triwaco

groundwater modelling software

9 Visualisation and Post-Processing: TRIPLOT

ROYAL HASKONING

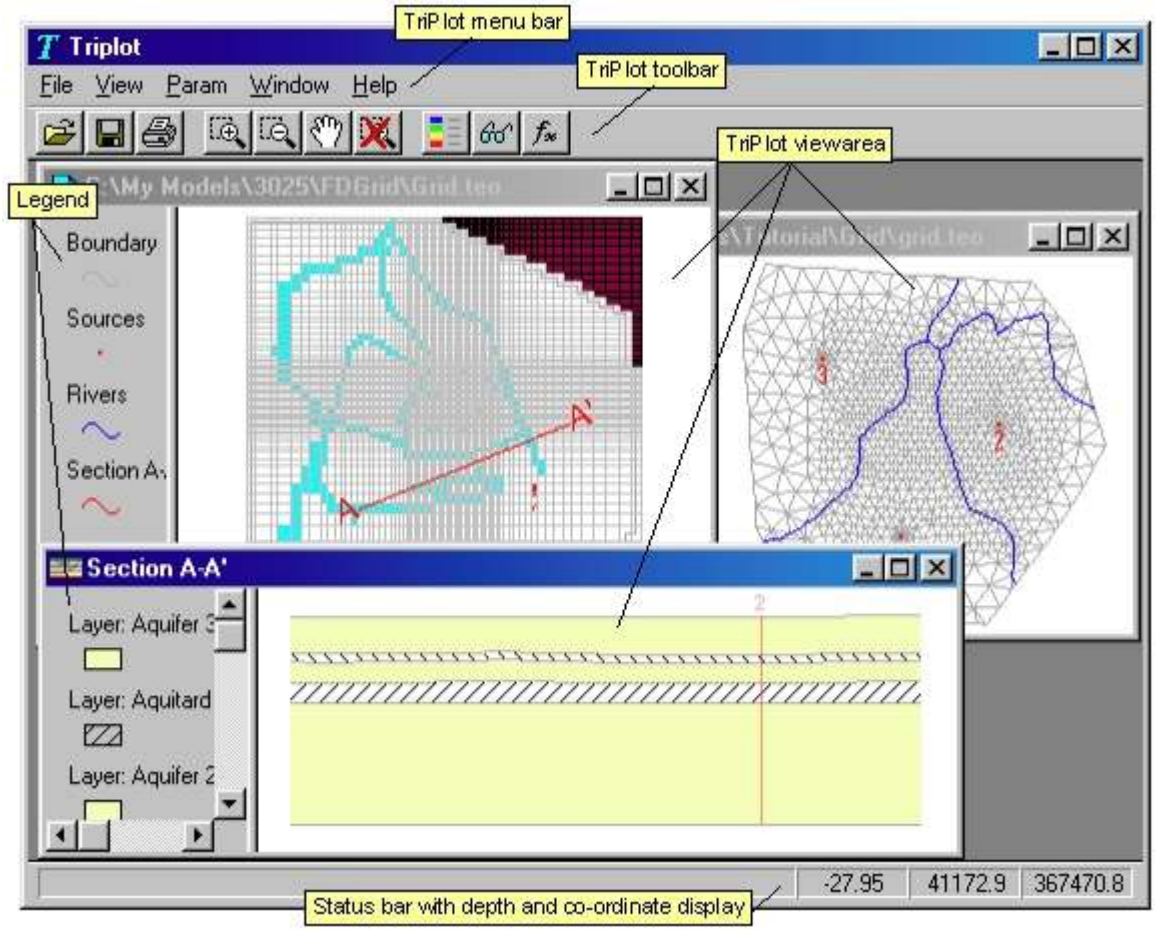
Chapter 9: Visualisation and post-processing: TriPlot

9.1 Introduction

Triplot is a Windows-based graphical viewing tool, included in the **Triwaco** package. **Triplot** can be called from within the **TriShell**, but can also be used stand-alone.

Triplot enables the user to view, change, save and export in- and output of **Triwaco** groundwater models both in plane view as in user defined cross-sections. The program displays a view with the layout of the model grid. In- and output parameters can be loaded from the **TriShell** or **TriPlot** and parameter values may be classified, contoured or used in an expression. Furthermore **Triplot** may display calculated groundwater path lines, read from the binary output file of the ***particle-tracking*** program **Trace**. **TriPlot** also provides the possibility to interactively start tracing of particle trajectories once a **Trace** configuration file has been defined and loaded. How to carry out particle tracking in **TriPlot** is explained in chapter 11.

Geographical oriented information not allocated to the grid may be displayed as a ***background map***. Various formats are supported. The resulting view can be ***printed*** and the various quantities, including classes and contours of the parameters loaded, may be ***exported*** and saved on disk.



9.2 Concepts and terms

The graphical presentation program **TriPlot** can be started from the various **Triwaco** data sets by highlighting a parameter and subsequently choosing 'View' 'Adore' from the Parameter pull-down menu or by selecting 'View Ado file' from the pop-up menu activated by the right-hand mouse button. The program will start up opening a view or window, displaying the model's boundary, rivers and sources read from the grid file. If no grid file exists or if the grid file does not correspond with the parameter file selected the program displays a warning.

Starting **TriPlot** as a stand-alone program, the user should specify an existing grid file to be opened and displayed.

A ***view*** consists of a number of ***layers***, which contain information on the layout of the Finite Element or Finite Difference grid and on parameter values. The grid properties are available in a grid file (***grid.teo***). Parameter values should be allocated to the grid and are stored in one or more ***ado***-files. An ado-file may contain one or more adore sets: arrays of grid related data that result from parameter allocation. After loading a parameter in **TriPlot**, its value may be displayed, classified or contoured.

Pathlines may be added to the view and are read from the unformatted output file of the particle-tracking program Trace.

Additional information that is not related to the grid may be added to the view by means of a ***background map***. By now the following ***background file types*** are supported by **Triplot**:

AutoCAD DXF files	*.dxf
ARC/INFO ungenerate files	*.ung
ArcView Shapefiles	*.shp
Dawaco maps	*.map; *.top
Windows Bitmap files	*.bmp
TIFF files	*.tif
GIF files	*.gif
PCX files	*.pcx
JPEG files	*.jpg
Comma delimited files	*.csv

It is possible to open several background maps at the same time. Each background map will be represented by a separate layer in the view. How to open a background map.

Layers, items or properties of a view may be visible or hidden. Hidden properties are loaded but will not be displayed in the view.

A large number of items may be available during a **TriPlot** session, depending whether or not the appropriate files have been loaded. Grid parameters are read from the **Triwaco** grid file (***grid.teo***). Input and output parameters are read from the corresponding adore sets. The following table gives an overview of items that may be available during a **Triplot** session.

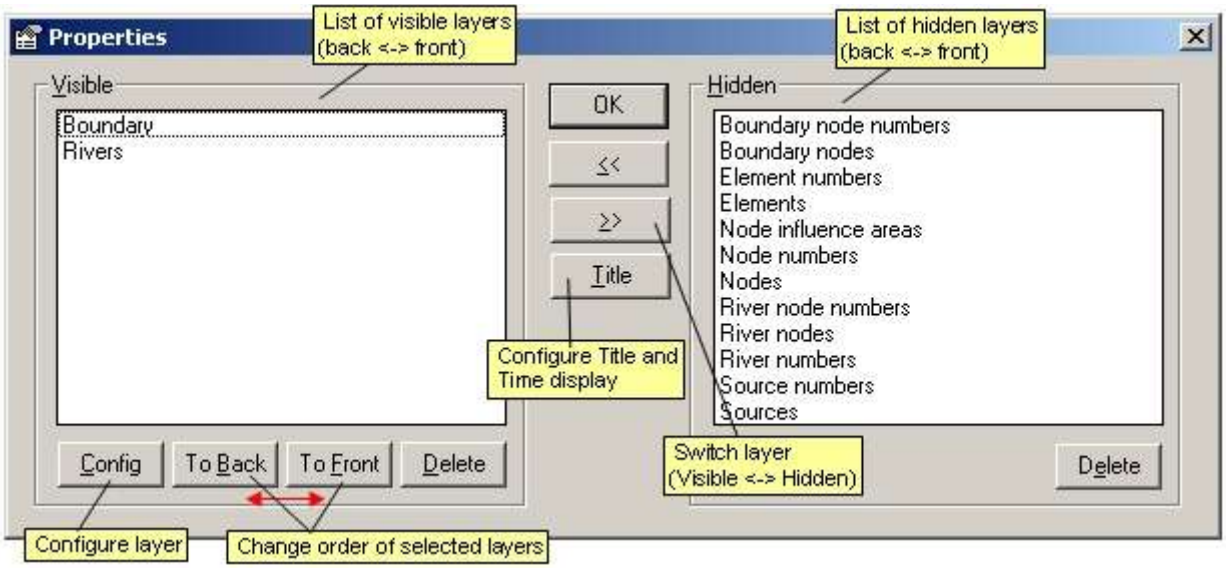
Here *Bpar* stands for any Boundary parameter, *Rpar* stands for any River parameter, *Spar* stands for any Source parameter and *Dpar* for any distributed parameter, that is a parameter that should have a value at each of the nodes of the Finite Element or Finite Difference Grid.

Item or property	**File containing parameter**
Grid parameters	
Boundary	Grid output file: Grid.teo
Boundary nodes	Grid output file: Grid.teo
Boundary node numbers	Grid output file: Grid.teo
Elements	Grid output file: Grid.teo
Element numbers	Grid output file: Grid.teo
Nodes	Grid output file: Grid.teo
Node numbers	Grid output file: Grid.teo
Rivers	Grid output file: Grid.teo
River nodes	Grid output file: Grid.teo
River node numbers	Grid output file: Grid.teo
Sources	Grid output file: Grid.teo
Source numbers	Grid output file: Grid.teo
Node influence areas	Grid output file: Grid.teo

Item or property	File containing parameter
Cells	ModFlow compatible grid output file: Grid.teo
Input and output parameters	
Boundary classes: *Bpar*	Boundary parameter file *Bpar*.ado
Boundary node values: *Bpar*	Boundary parameter file *Bpar*.ado
River classes: *Rpar*	River parameter file *Rpar*.ado
River node values: *Rpar*	River parameter file *Rpar*.ado
Source labels: *Spar*	Source parameter file *Spar*.ado
Classes: *Dpar*	Distributed parameter file *Dpar*.ado
Contour map: *Dpar*	Distributed parameter file *Dpar*.ado
Node labels: *Dpar*	Distributed parameter file *Dpar*.ado
Miscellaneous	
Background Map: C:\PATH\map.ext	Specified '*Background* ' map file
Streamlines: C:\PATH\trace.bin	Binary TRACE output file
Streamlines: C:\PATH\trace.tro	TRACE result file
Vector field	Triplot vector file: velocity.vec
Section aa-aa'	Triplot section file: aa-aa'.tps
For user specified sections and river sections	
Profile: *Dpar*	Distributed parameter file *Dpar*.ado
Profile: *Rpar*	River parameter file *Rpar*.ado
Streamlines: C:\PATH\trace.bin	Binary TRACE output file

9.3 General settings

The general settings and properties window can be accessed selecting 'Properties' from the 'View' pull-down menu, or right-mouse-button. The properties window contains two areas, listing the **Visible** items and the **Hidden** items:



Selecting either >> or << one can move an item from the list of hidden items to the list of visible items and vice versa.

To change the appearance of an item or layer one may choose Config after having selected the item. A dialog-box or configuration window will appear and the user may change the settings of the item selected. The type of dialog-box depends on the selected item. There are different configuration windows for lines, points, labels, contours, classes, background files and streamlines. After having modified the settings the user can save these to be added to the user's preferences or simply confirm them for the current **TriPlot** session.

9.4 Menu bar

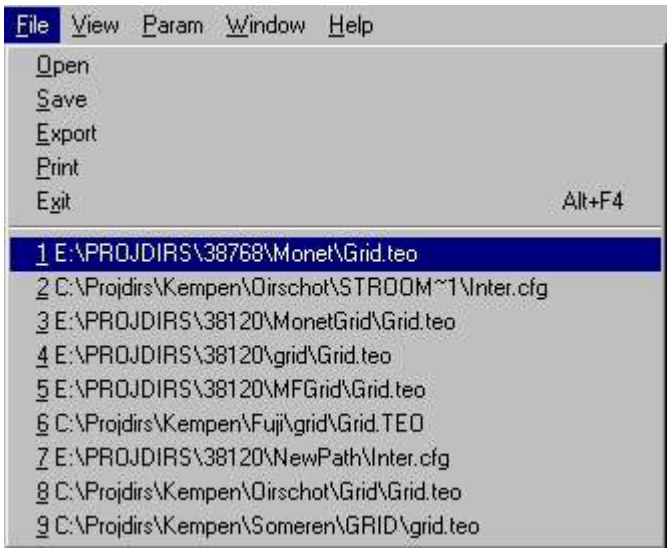
The menu bar displays the following five commands: File, View, Param, Window and Help (See graphical screen). Selecting one of these commands activates a pull-down menu with a number of sub-commands. Some of the sub-commands may be activated by a shortcut-key (generally the **Ctrl**-key and another character). Note that the Trace command, for pathline calculations, is described in chapter 11.

File

The File Commands pull-down menu is being activated selecting **'File'** from the menu bar.

The pull-down menu has two sections. The lower section lists recently opened **TriPlot** files. Selecting one of these files will open the corresponding Triplot session.

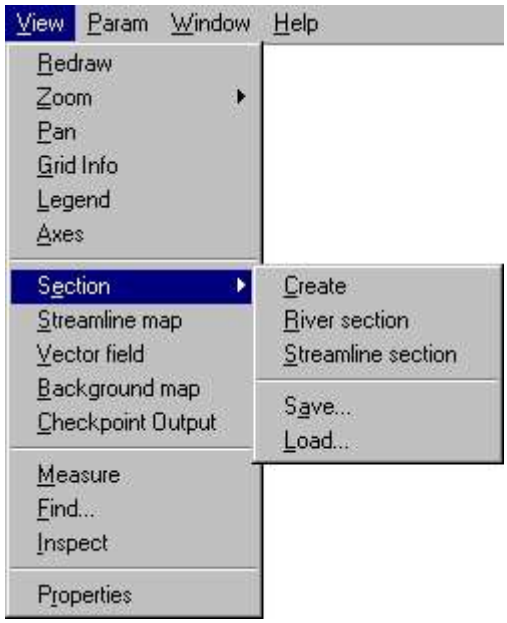
The upper section offers the sub-commands the user can choose from:



Open	Open the file selected and display the information. The following formats are supported: - a valid **Tesnet** grid-file (*.teo, *.grd file) - a previously saved **Triplot** view (*.tpp file) - a **Trace** configuration file (*.cfg file)
Save	Save the active view on disk for later use or for printing. The following formats are supported: - Triplot format (*.tpp file) - Bitmap format (*.bmp file)
Export	Export a layer (properties) of the active view. The following formats are supported: - ArcView Shapefile format (*.shp file) - ARC/Info ungenerated format (*.ung file) - AutoCAD DXF format (*.dxf files)
Print	Send the active view to the printing or plotting device. The program opens a print properties dialog box to select: - The printing device - The area to be printed - The scale of the map to be printed - Whether or not to print a legend and scale bar
Exit	Close all views and exit **TriPlot** Shortcut-key for this command: **Alt-F4**.

View

The View Commands pull-down menu is being activated selecting 'View' from the menu bar. The menu has four sections.



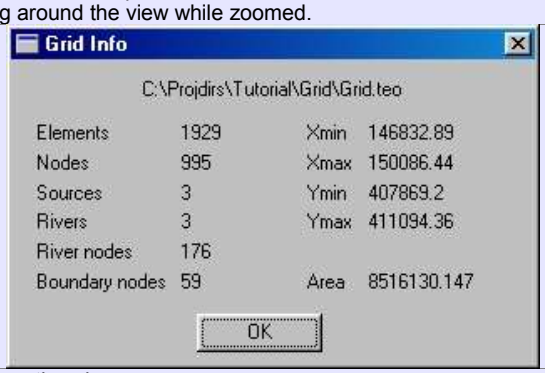
The first section contains commands that enable adjustment of the size and the area of the current map on the screen (zooming and panning) and that display general information on the grid or enable the display of a legend and axes.

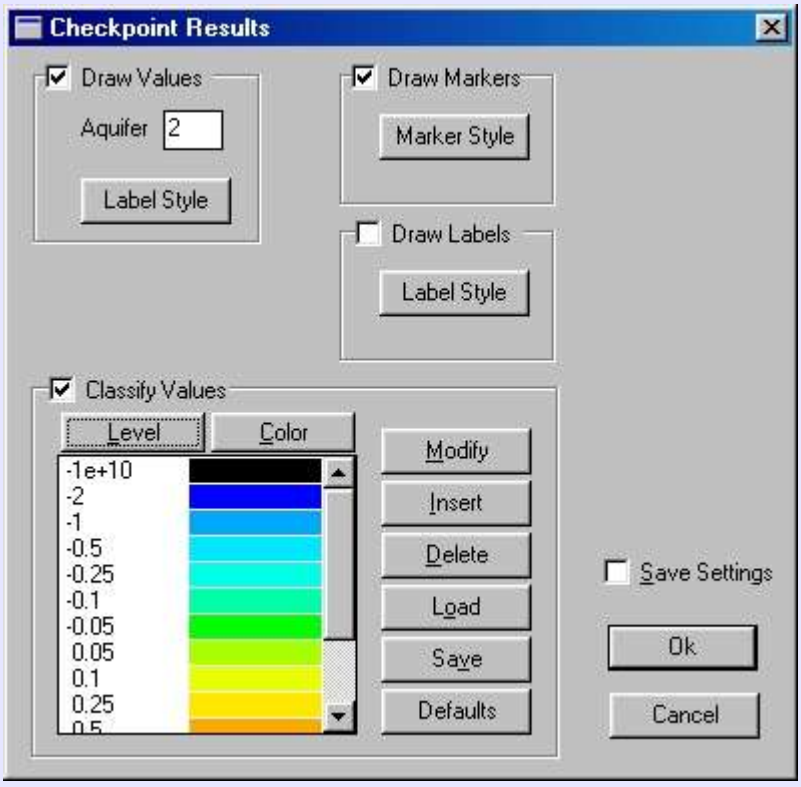
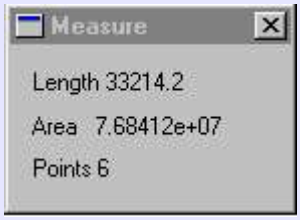
The second section enables the display of cross-sections, streamline maps, velocity vectors, background maps and calibration results.

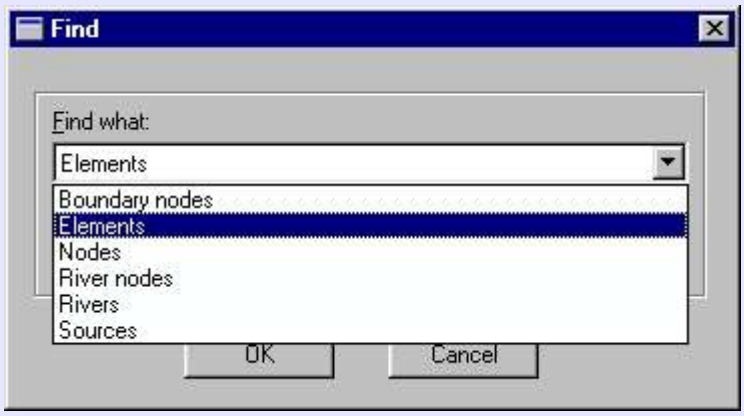
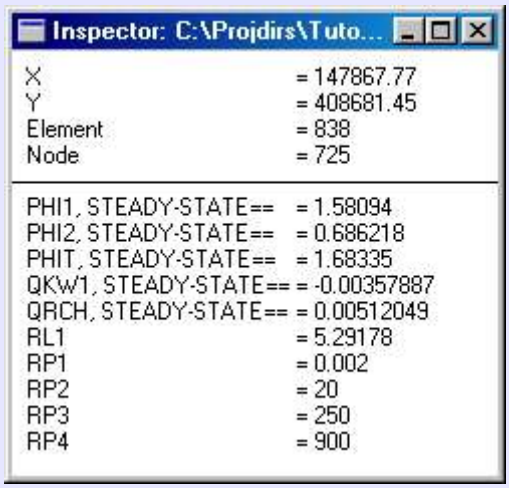
The third section allows the user to measure distances and areas, to find the location of certain items and to inspect parameter values.

The last section allows the user to access the properties window and to change the configuration settings.

The View commands can be issued via this pull down menu or via the Toolbar. The pull-down menu offers the following sub-commands the user can choose from:

Redraw		Refreshes the screen. Generally the program automatically refreshes the screen after issuing a command.
Zoom		
	In	Enlarge the scale of the view by drawing a box. Only part of the model area will be visible in the view.
	Out	Reduce the scale of the view by drawing a box. A part or the entire model area will be visible in the view.
	Reset	Fit the extent of the active view in the current window. The total model area will be visible in the view.
Pan		Point and drag (a point of) the view to a new position. This command facilitates moving around the view while zoomed.
Grid Info		Display information on the Finite Element grid of the active view. - Number of elements, nodes, rivers etc. - Minimum and maximum coordinates and total model area 
Legend		Display or hide the legend of the active view.
Axes		Allows the user to select the axes to be displayed, choosing from Left, Top, Right or Bottom. Default no axes are being displayed. 

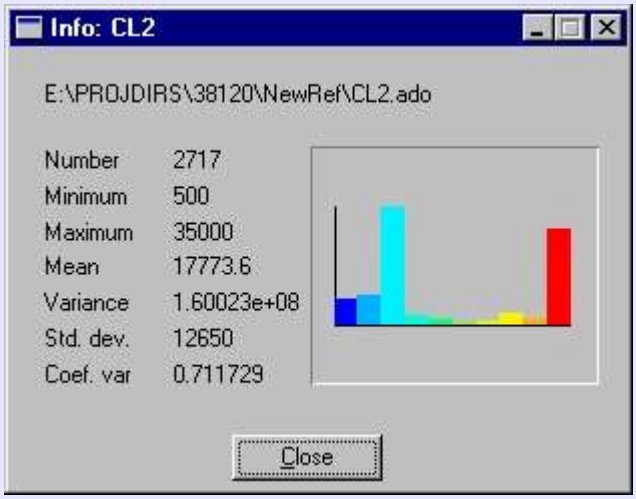
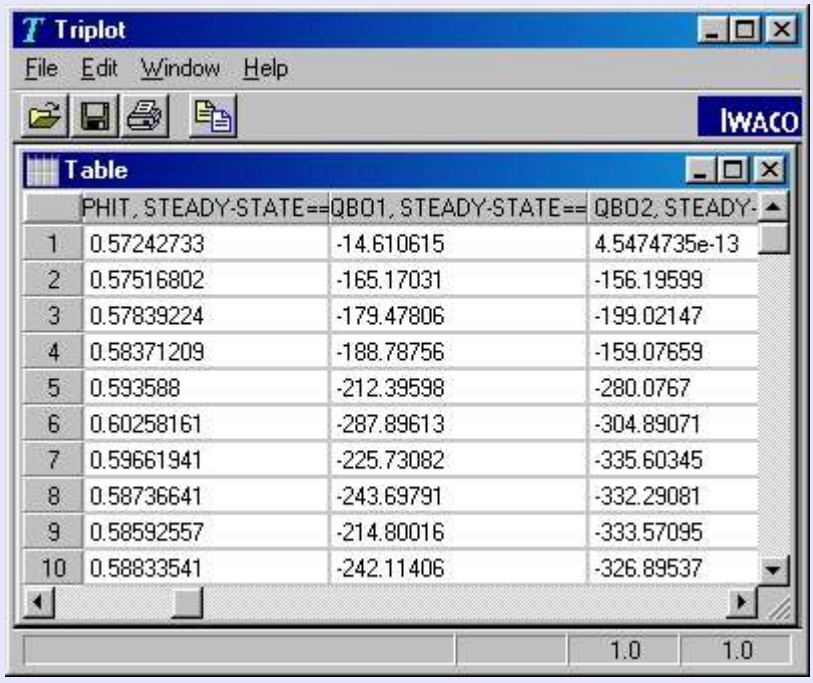
Section	
Create	Generate and display a cross-section along a user-defined extent. The extent of the section is defined by positioning the pointer and pressing the left-hand mouse button several times. The section will be closed pressing the right-hand mouse button.
River Section	Generate and display a cross-section along a river-stretch. The program prompts for the river ID for which a section will be generated.
Streamline section	Generate and display a cross-section along a streamline. The program prompts for the path line ID for which a section will be generated.
Save	Save a previously generated cross-section to a *.tps file
Load	Load and display a saved cross-section in a new window.
Streamline map	Display streamlines from a binary TRACE output file.
Vector field	Display velocity vectors from file.
Background map	Display background map from file. Several background file types are supported (such as: DXF, BMP, TIFF, GIF, PCX and JPEG files, ArcView Shapefiles, Arc/Info ungenerate files and Dawaco maps).
Checkpoint Output	Display calibration results from a checkpoint output file (*.cho). A pop-up window will appear in which to specify the aquifer for which results have to be displayed. 
Measure	Display measurement tool and pop-up window. Pointing and pressing the left-hand mouse button at various locations causes the program to compute and display the length of the line and the area enclosed within a polygon with the line as perimeter. Pressing the right-hand mouse button ends this action. 

Find …	Positions the pointer at the location of the selected item. Select item and number: e.g. Nodes, 567. One can choose from the following list of items: 
Inspect	Starts the inspection tool and pop-up window. Pointing and pressing the left-hand mouse button at various locations causes the program to compute and display coordinates, element and node numbers and values of all loaded parameters for the selected location. 
Properties	Display the properties or general settings window.

Param

The Param Commands pull-down menu is being activated selecting 'Param' from the menu bar.
The pull-down menu comprises commands that enable various ways of displaying (distributed) parameters, to retrieve general or detailed information and to combine loaded parameters to create new ones.
The pull-down menu offers the following sub-commands to choose from:

Load	Import parameter values from file
Expression	Combine loaded parameters to generate a new parameter, using simple mathematical expressions. The function is comparable to the expression allocator from the **TriShell**.
Save	Save a parameter from the active view. Usually only meaningful for new parameters generated with the **'Expression'** command.
Contour	Calculate and display contours for distributed parameters.

Classify	Assign parameter values to classes and display the classes of distributed parameters, river parameters or boundary parameters.
Polygonize	Joins adjacent node influence areas, for nodes having identical values, to larger polygons. The resulting polygons are displayed.
Shade	Generate and display a relief map of a distributed parameter.
Info	Display general information on the parameter loaded. Statistical information and a histogram of parameter values are displayed in a pop-up window. 
Table	Display a table of all loaded parameters of the active view. This table may be copied to a spreadsheet to be processed. 

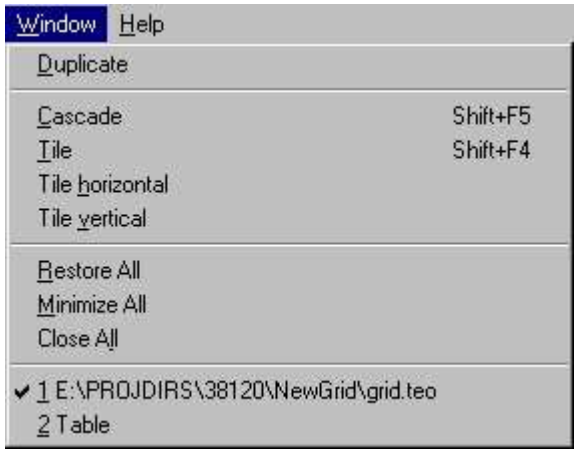
Time

The Time Commands pull-down menu is being activated selecting 'Time' from the menu bar. The pull-down menu offers the following sub-commands to choose from:

Starting date	Converts timesteps into dates. Date can be displayed in the title.
Animate	Generate and display an animation of contoured or classified transient parameters.
Time series	Display time series in a graph of multiple output parameters (*.ado, *.flo, *.fzo).
Observations	Display time series in a graph of multiple observed time series (*.csv).

Window

The Window Commands pull-down menu is being activated selecting **'Window'** from the menu bar.
The pull-down menu comprises commands that enable various ways of arranging the open windows. For some of the commands a Shortcut-key is available.
The lower part of the pull-down menu displays all windows available. The active window is marked with an .
The pull-down menu offers the following sub-commands to choose from:



Duplicate	Create a copy of the active view. The copy will have the same title, supplemented with a sequence number.
Cascade	Display all active windows (or views), partly overlying each other. Shortcut-key for this command: **Shift-F5**.
Tile	Display all active windows (or views) completely filling the Triplot window area, each view aside from the other. Shortcut-key for this command: **Shift-F4**.
Tile horizontal	Tile all active windows (or views), each view covering the width of the Triplot window area.
Tile vertical	Tile all active windows (or views), each view covering the height of the Triplot window area.
Restore All	Display all windows (or views), including those minimized.
Minimize All	Minimize all windows (or views) and display icons only.
Close All	Close all windows (or views) of the current Triplot session.
List of active windows or views in the current Triplot session	

Help

The Help Commands pull-down menu is being activated selecting 'Help' from the menu bar. The pull-down menu offers the following sub-commands to choose from:

Help	Activates the Triplot help file indicated in Triplot.ini. Shortcut-key for this command: **F1**.
About Triplot	A pop-up window, showing the current version of Triplot.
About Royal Haskoning	A pop-up window, showing general information on Royal Haskoning

9.5 The toolbar

The toolbar offers the following possibilities that also may be addressed using commands from the menu bar.

	Open and display an existing grid-file or a saved Triplot-view. Equivalent to the command 'Open' from the 'File' pull-down menu.
	Save the active view on disk for later use. The formats supported are: - Triplot format (*.tpp file) - Bitmap format (*.bmp file) Equivalent to the command 'Save' from the 'File' pull-down menu.
	Send the active view to the printing or plotting device. Equivalent to the command 'Print' from the 'File' pull-down menu.
	Enlarge the scale of the view by drawing a box. Equivalent to the command 'Zoom' 'In' from the 'View' pull-down menu.
	Reduce the scale of the view by drawing a box. Equivalent to the command 'Zoom' 'Out' from the 'View' pull-down menu.
	Point and drag (a point of) the view to a new position. This command facilitates moving around the view while zoomed. Equivalent to the command 'Pan' from the 'View' pull-down menu.
	Fit the extent of the active view in the current window. Equivalent to the command 'Zoom' 'Reset' from the 'View' pull-down menu.
	Toggle between showing or hiding the legend of the active view. Equivalent to the command 'Legend' from the 'View' pull-down menu.
	Generate and display a cross-section along a user-defined extent. The extent of the section is defined by positioning the pointer and pressing the left-hand mouse button several times. The section will be closed pressing the right-hand mouse button. Equivalent to the command 'Section' 'Create' from the 'View' pull-down menu.
	Display measurement tool and pop-up window. Pointing and pressing the left-hand mouse button at various locations causes the program to compute and display the length of the line and the area enclosed within the polygon with the line as perimeter. Pressing the right-hand mouse button ends this action. Equivalent to the command 'Measure' from the 'View' pull-down menu.
	Starts the inspection tool and pop-up window. Pointing and pressing the left-hand mouse button at various locations causes the program to display coordinates, element and node numbers and values of all loaded parameters for the selected location. Equivalent to the command 'Inspect' from the 'View' pull-down menu.
	Combine loaded parameters to generate a new parameter, using simple mathematical expressions. The function is comparable to the expression allocator from the **TriShell** Equivalent to the command 'Expression' from the 'Param' pull-down menu.
	Generate and display an animation of contoured or classified transient parameters. Equivalent to the command 'Time' from the 'Animate' pull-down menu.

9.6 How to ...

9.6.1 How to open a view

TriPlot will start and open a view or window, displaying the model's boundary, rivers and sources read from the grid file whenever it is started from the various **Triwaco** data sets. **TriPlot** is started by highlighting a parameter and subsequently choosing 'View' 'Adore' from the 'Parameter' pull-down menu or by selecting 'View Ado file' from the pop-up menu activated by the right-hand mouse button.

Alternatively **TriPlot** may be used as a stand-alone program by selecting the Triplot-icon from the **'Program Files\Triwaco'**-folder. Starting Triplot as a stand-alone program, the user should open an existing grid file first by selecting 'Open' from the 'File' pull-down menu or  from the toolbar. The user is prompted for a file name, either a Triwaco grid file or a saved Triplot-view.

9.6.2 How to open background maps

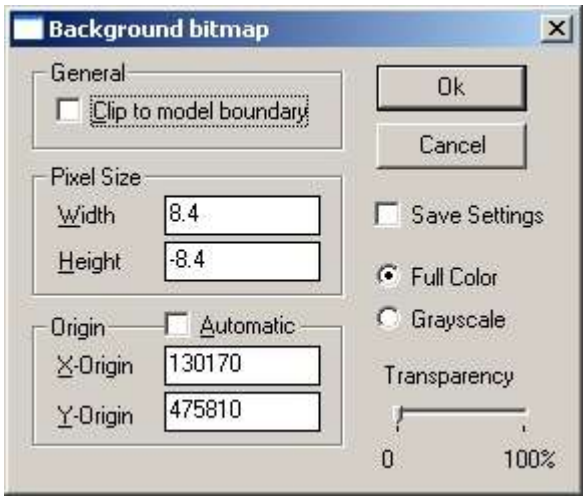
A background map, to be displayed together with grid and parameter information, may be added to the view selecting 'Background map' from the 'View' pull-down menu. The user is prompted for a file name; consistent with one of the supported background map file types.

AutoCAD DXF files	*.dxf
ARC/INFO ungenerate files	*.ung
ArcView Shapefiles	*.shp
Dawaco maps	*.map; *.top
Windows Bitmap files	*.bmp
TIFF files	*.tif
GIF files	*.gif
PCX files	*.pcx
JPEG files	*.jpg

While opening a bitmap as a background, **TriPlot** will display a dialog box, which can be used to scale the bitmap to the edited map. The size of the pixels can be changed as well as the origin of the bitmap (the upper left corner). Changing the width or the height independently from one another may result in a distorted bitmap. By pressing the Save button,**TriPlot** will save a world-file (bitmap.tfw) with the actual values for pixel size, origin and rotation shown in the dialog box. The next time **TriPlot** opens the bitmap (for example bitmap.bmp) the values stored in the associated world file will be given as default values. If no world file is present the width and height of the pixels are computed from the width and height of the area displayed in the bitmap (in user units) and the number of pixels in horizontal and vertical direction.

The world-file may also be edited or created using a text-editor. It is important that the worldfile (*.tfw) has the same name as the bitmap file. Below an example of a dialog box and next to it the same data shown in the tfw-file.

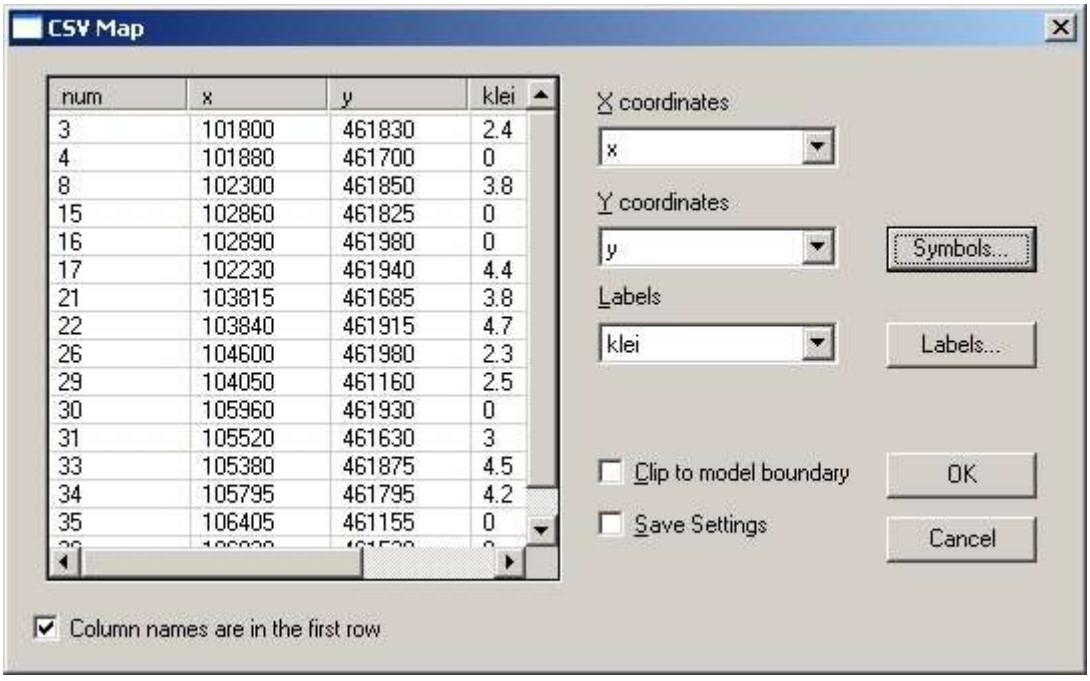
The transparency of the background map can be changed in the dialog box.




```
1.587
0
0
-1.587
146800
411200
```

9.6.3 How to display grid independent point values

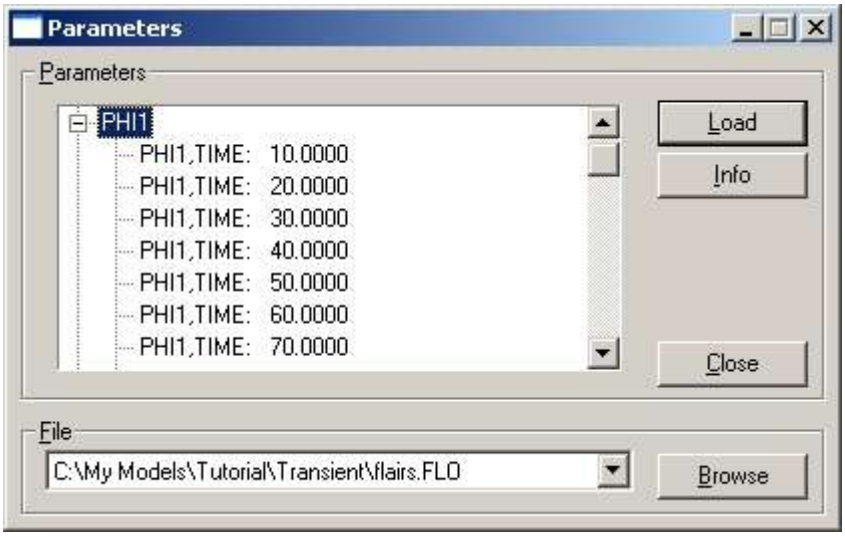
Grid independent point values, to be displayed together with grid and parameter information, may be added to the view selecting 'Background map' from the 'View' pull-down menu. The user is prompted for a file name consistent with .csv file type. The following dialog box will appear.



9.6.4 How to add a parameter (*.ado)

To add a parameter to the view one should select 'Load' from the 'Param' pull-down menu. The user is prompted for the name of a file containing parameter values in Adore format. Once this file is accepted a list of (valid) parameters will be displayed. Selecting one of these parameters will add the parameter to the view. More than one parameter may be selected.

Adding transient parameters works slightly different. An example is shown below. To add an entire timeseries of a parameter select the parameter without extension. Seperate timesteps can be selected as well (select parameter with extension TIME: .....).

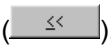


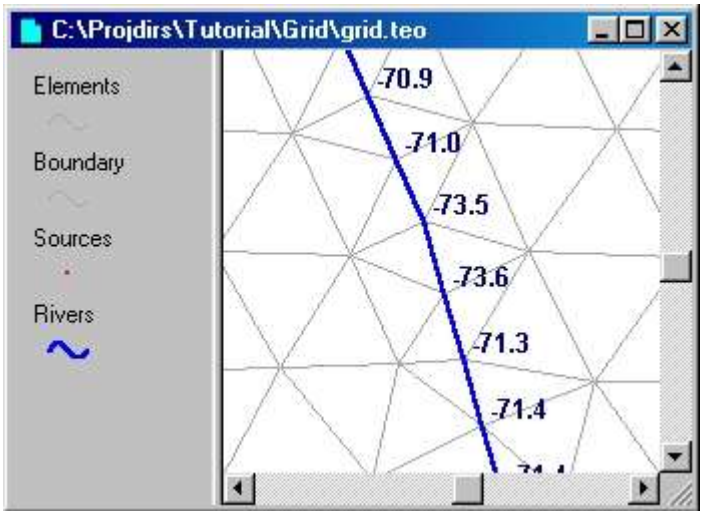
9.6.5 How to add a grid information as a parameter (*.teo)

Grid information, for instance node influence area, can be loaded as an paramter as well since it has the standard adore file format. To add a grid parameter to the view one should select 'Load' from the 'Param' pull-down menu. The user is prompted for the name of a file containing parameter values in Adore format. No select All files (*.*) and select grid.teo froma grid data set. Once this file is accepted a list of (valid) parameters will be displayed. Selecting one of these parameters will add the parameter to the view. More than one paramter may be selected.

***Note** that, depending on the general settings, the parameter may remain invisible!*

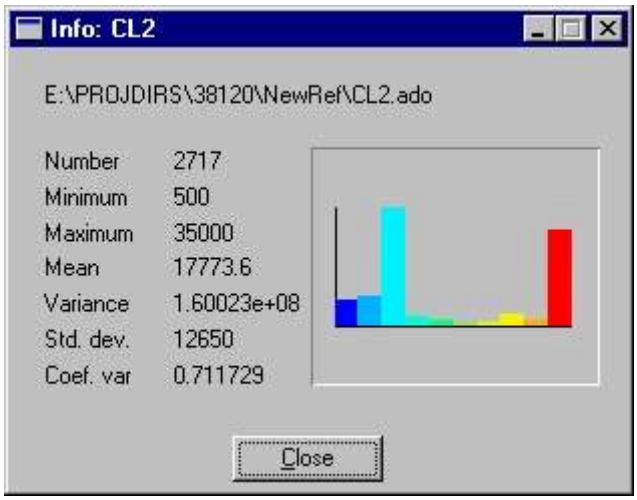
9.6.6 How to display parameter values

To display parameter values, plotted as labels, select 'Properties' from the 'View' pull-down menu or press the **right-hand mouse button**, while pointing somewhere in the active view. The General settings or properties pop-up window will appear. Parameter values may be displayed moving the items 'Node labels: PARNAME', 'Boundary node values: BNDPAR', 'River node values: RIVPAR' or 'Source labels: SRCPAR' from the **Hidden** parameter list to the **Visible** parameter list ().



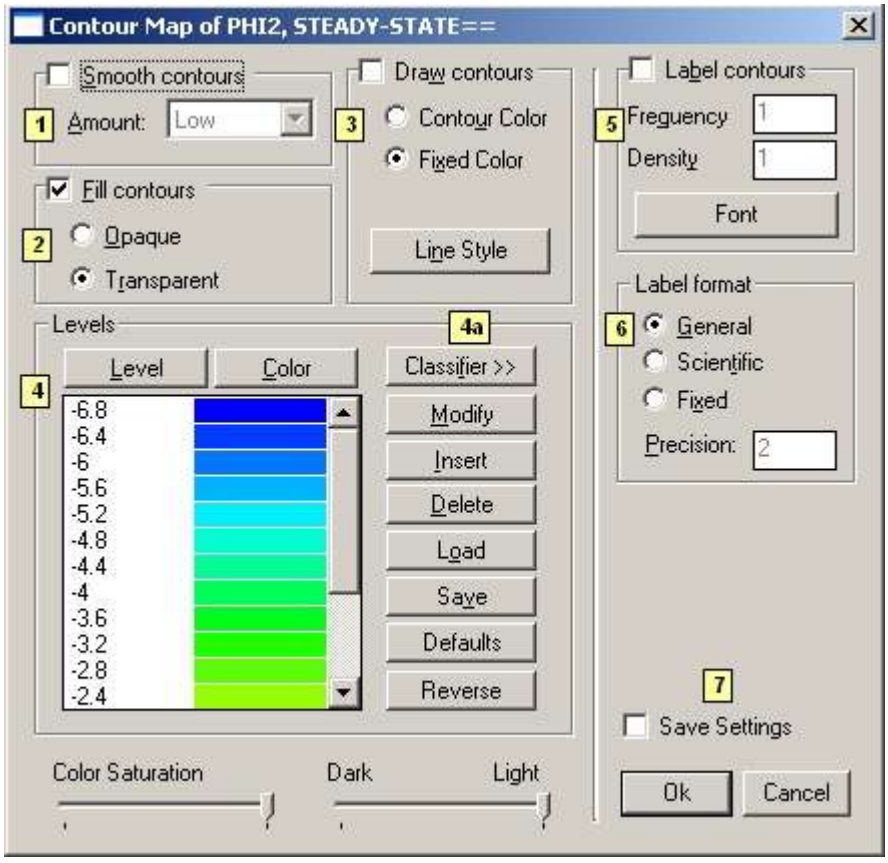
9.6.7 How to retrieve parameter info

Statistical information and a histogram of parameter values are displayed in a pop-up window selecting 'Info' from the 'Param' pull-down menu. The user is prompted to select the parameter he wants information about. The pop-up window displays the parameter name, the file name, the number of values in the Adore set, the minimum and maximum values encountered and some more statistical information. Moreover, a histogram of the parameter's value is displayed. The histogram contains ten classes, distributed evenly between the minimum and maximum values encountered.

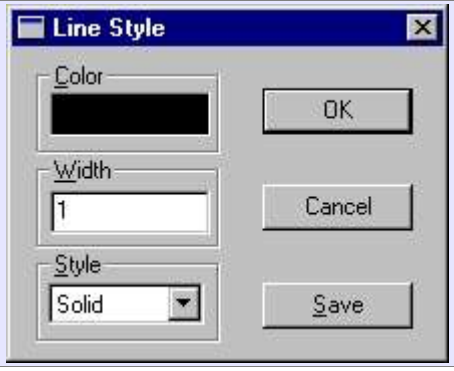


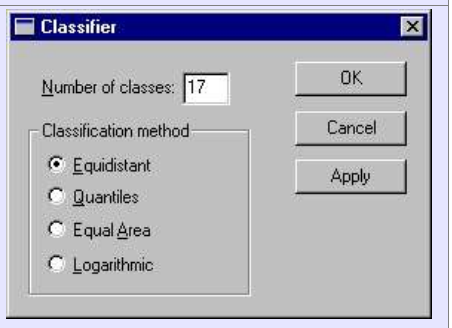
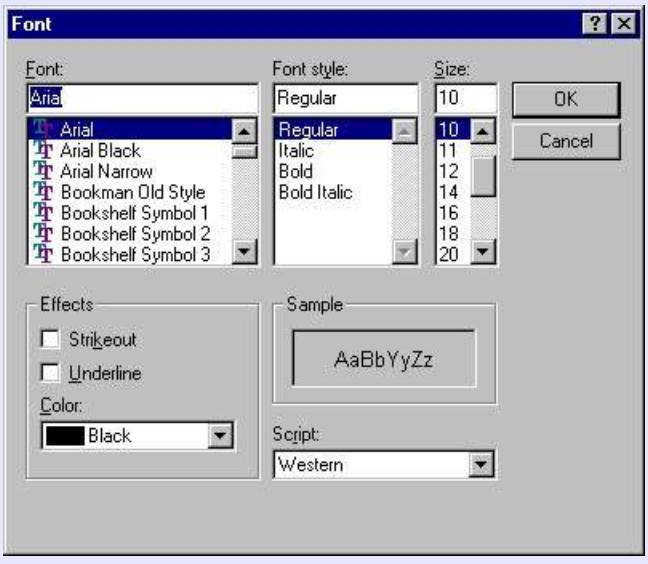
9.6.8 How to contour parameter values

To display contours of parameter values select 'Contour' from the 'Param' pull-down menu. The user will be prompted to select one of the parameters loaded. The program displays the contour configuration window to define the levels to be contoured and the appearance of the contours.



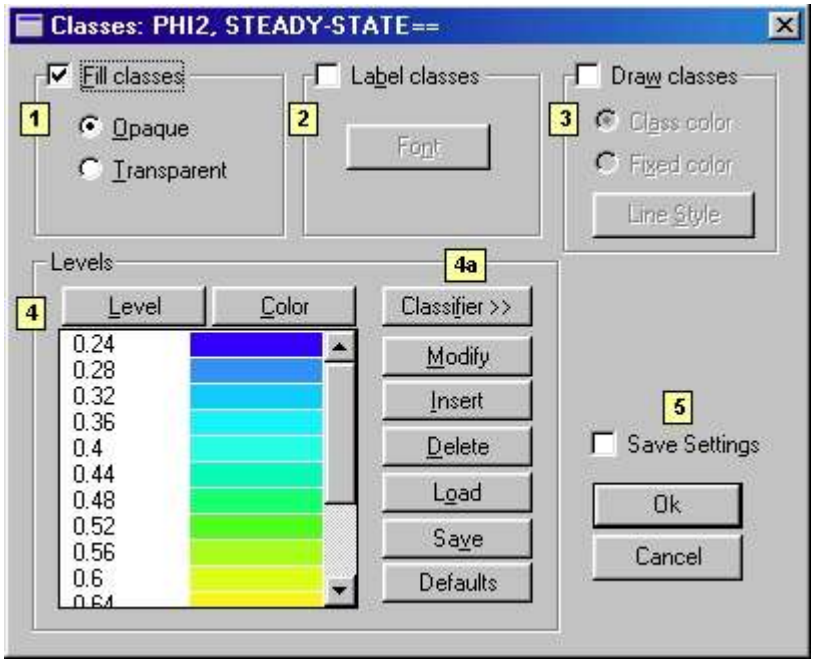
Within the configuration window the following areas may be distinguished to define various aspects of the contour plot:

1. Smooth contours	Option for smoothing contours. Three degrees of smoothing may be selected: Low, Medium or High. If no smoothing is required clear the tick-box. (☐)	
2. Fill contours	Select whether or not to apply a color fill. Two types of color fill are supported: Opaque and Transparent. If no color fill is required clear the tick-box. (☐)	
3. Draw contours	Select whether or not to accentuate the computed contours with a drawn line. Selecting [Line Style] displays a pop-up window for the definition of the color, width and style of the line the contours are drawn with.	

4. Levels		Define values and colors of the levels to be contoured. The left part of the area displays the levels and colors selected. The right part (**4a**) displays a number of commands for manipulating the selected levels and colors.
	Level	Displays a pop-up window for the definition of the start and stop value and the increment of the parameter to be contoured.
	Color	Displays a pop-up window for selection of the color gradient to be used filling the area between contours.
4a. Levels		The commands in the right-hand side of the area are:
	Classifier	Defines the type of classifier to be used for automatic generation of contours or classes. Four types of classifiers are supported: Equidistant, select Quantiles, select Equal Area and Logarithmic. 
	Modify	Changes the properties (Level and/or Color) of the contour selected in the list at the left-hand side of this area.
	Insert	Adds a contour to the list of contours, between the selected one and the one above, having an average value and intermediate color.
	Delete	Deletes the selected contour from the list at the left-hand side of the Levels-area.
	Load	Loads an earlier saved level-configuration file (*.lvl) with level and color definition.
	Save	Saves the actual level-configuration (values and colors of selected levels) in a file for future use.
	Defaults	Restores the default settings for the contour map. Default settings are the use of the 'rainbow color gradient' and a restricted number of levels (equidistant, the number depending on the difference between maximum and minimum value encountered).
	Reverse	Reverse coulour scheme
5. Label contours		Option for adding labels to the contours. Both the number of contours to be labeled (Label every 1, 2,…) and the number of labels for each contour (Density) may be defined.
	Font	Displays a pop-up window for the definition of the appearance of the labels. 
6. Label format		Defines the way labels are displayed on screen.
7. Save settings		Option for saving contour settings for future use. The settings are added to the user's preferences in the General settings file.

9.6.9 How to classify parameter values

To display parameter classes select 'Classify' from the 'Param' pull-down menu. The user will be prompted to select one of the parameters loaded. The program displays the following configuration window to define the levels for classification and the appearance of the classes.

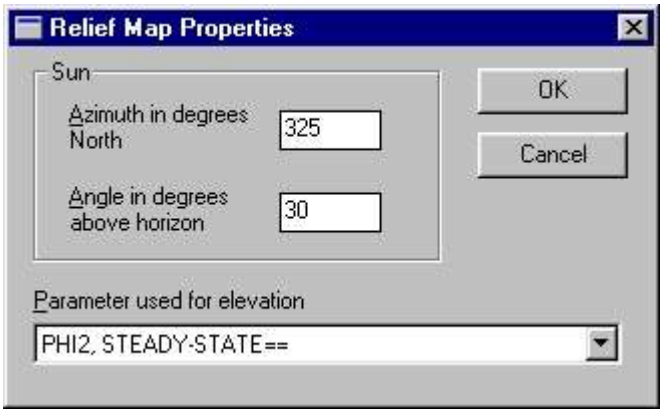


Within the configuration window the following areas may be distinguished to define various aspects of the classified parameter plot:

1. Fill classes	Select whether or not to apply a color fill. Two types of color fill are supported: Opaque and Transparent. If no color fill is required clear the tick-box. (☐)
2. Label classes	Option for adding labels to the classes. Whenever the tick-box is checked all classes will labeled.
Font	Displays a pop-up window for the definition of the appearance of the labels; similar to the one used for the contour labels.
3. Draw classes	Select whether or not to accentuate the computed class-boundaries with a drawn line.
Line Style	Displays a pop-up window for the definition of the color, width and style of the line the class-boundaries are drawn with.
4. Levels	Define class-boundaries (levels) and colors of the parameter to be classified. The left part of the area displays the levels and colors selected. The right part (**4a**) displays a number of commands for manipulating the selected levels and colors, similar to those used for contouring of parameter values.
Level	Displays a pop-up window for the definition of the start and stop value and the increment of the parameter to be classified.
Color	Displays a pop-up window for selection of the color gradient to be used filling the classes.
5. Save settings	Option for saving classification settings for future use. The settings are added to the user's preferences in the General settings file.

9.6.10 How to create a relief map

To display a relief map of parameter values select 'Shade' from the 'Param' pull-down menu. The user will be prompted to select one of the parameters loaded. The program displays the following window to define the values for Azimuth and Angle that define the appearance or intensity of the shading.

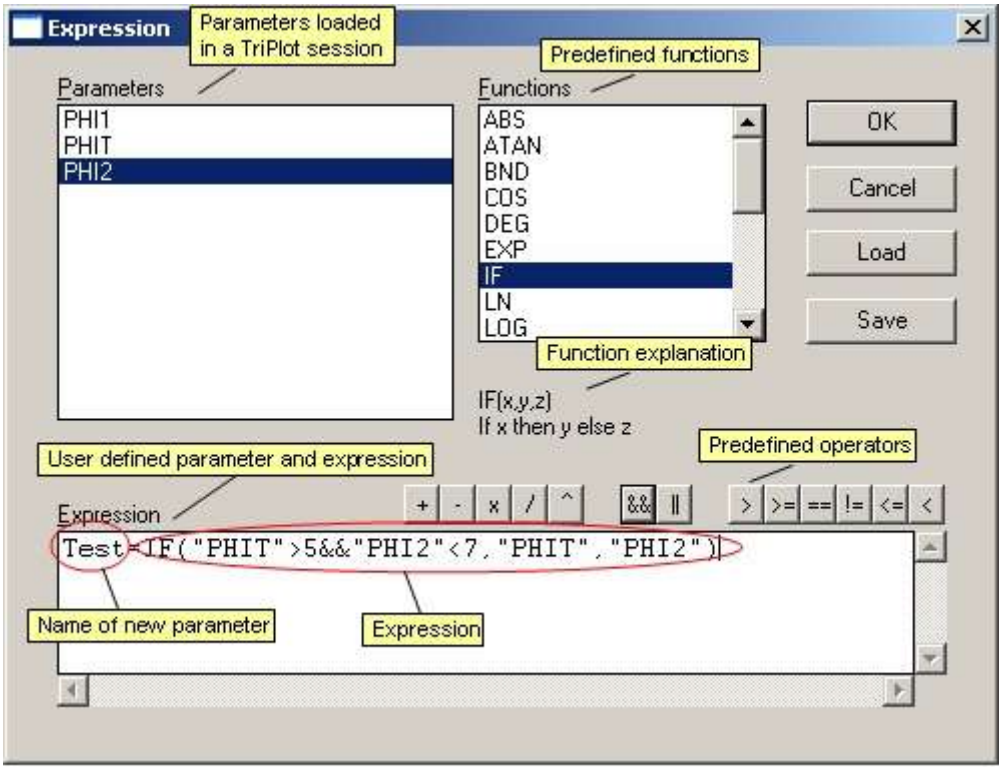


9.6.11 How to create new parameters (expression)

After having loaded one or more parameters the user may create a new parameter by combining these parameters using the set of predefined mathematical expressions. To do so one has to select 'Expression' from the 'Param' pull-down menu or  from the toolbar. The program displays the following window to define the new parameter and the expression to be used.

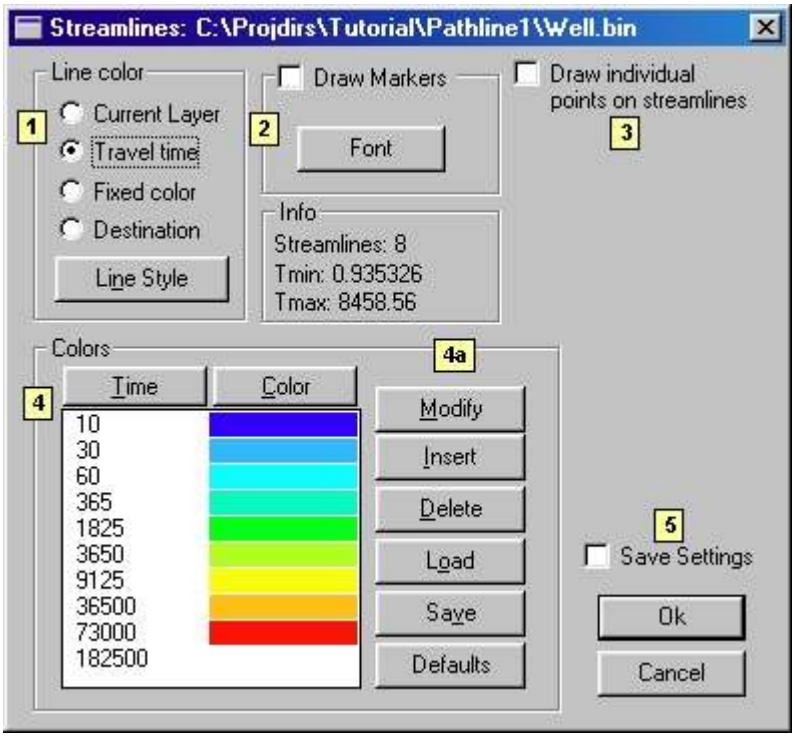
The left part of the window lists all parameters that are loaded in the active Triplot view. To the right the user finds a list of predefined mathematical functions, a short description of the highlighted function and a list of operators that can be used (presented by buttons in a tool-bar).

The lower part of the window allows the user to define a new parameter by entering a mathematical expression. Double clicking on one of the parameters, a function or an operator adds the selected item to the expression. The values for the parameter will be computed after confirmation of the expression entered.



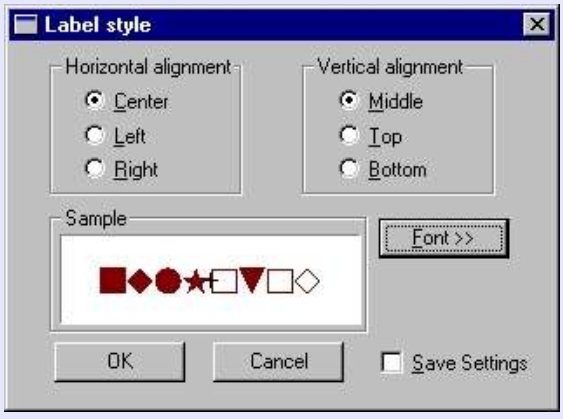
9.6.12 How to display streamlines

To display a streamline map select 'Streamline map' from the 'View' pull-down menu. The user will be prompted to select a Trace unformatted output file (*.bin). Subsequently the user will be prompted to enter a ***Snap distance***. Points belonging to the same streamline and lying within this distance from each other will be omitted from the view.



The way the streamlines appear in the view can be modified through the general settings window that is accessed by the commands 'Properties' from the 'View' pull-down menu.

The central area of the window (**Info**) displays information on the total number of streamlines loaded from the streamline file and on the minimum and the maximum residence time encountered. Similar to the contour settings window several areas may be distinguished to define the appearance of the streamlines.

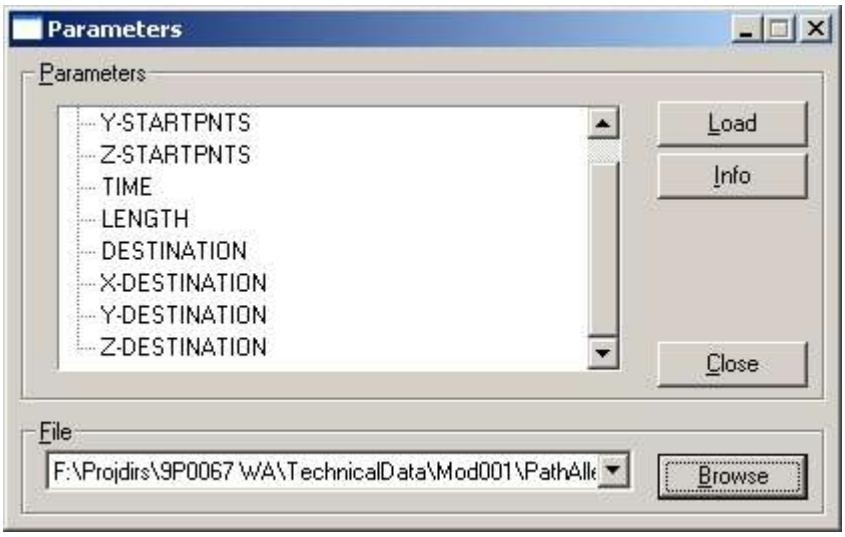
1. Line color		Option for defining the color of the streamlines. Streamline colors may vary with the layer the streamline crosses, with the travel time or with the destination of the streamline. Also a single fixed color may be selected. The user chooses one of the options by checking the appropriate box (◉).
	Line Style	Displays a pop-up window for the definition of the color, width and style of the line the streamlines are drawn with.
2. Draw Markers	Option for adding time markers to the streamlines. Whenever the tick-box is checked time markers will be added to the streamlines. The travel times for which markers are added are those defined in the **'Times area'**. The appearance of the markers depends on the font selected.	

	Font	Displays a pop-up window for the definition of the appearance of the markers. Selecting font again will result in the well-known Font-style window. If one prefers marker symbols in stead of the numbers one could select one of the ESRI-fonts.
3. Draw individual		If the tick-box is checked (☑) each individual point of the calculated streamlines will be displayed.
4. Colors		Define the class-boundaries and colors for the streamline parameter to be classified. The left part of the area displays the parameter values and colors selected. Depending on the parameter selected one the following three labels will be displayed:
		Time Color | Layers Color | Dest Color
		The right hand part (**4a**) displays a number of commands for manipulating the selected times and colors, similar to those used for contouring and classifying parameter values.
	Time, Layers or Dest	Displays a pop-up window for the definition of the start and stop value and the increment of the parameter to be classified.
	Color	Displays a pop-up window for selection of the color gradient to be used to display the classes.
5. Save Settings		Option for saving classification settings for future use. The settings are added to the user's preferences in the General settings file.

9.6.13 How to display results from an influence area simulation (*.tro)

Results from an influence area simulation is stored in an adore type format file. To add a parameter to the view one should select 'Load' from the 'Param' pull-down menu. The user is prompted for the name of a file containing parameter values in Adore format. No select All files (*.tro) and select grid.teo froma grid data set. Once this file is accepted a list of (valid) parameters will be displayed (see below). Selecting one of these parameters will add the parameter to the view. More than one parameter may be selected.

***Note** that, depending on the general settings, the parameter may remain invisible!*



9.6.14 How to execute streamline calculations

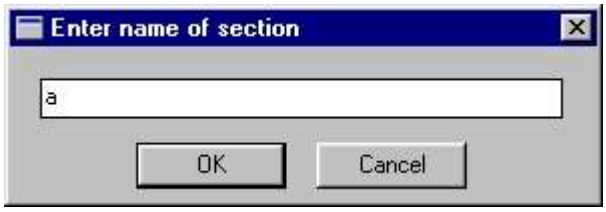
How to execute streamline calculations in **TriPlot** is explained in Chapter 11 Calculation of pathlines and particle tracking; **Trace**.

9.6.15 How to create a cross-section

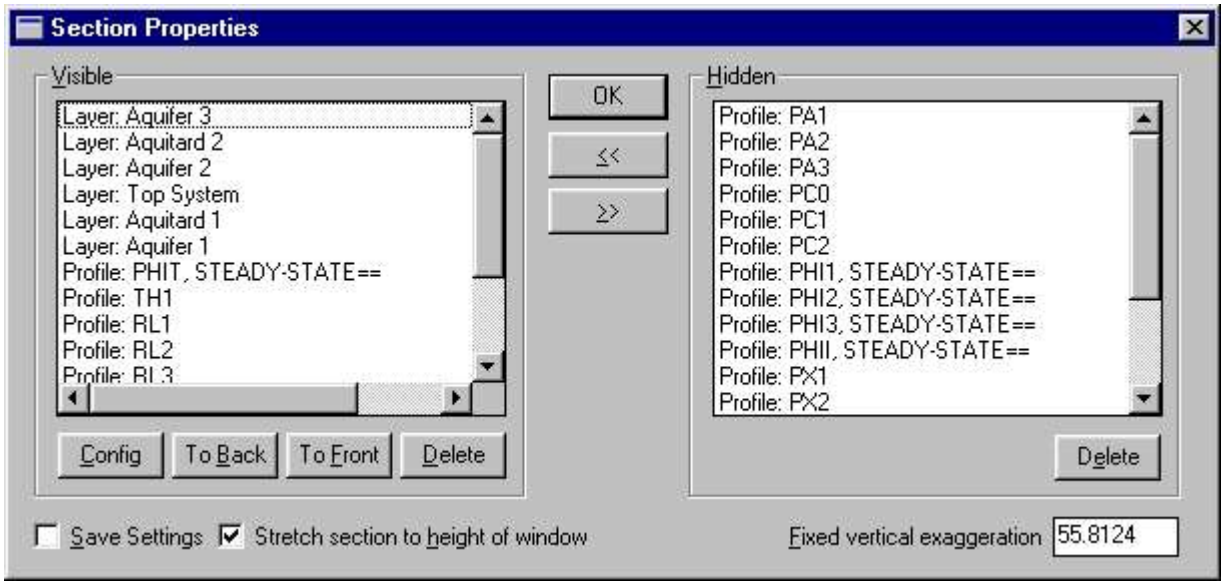
Triplot recognises the following types of cross-sections: the user defined cross-section, the river section and the streamline section. A cross-section consists of a line, defined by two or more points. Aquifers and aquitards will be displayed once the corresponding parameters (**RL*i*** and **TH*i***) have been loaded. In a cross-section hydrogeological parameters for the various aquifers and aquitards may be contoured or classified, groundwater heads may be plotted and streamlines may be projected on the plane of the (vertical) cross-section.

User defined cross-section

To create a user defined cross-section select 'Section' 'Create' from the 'View' pull-down menu or  from the tool-bar. A special (pencil-like) cursor appears on the screen and the user defines the section by drawing a line, pressing the left-hand mouse button to start the line and to define intermediate points and pressing the right-hand mouse button to define the end point of the section. The user is prompted to enter a section name.



Confirming the name, **TriPlot** opens a new window to display the cross-section. The parameters from the original window that define the hydrogeological layers will automatically be added to the cross-section, as will be any streamline map present in the original window. The program also displays the section properties window so the user can select the layers to be displayed in the cross-section.

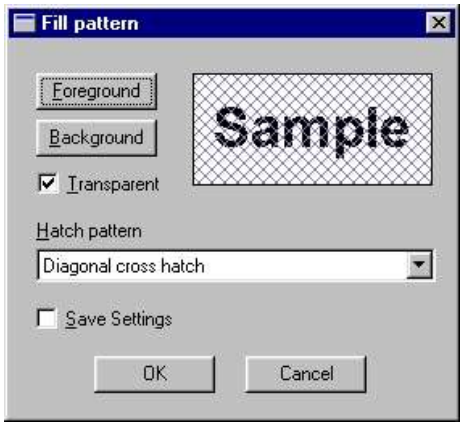


Whenever the parameter **RL*i*** and **TH*i*** are loaded the program will create and add the following items to the view:

Layer: Aquifer ***i***	Between **RL*i*** and **TH*i***
Layer: Aquitard ***i***	Between **TH*i*** and **RL*i+1***
Layer: Top System	Between **TT** and **RL1**

Items or parameters that are not required can be moved from the list of visible items to the list of hidden items.
The appearance of the items can be adjusted and one can use vertical exaggeration.

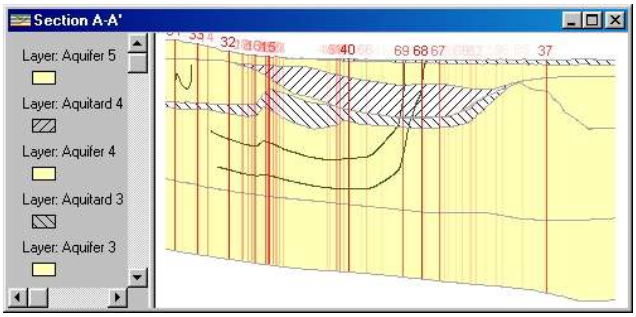
Selecting  the configuration window appears and the user can modify the appearance of the parameter selected.



New parameters may be added to the cross section selecting 'Profile' from the 'Parameter' pull-down menu. A list of the parameters that have been loaded appears to choose from.

A parameter that has not been loaded yet can be added to the cross-section selecting 'Load' from the 'Parameter' pull-down menu.

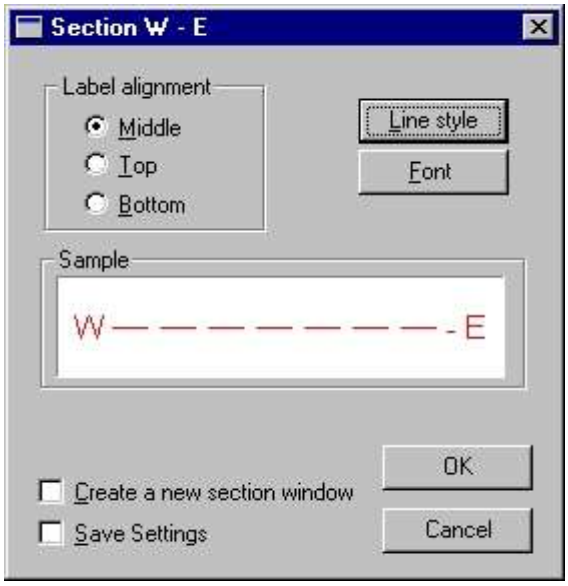
After having made a selection of layers, parameters and streamlines the cross-section will be displayed.



Once a section has been defined the section can be saved from the properties window of the main Triplot view (e.g. the plane view Triplot starts with). To save a section to a *.tps file select 'Section' 'Save …' from the 'View' pull-down menu. The program prompts for a name and for the section to be saved (if there is more than one section defined in the view).

To load a previously saved cross-section select 'Section' 'Load …' from the 'View' pull-down menu. The program prompt for a *.tps file to open and will display the saved cross-section in a new window.

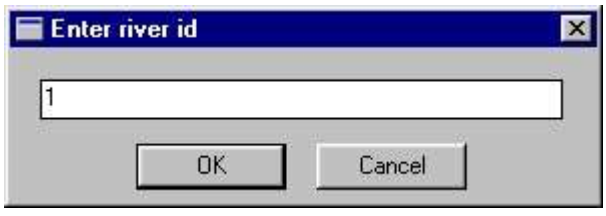
The user-defined cross-section will be displayed in plane view by a (red) line, with the section's name next to it.



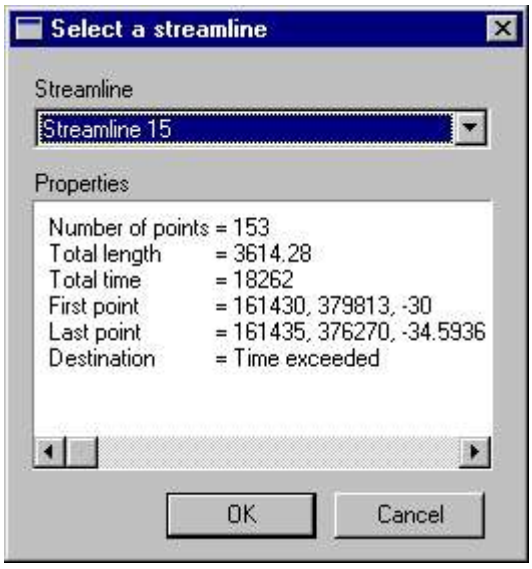
Selecting  while highlighting the section in the visible items list the program displays the configuration window, which allows the user to change the appearance of the section in plane view and to save these settings.

River section

To create a river section select 'Section' 'River section' from the 'View' pull-down menu. The user is prompted to enter a river number. In addition to the distributed parameters also river parameters may be added to the view. The selection of parameters to add to the river-section is the same as with the user defined cross-section.



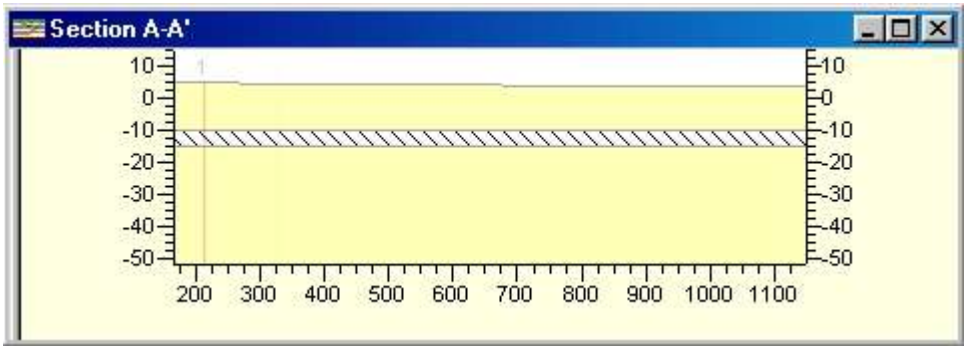
Streamline section
To create a streamline section select 'Section' 'Streamline section' from the 'View' pull-down menu. The user is prompted to select the streamline number from a pop-up window. This window displays information on the selected streamline while this line is flashing in the **TriPlot** view.



***Note** River sections or streamline sections can NOT be saved to a separate *.tps file.*

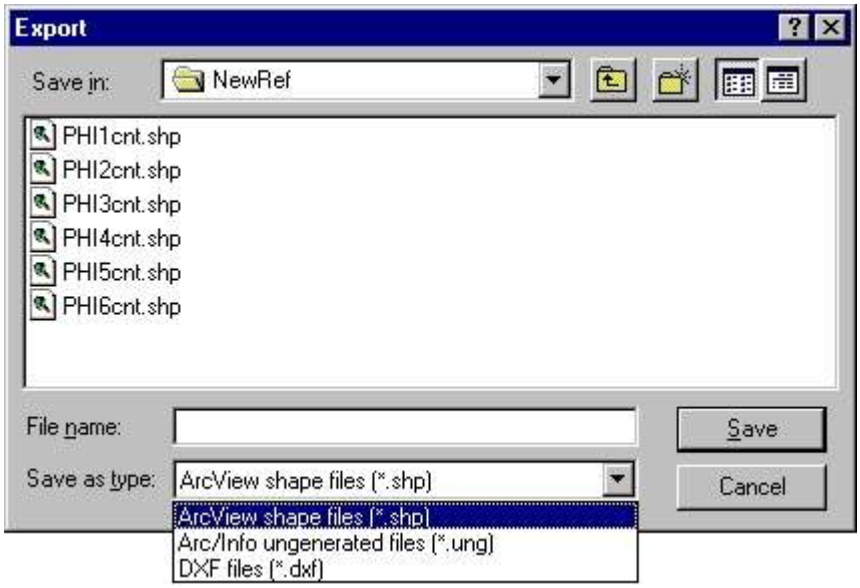
9.6.16 How to add axes to an area view or cross-section

To add axes to an area view or cross-section select "Axes' from the 'View' pull-down menu. The user is prompted to select the axes from a pop-up window. The result in a cross-section may look something like this.



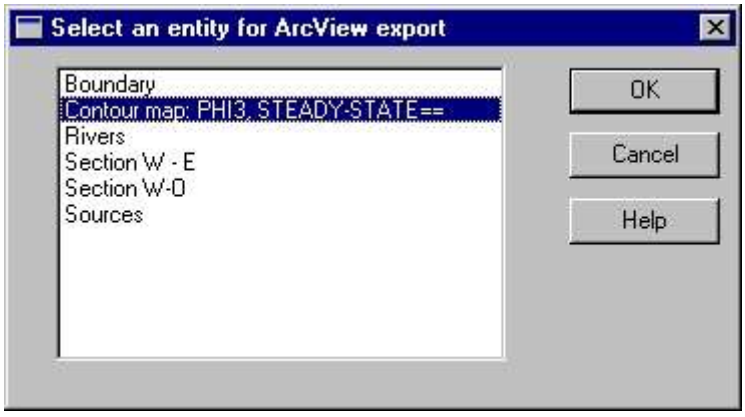
9.6.17 How to export a parameter map

To export a parameter map, either defined by classes or by contours, one should select 'Export' from the 'File' pull-down menu. The user is prompted for a file name. The following file types are supported by Triplot for saving a parameter map: the ARC/Info ungenerated file format, that can also be read by **DigEdit**, the ArcView shape file format and the AutoCAD DXF file format.



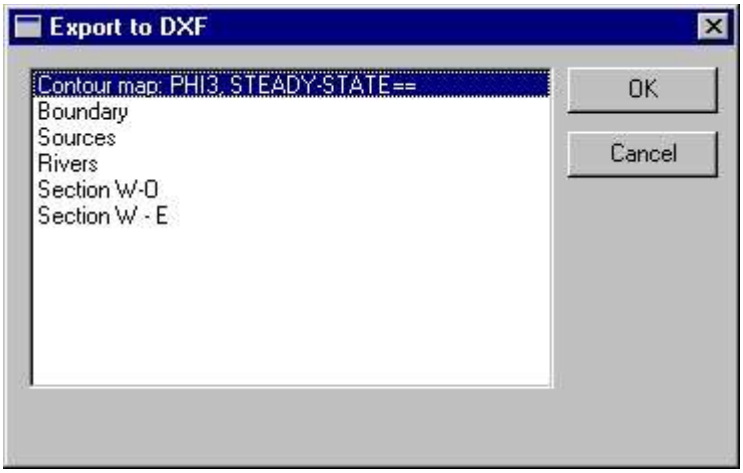
Choosing either one of these formats and supplying a file name, the program prompts for the parameter to be exported. Only for export to Arc/Info ungenerated format some additional export options may be selected. For export to an ArcView shape file or an AutoCAD DXF file no additional information is needed.

ArcView shape file



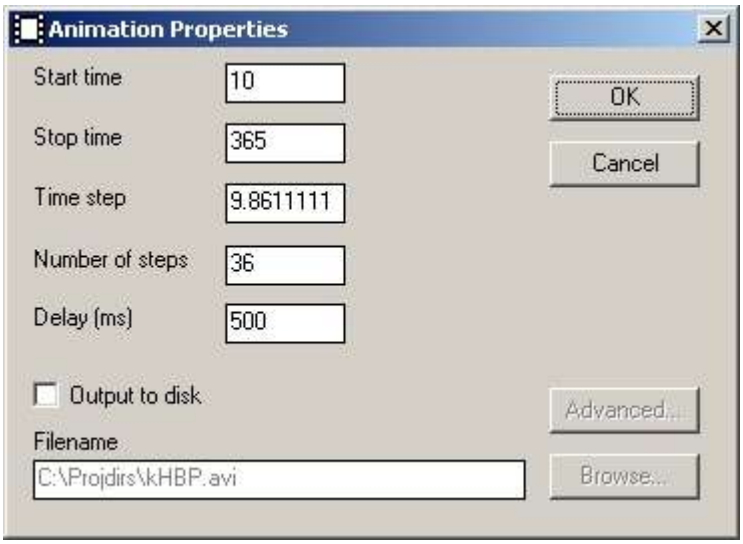
ARC/Info ungenerated file

AutoCAD DXF file

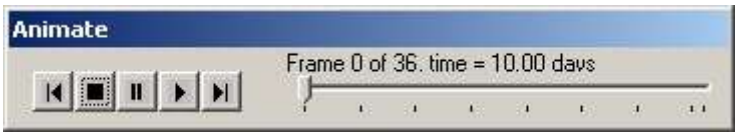


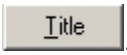
9.6.18 How to create an animation

An animation can be created provided that a transient parameters or transient simulation results are loaded. To create an animation first create an contour or classified map of the parameter used in the animation. Then select 'Time', 'Animate' from the menu bar (or simply ). The following dialog box will appear.

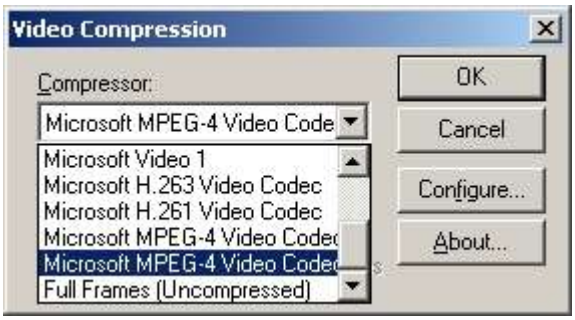


Specify the start time, stop time, time step or number of steps. Delay is used to set the delay time for the frames. Close the dialog box by OK. To start the animation push the play button in the following box. One may also use the time bar to show individual frames.



To create a title and to insert dates in each frame first define the start time by 'Time', 'Starting date' . One is prompted to define date and starting time. Next step is to open the properties window which can be accessed selecting 'Properties' from the 'View' pull-down menu, or right-mouse-button. Select  . The following dialog box will appear. Check 'Show title'. By adding title to the view the date progress is also added to the view. The title will appear at the bottom left and the time progress at the bottom right (often behind the frame dialog box).

An animation may also be saved to disk by selecting output to disk. Various file compression formats are available.



***Note** that, in some cases the transient parameter may exceed the standard windows filesize. For these instances a adore splitting tool is available in the triwaco3\tools directory (adosplitter).*

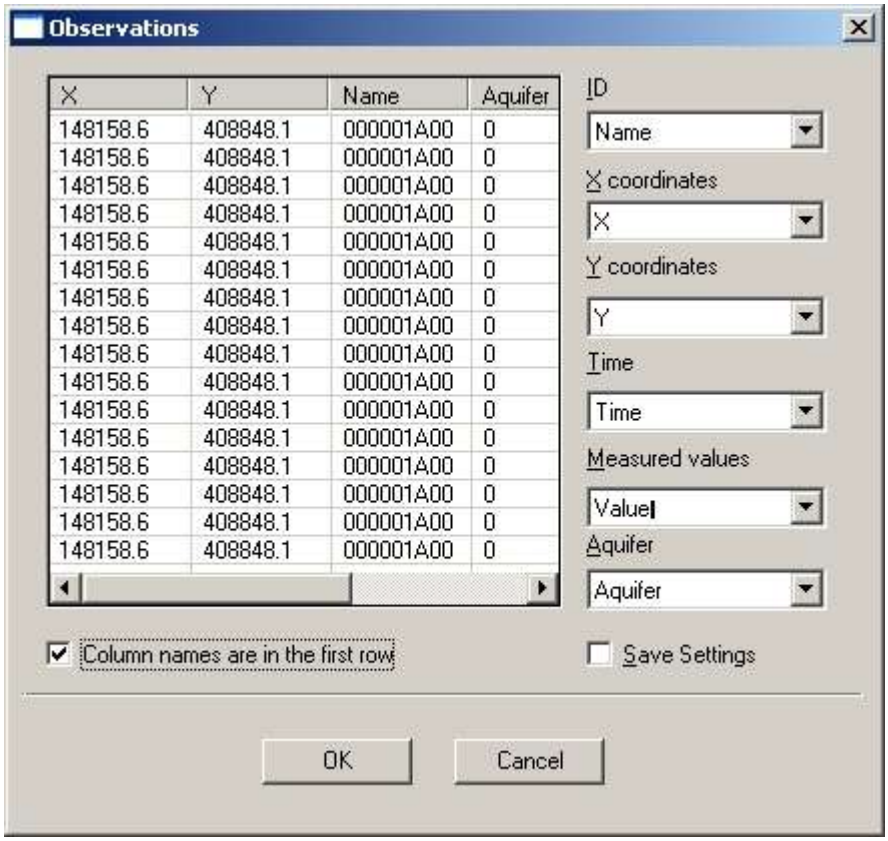
9.6.19 How to create time series (graphs)

Time series from output parameters (*.ado, *.flo, *.fzo)
A time series graph can be created provided that a transient parameters or transient simulation results are loaded. To create a time series graph 'Time', 'Time series' from the menu bar. Select parameters for time series. The time series for each selected parameter is loaded from the transient file. Then point and select the location to create the time series graph. To create another graph for another location simply point and select that location. The current time series graph will be refreshed.

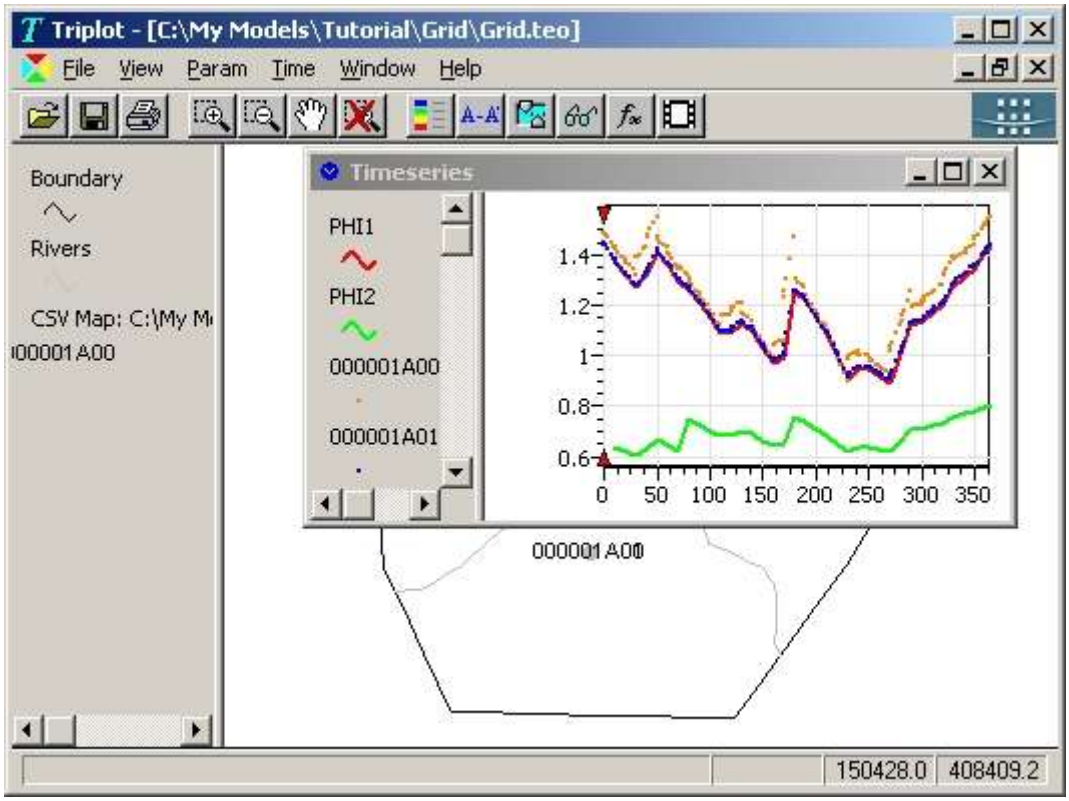
Time series from observed time series (*.csv)
Added to a time series graph or as a seperate graph time series from observation point can be visualised. To add or create a time series from observed heads select 'Observations' from the pulldown menu. Select the csv file which at least should contain X-coordinat, Y-coordinate, Name, Time and Value as shown below. A CSV file can easily be created in Excel or other spreadsheet program. Note that is has to comma seperated and not ; seperated.

```
X,Y,Name,Aquifer,Time,Value
148158.6,408848.1,000001A00,0,0.00E+00,1.53E+00,
148158.6,408848.1,000001A00,0,5.00E-01,1.49E+00,
148158.6,408848.1,000001A00,0,1.49E+00,1.48E+00,
148158.6,408848.1,000001A00,0,3.44E+00,1.47E+00,
...
```

The program opens a pop up window where each column from the observations file can be assigned. Once loaded the observations points are displayed in the model area (similar as for the calib.chi). To display the time series select 'Time', 'Time series' from the menu bar. Select parameters for time series. The time series for each selected parameter is loaded (either from the transient output file or fro observations). Then point and select the location to create the time series graph. The time series of an observation point will only be displayed when a point is selected near to an observation point. To create another graph for another location simply point and select that location. The current time series graph will be refreshed.



The appearance of time series can easily be changed by right-click mouse which opens a menu. One may change the thickness of the lines and change the time of markers.



***Note** that, more than one time series graphs can be opened with different parameters.*

9.6.20 How to save a triplot session

A Triplot view may consist of one or more windows, of which at least one contains the model area in plane view.

The view, with all associated settings, may be saved in a so-called Triplot project file (*.tpp). Selecting 'Save' from the 'File' pull-down menu or  from the tool-bar will cause the program to prompt for a file name. Having supplied this information the view is saved for future use in the selected *.tpp file. A saved view can be opened by Triplot selecting 'Open' from the 'File' pull-down menu or  from the tool-bar.