. The wizard to create such a grid will appear.

Local Corrections can be made either through use of gridded local terrain data (*.egr) or Local Terrain Correction (stations)

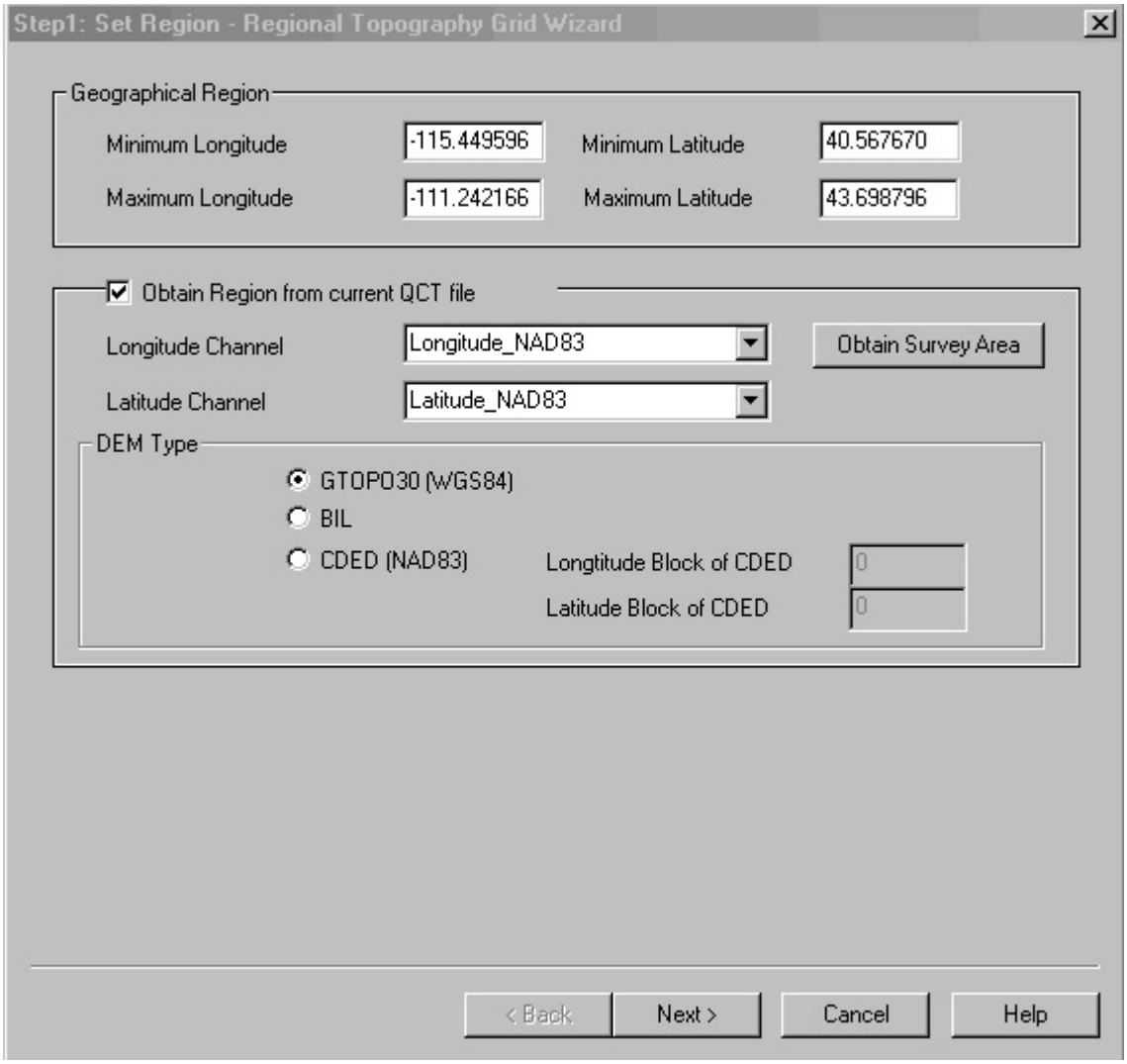
Step 1

Step 2 (GTOPO30 or BIL)

Step 2 (CDED)

Step 3

Regional Topography Grid Wizard- Step 1



- On the first page of the wizard, specify the geographical region by setting the minimum and maximum values for the latitude and longitude.

- If you want to cover the region of your survey data simply check the **Obtain Region from current QCT file** check box, specify channels for latitude and longitude and click on the **Obtain Survey Area** button. The minimum and maximum values for latitude and longitude will be updated.

- The outer radius can be edited. Click **Update** to display the new latitude and longitude limits if you change the outer radius value

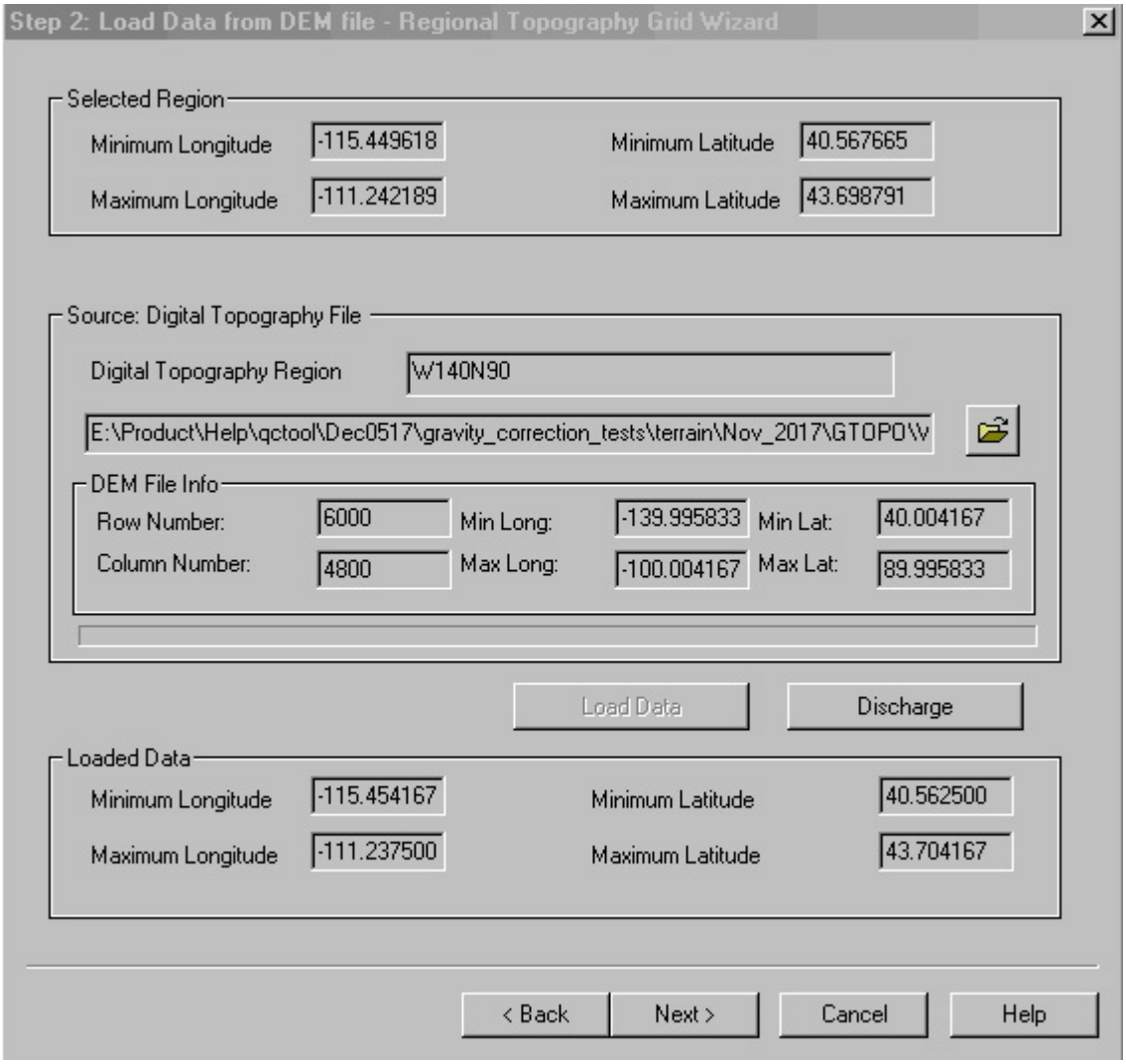
- Either the GTOPO30, CDED or BIL format can be used for the DEM files. Select.

- For GTOPO30, lat/lon must be in WGS84 and for CDED in NAD83. For .bil this will depend upon the source of the data but be sure to verify that the survey lat/log is consistent with the DEM lat/lon. If the survey datum is not consistent with the DEM datum, then the datum and coordinates can be translated as required via **Tools/Coordinates**.

  Longitude Block of CDED indicates the number of CDED files required to cover the selected longitude span. A maximum of two files are allowed.

  Latitude Block of CDED indicates the files needed to cover the selected latitude span. A maximum of two files are allowed.

Regional Topography Grid Wizard - Step 2 (GTOPO30)



- On the second page of the wizard, specify the appropriate digital topography file. The digital topography region provided by the USGS is specified.

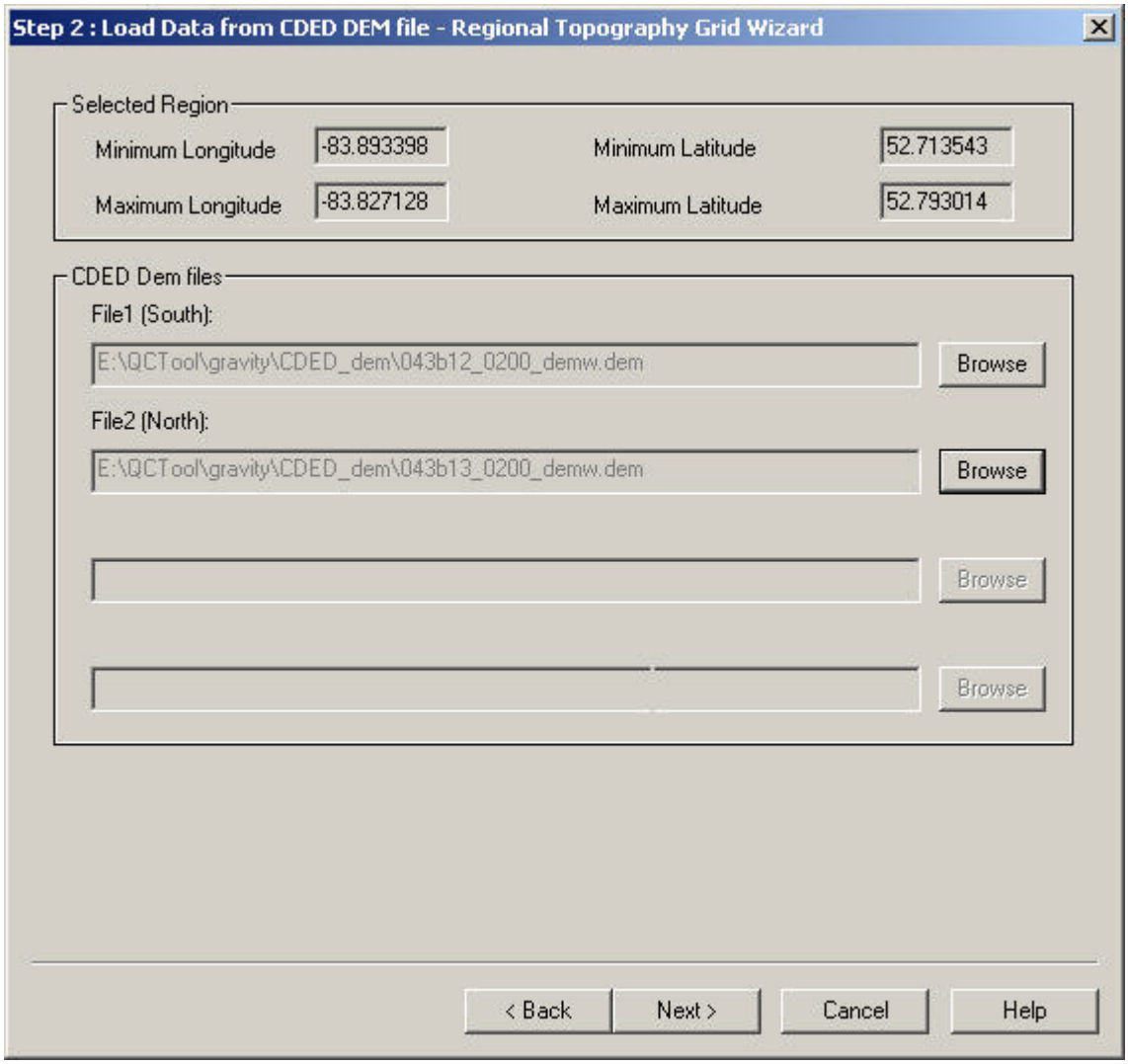
  For BIL files, change the file type to **BIL Files** on the Open File window you see after clicking the  button.

  For GTOPO30, the directory of the DEM file should have a header file containing size and coordinate information for the DEM. This file is expected to have the same name as the DEM file with a .hdr extension.

- Click on the **Load Data** button to load the elevation data. Check that the Loaded Data Lat/Lon extremes are consistent with the Selected Region extremes

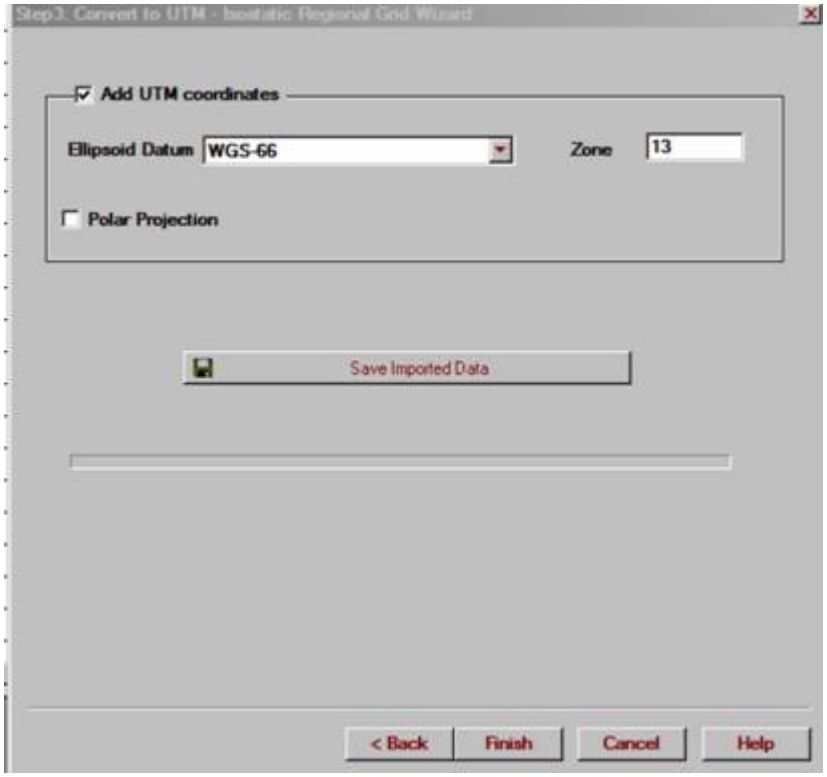
- You can reset the **Loaded Data** section with the **Discharge** button.
- Click on **Next**.

Regional Topography Grid Wizard - Step 2 (CDED)



- On the second page of the wizard, specify the appropriate digital topography file(s).
- The **Selected Region** section displays the latitude and longitude boundaries for the selected region.
- The required files needed to cover the selected region will be determined and labels are displayed to indicate the files that need to be loaded. e.g. if two files are needed to cover the selected region, **North** and **South** labels are displayed to show which file contains the data for the northern region and which file contains the data for the southern region.

Regional Topography Grid Wizard - Step 3



- On the third page you will create a grid with topography data file in the QCTool grid format. Two steps will be processed to get such a grid.

- First, the latitude/longitude coordinates are transformed to UTM X and Y. Select desired Ellipsoid Datum to do this and the Zone should be consistent with the survey data zone.
- Select the Polar Projection checkbox if the survey area is inside the polar bound. Note: The Add UTM coordinates checkbox must be selected to use the grid with the Regional Terrain Correction tool.

- Second, the regional grid is created. Click the Save Imported Data button and specify the name of output file. *.qdem. This file is used to perform the regional corrections.

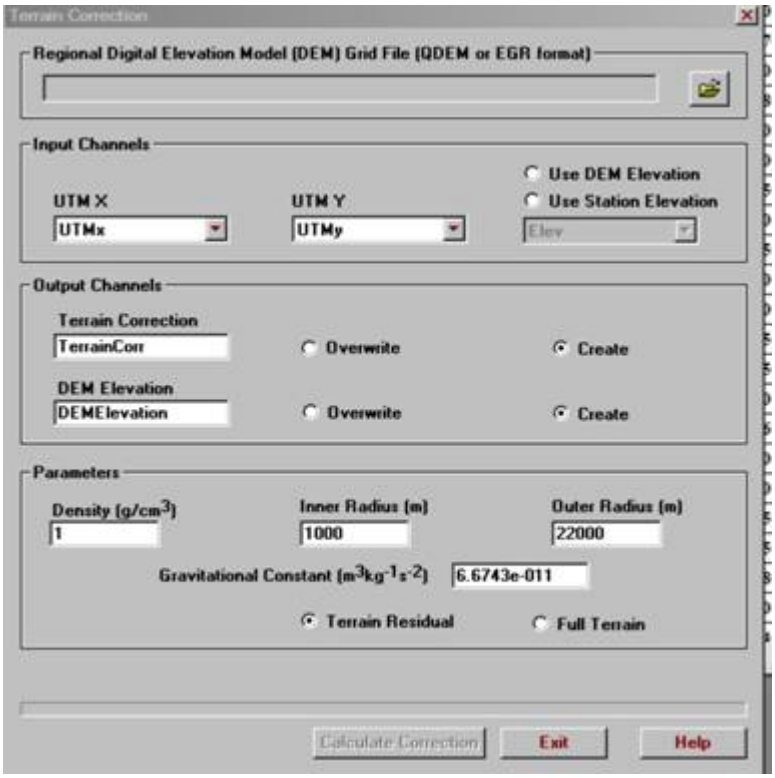
- Click the Finish button to produce the grid file.

Regional Terrain Correction

Select **Processing/Gravity Data Processing/Corrections** from the main menu.

Then select either **Continuous Surface Topography** or **Discontinuous Prismatic Topography** under the **Terrain Correction from Grid** heading.

The **Terrain Correction** window appears for a continuous surface: A DEM grid must first be prepared.
Note: If in addition to a large regional grid covering the survey area, you have a smaller higher resolution grid over the survey then this process is applied twice.



- Specify the DEM Grid file. Valid file formats are EGR or QDEM files, both of which are QCTool file formats. .*qdem* maintains the exact sampling of the original DEM data while .*egr* uses your interpolation density when the data is gridded.

  EGR files are created after gridding the topographic data and then saving the grid using the grid mesh tab. QDEM files are native QCTool files built with the Regional Topography Grid Wizard. You may create a *qdem* file

without using the wizard by creating a qct file insuring there are channels named UTM_X, UTM_Y and ELEVATION. Then simply change the file extension from *.qct* to *.qdem*

- Select file columns for **UTM X** and **UTM Y** in the respective drop down menus and the elevation channel.
- Specify the **Output Channel**.

- Two elevations are allowed. The elevation provided by the survey or the elevation provided by the interpolated DEM grid at each station. If you wish to compare your survey elevation with the elevation provided by interpolating the DEM grid this information is output as DEMElevation. For us, comparison of your survey elevations and the interpolated DEM elevation is critical to good topographic corrections.
- Specify the **Output Channel**
- Set **Parameters**: density, inner and outer radii. Suggested value for inner radius is the resolution of the grid. e.g. Enter 1000 for grid with cells measuring 1x1 km. Outer radius should be less than the distance between the outer edge of the DEM and the outer edge of the survey area. For a SRTM gird, for example, the inner radius would be 30m.

- Note: default density is 1 (one) which allows you to create easily new channels with varying density via the QCTool calculator tool.
- Two corrections can be made. First, only the Terrain Residual correction and second the Full Terrain which means the Bouguer slab minus the Terrain Residual correction
- Click on the **Calculate Correction** button to generate a column with local terrain correction values.

Prism Correction

If you selected **Discontinuous Prismatic Topography**, a **Prism Correction** checkbox will be visible in the bottom left corner of the interface. If you deselect the checkbox, the correction will be calculated using continuous surface topography. (see manual)

Building a Grid from User Elevation Data

If you have your own elevation data, you may use it to build a grid and use this grid to perform the correction.

- Import your data into QCTool. You must have UTM X and Y channels in addition to elevation data.
- Grid the elevation channel in QCGrid  . When you click QCGrid, a Set Channels dialog appears. Ensure that the correct channels are selected for X,Y, and data.
- Interpolate the data using  . You can set an appropriate cell size. Note that the rotation angle must be zero to use the grid for a terrain correction.
- Click the grid mesh button  , then click 

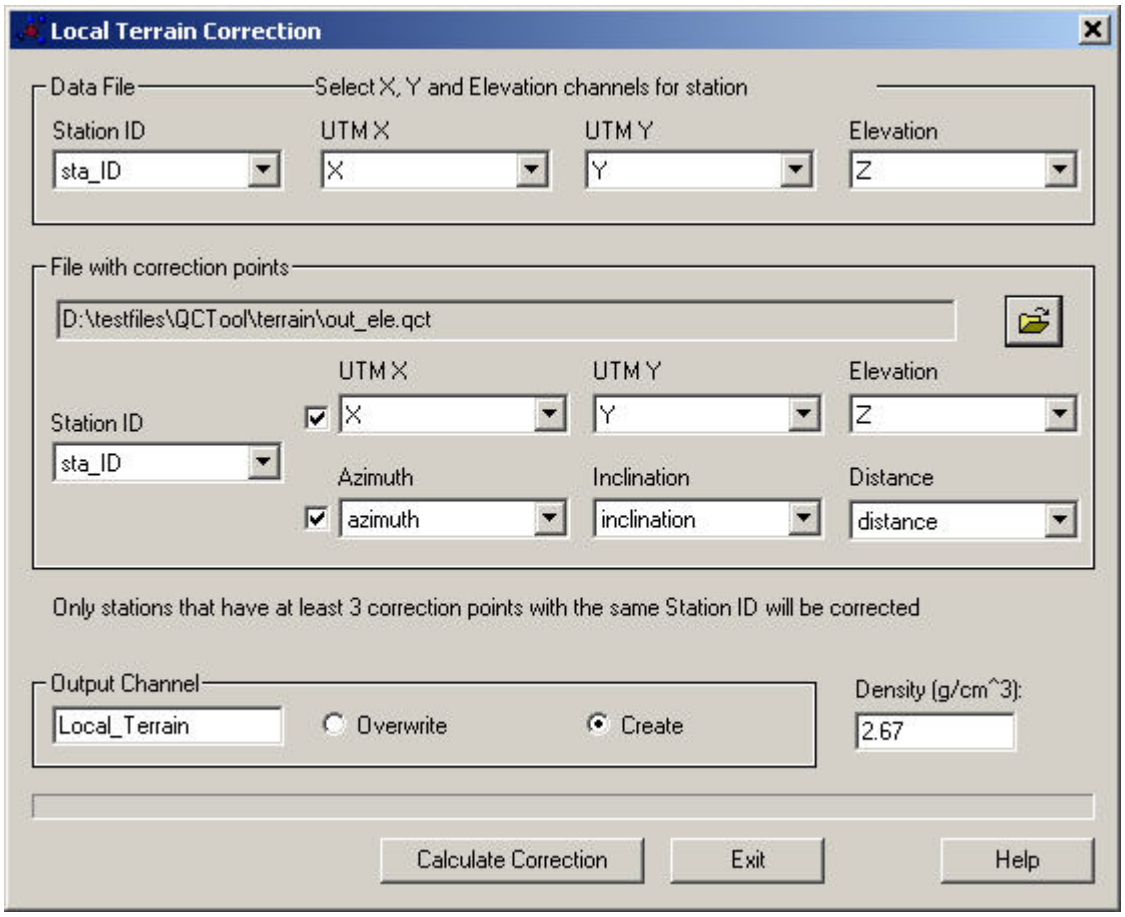
to save the grid to an EGR file.

If this elevation data is in a addition to a larger elevation grid then this is applied before the larger elevation grid. In this case, for the local, high resolution grid, the inner radius would be the size of the interpolation cells. Then for the larger grid, the inner radius is the outer radius of the local grid.

Local Terrain Correction (Stations)

Select **Local Terrain Correction** from the **Processing/Gravity Data Processing/Corrections** menu. Then select **Local Terrain Correction** from the submenu.

The **Local Terrain Correction** window appears:



- A correction point file in .qct format must first be created. If your correction data are in Excel, simply export to .csv and open into QCTool and save.
- Select data file columns for Station ID, UTM X, UTM Y, and Elevation in the respective drop down menus.
- Specify the qct file with correction points. This file should be already created with Station ID, UTM X, UTM Y, and Elevation channels. There should be at least 3 correctional station points for each station of the original

file. You may instead use a file with azimuth, inclination and distance. Azimuth is measured in degrees east of north. Inclination is measured in degrees upward from the horizontal. The file should look similar to the image below:

N	A: StationID	B: X	C: Y	D: Z
1	A	380669.88	4439796.51	2000.50
2	A	380667.58	4439794.51	2001.00
3	A	380666.88	4439790.00	2000.00
4	A	380670.00	4439795.00	2000.40
5	A	380669.00	4439795.51	2000.30
6	B	374649.08	4436562.84	1900.50
7	B	374648.08	4436560.84	1900.00
8	B	374647.58	4436561.34	1901.00
9	B	374647.28	4436561.24	1900.80

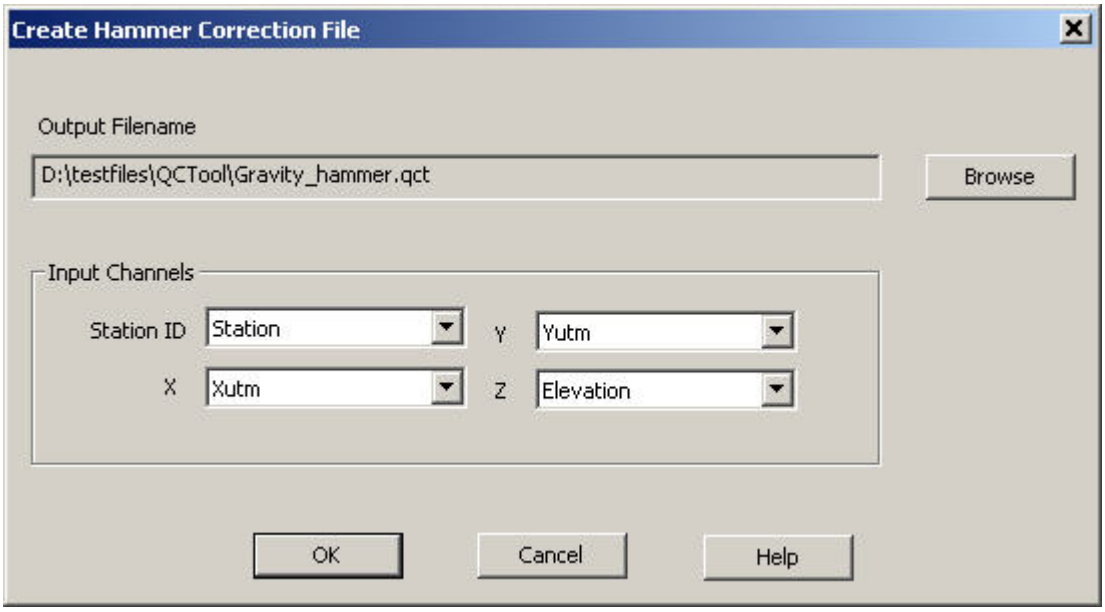
- Select columns for Station ID, UTM X, UTM Y, and Elevation in the respective drop down menus for the file with correction points and select the checkbox to the left of the drop menus. If your correction file has azimuth, inclination and distance channels, you may use these channels instead of UTM X, UTM Y and Elevation by deselecting the checkbox to the left of X, Y and Elevation and selecting the checkbox to the left of Azimuth, Inclination and Distance. If both sets of channels are selected with the checkbox, the values in the channels will be checked to confirm they contain the same information.
- Specify the output channel and density.

- Click the **Calculate Correction** button.
- The Local Terrain Correction is calculated to be a correction to the Bouguer corrections which overcompensate for topography. Thus, Local Terrain and Regional Terrain are to be added back into the final data.

Hammer Correction File

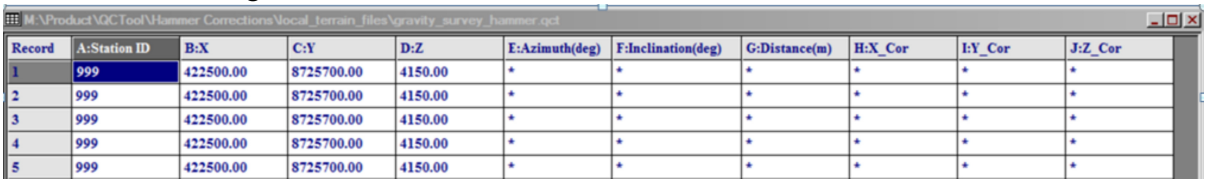
Select **Local Terrain Correction (Stations)** from the **Processing/Gravity Data Processing/Corrections** menu.

Then select **Create Hammer Correction File**. The **Hammer Correction File** window appears:



- Select columns for Station ID, X, Y, and Z in the respective drop down menus.

- You may change the path of the output file by clicking the **Browse** button
- Click the **OK** button.
- The columns you selected will be copied to the output file and new columns for Azimuth, Inclination and Distance are created. A separate sheet of five rows is assigned to each station.



Record	A:Station ID	B:X	C:Y	D:Z	E:Azimuth(deg)	F:Inclination(deg)	G:Distance(m)	H:X_Cor	I:Y_Cor	J:Z_Cor
1	999	422500.00	8725700.00	4150.00	*	*	*	*	*	*
2	999	422500.00	8725700.00	4150.00	*	*	*	*	*	*
3	999	422500.00	8725700.00	4150.00	*	*	*	*	*	*
4	999	422500.00	8725700.00	4150.00	*	*	*	*	*	*
5	999	422500.00	8725700.00	4150.00	*	*	*	*	*	*

- At least 3 correction stations must be defined for each gravity Station ID. The azimuth, inclination and distance from the gravity station to the correction station should be inserted.
- Populate these new columns (azim,incl,dist) on each sheet with the desired values then click the calculator button  on the toolbar
- The following steps will convert the Azimuth, Inclination and Distance columns to coordinates in the X_Cor, Y_Cor and Z_Cor columns
- Under **Result**, select the column for X_Cor which is H by default. Select **All Rows** and **All Lines**. Make sure degrees is selected for the angle units. and select Load Channel formula to load the correct formula
- Click **Apply**
- Repeat the above two steps for the Y_Cor and Z_Cor columns which are I and J respectively.
- These steps will produce the coordinates of the correction stations
- Save the file and it can be used for Local Terrain Correction

Isostatic Correction

Regional Grid

This application has been designed to create a regional grid from the f4 ice surface version of the NOAA ETOPO1 global relief model. The grid file etopo1_ice_g_f4.flt can be extracted from a zip file available at https://www.ngdc.noaa.gov/mgg/global/relief/ETOPO1/data/ice_surface/grid_registered/binary/

The "Regional topography grid wizard" extracts from the ETOPO1 global map a portion covering the area of current the .qct file with a 166.7 km overlap on each side, and saves it as a smaller grid. The extracted local elevation map is saved with a .isgr extension.

In the wizard, the user is directed to set a path to the ETOPO1 grid, or to download it from NOAA site if it can't be found on the computer. It is too large and not included in the setup package.

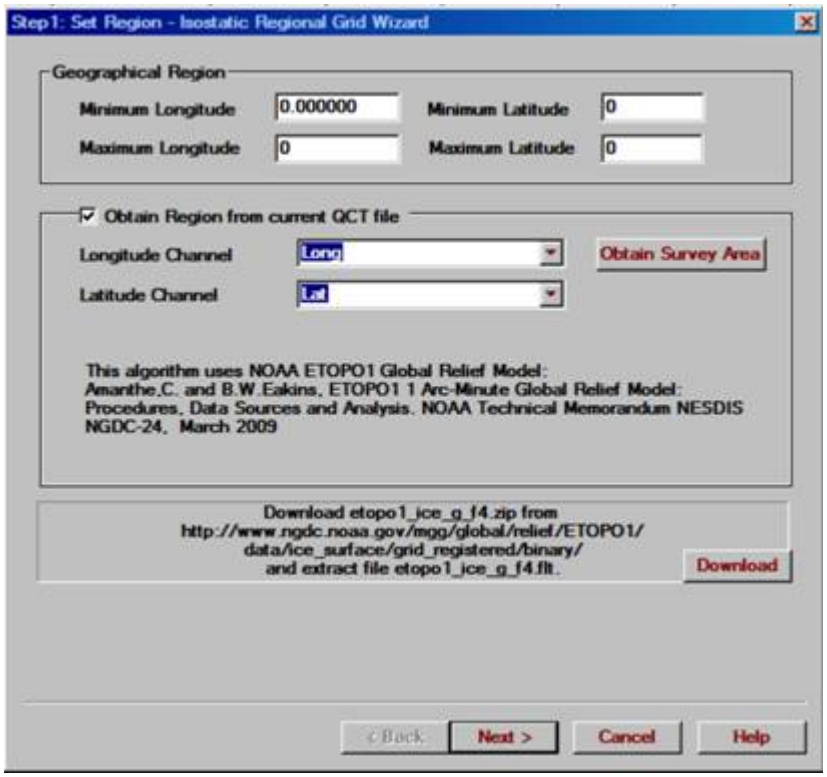
Select **Processing/Gravity Data Processing/Corrections** from the menu then **Create Grid for Isostatic Correction**. The wizard to create such a grid will appear.

Step 1

Step 2

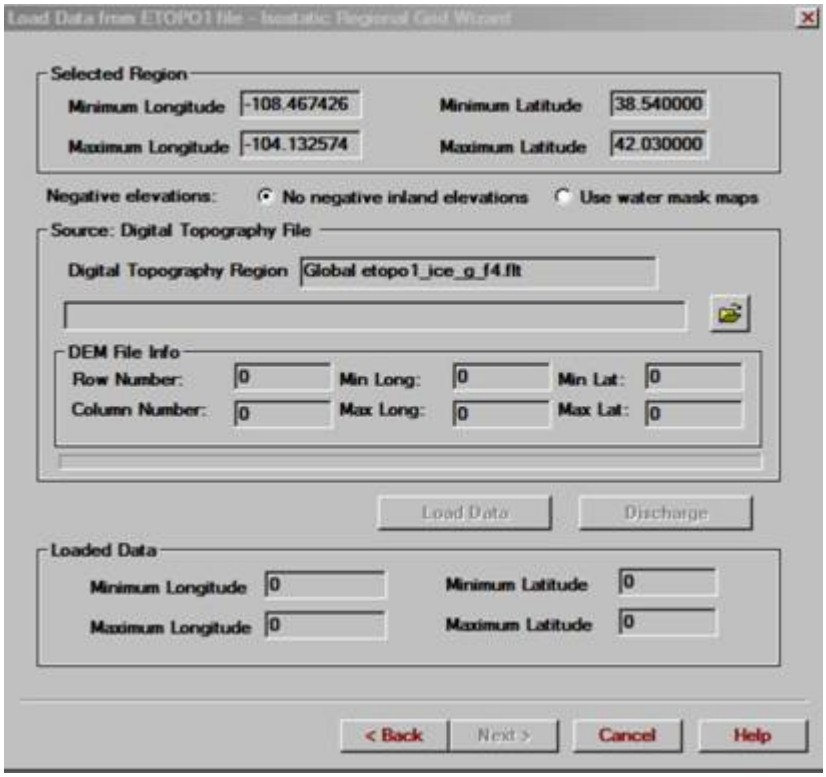
Step 3

Regional Grid Wizard - Step 1



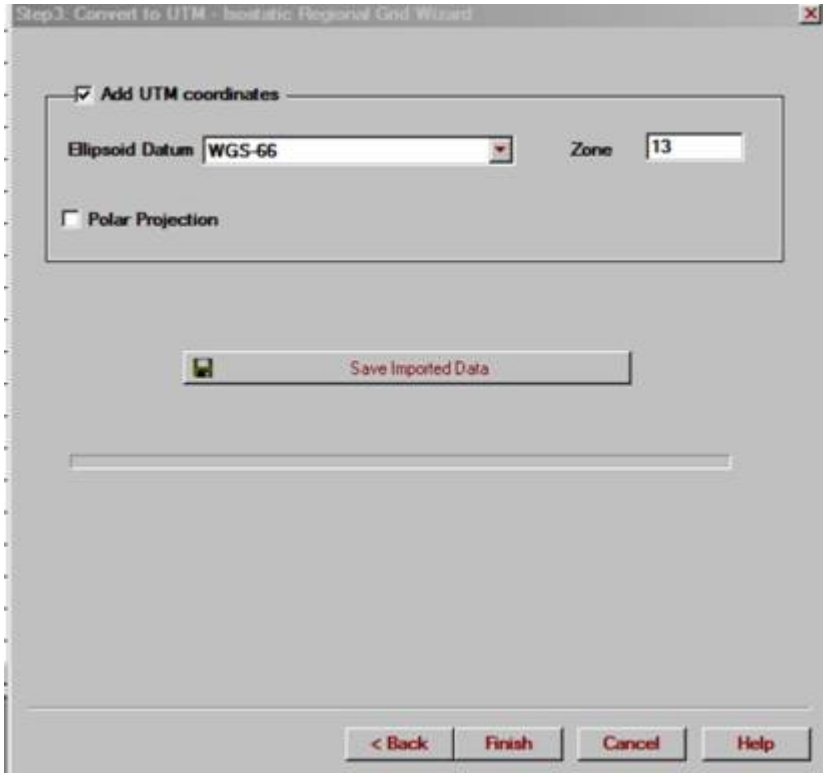
- On the first page of wizard specify the geographical region by setting the minimum and maximum values for the latitude and longitude.
- If you want to cover the region of your survey data simply check the **Obtain Region from current QCT file** check box, specify channels for latitude and longitude and click on **Obtain Survey Area** button. The minimum and maximum values for latitude and longitude will be updated.
- The outer radius can be edited. Click **Update** to display the new latitude and longitude limits if you change the outer radius value
- The algorithm uses the grid-registered binary ice surface version of the NOAA ETOPO1 Global Relief Model.
- You may click the **Download** button to send your browser to the URL http://www.ngdc.noaa.gov/mgg/global/relief/ETOPO1/data/ice_surface/grid_registered/binary/ and download the required file etopo1_ice_g_f4.zip The zip file contents can be extracted to any location and this location will need to be entered on the next page.

Regional Grid Wizard - Step 2



- On the second page of the wizard, specify the location of the digital topography file by clicking the  button.
- Select **Use water mask maps** if you need to differentiate between negative elevations on land and negative elevations in water. Otherwise, select **No negative inland elevations**
- Click on the **Load Data** button to load the elevation data.
- You can reset the **Loaded Data** section with the **Discharge** button.
- Click on **Next**.
- The **DEM File Info** section is not used for isostatic correction.

Regional Topography Grid Wizard - Step 3



- On the third page you will create a grid with topography data file in the QCTool grid format. Two steps will be processed to get such a grid.

- First, the latitude/longitude coordinates are transformed to UTM X and Y. Select desired Ellipsoid Datum to do this. Select the Polar Projection checkbox if the survey area is inside the polar bound.
  Note: The Add UTM coordinates checkbox must be selected to use the grid with the Isostatic Correction tool.

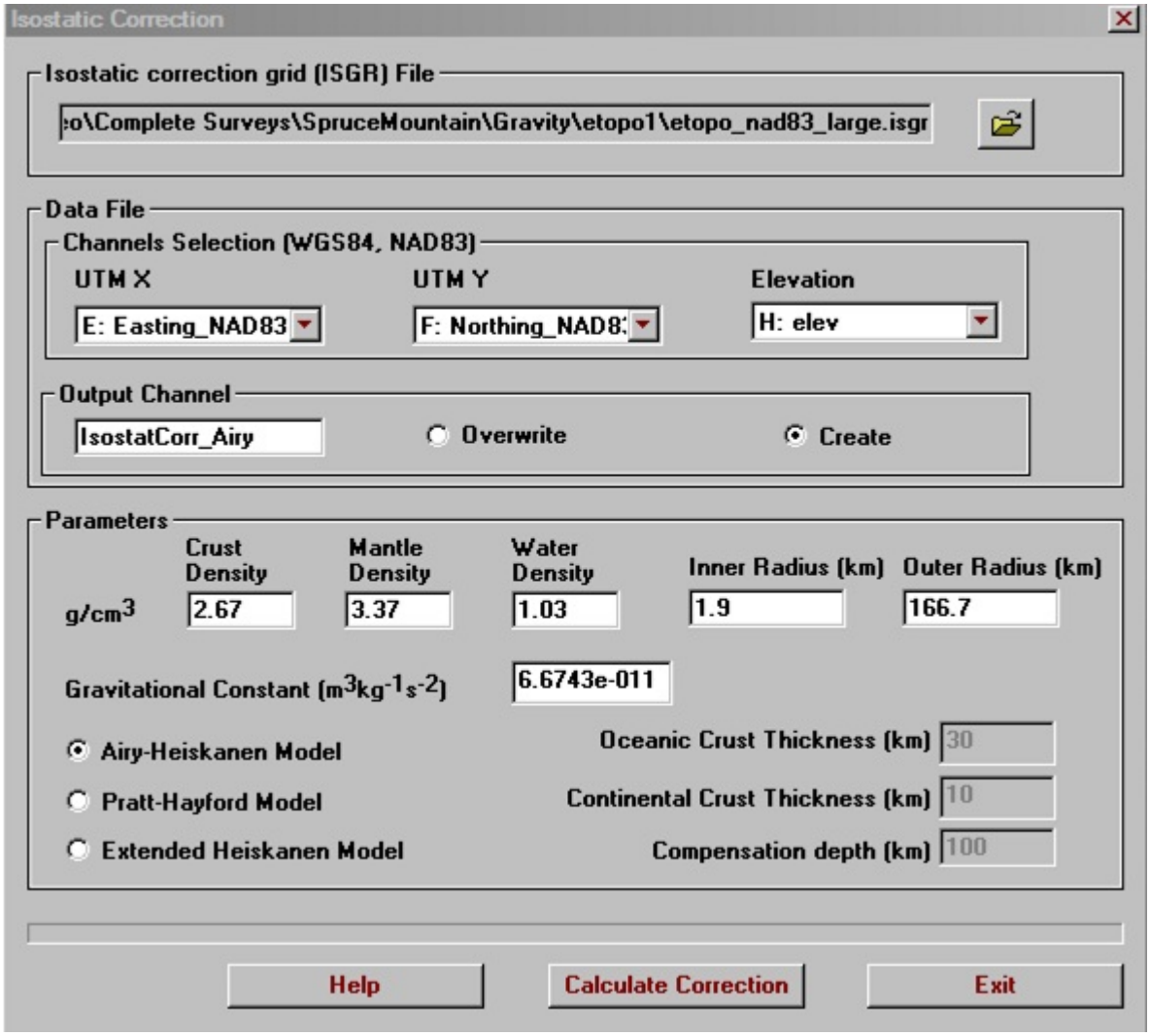
- Second, the regional grid is created. Click the Save Imported Data button and specify the name of output file.

- Click the Finish button to produce the grid file.

- You may use the produced grid file to perform isostatic correction by selecting **Isostatic correction** from the **Processing/Gravity Data Processing/Corrections** menu.

Isostatic Correction

Select **Isostatic Correction** from the **Processing/Gravity Data Processing/Corrections** menu. The **Isostatic Correction** window appears:



- Specify the Isostatic correction grid file. The file needs to be an ISGR file built with the Isostatic Regional Grid Wizard.
- Select file channels for **UTM X** and **UTM Y** in the respective dropdown menus.
- Specify the **Output Channel**.

- Choose between either the Airy-Heiskanen model proposed by Airy with equations developed by Heiskanen, the Pratt-Hayford model which was proposed by Pratt with equations by Hayford or the what we term the Extended Heiskanen model which was proposed by Heiskanen as a compromise between the Airy approach and the Pratt approach.

- Set **Parameters**: densities, inner and outer radii. Suggested value for inner radius is the resolution of the grid. e.g. Enter 1000 for grid with cells measuring 1km. Outer radius should be less than the distance between the outer edge of the grid and the outer edge of the survey area.

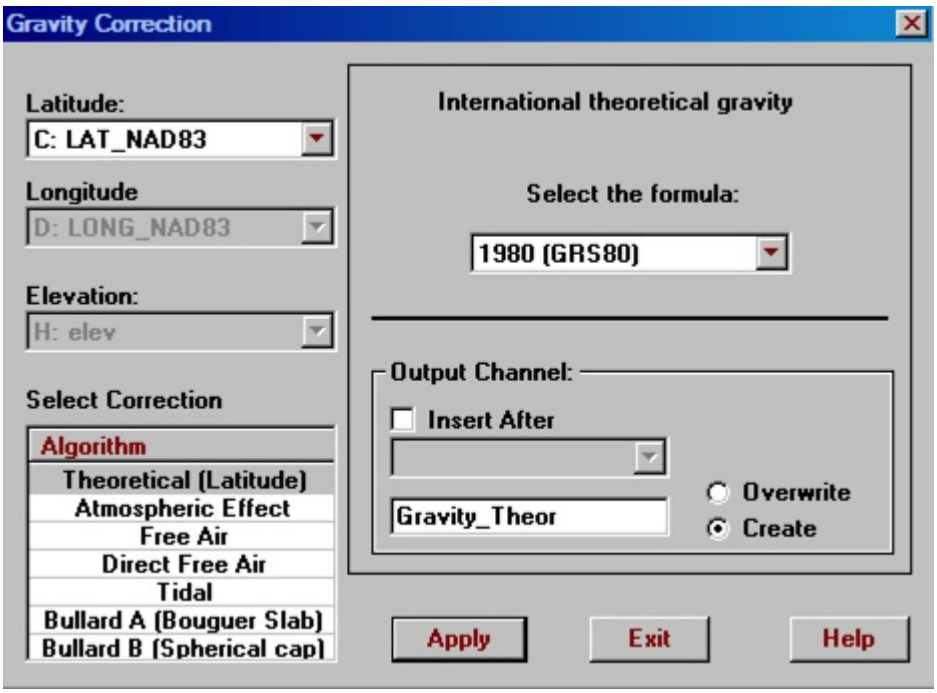
- The value you choose for **compensation depth** will be the assumed thickness of the crust.

- Click on the **Calculate Correction** button to generate a column with local isostatic correction values. These values need to be subtracted from your instrument readings to apply the correction

Latitude, Free Air, Atmospheric

It is suggested if your working coordinates are not in a modern datum (e.g. NAD27) that you create new channels with coordinates in more modern datums such as NAD83 or WGS84 and that the processing be done with regard to these coordinates, so as to be able to utilize modern ellipsoid models. At completion, you may then utilize your original coordinates.

Select **Latitude, Free Air, Tide, Bouguer, Atmospheric Effect** from the **Processing/Gravity Data Processing/Corrections** menu. The following window appears:



- Select Theoretical, Free Air Correction, Direct Free Air or Atmospheric algorithm from the Correction Algorithm list
- Specify Latitude, Longitude, Elevation channels in the respective drop down menus as required.
- Select the desired formula from the list.
- Set the additional parameters for each algorithm as requested (e.g. theoretical gravity correction system for Latitude correction, density value for the Bouguer slab correction.
- Confirm the output channel name for each algorithm or rename.

- Click the **Apply** button. The new channels with theoretical gravity data, atmospheric, free air correction and simple Bouguer corrections data will be created. The output channels are the gravitational effects of each effect and NOT corrected data.

- Note: The theoretical or Latitude correction not only removes the gravity response for the theoretical ellipsoid with a uniform density for a station on the geoid but also the gravity effects due to the rotation of the earth at the station's latitude.

- The newly constructed channels are to be subtracted from the Observed data.

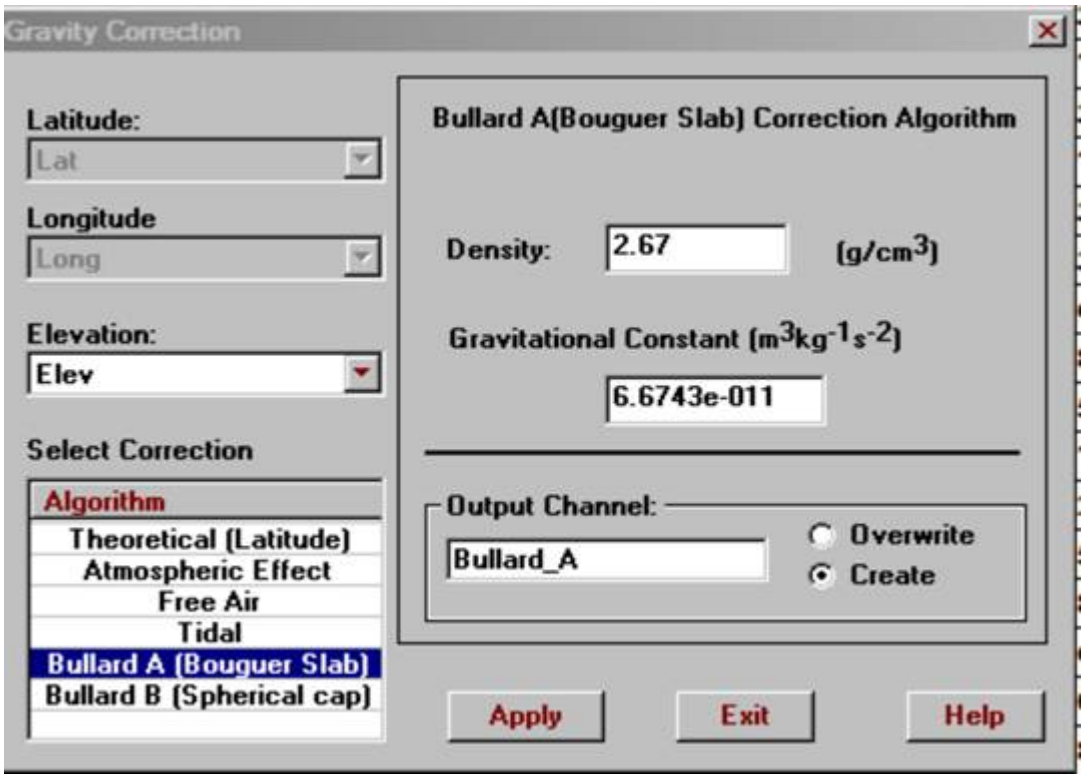
- For example, to calculate the reduced gravity value with simple Bouguer correction, use the formula *Corrected_Gravity* = *Gravity_Observation_Final - Gravity_Theoretical - Free Air correction - Atmospheric Effect - Bouguer correction* .

- **NEW** Direct Free Air result: The final free air by definition is the observed data minus the theoretical value which is calculated at the surface of the ellipsoid mode minus a correction for the excess theoretical value between the height of the station and the geoid.

  The gravity value due to the geoid at the station elevation (theoretical at the station) is less than that at the geoid if the station is above the geoid and greater if above the geoid. The traditional approach is to calculate the strength at the geoid and then the differential gravity between the elevation and the geoid and then use the two values to create the final theoretically reduced data at the station elevation. This is an unnecessary strep as the final corrected Free Air can be calculated easily directly. In this case, the output channel is the final Free Air calculation.

Bouguer Slab, Spherical Cap

Select **Latitude, Free Air, Tide, Bouguer, Atmospheric Effect** from the **Processing/Gravity Data Processing/Corrections** menu. The following window appears:

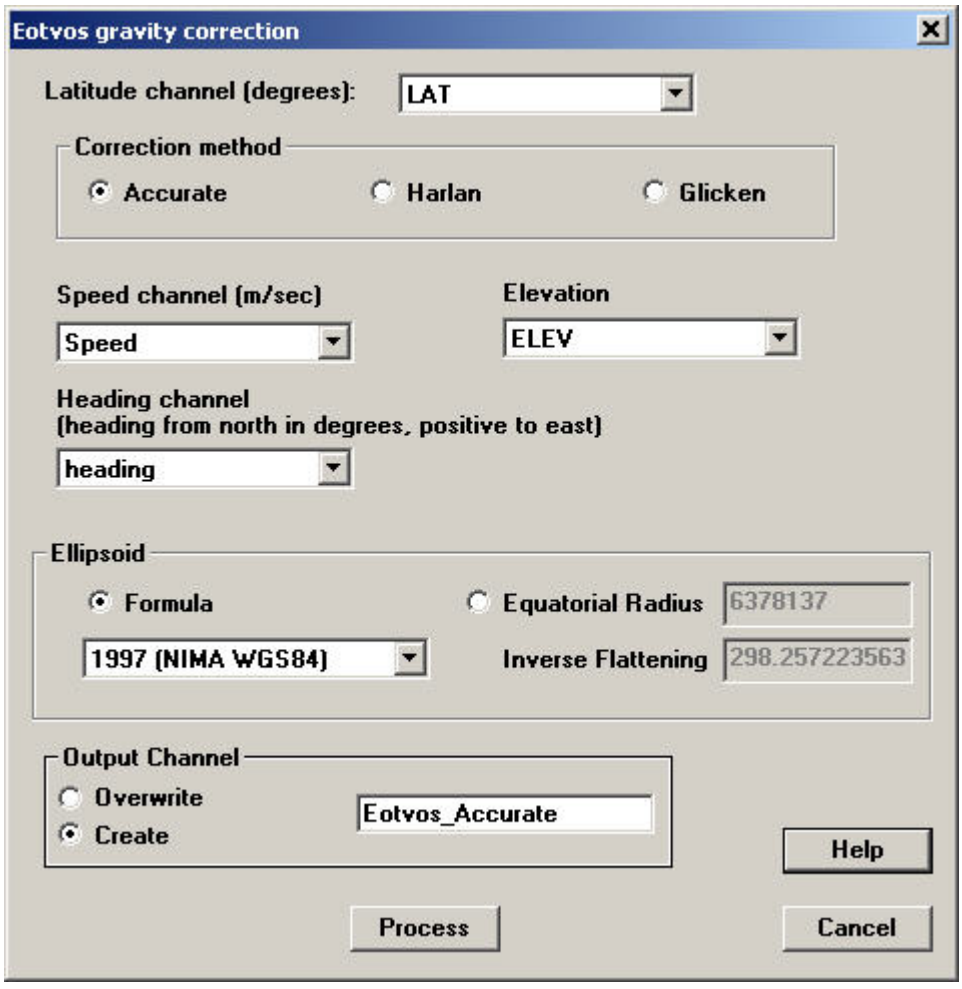


- Select the Bullard A (Bouguer slab) or Bullard B (Spherical cap) algorithm from the Correction Algorithm list
- Specify the Elevation channel in the respective drop down menu as required
- Select the desired formula from the list
- Set the additional parameters for each algorithm as requested (e.g. density value for the Bouguer slab correction)
- The Gravitational constant may be adjusted if you desire
- Confirm the output channel name for the algorithm or rename.
- Click the **Apply** button. The new channels with theoretical gravity data, atmospheric, free air and simple Bouguer corrections data will be created. The output channels are the gravitational effects of each effect and NOT corrected data.

- To calculate the reduced gravity value with simple Bouguer correction, use the formula *Corrected_Gravity = Gravity_Observation_Final - Gravity_Theoretical - Free Air correction - Atmospheric Effect - Bouguer correction*.
- Bullard B is the difference between the spherical cap and the slab. Thus, the total effect of the cap is Bouguer Slab plus Bullard B. Thus, Bullard A and Bullard B are both removed from the Observed data.
- The Bouguer slab and Spherical Cap corrections both overestimate the effects of the local topography and thus the local topography correction is be added back into the above formula.

Eötvös Correction

Select **Eötvös Correction** from the **Processing/Gravity Data Processing/ Corrections** menu. The **Eötvös gravity correction** window appears.



Three algorithms are available. **Glicken**, (Glicken, M. 1962) is the standard Eötvös including the vertical component of the Coriolis as well as the second centrifugal term. This assumes a constant elevation above the ellipsoid and low velocities and thus suitable for marine gravity. **Harlan** (Harlan, R.B. 1968) is an improvement to account for higher velocities and incorporates the effects of the earth's flattening. It does not account for accelerations due to

variations in the vehicle altitude above the ellipsoid surface. **Accurate**, is our modification of Harlan which are improvements in his approach to his approximations in the terms dependent upon the ellipticity.

Select channels from the respective dropdown menus. Glicken does not require an elevation channel.

Speed is the total speed along the surface and heading provides the vector direction.

Define the **Ellipsoid** by either selecting a **Formula** or entering values for the **Equatorial Radius** and **Inverse Flattening**

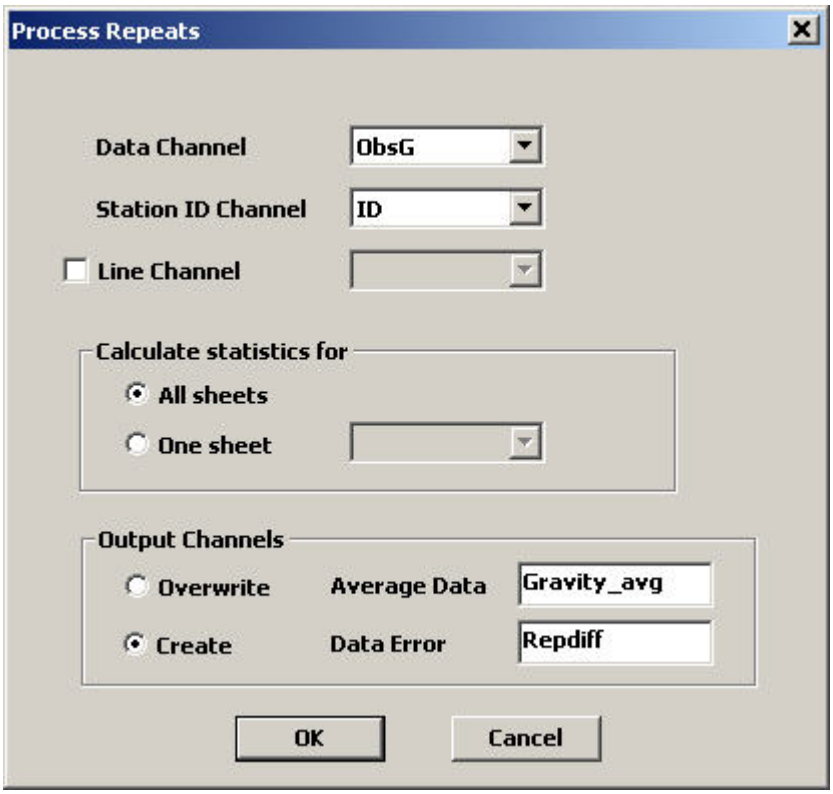
Specify the **Output Channel**. You may overwrite an existing channel or create new one.

Click the **Process** button to calculate the correction for the Eötvös effect. The output channel by our convention is to be removed from the processed gravity channel to be consistent with the other corrections.

Process Repeats

This tool generates statistics for repeated stations.

Select **Process Repeats** from the **Processing/Gravity Data Processing** menu. The **Process Repeats** dialog appears.



Select channels from the respective drop down menus.

- **Data Channel** - channel with data values to be averaged
- **Station ID Channel** - channel with station label
- **Line Channel** - this optional channel is only needed if the same station label is used at different locations

In the **Calculate statistics for** section, you select the sheet for which statistics will be calculated by selecting **One Sheet** and the sheet name in the combo box. You may also select **All sheets**. This selection only affects the results on the repeat statistics window that is displayed when after clicking the OK button. Results saved to the data channels will always analyze all stations in all sheets.

Average data and **Data error** results will be output to two channels. If a station has at least one duplicate, the average data value for all duplicates of that station is saved. A dummy

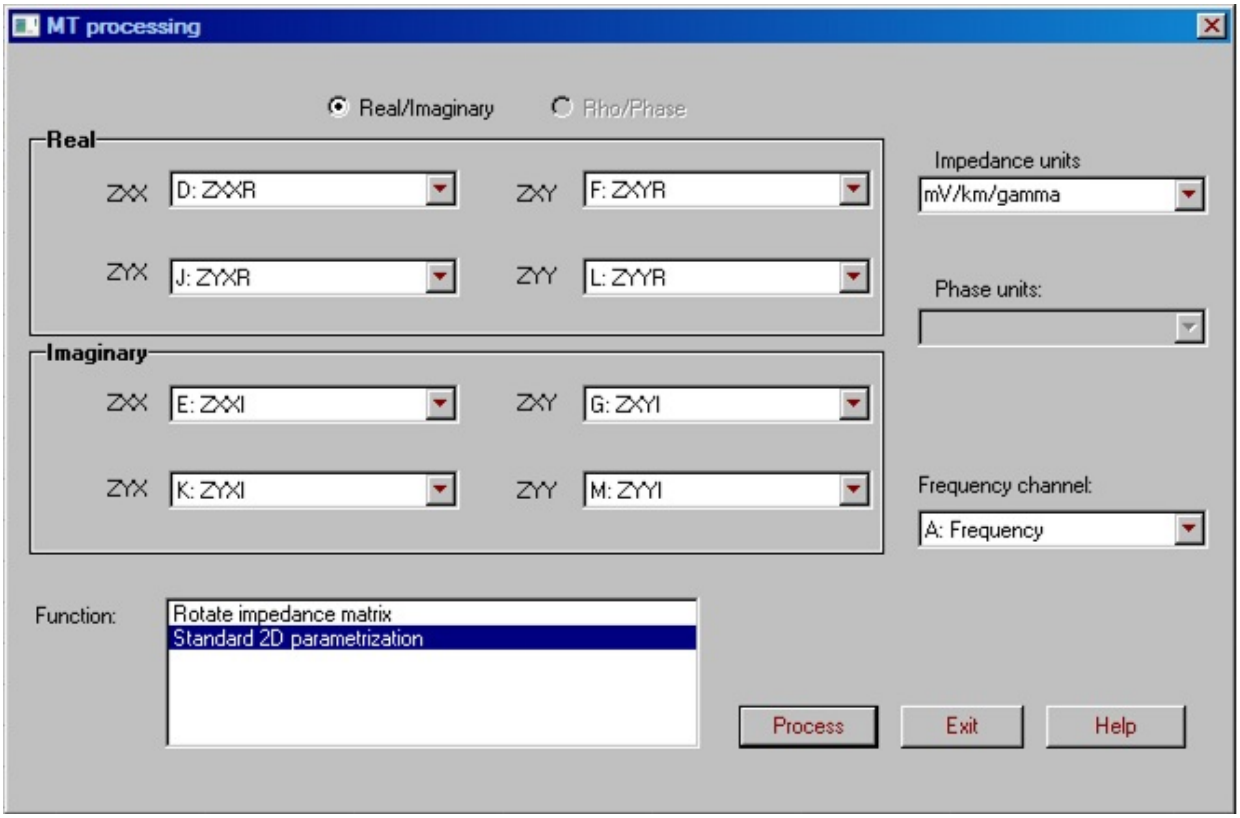
data value will be saved otherwise. The error is equal to the difference between the average and the data value measured at the station. Select whether existing channels should be overwritten or new channels created in which case, the names for the new channels may be edited.

Repeat statistics
Click the **OK** button to generate the data and display a page of statistics. You may save these results to a text file by clicking **Save**

- **Total number of stations** - Number of unique stations
- **Number of repeated stations** - Number of unique stations that have duplicates
- **Percentage of repeat stations** - Calculated using A/B*100 where A is the number of repeated stations and B is the total number of stations
- **Total number of readings** - Number of readings including duplicates
- **Number of repeated readings** - Number of readings that have duplicates, readings with no duplicates are not included
- **Percentage of repeat readings** - Calculated using C/D*100 where C is the number of repeated readings and D is the total number of readings
- **Maximum of repeat difference** - Largest of the values in the **Data error** channel
- **RMS difference** - root mean square of all the data error values

MT Processing

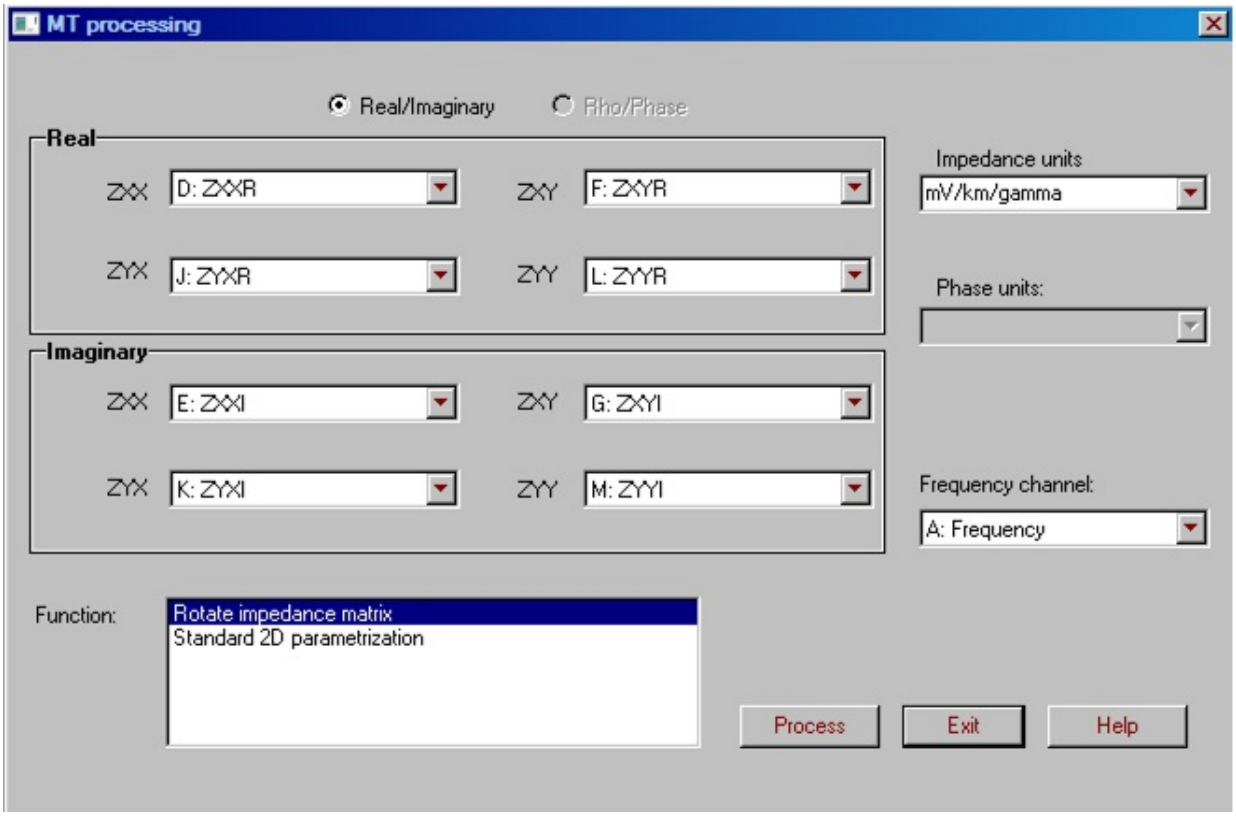
Select Processing -> Magnetotelluric Data Processing -> 2D MT Parameters from the menu. Produces 2D Strike Angle, Skew Angle and the 2 principal impedances (Rho/Phase)



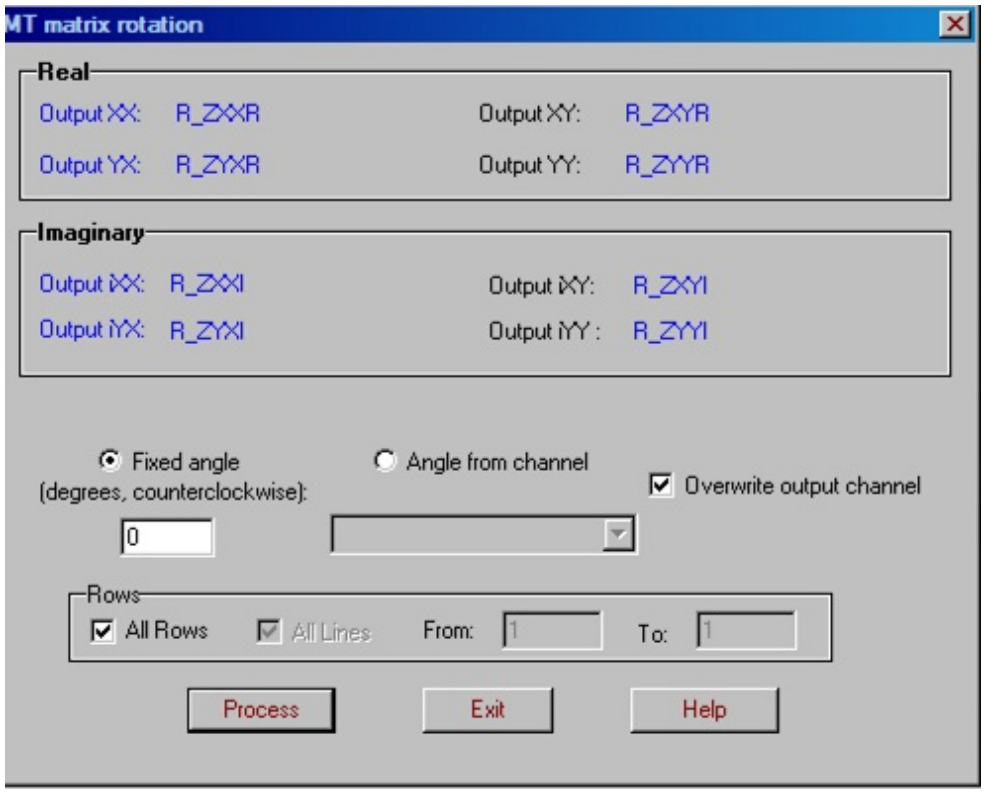
- Confirm that the channels selected for the real and imaginary parts of the impedance tensors are correct.>
- Also check that the channels for frequency and impedance units are correct.
- From this interface you can perform any of the operations listed

MT Matrix Rotation

Select Processing -> Magnetotelluric Data Processing -> MT Matrix Rotation from the menu.



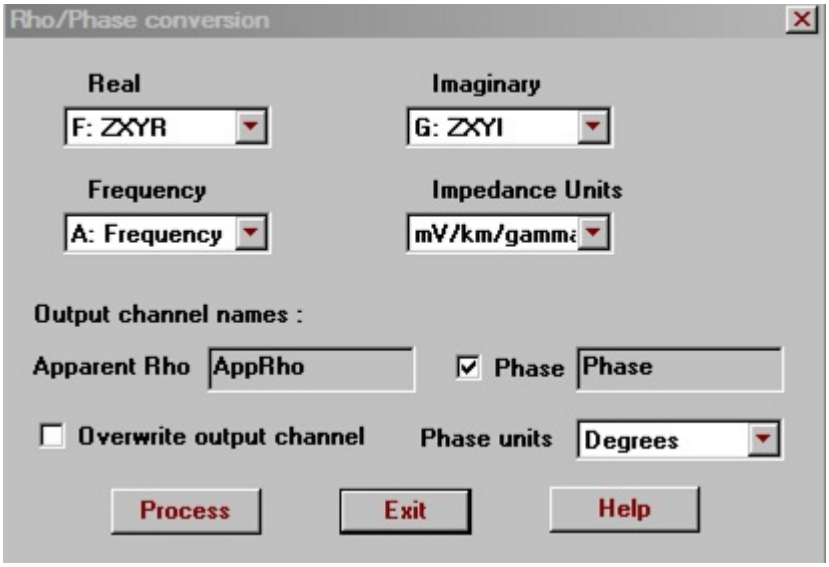
- Select your channels for real and imaginary of the 4 impedance elements plus frequency channel and your impedance units
- Select **Rotate impedance matrix** then **Process**



- The names of the output channels that will be created are displayed in the top half of the window
- Any channels that exist with the displayed output names will be overwritten unless the checkbox labelled **Overwrite output channel** is deselected.
- If **Fixed angle** is selected, enter the rotation angle in the box below the label. The output channel names are updated to include the angle as part of the name.
- If **Angle from channel** is selected, select a channel that contains the rotation angles you would like to use.
- In the **Rows** section, you can specify which records will be processed.
- When all selections are correct, click **Process** to create the new channels.

Apparent Rho/Phase conversion

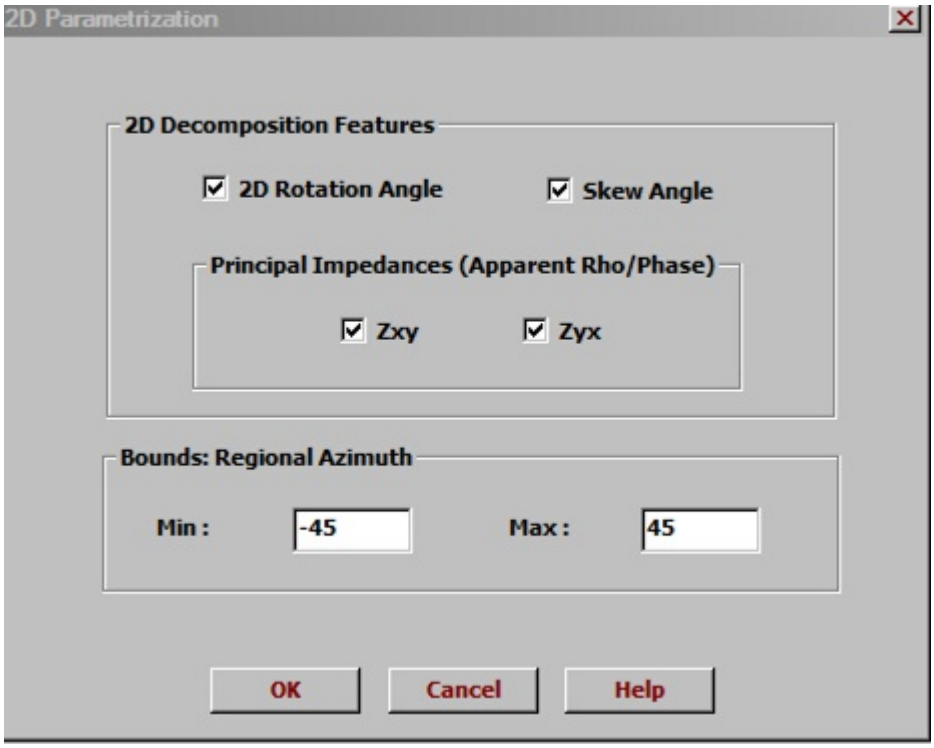
Selecting **Magnetotelluric Data Processing|Apparent Resistivity/Phase Covnersion** from the **Processing** menu launches the following interface:



- Select the impedance tensors you would like to convert in the **Output channels** section
- The names of the output channels that will be created are displayed in the **Output channel names** section.
- Any channels that exist with the displayed output names will be overwritten unless the checkbox labelled **Overwrite output channel** is deselected.
- The desired units for the phase can be selected in the **Phase units** box.
- When all selections are correct, click **Process** to create the new channels.

Standard 2D parametrization

Select **Standard 2D parametrization** from the main **MT Processing** window and continue to launch the following interface:

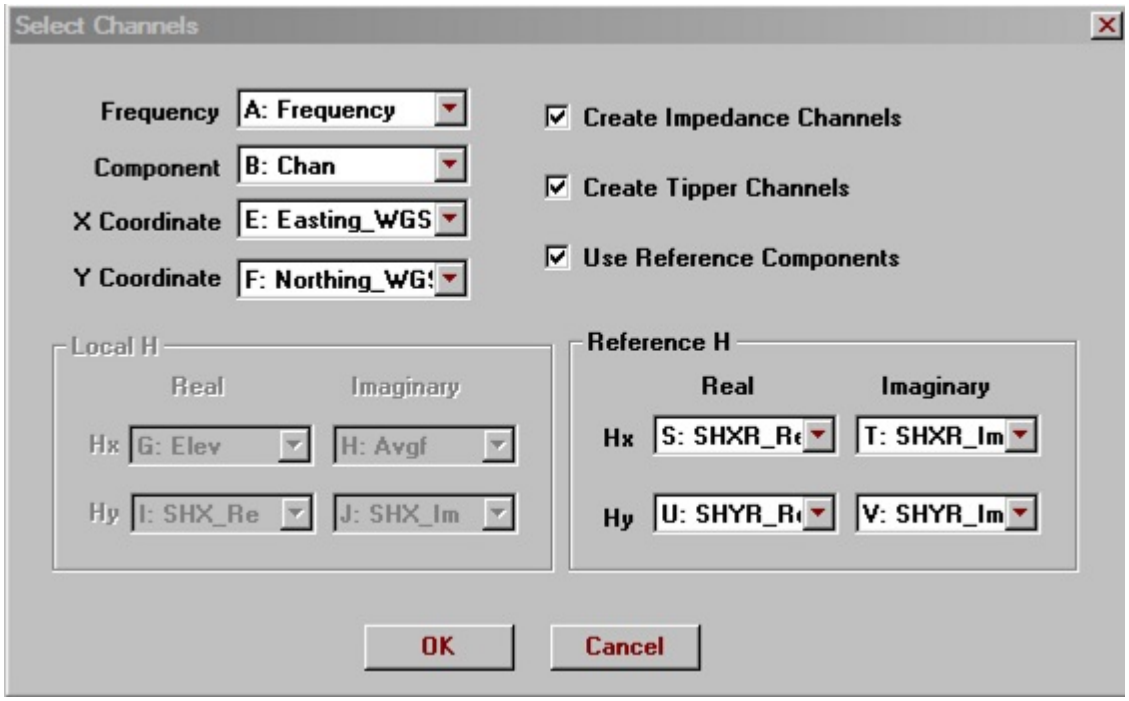


Select the **2D Decomposition Features** for which you would like to create channels. There are four different items to choose from.

Spectral to Impedance Conversion

Select Processing -> Magnetotelluric Data Processing -> Spectral to Impedance Conversion from the menu. You may choose either the standard or theoretical version of the algorithm.

If the spectral data is in .edi format then using QCTool's MT Spectral EDI import produces default channel names. Otherwise, some re-organizing and naming may be required.



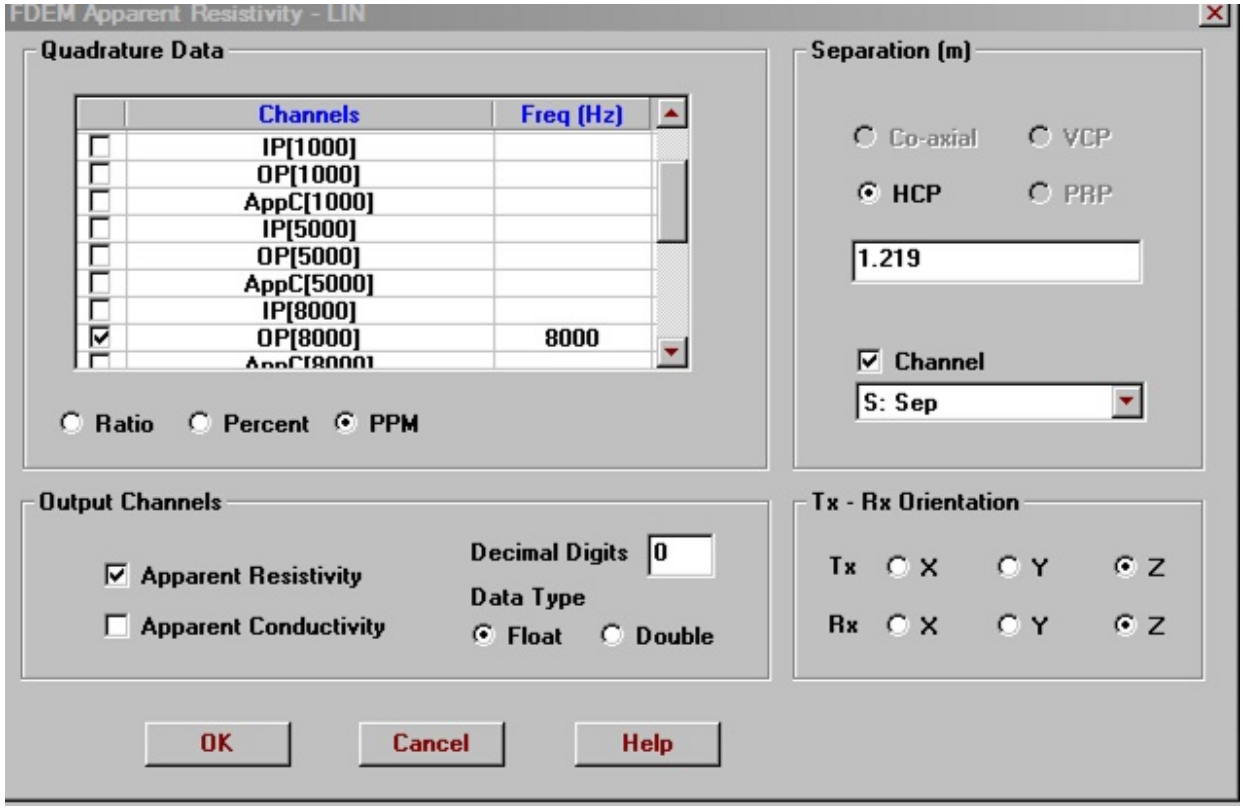
- Within the spreadsheet are 7 columns (records), one for Hx,Hy,Hz,Ex,Ey, and two additional ones Hx, Hy which can be remote channels
- Confirm that the channels selected for the frequency, component and coordinates are correct. Also check that the channels for the reference real and imaginary parts of the Hx and Hy components are correct.
- You may select whether you would like impedance, tipper or both channels created. At least one must be selected. The function will produce a new file.

- You may also select whether reference or local measurements will be used for the conversion.
- The function expects the magnetic reference channels be named SHXR and SHYR
- The data channels are named SHX, SHY, SHZ, SEX, SEY, for example, SHX_Re and SHX_Im

FDEM Apparent Resistivity - LIN

Based upon the single term LIN approximation for short offset dipole-dipole FDEM systems – EM31, EM38, GSSI Profiler, etc

Select **FDEM Processing** from the **Processing** menu. Then select **Apparent Resistivity - LIN**. The following window appears:



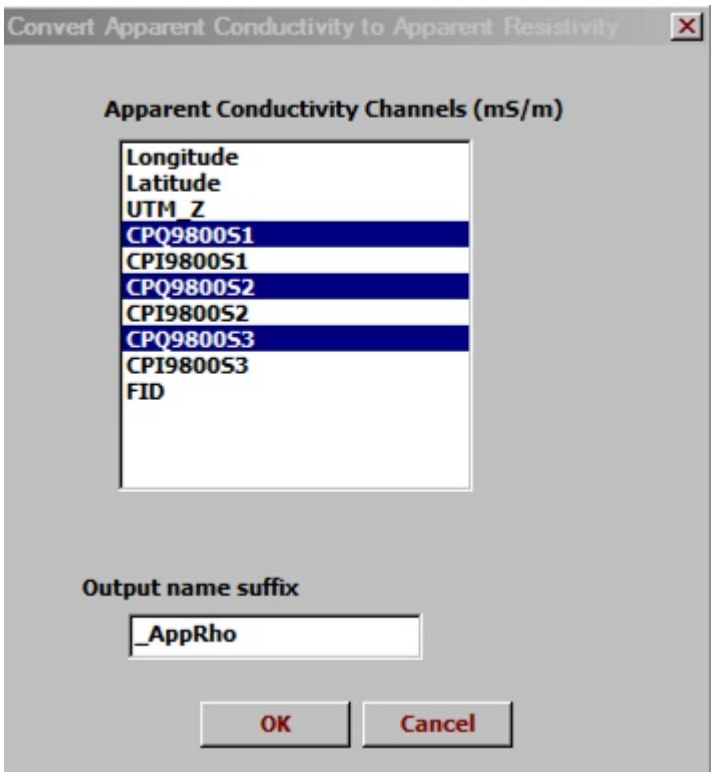
- This example is for the GSSI Profiler which outputs the Quadrature data in units of PPM.
- Click the checkbox beside the channels for the quadrature data in the **Quadrature Data** section.
- Units for the quadrature data can be specfied below the channel list
- Double click the relevant frequency cell to enter the value in Hertz if not automatic.
- The orientations of the transmitter and receiver can be selected in the **Tx-Rx Orientation** section.

- The value for the separation is entered in the **Separation** section as well as the array type of the system. You may choose between Co-axial, vertical coplanar (VCP), horizontal coplanar(HCP) or perpendicular (PRP). Different options will be available depending on what has been selected for the Tx-Rx Orientation.
- A channel for separation may be selected instead of entering a value by clicking the **Channel** checkbox in the **Separation** section and selecting a channel from the drop down list below.
- In the **Output Channel** section, select if apparent resistivity, apparent conductivity or both need to be calculated. Also, select the **Data Type** and number of **Decimal Digits** for the values in the output channel.
- Click the **OK** button when all the settings are correct.

Convert Apparent Conductivity to Apparent Resistivity

Many FDEM instruments output apparent conductivity as data units. These values are based upon a simple, very approximate formula which takes the relative data and converts to conductivity. In most cases, it is more useful to examine the apparent resistivity.

Select **FDEM Processing** from the **Processing** menu. Then select **Convert Apparent Conductivity/to ApparentResitivity**. The following window appears:

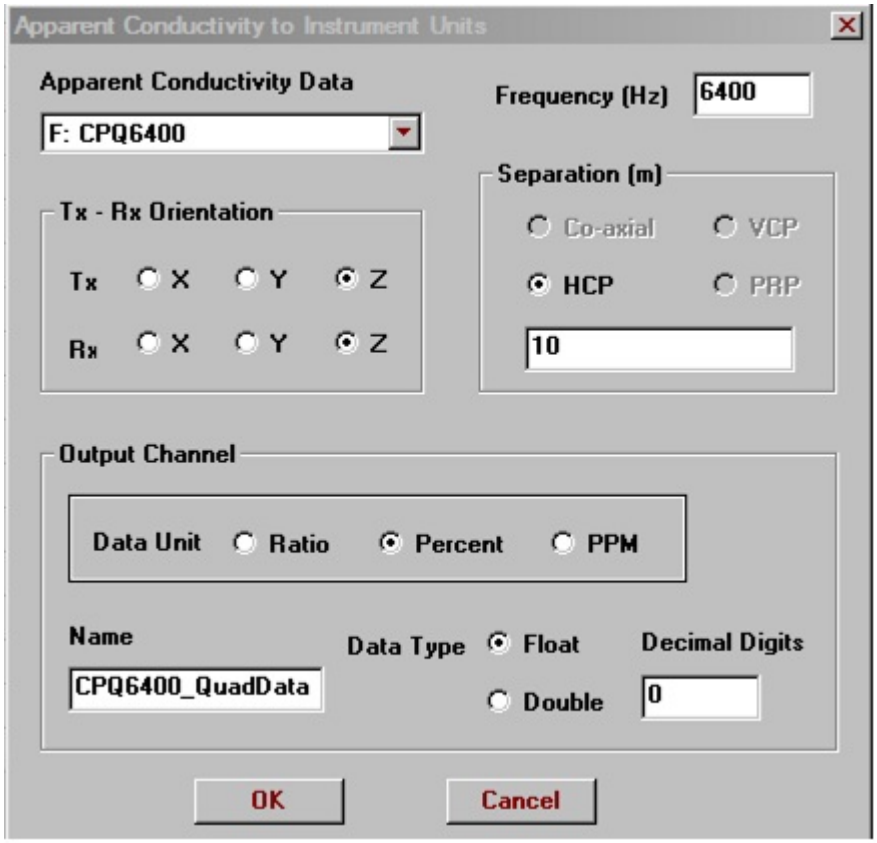


- Select the channels to be converted in the **Apparent Conductivity Channels** list box.
- EM data needs to be in units of mS/m.
- New channels will be created containing apparent resistivity values. The channel names will be constructed using the apparent conductivity channel name and the text entered in the **Output name suffix** box.

- Click the **OK** button to convert the data.

FDEM Apparent Conductivity to Instrument Units

You can convert FDEM data from apparent conductivity to instrument units. Select **FDEM Processing** from the **Processing** menu. Then select **Convert Apparent Conductivity/to Instrument Units**. The following window appears:



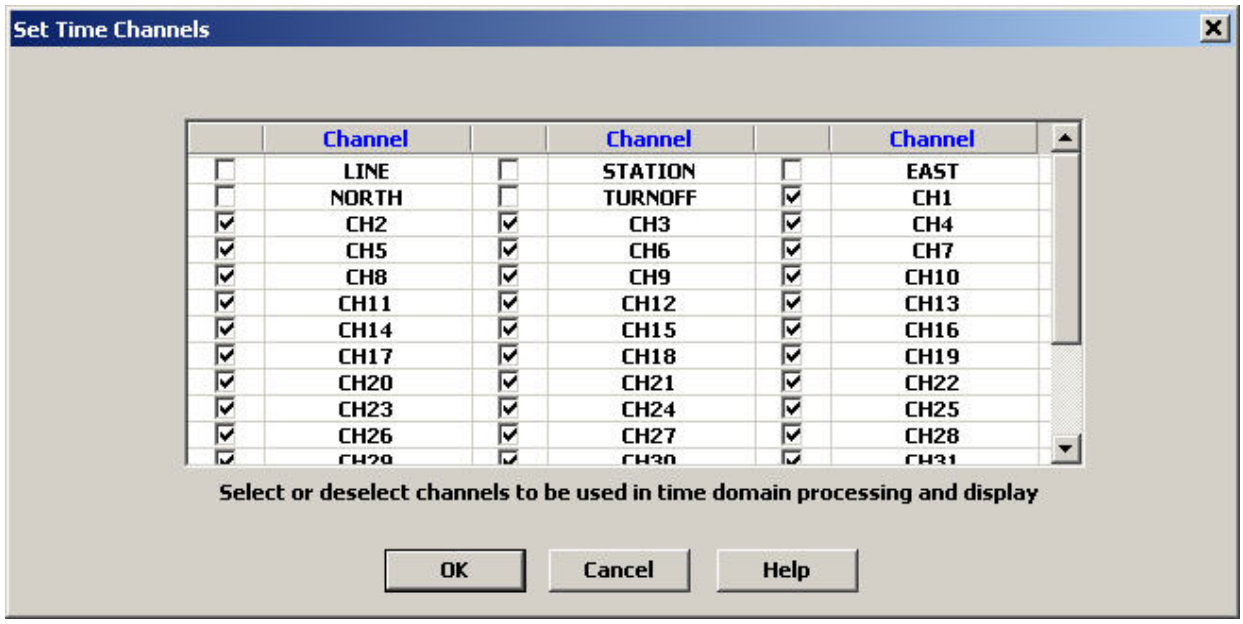
- This example is for EM34 data collected at 10m separation for which 6400Hz is used.
- Select an input column for the **Apparent Conductivity Data** in drop down box in the upper left corner of the window.
- Enter the frequency in Hertz.
- The orientations of the transmitter and receiver can be selected in the **Tx-Rx Orientation** section.
- The value for the separation is entered in the **Separation** section as well as the array type of the system. You may choose between Co-axial, vertical coplanar (VCP), horizontal coplanar(HCP) or perpendicular (PRP). Different options will be

available depending on what has been selected for the Tx-Rx Orientation.

- In the **Output Channel** section, enter a **Name** for the new channel that will be created for the output data. Also, select the **Data Units**, **Data Type** and the number of **Decimal Digits** for the values in the output channel.
- Many instruments use either Percent or PPm as raw data units.
- Click the **OK** button when all the settings are correct.

Set Time Channels

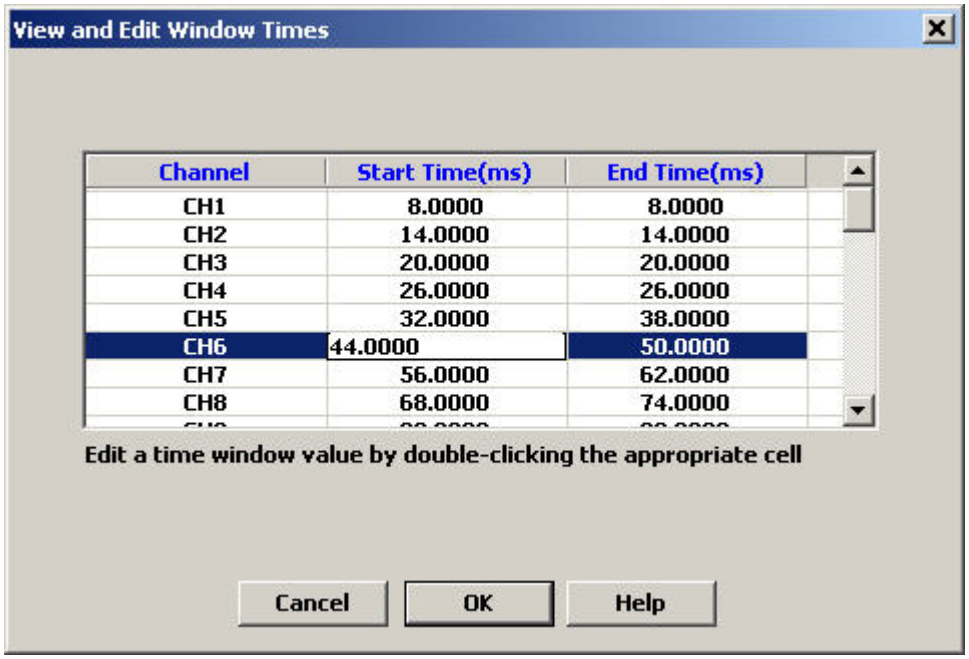
Select **Set Time Channels** from the **Processing/TDEM Processing** menu. The **Set Time Channels** interface appears.



- Channels with associated time windows are selected by default.
- Select the channels to which you would like to assign time windows.
- The time window values can be entered using either Load Window Times or View and Edit Window Times.
- Deselect any channels from which you would like to remove its associated time window.

View and Edit Time Windows

Select **View and Edit Window Times** from the **Processing/TDEM Processing** menu. The **View and Edit Window Times** interface appears.

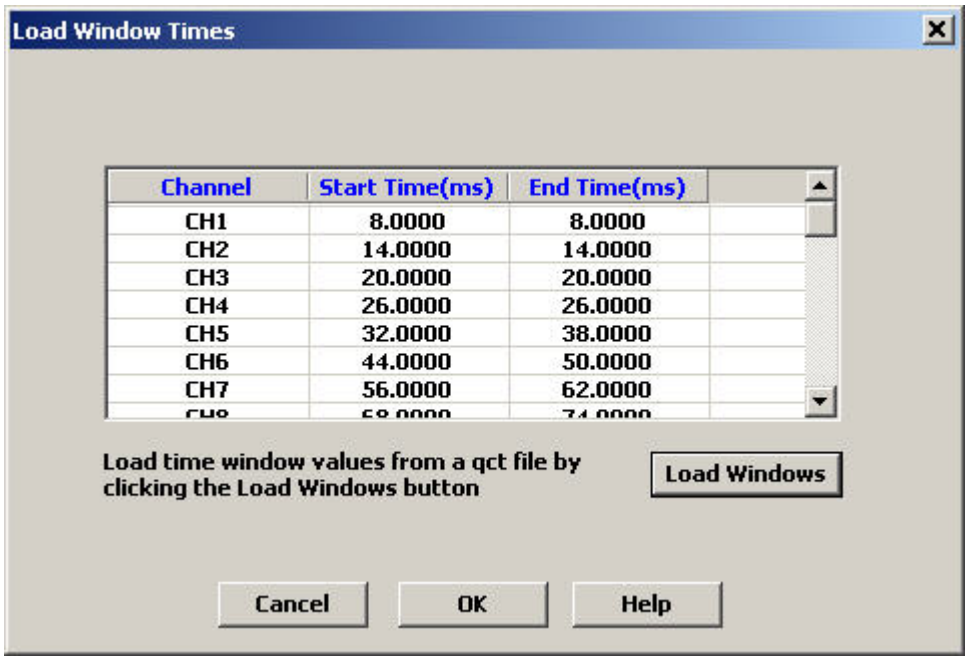


Existing start and end times are displayed for all channels specified as time window channels. Time window channels are defined using Set Time Channels.

- Any start time or end time can be edited by double clicking on the desired cell.
- Mid times will be displayed for files defined in that format.

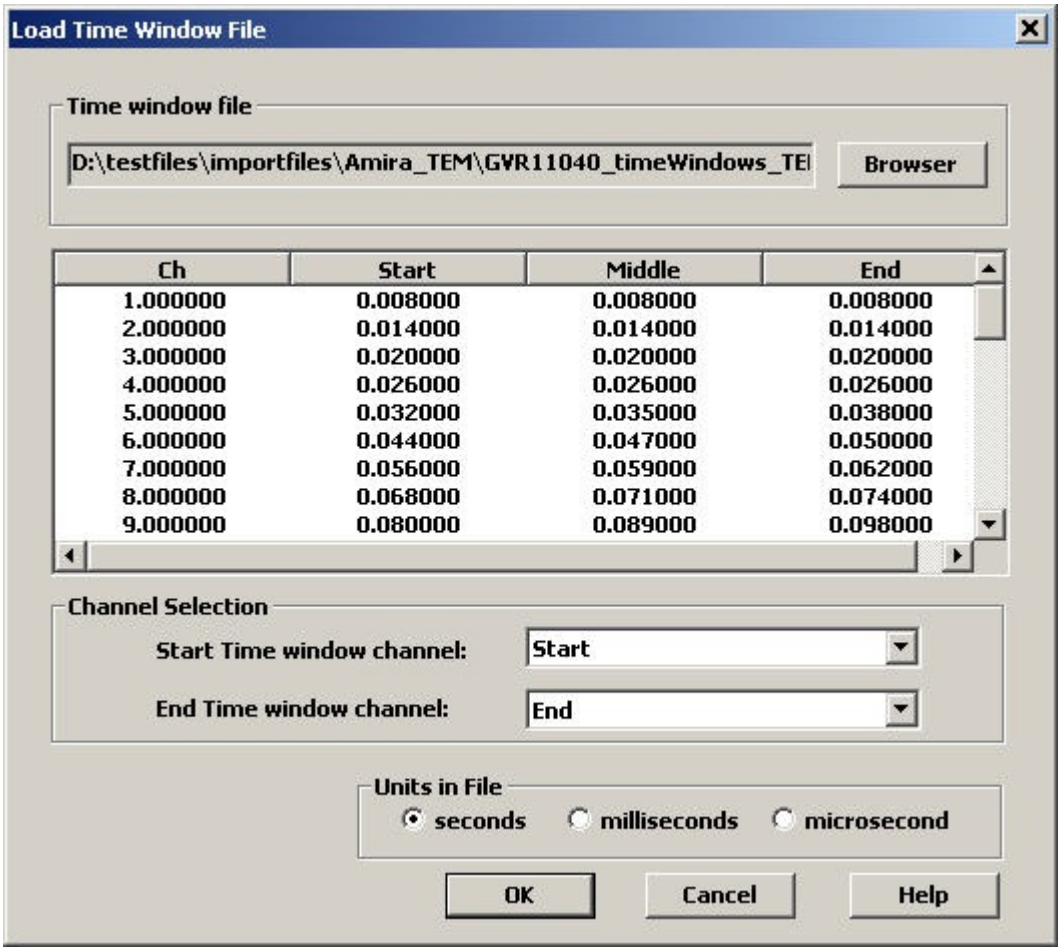
Load Window Times

Select **Load Window Times** from the **Processing/TDEM Processing** menu. The **Load Time Windows** interface appears.



Existing start and end times are displayed for channels specified as time window channels.

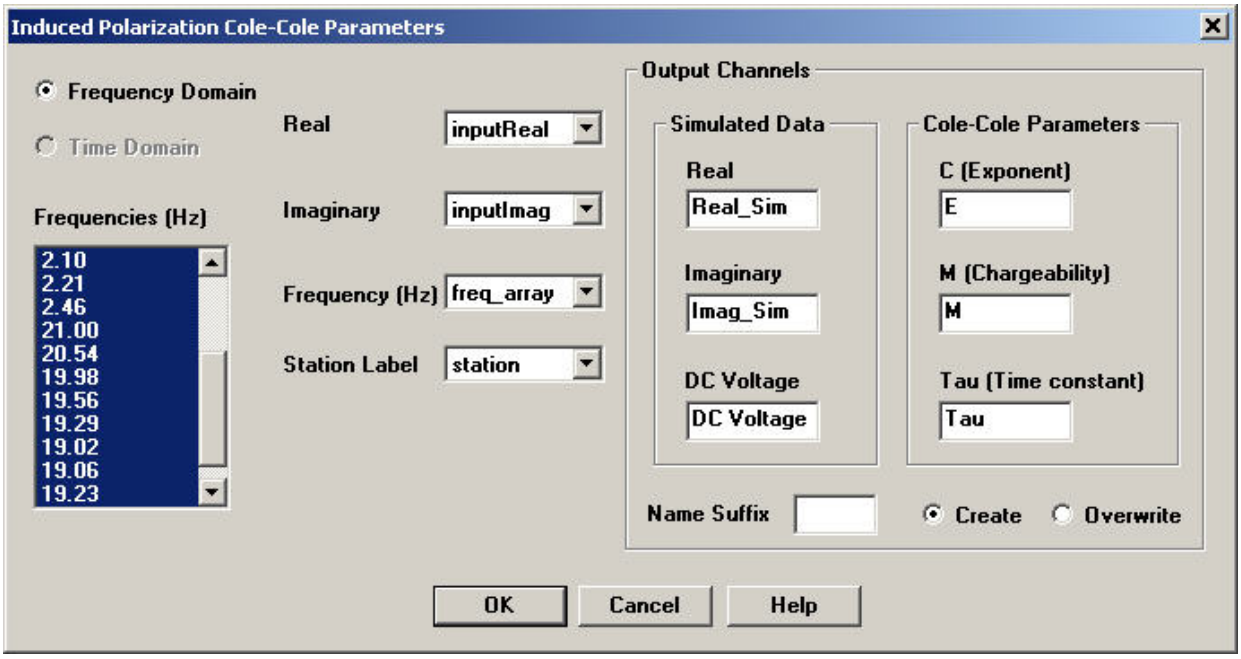
- Click the **Load Windows** button to replace the time values with value saved in a qct file.
- The interface to select the qct file appears



- Click browse to select the qct file containing the time window values. The file needs to have channels for start time and end time.
- The channels can be selected in boxes beside the **Start Time** and **End Time** labels
- Specify the units of the time values in the file in the **Units** section
- Click **OK** when all the settings are correct
- The values from the file are now applied to the time window channels
- The number of time windows in the file does need to be the same as the number of time window channels. For example, if the file only contains ten time windows, the time window values will only be assigned to the first ten time window channels.

Induced Polarization Cole-Cole Parameters

Select **IP/Resistivity Parameters** from the **Processing** menu. Then select **IP Cole-Cole**. The following window appears:

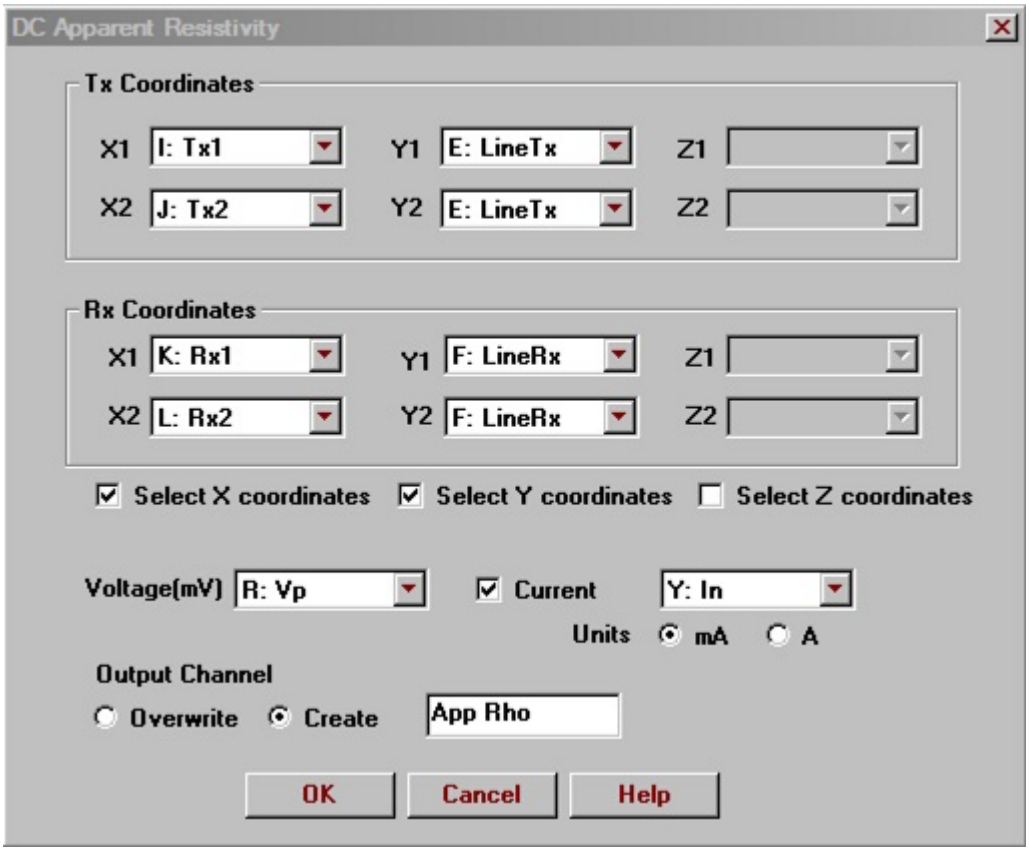


- Select input columns for real and imaginary parts of the data in the respective drop down menus. A column for the station label may also be selected.
- The **Frequencies** list box updates when the frequency column is changed. Unwanted frequencies can be deleted by clicking the appropriate value with the mouse while holding the control key.
- Six channels of data displayed in the **Output Channels** section are calculated with this tool. New channels can be created using the names entered or if the **Overwrite** option is clicked, existing channels can be selected to be overwritten.
- A **Name Suffix** can be entered that will be added to the names of each of the output channels if using the **Create** option
- Click the **OK** button when all the settings are correct.

DC Apparent Resistivity

Select **IP/Resistivity Parameters** from the **Processing** menu. Then select **DC App Rho**. The following window appears:

Note: for remote POLE electrodes, "*" will be recognized as infinite.



- Select input columns for transmitter and receiver electrode coordinates in the respective drop down menus. Any detected columns will be selected automatically. Indicate if any coordinates are or are not available by using the checkboxes labelled **Select X coordinates**, **Select Y coordinates**, or **Select Z coordinates**
- Select the channel for the voltage which must be in milliVolts.
- Select the channel for the current which must be in units of Amps or milliAmps. If a current channel is not available,

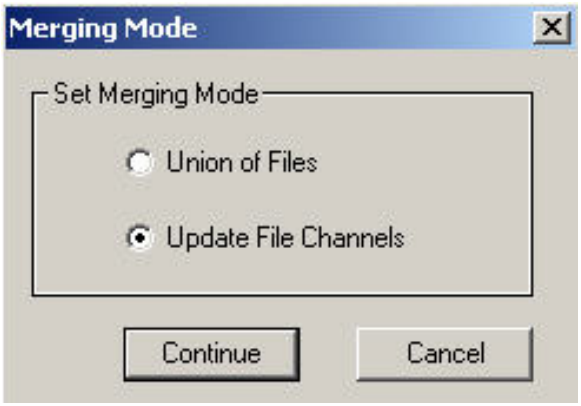
deselect the **Current** checkbox and a current of 1 Amp will be used for the calculation.

- Select **Create** to place the output data in a new channel with the entered name.
- Select **Overwrite** to overwrite an existing column that can be selected in the drop down list.
- Click the **OK** button when all the settings are correct.

Merging Files

QCTool allows two ways of merging files: updating one file with the data from another and merging two files using the same base channel (usually, the Fiducial channel). In both cases:

- Close any opened files.
- Select **Tools/Files/Merging Files** on the QCTool menu. The **Merging Mode** dialog opens, with the **Update File Channels** option selected by default:



- Leave the **Update File Channels** option selected if you want to update your file with channels from the 2nd file or select **Union of Files** to merge the two files, and click **Continue**.

See also:

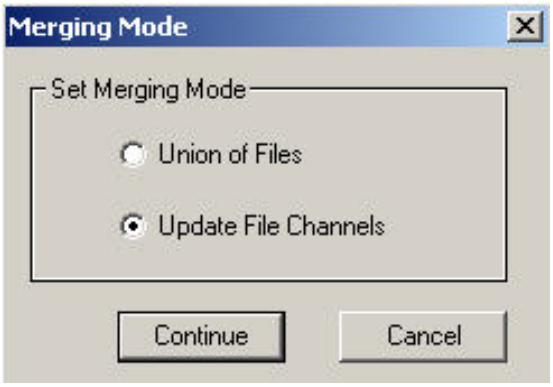
Union of Files

Updating Files

Updating Files

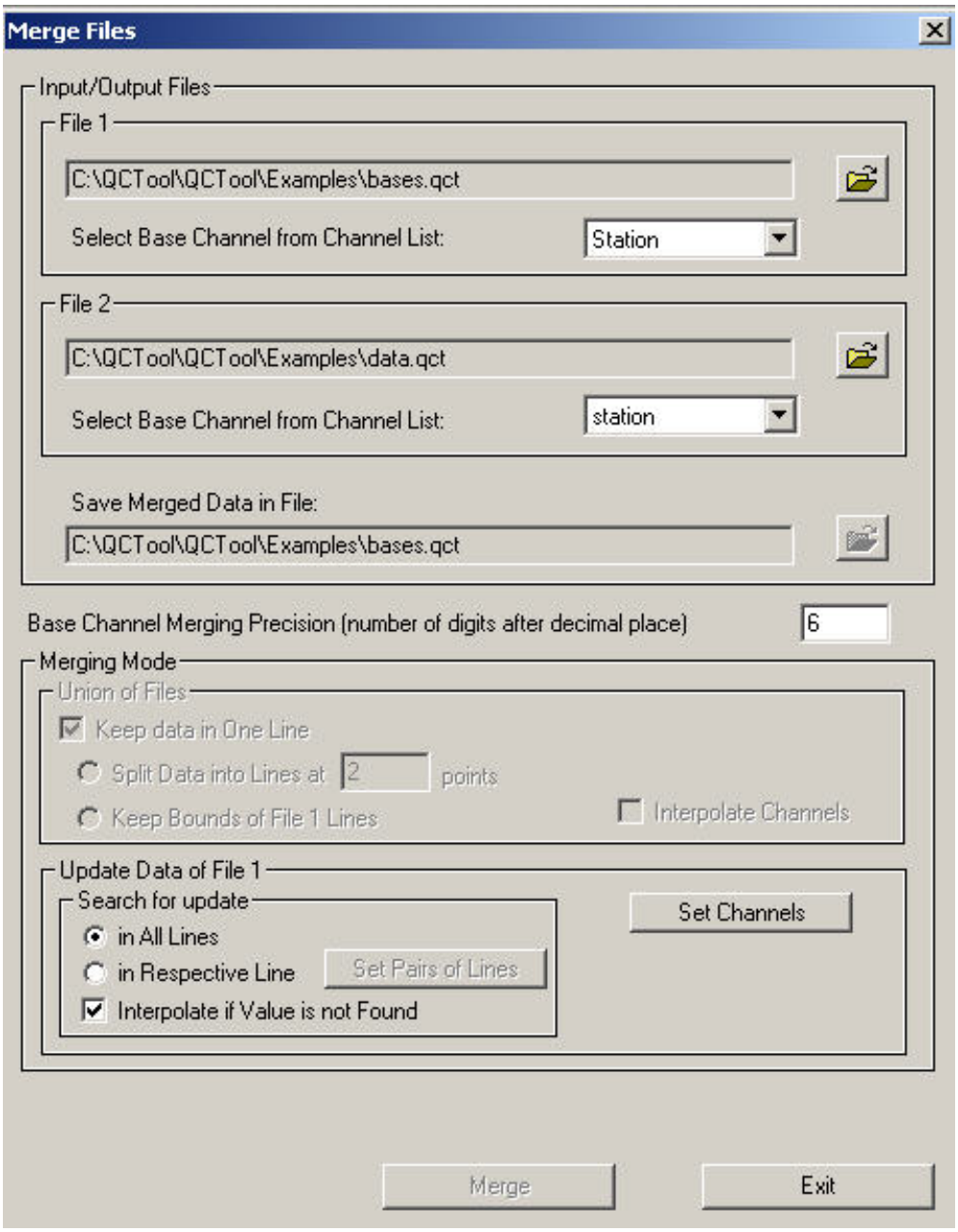
In some cases, you may need import separate columns from one file to another. This option is especially useful when you have two different files containing data of one survey; for example, one of the files contains the coordinates of your measuring stations and the other - the measured data.

- Select **Tools/Files/Merging Files** on the QCTool menu. In the **Merging Mode** dialog that appears:



- Leave the **Update File Channels** option selected and click **Continue**.

The **Merge Files** dialog opens:



In the **File 1** section of this dialog:

- Click the **Open** button to browse for the file to update.
- Select the base channel from the respective list below.

In the **File 2** section of this dialog:

- Click the **Open** button to browse for the file, from which you want to import an update.

- Select the base channel from the respective dropdown list below.

The **Save Merged Data in File** field in the bottom of the **Input/Output Files** section contains the location of the result file.

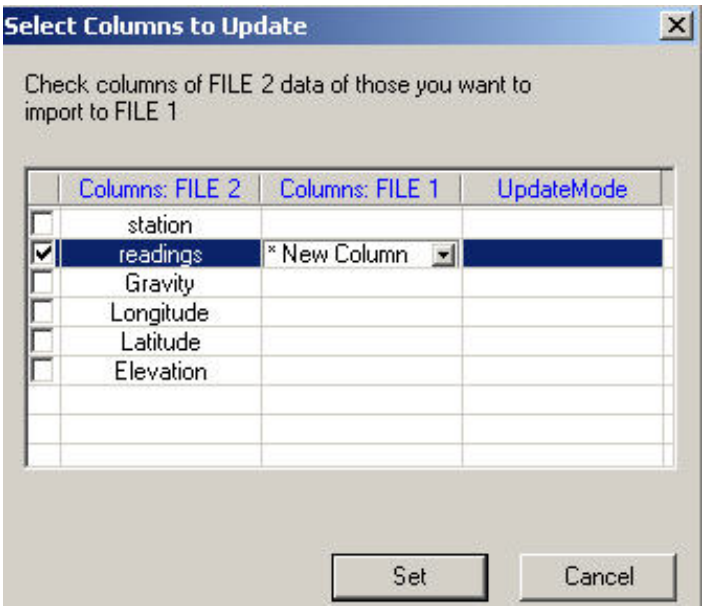
In the box below the **Input/Output Files** section:

- Select the number of digits after the decimal point. This is a precision order QCTool uses to merge base channel data from different files.

In the **Merging Mode** section:

- Click the **Set Channels** button. The **Select Colums to Update** dialog appears, with the first column containing the data from the file to import. There are two ways of import. The first imports data into a new column of the file to be updated and the second updates an existing column.

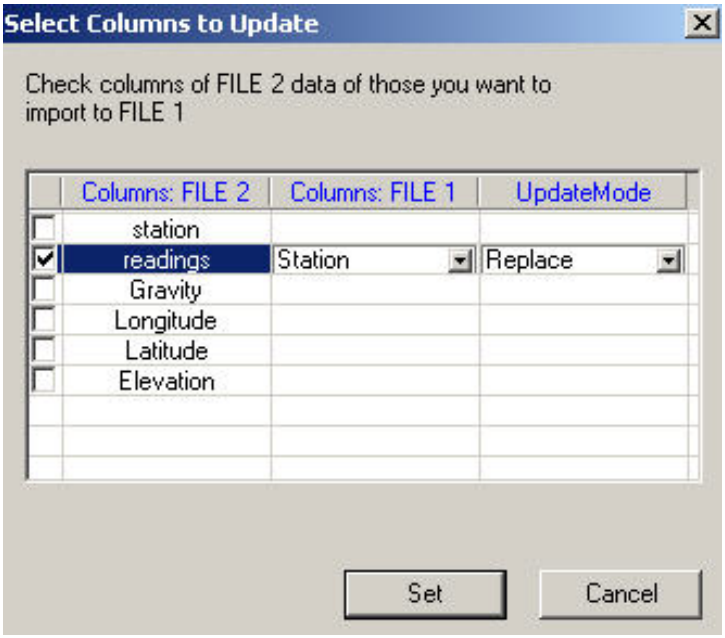
To import data into a new column:



- Click in the checkbox next to the column to import and leave **New Column** selected in the dropdown list that becomes active in the **Columns: File 1** column.

- Click **Set**

To update an existing column:



- Click in the checkbox next to the column to import and select the channel to update from the dropdown list in the **Columns: File 1** column.

- In the dropdown list that appears in the **Update Mode** column upon your selection, choose between three update modes: **Replace** (overwrites an existing column), **Average** (overwrites an existing column with the average calculated from the respective values in File 1 and File 2), and **Replace No Data** (overwrites only dummy values).

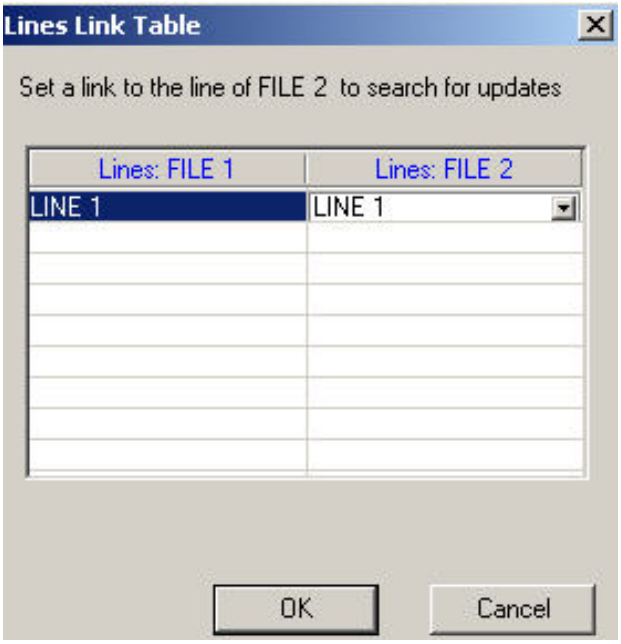
- Click **Set**.

Back in the **Search for Update** section of the **Merge Files** dialog:

- Select between two methods of searching for the update: **In All Lines** (this option searches data in all available lines and sorts them in order) and **In Respective Line** (this option searches and sorts data within each separate line). If you select the second option, you can also specify the pairs of lines from File 1 and File 2.

To do this:

- Click the **Set Pairs of Lines** button. In the **Lines Link Table** dialog that appears:

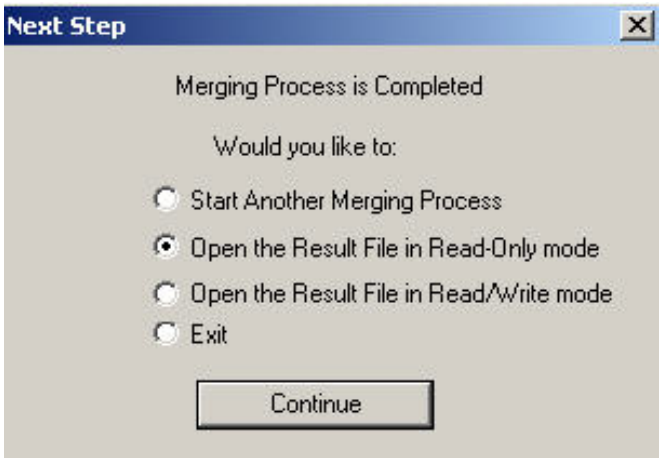


- Select the line to update in the first (File 1) column. The second column (File 2) now contains a dropdown list with all the lines available in File 2.
- Select a required line and click **OK**.

Back in the **Search for Update** section of the **Merge Files** dialog:

- Check the box next to **Interpolate if Value is not Found** if the column to import lacks a base channel vaue requested by the file to update.
- Click **Merge** in the bottom of the **Merge Files** dialog.

The **Next Step** dialog appears:



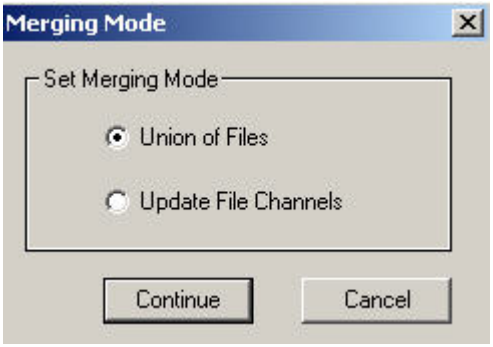
In this dialog:

- Select **Start Another Merging Process** if you want to import new updates into the same file or update another file. This brings the **Merge Files** dialog back. Repeat the steps described above.
- Select **Open the Result File in Read-Only Mode** if you do not want to change the updated file.
- Select **Open the Result File in Read/Write Mode** if you want to change the updated file.
- Select **Exit** to quit the **Merge Files** application.
- Click **Continue** to close the dialog.

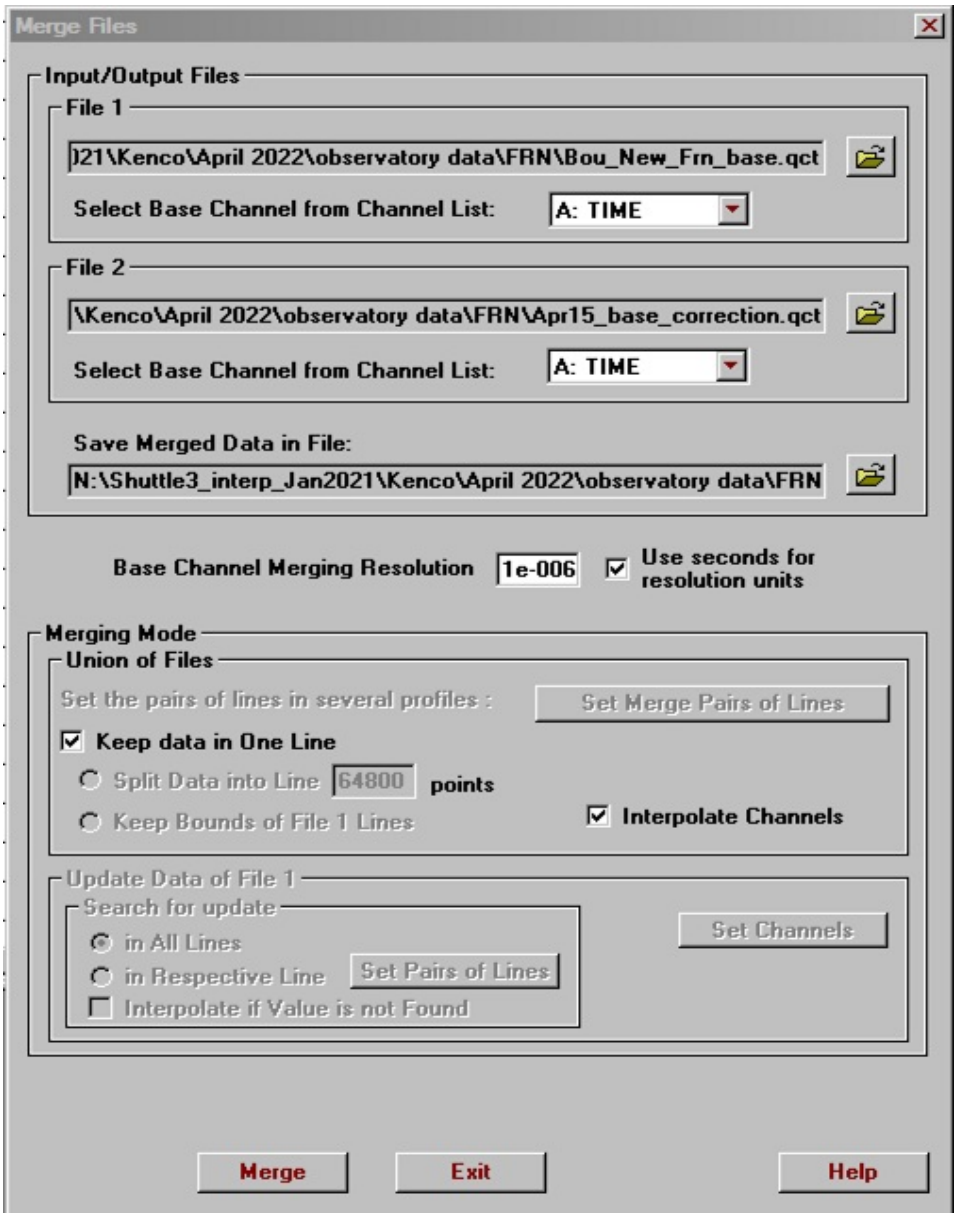
Union of Files

In case your file contains a base channel, which is a column with values sorted in a strictly ascending order (e.g. Fiducial Channel), you can merge it with one or more other similar files. For this purpose, you do not need to open these files.

- Select **Tools/Files/Merge Files**. In the **Merging Mode** dialog that appears:



- Select **Union of Files**. The **Merge Files** dialog opens:



In the **File 1** section of this dialog:

- Click the **Open** button  to display the standard **Open** dialog.
- Select the file you want to merge and click **Open**. Its name and path appear in the **File 1** field.
- Select the base channel from the dropdown list below, containing all channels from your file.

In the **File 2** section:

- Click the **Open** button  to bring up the standard **Open** dialog.

- Select the second file to merge with the one in the **File 1** section and click **Open**. Its name and path appear in the **File 2** field.

- Select the base channel from the respective dropdown list below.

In the **Save Merged Data in File** field:

- Click the **Open** button to bring up the standard **Save As** dialog, select the folder to save the new (merged) file into, and enter its name in the **File Name** field.

- Click **Save** to return to the **Merge Files** dialog. The name of the new file and its location appear in the **Save Merged Data** field.

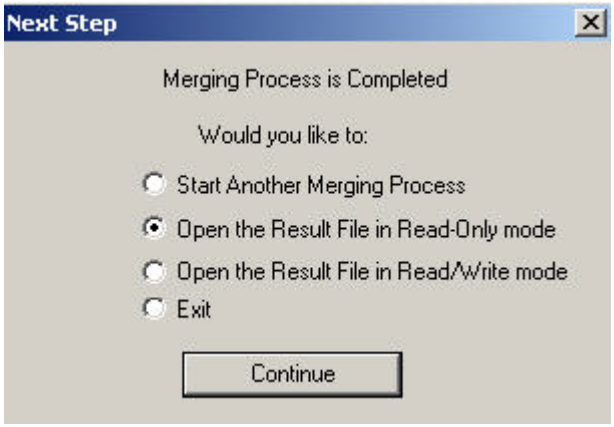
In the **Base Channel Merging Precision** field:

- Specify the number of digits after the decimal point for the base channel in the box on the right. This is a precision order QCTool uses to merge base channel data from different files.

In the **Merging Mode** section:

- The **Interpolate Channels** box is checked by default. De-select it to cancel interpolation. The cells, which otherwise would have been filled with the results of linear interpolation, will contain asterisks.
- To merge all data into one table without dividing it into lines, leave the **Keep Data in One Line** box checked (it is checked by default).
- To divide a table into lines, de-select the **Keep Data in One Line** box and specify the number of points per line in the **Split Data into Lines at Points** field.

Click **Merge** in the bottom of the **Merge Files** dialog. In case any of your files has no base channel, a message will warn you that the merging has failed. If the merging is successful, the **Next Step** dialog appears:



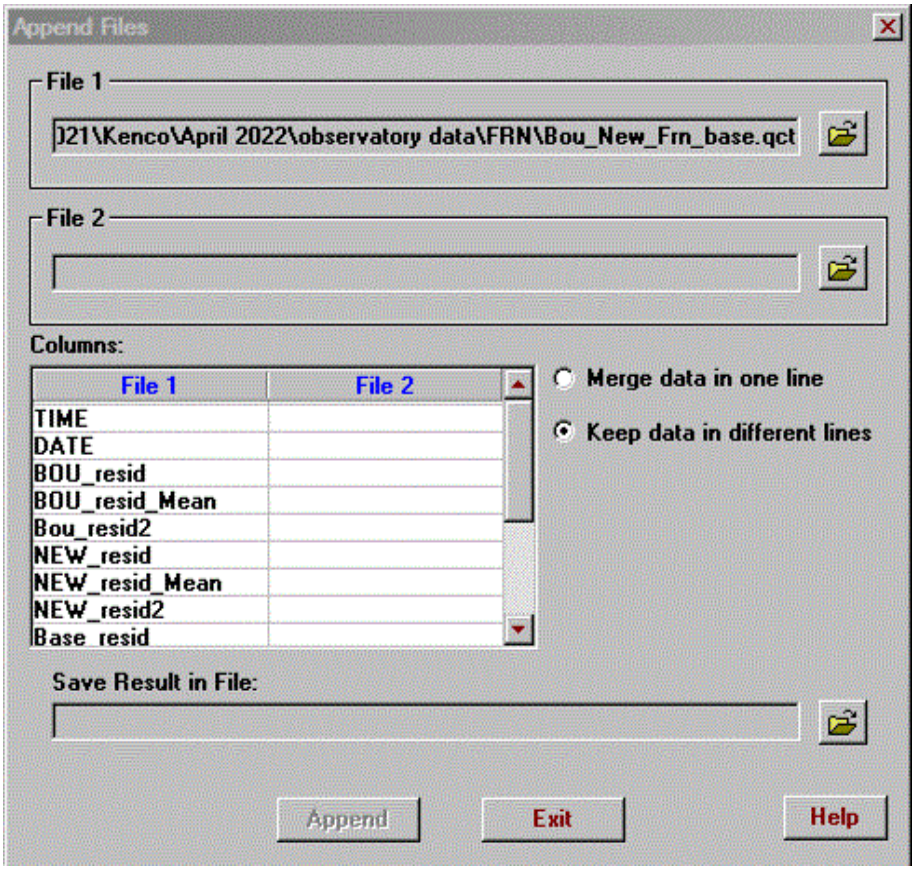
In this dialog:

- Select **Start Another Merging Process** if you want to merge one more file with the two already merged or to merge another pair of files. This brings the **Merge Files** dialog back. Repeat the steps described above as many times as needed.
- Select **Open the Result File in Read-Only Mode** if you do not want to change the merged file.
- Select **Open the Result File in Read/Write Mode** if you want to change the merged file.
- Select **Exit** to quit the **Merge Files** application.
- Click **Continue** to close the dialog.

Appending files

In some cases, you need to append, or attach, one file to another.

Select **Tools/Files/Append Files** on the QCTool menu. In the window that appears:



- Click the **Open** button in the **File 1** section to browse for the first file.
- Click the **Open** button in the **File 2** section to browse for the file to append.
- Compare the **File 1** and **File 2** columns in the respective table below.

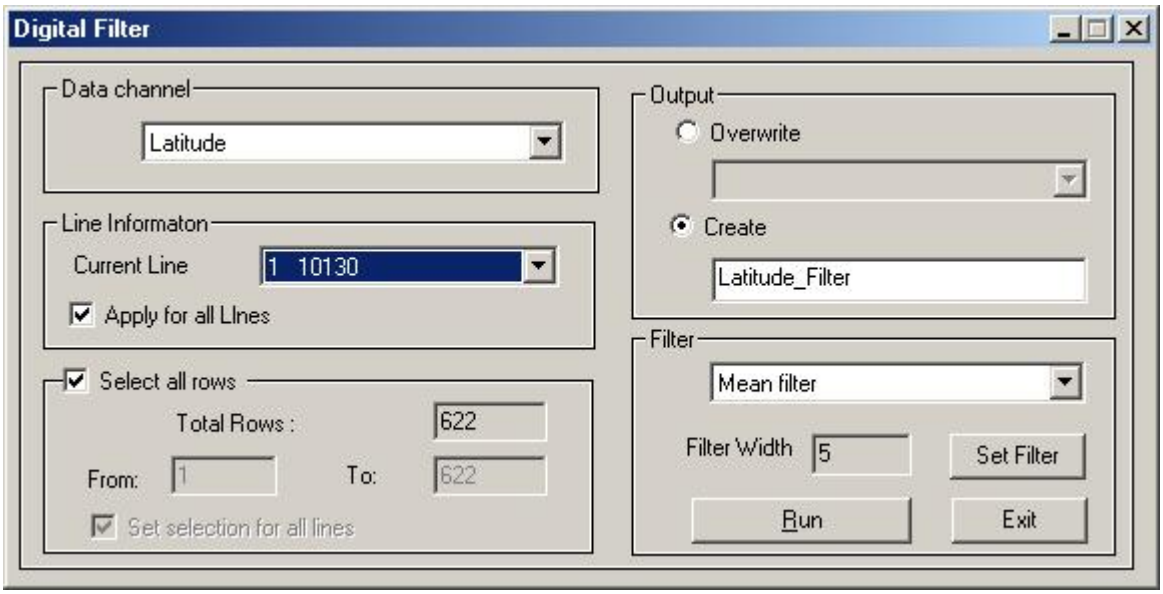
  If a **File 2** column does not coincide with a **File 1** column, you can always change the former to match the latter. Click on it and select a required column from the dropdown list to appear.

- Click the **Open** button in the **Save Results in File** section to specify the file to save your newly created data set in.

- Click **Append**. Your new file appears in the current window.

Digital Filter

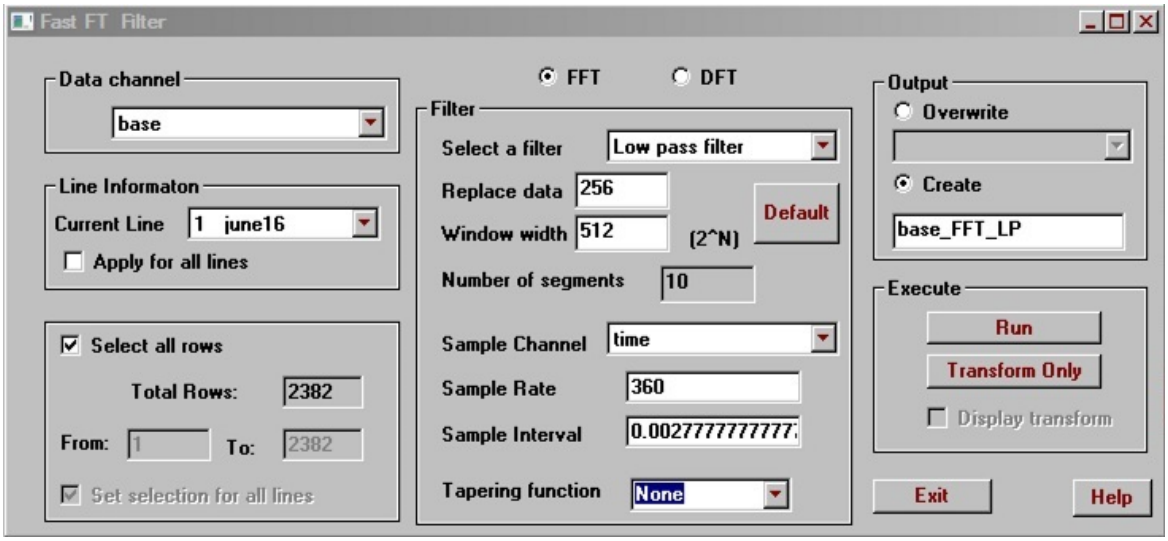
Click the **Processing -> Filters -> Digital Filter**. The dialog appears.



- Select a data channel for filtering
- Select a data range if needed in the **Select all rows** section. As well as **Apply for all Lines**
- Select output mode. If create mode is chosen, a new channel will be inserted after base channel
- Select filter type: Mean filter, Median filer, Gaussian filter, Decimation filter, Fraser filter, or Gaussian High Pass
  Decimation filter also is available as a separate menu item in **Processing -> Filters** submenu list.
- **Set Filter to** define the parameters for the filter
- Give the output channel a name
- Click the Run button to filter data

Fast FT Filter

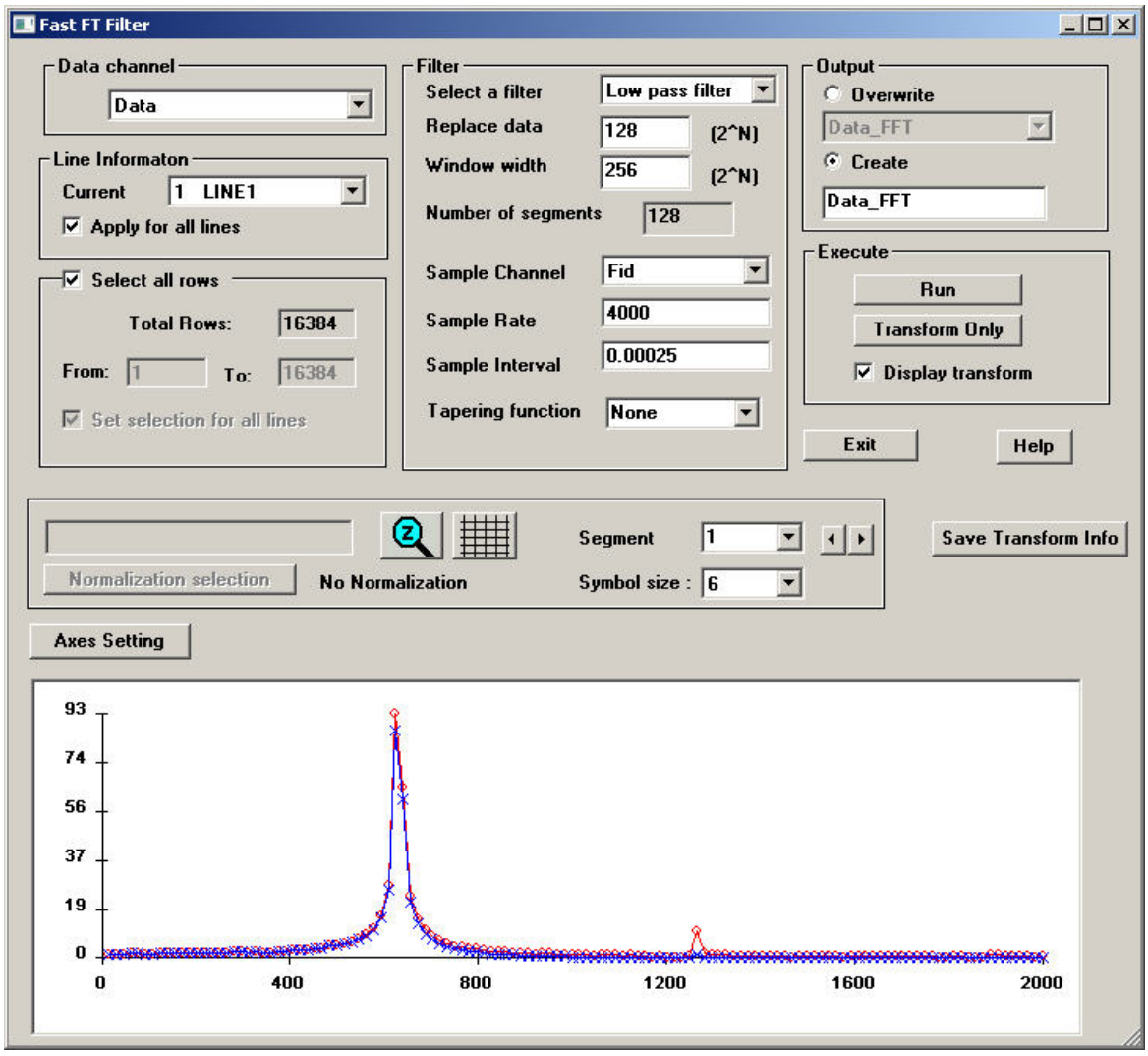
Click the Processing -> Filters -> FFT Filter. The dialog appears.



- Select FFT or DFT. Window width for FFT must be 2^N
- Select a data channel for filtering
- Select a data range if needed
- Select output mode. If create mode is chosen, a new channel will be inserted after base channel
- Select filter: Low pass filter, high pass filter, Band pass, Band remove and notch filter
- The data will be divided into segments of **Window width** length and sent through the FFT a segment at a time. **Replace data** specifies the number of values in each segment that are overwritten by neighbouring segments in order to avoid discontinuities. The **Window width** and **Replace data** values need to be powers of two.
- The **Sample Channel** needs to have data with a regular interval in units of seconds. The **Sample Rate** is the inverse of the **Sample Interval**
- Click the Run button to filter data

- Click the Transform only button to transform data without filtering

- Check the Display transform. The window changes to



- Click **Save transform Info** to save the transform information to a QCT file

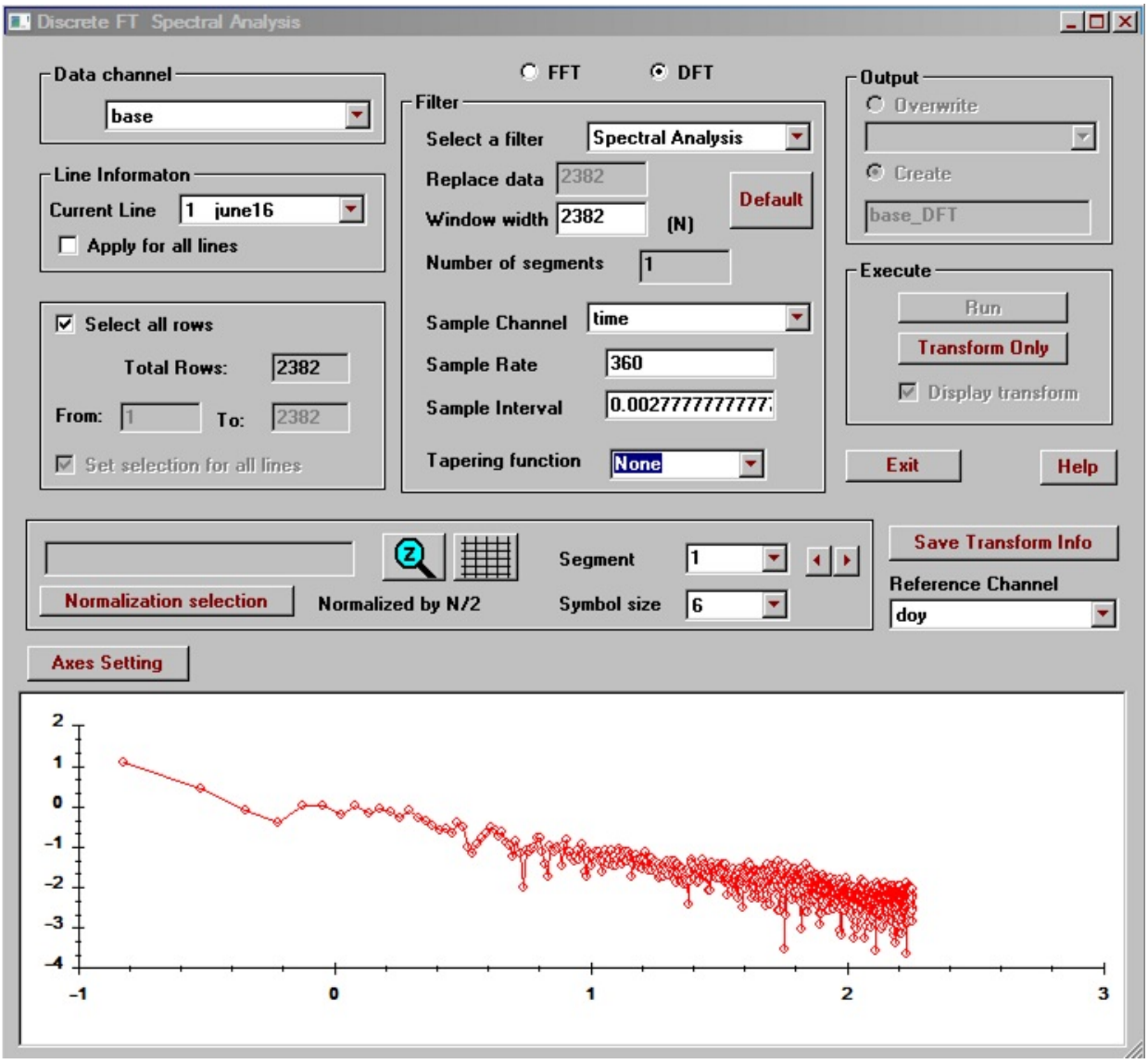
- The filtered data is plotted in blue. The unfiltered data is plotted in red.

- Click the  button to zoom in on a section of the plot. Choose the area you want to see by selecting rectangle on the plot with the mouse

- Click the  button to toggle the display of a mesh on the plot
- The appearance of the plot can be controlled by clicking the **Axes Settings** button

FFT Spectral Analysis

Click the Processing -> Filters -> FFT Spectral Analysis menu item.



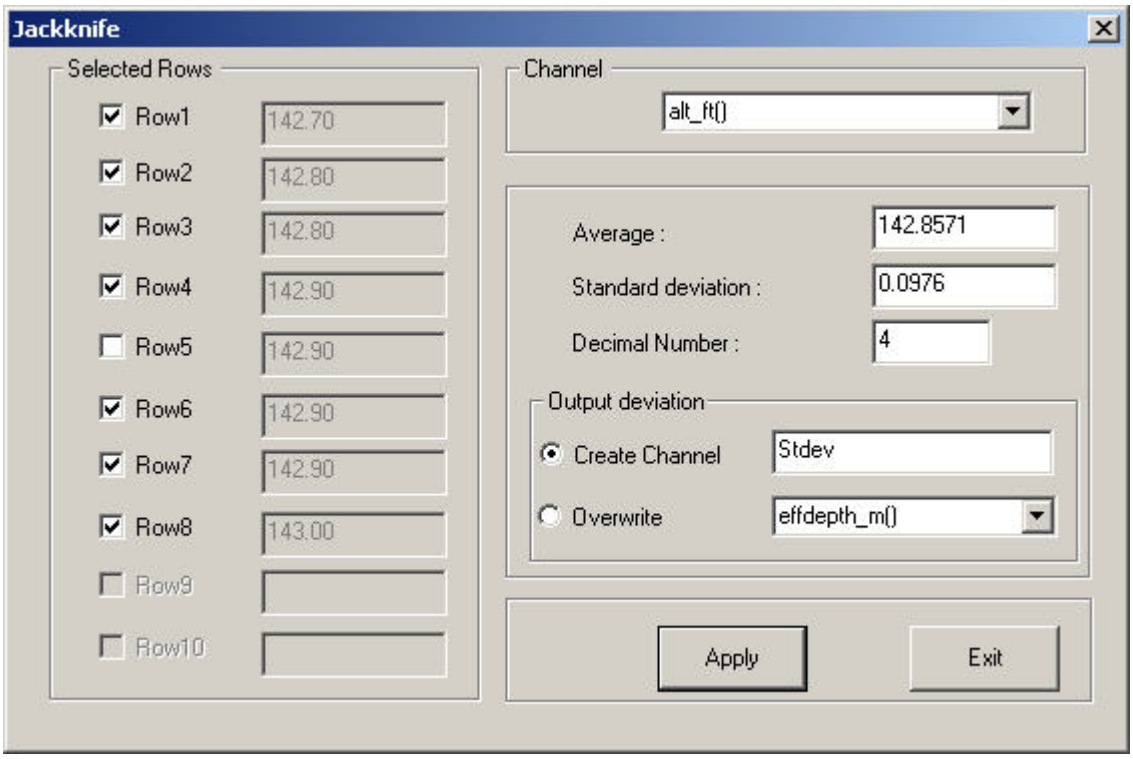
- Select FFT or DFT for Analysis
- FT requires window width to be 2^N but this is not required for DFT
- Select a data channel for spectral analysis

- Select number of rows if required
- The value for **Window width** will be the number of frequencies used in the spectral plot.
- The **Sample Channel** needs to have data with a regular interval in units of seconds. The **Sample Rate** is the inverse of the **Sample Interval**
- Click the Run button to filter data
- Click the Transform only button to display a spectral plot
- Select one of the tapering functions if desired. The data will be multiplied by this function before applying the FFT/DFT.
- Check the **Display transform** button to toggle the spectral plot.
- Click **Save transform Info** to save the transform information to a specified QCT file The file has channels for real, imaginary, amplitude and phase. If a tapering function has been selected, results with and without the use of the tapering function will be saved in the same file.
- Click the  button to zoom in on a section of the plot. Choose the area you want to see by selecting rectangle on the plot with the mouse
- Click the  button to toggle the display of a mesh on the plot
- The appearance of the plot can be controlled by clicking the **Axes Settings** button
- Click the **Normalization selection** button to normalize the data by the number of data values. You may also select between display of real, imaginary, amplitude and phase.

Stack Utilities

Jackknife

The purpose of the jackknife function is to facilitate processing of repeat stations in a dataset. To use this function, select up to 10 rows, right-click and select **Jackknife**. The following window appears:

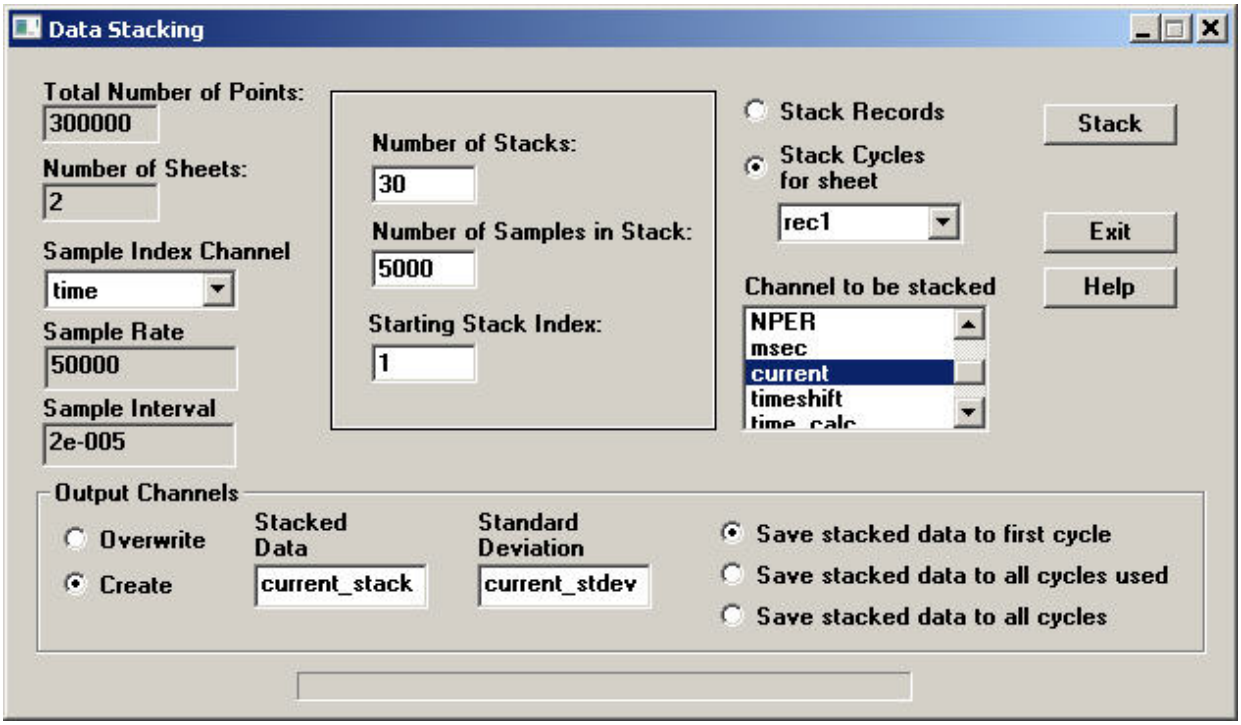


- Make a selection in the **Channel** section and you will be able to view the average and standard deviation for these rows. **Decimal number** specifies the number of decimal places for the displayed values.
- Deselect entries in the **Selected Rows** section as desired to observe the effect on the statistics. When you click **Apply** the average value of the selected rows will be saved for each channel in a new row.

- The standard deviation will also be saved. Select **Create Channel** and enter a new channel name to save the standard deviation in a new channel. Select **Overwrite** to replace an existing channel that you may also select.

Stack Data

Use this tool to average groups of data. Select **Processing/Stack Utilities/Stack Data** to open the following window:



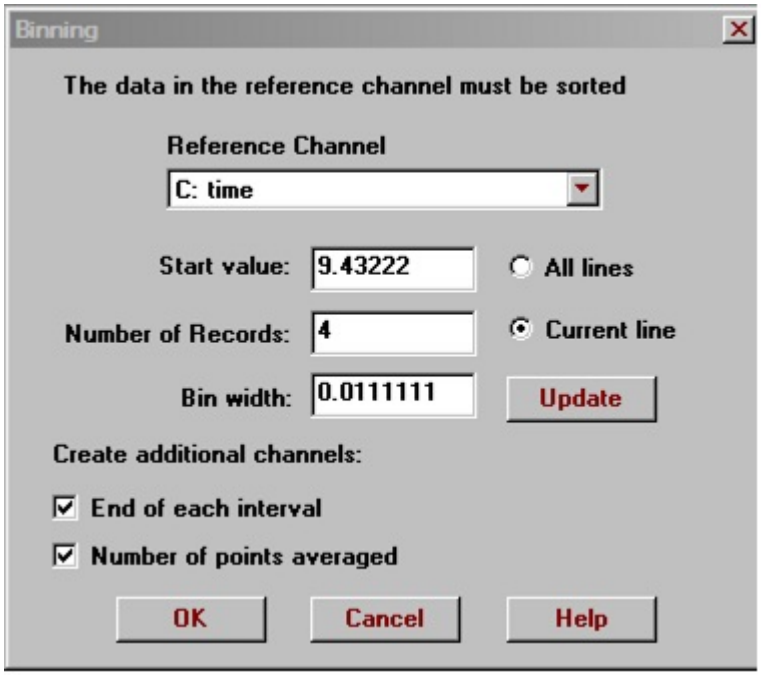
- Enter the number of groups of data values to be averaged in the box labelled **Number of Stacks**. and set the **Number of Samples in Stack**.
- Initial data values can be skipped by entering a value for the **Starting Stack Index**. E.g. entering 10 will not include the first nine values in the stack result.
- Select a channel for the **Sample Index Channel**. Values in this channel need to be increasing at a regular interval.
- Select the channel you would like to process in the **Channel to be stacked** list.
- Processing may be performed on a single sheet by selecting **Stack Cycles for Sheet**. Stacking will only be done for the selected sheet.

- The **Stack Records** option will be enabled if there is more than one sheet. The number of samples will be set to the maximum number of values in a sheet when this option is selected.
- Start the processing by clicking the **Stack** button
- New channels will be created for **Stacked Data** and **Standard Deviation** using the specified channel names. Select **Overwrite** if you would prefer to replace existing channels.
- Data results can be saved to the **first cycle** only, **all cycles used** or **all cycles**.

Bin Data

To perform data binning on a channel, select **Processing/Stack Utilities/Bin Data** to open the following window:

This application bins consecutive records by averaging each channel through each bin for all channels and producing new averaged records and compressing the data simultaneously.

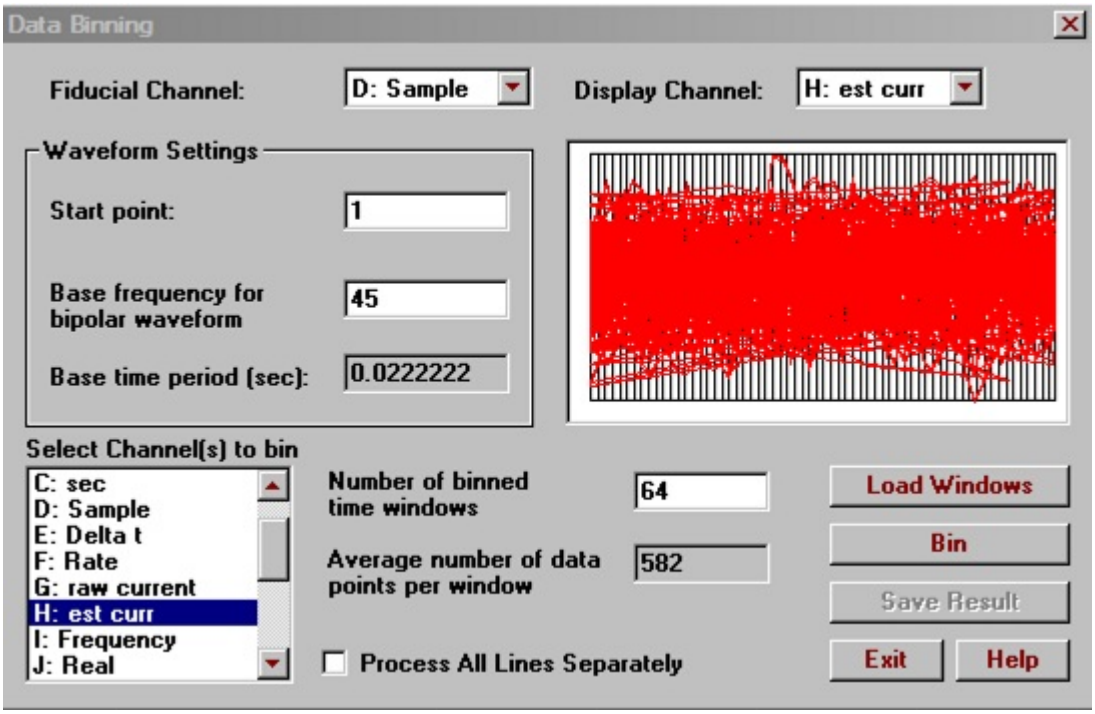


- For this tool to work correctly, the channel you select in the **Reference Channel** drop down menu needs to be sorted to be monotonic increasing or decreasing (use SORT function).
- You can process **All lines/spreadsheets** or the **Current line** only.
- A number of data bins are created. The first bin will contain the values in the channel that fall beteween two values. Those values in the units of each channel are the **Start value** and the **Start Value** plus the **Bin width**. The second bin starts where the first bin ends and uses the same bin width. The range of the next bin to the end of each channel are calculated in the same fashion.

- The bin width is equal to the sampling rate of the reference channel times the number of records.
- Select **End of each interval** to create a channel containing the end values of the bins
- Select **Number of points averaged** to create a channel containing the number of records averaged in each bin.
- Click OK and the first row in each bin will contain an average of all the rows in the bin. The other rows are deleted.

Bin TEM Data

To perform binning of digitally sampled full waveform data into windows, select **Processing/Stack Utilities/Bin TD Data** to open the following window:



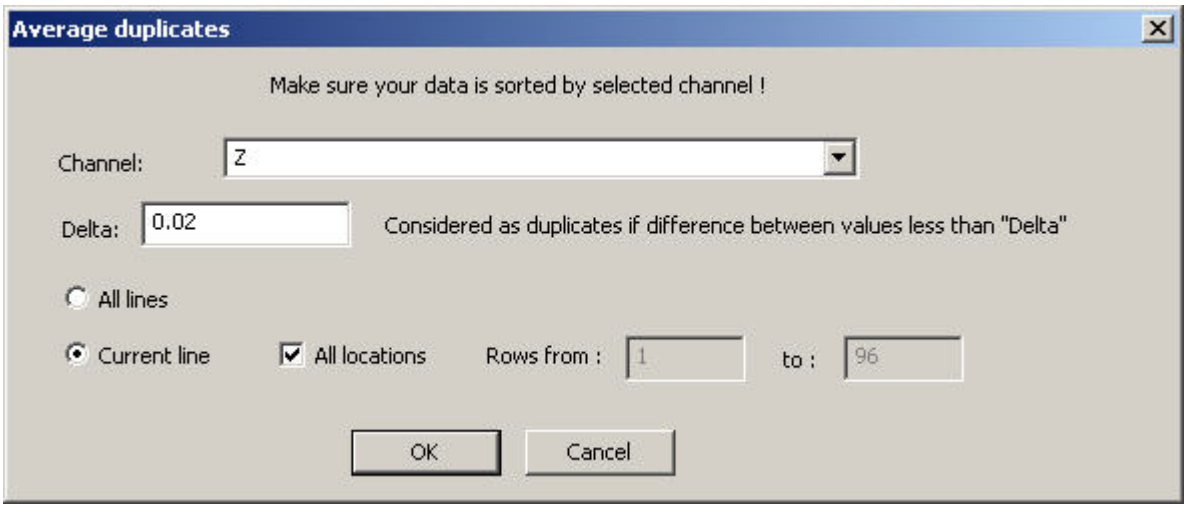
- If the **Process All Lines Separately** checkbox is not checked, all lines will be merged into one line.
- Select a **Fiducial Channel** and a **Display Channel** to display a small plot of the **Display Channel**
- **Start point** specifies the row that contains the first time value you want to use in the **Fiducial Channel**.
- Enter a **Base frequency** and the **Base time period** will be updated. The value for the **Base time period** must be less than the largest value in the **Fiducial channel**.
- Select a display channel and it will be plotted in red and the processed data is plotted in blue.
- Click on the channels on which you would like to perform processing in the **Select Channel(s) to bin** box

- Load your windows definition files
- The **Base time period** will be divided by the **Number of binned time windows**. All the values that fall within a certain data bin will be averaged.
- Click **Bin** to begin processing
- Click **Save Results** to save the processed data to a qctool file.

For a base time period, each averaged value will be given its own channel.

Average Duplicates

To average rows that are duplicates, select **Processing/Stack Utilities/Average Duplicates** to open the following window:



- The channel indicated in the **Channel** drop down menu will be analyzed for duplicate values. Starting from the first value, each subsequent value that is within the range indicated by the **Delta** value will be added to the group to be averaged until a value is detected that falls out of the **Delta** range. Then a new group of values to averaged is created.
- You can specify whether you want averaging to be performed on **All lines**, only the **Current line** or for a specific range of rows on the current line. Specify the range in the boxes labelled **Rows from:** and **to:**
- In order for this tool to work correctly, the selected channel needs to be sorted. Click **OK** to start processing. The first row in each averaging group will contain the results. The other rows retain their original values. However, in the selected channel, the entries not averaged are set to a dummy value.

QCTool Reference

File menu commands

The File menu offers the following commands:

	New QCT File	Creates a new qct-file.
	Open QCT File	Opens an existing qct-file.
	Close	Closes an opened qct-file.
	Save QCT file	Saves an opened qct-file using the same file name.
	Save As	Saves an opened qct-file to a specified file name.
	Import File	Import data from a specified file name.
	Print	Prints a map from the Grid Mapping Window 
	Print Preview	Displays the document on the screen as it would appear printed.
	Print Setup	Selects a printer and printer connection.
	Print Selection Area	Selects a portion of channels for printing.
	File Manager	Invokes a dialog to manage the opened qct-files.
	Exit	Exits QCTool

Edit menu commands

The Edit menu offers the following commands:

	Undo	Reverse previous editing operation.
	Cut	Deletes data from the spreadsheet and moves it to the clipboard.
	Copy	Copies data from the spreadsheet to the clipboard.
	Paste	Pastes data from the clipboard into the spreadsheet.

Note: Edit menu functions not implemented in the current version

View menu commands

The View menu offers the following commands:

Toolbar	Shows or hides the toolbar.
Status Bar	Shows or hides the status bar.
Spreadsheet Font	Displays the interface to specify the font for the spreadsheet.

Window menu commands

The Window menu offers the following commands, which enable you to arrange multiple views of multiple documents in the application window:

	New Window	Creates a new window that views the same document.
	Cascade	Arranges windows in an overlapped fashion.
	Tile Horizontally	Arranges windows Horizontally.
	Tile Vertically	Arranges windows Vertically.
	Arrange Icons	Arranges icons of closed windows.

Help menu commands

The Help menu offers the following commands, which provide you assistance with this application:

	QCTool Help	Offers you an index to topics on which you can get help.
	About QCTool	Displays the version number and license type for this application.
	Register a QCTool license	If using QCTool in evaluation mode, select this option to enter your serial number. Registration can also be done by selecting the Register option from the QCTool folder on the Windows Start menu. You must have administrative privileges to do this task.
	Deregister a QCTool license	Should you wish to move a registration from one computer to another, simply select "Deregister". Then you may install and register on another computer.
	Check for Updates	Updates are periodically made available for current licenses. Use this option to see if there are any updates for which you are eligible. Updates can also be checked by selecting the Update option from the QCTool folder on the Windows Start menu. You must have administrative privileges to do this task.
	Use Proxy to get Updates	Select this option when having problems connecting to the update server to check for updates.
	Extend Update Period	When your license maintenance has expired you may purchase a one year license renewal. After you see a window displaying your serial number, your browser will be launched to a site where you will be able to pay for your renewal.

Toolbar

The toolbar is displayed across the top of the application window, below the menu bar. The toolbar provides quick mouse access to many tools used in QCTool,

To hide or display the Toolbar, choose Toolbar from the View menu (ALT, V, T).

 Create a new qct file. QCTool displays the interface to specify the channel and line information.

 Open an existing qct-file. QCTool displays the Open dialog box, in which you can locate and open the desired file.

 Save the active qct-file with its current name.

 Import data from a file. QCTool displays the supported file formats you can choose from. Then, you can locate the desired file and save the data to a qct format.

 Refresh. Effectively, saves then reopens a file.

 Removes all plots and saved plot settings.

 Invoke a dialog to manage the opened qct-files. You can close, hide or activate different view windows of the opened file.

 Show/hide the spreadsheet window of active qct-file.

 Show/hide the plotter window of active qct-file. If plotter is being displayed for the first time, the data from the selected column will be plotted

 Show/hide the grid window of active qct-file.

Make a selection from a list of existing lines.

 Duplicate the current line.

 Invoke a dialog with line information.

 Cut the current line at a selected point.

 Mark a selected line as deleted.

 Mark selected points (locations) as deleted.

 Insert a new point (location).

 Insert points (locations) .

 Undo last action on the spreadsheet.

License and Ordering Information

QCTool(R)

QCTool SOFTWARE LICENSE

EIKON TECHNOLOGIES LIMITED

You should carefully read the following terms and conditions before using this software. Unless you have a different license agreement signed by Eikon Technologies Ltd("PEI"), by using, copying, transmitting, distributing or installing QCTool, you agree to all of the terms of this license.

If you do not agree to any of the terms of this license, then do not install, distribute, transmit, copy or use QCTool. If you have paid PEI or an authorized PEI distributor for this copy individually, you may return it unused, within thirty (30) days after purchase, for a full refund of your payment minus costs.

When this copy of QCTool has been registered, it may be installed on a single workstation used by a single person or used non-simultaneously by multiple people. This is not a concurrent use license.

The license is granted to the purchaser and is non-transferable to either another individual, company, organization or department within an organization.

Until QCTool has been registered, it is a free evaluation software. Subject to the terms below, you are hereby licensed by PEI to use this software for evaluation purposes for up to seven (7) days without charge.

You may, without making any payment to PEI or its agents:
a) give exact copies of the evaluation version of QCTool personally to anyone;
b) distribute exact copies of the evaluation version of QCTool, if done exclusively through electronic channels;
and
c) produce as many exact copies of the evaluation version of QCTool

as you wish, for purposes of distribution as described in (a) and (b) above.

YOU ARE SPECIFICALLY PROHIBITED FROM CHARGING, OR REQUESTING DONATIONS, FOR ANY COPIES, HOWEVER MADE, AND FROM DISTRIBUTING SUCH COPIES WITH OTHER PRODUCTS OF ANY KIND, COMMERCIAL OR OTHERWISE, WITHOUT PRIOR WRITTEN PERMISSION FROM PEI.

PEI RESERVES THE RIGHT TO REVOKE THE ABOVE DISTRIBUTION RIGHTS AT ANY TIME, FOR ANY OR NO REASON.

You may not alter or modify the Software, nor give anyone permission to do so.

THIS SOFTWARE, AND ALL ACCOMPANYING FILES, DATA AND MATERIALS, ARE DISTRIBUTED "AS IS" AND WITH NO WARRANTIES OF ANY KIND, WHETHER EXPRESS OR IMPLIED. Good data processing procedure dictates that any program be thoroughly tested with non-critical data before relying on it. The user must assume the entire risk of using the program. THIS DISCLAIMER OF WARRANTY CONSTITUTES AN ESSENTIAL PART OF THE AGREEMENT.

ANY LIABILITY OF PEI WILL BE LIMITED EXCLUSIVELY TO REFUND OF PURCHASE PRICE. IN ADDITION, IN NO EVENT SHALL PEI, OR ITS PRINCIPALS, SHAREHOLDERS, OFFICERS, EMPLOYEES, AFFILIATES, CONTRACTORS, SUBSIDIARIES, OR PARENT ORGANIZATIONS, BE LIABLE FOR ANY INCIDENTAL, CONSEQUENTIAL, OR PUNITIVE DAMAGES WHATSOEVER RELATING TO THE USE OF QCTool, OR YOUR RELATIONSHIP WITH PEI.

IN ADDITION, IN NO EVENT DOES PEI AUTHORIZE YOU TO UTILIZE QCTool IN APPLICATIONS OR SYSTEMS WHERE QCTool's FAILURE TO PERFORM CAN REASONABLY BE

EXPECTED TO RESULT IN A SIGNIFICANT PHYSICAL INJURY, OR IN LOSS OF LIFE. ANY SUCH USE BY YOU IS ENTIRELY AT YOUR OWN RISK, AND YOU AGREE TO HOLD PEI HARMLESS FROM ANY CLAIMS OR LOSSES RELATING TO SUCH UNAUTHORIZED USE.

This Agreement is the complete statement of the Agreement between the parties on the subject matter, and merges and supersedes all other or prior understandings, purchase orders, agreements and arrangements. This Agreement shall be governed by the laws of the Province of Ontario. Exclusive jurisdiction and venue for all matters relating to this Agreement shall be in courts and forums located in the Province of Ontario, and your consent to such jurisdiction and venue is implied.

Ordering Information.

All orders are subject to the QCTool License Agreement.

To order QCTool online, please click the "Buy Now" button on the prices page of the QCTool website.

You will receive your Registration Number by email within one business day after the payment is processed.

Website: www.qctool.ca

License Registration/Deregistration

To Register a QCTool license If using QCTool in evaluation mode, select **Help/Register a QCTool license** from the main menu to enter your serial number. Registration can also be done by selecting the **Register QCTool** option from the QCTool folder on the Windows Start menu. You must have administrative privileges to do this task.

To Deregister a QCTool license
Should you wish to move a registration from one computer to another, simply select **Help/Deregister a QCTool license** from the main menu. Then you may install and register on another computer. This feature is also available by selecting the **Deregister QCTool** option from the QCTool folder on the Windows Start menu.

Open command (File menu)

Use this command to open an existing qct-file in a new spreadsheet window.

Shortcuts

Toolbar: 

Keys:CTRL+O

Close command (File menu)

Use this command to close all windows containing the active file. QCTool suggests that you save changes to your file before you close it. If you close a file without saving, you lose all changes made since the last time you saved it.

Save command (File menu)

Use this command to save the active file to its current name and directory. If you want to change the name and directory or file format of an existing file before you save it, choose the Save As command.

Shortcuts

Toolbar: 

Keys:CTRL+S

Save As command (File menu)

Use this command to save and name the active file. QCTool displays the Save As dialog box so you can name your document.

To save a document with its existing name and directory, use the Save command.

Print command (File menu)

Use this command to print a document. This command presents a Print dialog box, where you may specify the range of pages to be printed, the number of copies, the destination printer, and other printer setup options.

Shortcuts

Toolbar: 

Keys:CTRL+P

Print Preview command (File menu)

Use this command to display the active document as it would appear when printed. When you choose this command, the main window will be replaced with a print preview window in which one or two pages will be displayed in their printed format. The print preview toolbar offers you options to view either one or two pages at a time; move back and forth through the document; zoom in and out of pages; and initiate a print job.

Print Setup command (File menu)

Use this command to select a printer and a printer connection. This command presents a Print Setup dialog box, where you specify the printer and its connection.

Exit command (File menu)

Use this command to end your QCTool session. You can also use the Close command on the application Control menu. QCTool prompts you to save documents with unsaved changes.

Shortcuts

Mouse:Double-click the application's Control menu button.

Keys:ALT+F4

File Save As dialog box

The following options allow you to specify the name and location of the file you're about to save:

File Name

Type a new filename to save a document with a different name. QCTool adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the document.

Directories

Select the directory in which you want to store the document.

Network...

Choose this button to connect to a network location, assigning it a new drive letter.

Save As Type

You can choose different file type from the list:

- QCT-file;
- XYZ ASCII file;
- Microsoft Excel;
- Geosoft GBN file.

See Also

Saving Files

Print dialog box

The following options allow you to specify how the document should be printed:

Printer

This is the active printer and printer connection. Choose the Setup option to change the printer and printer connection.

Setup

Displays a Print Setup dialog box, so you can select a printer and printer connection.

Print Range

Specify the pages you want to print:

All	Prints the entire document.
Selection	Prints the currently selected text.
Pages	Prints the range of pages you specify in the From and To boxes.

Copies

Specify the number of copies you want to print for the above page range.

Collate Copies

Prints copies in page number order, instead of separated multiple copies of each page.

Print Quality

Select the quality of the printing. Generally, lower quality printing takes less time to produce.

Print Preview toolbar

The print preview toolbar offers you the following options:

Print

Bring up the print dialog box, to start a print job.

Next Page

Preview the next printed page.

Prev Page

Preview the previous printed page.

One Page / Two Page

Preview one or two printed pages at a time.

Zoom In

Take a closer look at the printed page.

Zoom Out

Take a larger look at the printed page.

Close

Return from print preview to the editing window.

Print Setup dialog box

The following options allow you to select the destination printer and its connection.

Printer

Select the printer you want to use. Choose the Default Printer; or choose the Specific Printer option and select one of the current installed printers shown in the box. You install printers and configure ports using the Windows Control Panel.

Orientation

Choose Portrait or Landscape.

Paper Size

Select the size of paper that the document is to be printed on.

Paper Source

Some printers offer multiple trays for different paper sources. Specify the tray here.

Options

Displays a dialog box where you can make additional choices about printing, specific to the type of printer you have selected.

Network...

Choose this button to connect to a network location, assigning it a new drive letter.