MIKE View

A Results Presentation Tool for
MIKE URBAN and MIKE 11

User Guide

PLEASE NOTE

COPYRIGHT

This document refers to proprietary computer software which is protected by copyright. All rights are reserved. Copying or other reproduction of this manual or the related programs is prohibited without prior written consent of DHI. For details please refer to your 'DHI Software Licence Agreement'.

LIMITED LIABILITY

The liability of DHI is limited as specified in your DHI Software Licence Agreement:

In no event shall DHI or its representatives (agents and suppliers) be liable for any damages whatsoever including, without limitation, special, indirect, incidental or consequential damages or damages for loss of business profits or savings, business interruption, loss of business information or other pecuniary loss arising in connection with the Agreement, e.g. out of Licensee's use of or the inability to use the Software, even if DHI has been advised of the possibility of such damages.

This limitation shall apply to claims of personal injury to the extent permitted by law. Some jurisdictions do not allow the exclusion or limitation of liability for consequential, special, indirect, incidental damages and, accordingly, some portions of these limitations may not apply.

Notwithstanding the above, DHI's total liability (whether in contract, tort, including negligence, or otherwise) under or in connection with the Agreement shall in aggregate during the term not exceed the lesser of EUR 10.000 or the fees paid by Licensee under the Agreement during the 12 months' period previous to the event giving rise to a claim.

Licensee acknowledge that the liability limitations and exclusions set out in the Agreement reflect the allocation of risk negotiated and agreed by the parties and that DHI would not enter into the Agreement without these limitations and exclusions on its liability. These limitations and exclusions will apply notwithstanding any failure of essential purpose of any limited remedy.

1　Getting Started

1.1　How to Read this User Guide

MIKE View is a map-based GUI for the presentation of results from MIKE URBAN and MIKE 11 one dimensional, looped network, software packages.

MIKE View is designed to service several different MIKE by DHI software packages. As the content of result files for these different software packages varies, the features of MIKE View will also vary with the type of model result file being viewed.

In this user guide both the generic MIKE View features used to display results from the many different model result files and the product specific MIKE View features are described. The user guide follows the format of providing general information on features applicable to all model result files followed by product specific features described in relevant subsections.

1.2　Introduction

MIKE View is a standard part of the MIKE URBAN and MIKE 11 packages.

MIKE View can display results from

- MOUSE Runoff, HD, LTS, AD, WQ and ST simulations
- SWMM 5 (Results from Runoff, EXTRAN and TRANSPORT are converted to MOUSE result files and then loaded into MIKE View)
- MIKE 11 NAM, HD, AD, WQ and ST simulations
- EPANET
- MIKE 1D RR, HD, AD

All these results can be displayed in combination with the time series taken from other sources external to the result file such as ASCII files, MOUSE and MIKE 11 time series databases, etc. Please note that since the SWMM 5 results need to be converted to MOUSE result files before they are loaded into MIKE View all features valid for MOUSE are also valid for SWMM 5.

The results can be presented as:

- Color animations or static plots on a Horizontal Plan
- Animated longitudinal profiles
- Time series and duration curve graphs
- Q-H relations

MIKE View is undergoing continuing development. We would welcome your comments and suggestions for future versions. Please feel free to pass on your comments to DHI directly or via your local DHI agent.

1.3   Installation of MIKE View

When installing MIKE ZERO or MIKE URBAN, then MIKE View will be installed automatically with the installation of these software packages.

1.4   Starting the MIKE View GUI

The MIKE View GUI can be activated from various locations. The program is activated from Windows by double clicking on the 'MIKE View' icon in the MIKE VIEW program group.
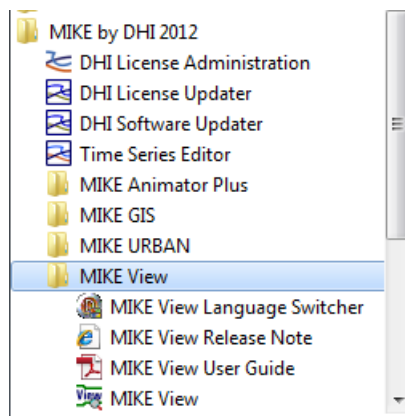
Figure 1.1   Example of how to start MIKE View from the program group

Once activated, the program prompts the user to open a result file and select the data you want to load. Choose the file you want to view and click the 'OK' button.
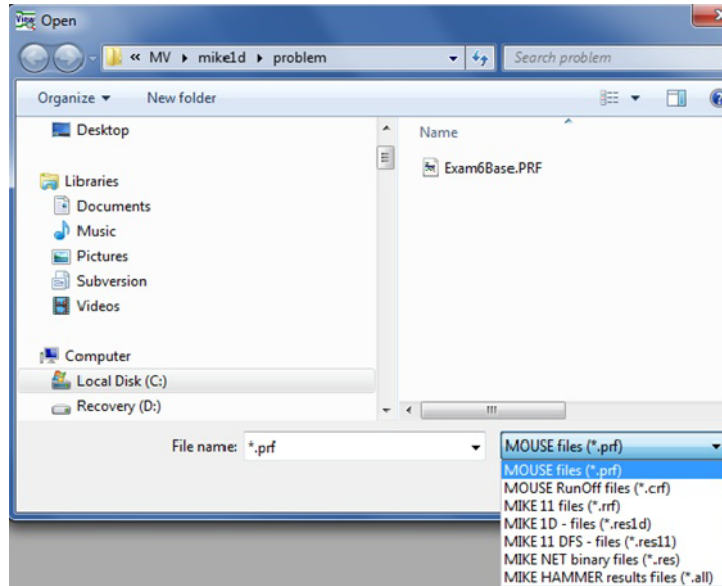
Figure 1.2   Choosing between the different result files

1.5   Content of the MIKE View Main Window
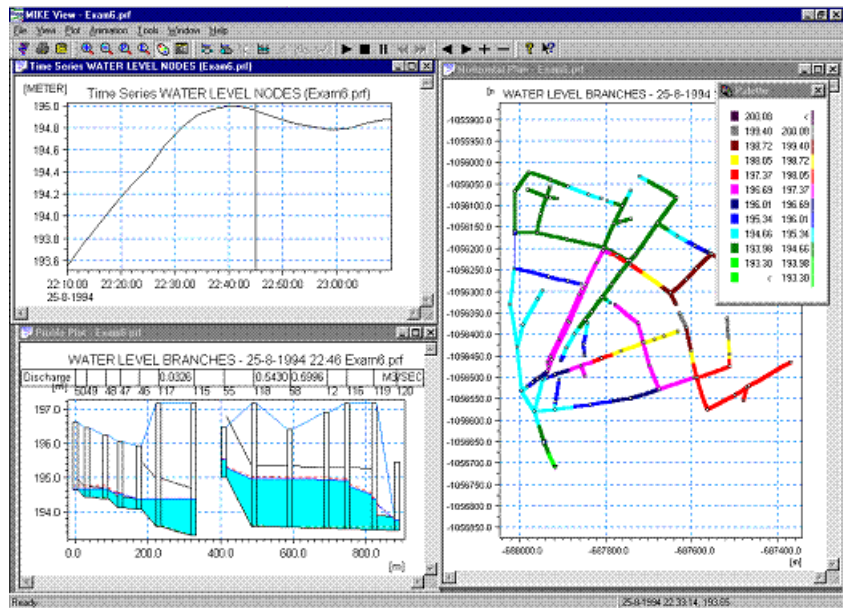
Figure 1.3   The MIKE View main window

1.5.1 Graphical windows and dialogs

The following types of graphical windows exist in MIKE View:

- Horizontal Plan window, opens automatically when a MOUSE/MIKE URBAN, MIKE 11 or MIKE 1D network result file is loaded. Displays various types of data in different presentation modes in a plan view of the urban drainage or river network. Only one Horizontal Plan window can be associated with each of the loaded network result files. The local menu is slightly different depending on the type of result file. The Horizontal Plan window acts as the 'main' window - if it is closed, all other windows associated with the result file will be closed too.
- Plan Overview window, opens automatically when a network flow result file is loaded. It displays an overview of the network plan plot, with the current area in the Horizontal Plan window shown as a frame.
- Palette window opens automatically with some presentation modes. Displays the color palette for the Horizontal Plan.
- Longitudinal Profile window, opens upon the user's selection of the Longitudinal Profile on the Horizontal Plan. Displays various types of data in different presentation modes in a profile view along the selected network branches. An arbitrary number of Longitudinal Profiles can be associated with one Horizontal Plan.
- Time Series window, opens upon the user's selection of a specific item on the Horizontal Plan. Displays various types of data as time series graphs. Each time series window contains one graph, however multiple time series can be presented in the same window. An arbitrary number of time series can be associated with one Horizontal Plan.
- Duration Curve window, opens upon the user's selection of the "Duration Curve" option for the currently active time series plot. Displays the time series and the duration curves for the time series. In the right-hand side of the window, tables with numerical values are presented.
- Q-H Relation window, opens upon the user's selection of a specific location on the Horizontal Plan. Displays Q-H relations for a network branch at the selected location. An arbitrary number of time series can be associated with one Horizontal Plan.

Apart from graphical windows, MIKE View contains a number of dialogs for display configuration and for tabular data display.

1.5.2 Menus

The MIKE View main menu contains the following items:

- File: files can be opened, added, closed, presentation layout loaded, the program terminated, graphics from the active window printed
- View: elements of the MIKE View outlook can be switched 'ON' or 'OFF', and the appearance of the Horizontal Plan can be controlled
- Plot: selection of various items to be viewed

- Animation: control of the animated presentation
- Tools: activation of various accessories
- Window: control of the windows arrangement
- Help: access to MIKE View online help facility

Additionally, local menus can be activated for any active window by clicking with the right mouse button inside a window. This is available for each of the graphical windows (Horizontal Plan, Longitudinal Profile, Time Series and Q-H). The number of menu items varies for each window type.

The MIKE View online help can also be activated directly by pressing the 'F1' button on the keyboard, this will open the relevant help page (where available) to the active window.

1.5.3 Toolbars

The MIKE View toolbars are located under the main menu. Each toolbar contains several tool icons, all of which serve as shortcuts for important program functions. The same functionality for each tool in the toolbars can also be found in the main menu.

The active tools on the displayed toolbars adapt as the user navigates between different windows. As a result, each MIKE View window has its own special toolbar, optimized for the specific functions associated with the window.

Individual toolbars can be switched 'ON' and 'OFF' using the "View | Toolbars" option.

1.5.4 The status bar

The MIKE View status bar is located in the bottom of the MIKE View window.

The left side of the status bar provides information related to the program mode, the expected user actions with tips for shortcut keys, etc.

In the right section of the status bar, the co-ordinates of the cursor are traced and displayed. Depending on the type of the active window, the co-ordinate system may be X-Y, X-Time, Time-Value or Q-H.

The status bar can be switched 'ON' and 'OFF' under the "View" menu.

2 Working with Result Files

2.1 Selecting a File for the Presentation

When the program is started an 'Open File' dialog appears. It is important to remember that only those result files, which contain the geographical information of the modelled network, can be opened directly by MIKE View (other file types have to be added).

Several file types can be opened from this view:

.PRF	MOUSE HD result file (binary)
.CRF	MOUSE runoff file (binary)
.RRF	MIKE 11 HD result file (binary)
.RES11	MIKE 11 DSF result file (binary)
.RES1d	MIKE 1D result file (binary)
.RES	EPANET Result file (binary)

As many different result files as the available computer hardware capacity allows may be opened by selecting "File |Open".

2.2 Selecting Data for the Presentation

By default, all available information saved in the result file is selected for loading.

You can reduce the amount of data to be loaded by excluding some of the available data types, by truncating the time period to be presented (choosing the appropriate FIRST and LAST time step), or by increasing the 'Step for loading' factor to an integer value larger than one.

The Data Load Selection view opens up automatically after you select a file to be loaded.
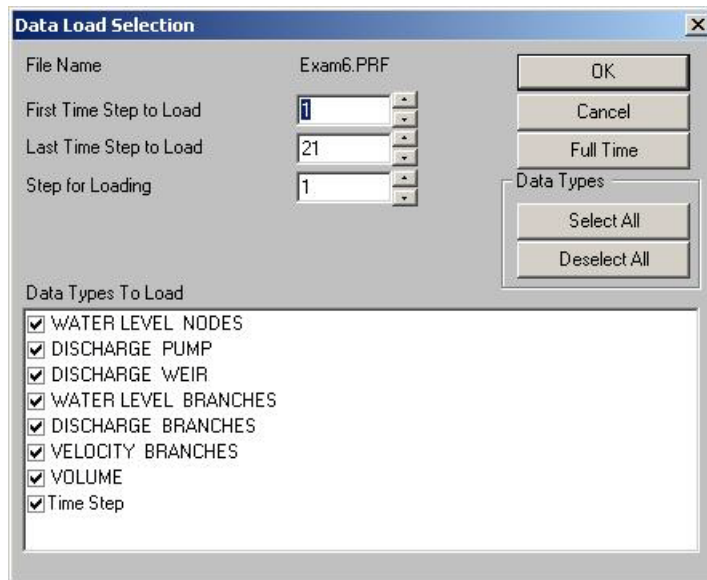
Figure 2.1   Example of the Data Load Selection view for a MOUSE result file

2.3   Adding a File

Some types of result files contain result data time series, but do not contain the necessary geographical information needed for plotting the model network. Such files can be 'added' to, and viewed from, an already opened MIKE View presentation.

Adding a file to an existing MIKE View presentation does not open a new Horizontal Plan window. The already opened Horizontal Plan window is reused for viewing of the added data. The program provides a list of file types eligible for "Add"-function automatically.

As many files as required may be loaded to the originally loaded file, as long as they have been generated by the same network file.

For example, you can also add files (like PRF files), which you would normally view using 'Open'.

2.4   Closing a File

You can close the loaded file by closing the active Horizontal Plan window. When you close a file, all associated windows derived from this file are automatically closed as well.

3 Viewing Data in a Horizontal Plan

3.1 The Local Menu for the Horizontal Plan

Clicking on the right mouse button when the Horizontal Plan is the active window activates the local menu. Please note that it is slightly different for the different type of result files.
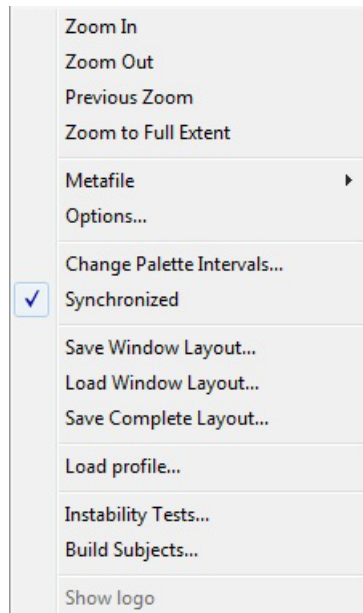
Figure 3.1   The Local Menu for the Horizontal Plan, example shown for prf-file

The local menu for the Horizontal Plan includes the following sections:

- Zooming facilities (Zoom In, Zoom Out, Previous Zoom, Zoom to Full Extent)
- Metafile (Horizontal plan copied to clipboard or saved as an Enhanced Meta File (*.emf))
- Options (Selecting the items to be displayed in the Horizontal Plan)
- Synchronization (Allows synchronization with other open windows when animated)
- Layout facilities (Window Layout (saving and loading) and the Complete Layout (saving))
- Load profile (Loading a longitudinal profile (*.lpf), which enables fast drawing of a saved longitudinal profile). Note: This option is not available for MIKE 11 and MIKE 1D.

3.2   Selecting the Horizontal Plan Type

MIKE View can display various static or dynamic items from the loaded (or added) result files in the Horizontal Plan. However, only one item can be shown at a time. The selection of the required item and the control of the Horizontal Plan type take place in the Horizontal Plan local menu under "Options | Plan Type"

The Options View is opened by clicking on the 'Options' button in the

tool bar 🖼 or alternatively by opening the local menu (click on the right mouse button) and choosing 'Options'.
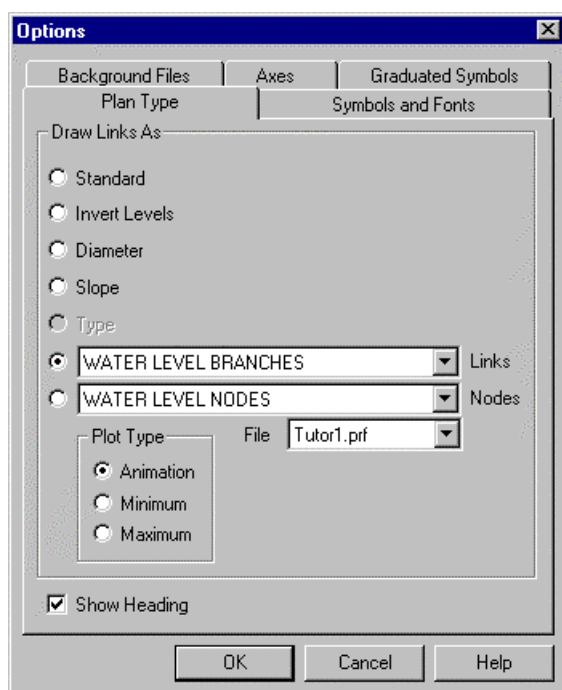
Figure 3.2   Options View for MOUSE/MIKE URBAN Horizontal Plan

The static (structural) items can only be presented as static horizontal plots, displayed color coded according to the associated palette.

Dynamic items can be shown as the envelope of minimum or maximum values, or as an animated presentation.

Presentation of negative flows as absolute values (e.g. to avoid confusion due to inconsistent specification of the upstream and downstream nodes for some links), can be toggled by ticking the 'Absolute values for Q' field.

Simultaneous animations in multiple windows can be fully synchronized by choosing "Animation |Synchronize All Plots".

3.3   Controlling Symbols and Fonts

Control of symbols and fonts shown in the Horizontal Plan is accessed in "Options | Symbols and Fonts" menu.
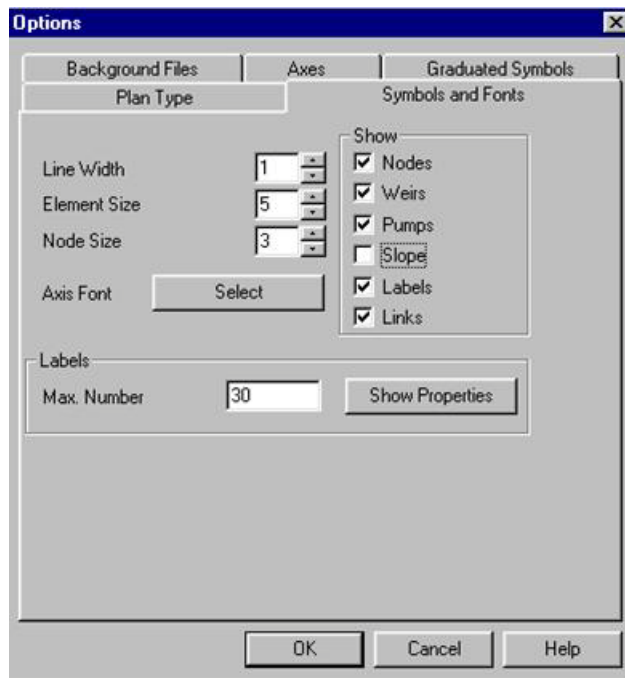
Figure 3.3   The Options View showing Symbols and Fonts settings

In this menu you can control the dimensions of the network graphical symbols, select fonts for the plan axes and labels, select the label properties and switch each of the available symbol types ON/OFF.

When the nodes are switched 'OFF', it is not possible to select Longitudinal Profiles or time series in nodes graphically. Switch the nodes 'ON' temporarily if you want to make a selection this way.

When pressing the 'Show Properties' button in the 'Labels' section it is possible to add predefined labels to the horizontal plan when working with prf files (e.g. in MOUSE result files, the diameter for links and the invert level for nodes).
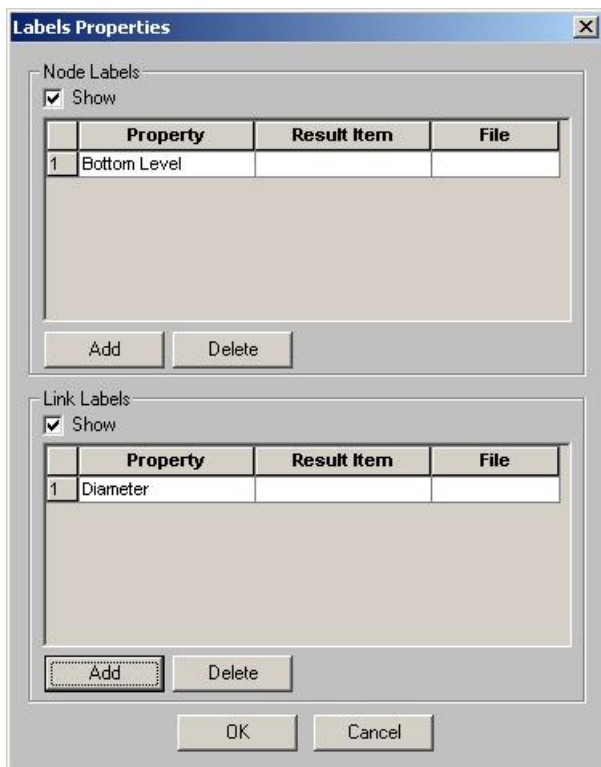
Figure 3.4   Adding (predefined) labels to the horizontal plan

3.4   Importing a Background Map
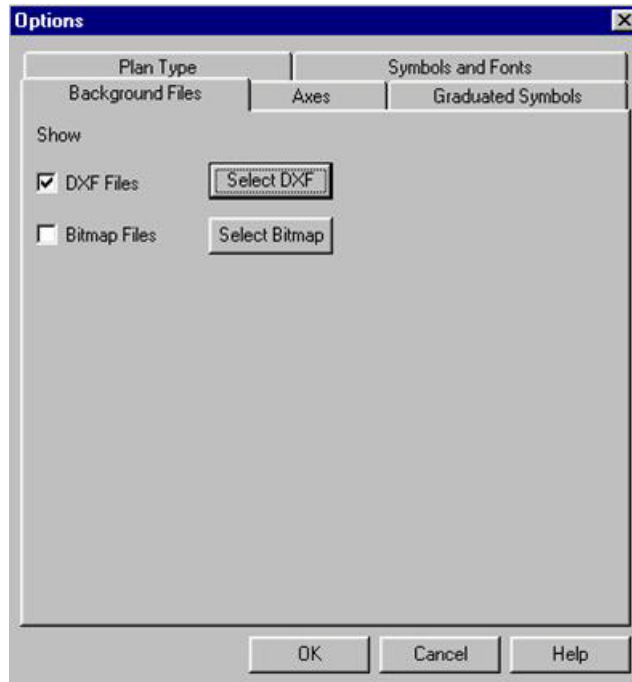
Figure 3.5   The Options View showing Background Files settings

Working with the Horizontal Plan can be easier if you import a background map image. MIKE View supports 3 standard graphical file formats (DXF, TIFF and BMP) for this feature. The background map selection is found under "Options | Background Files".

Loading DXF files is straightforward, assuming full consistency between the co-ordinate systems used in the model and in the DXF file.

If a DXF file is based on a co-ordinate system different from the MOUSE network model and/or if it is based on other units than SI, the MIKEVIEW.INI file (located in the Windows directory) must be manually edited, to compensate for the co-ordinate offset, and/or for the scaling in relation to SI.

The section of the MIKEVIEW.INI file controlling the DXF image offset and scaling is shown below:

```
[DXF_transform]
Xoff=0.00
Yoff=0.00
Xmulti=1.0000
Ymulti=1.0000
```

It is possible to add several BMP and TIFF files to the background of a single Horizontal Plan window. The visibility of these can be set individually to 'Full' (the entire image is drawn), 'Name+Border' (only the name of file and the border of the image will be shown), 'Border' (solely the border of image is shown) and 'None'. The images will be drawn in the sequence shown in the list of images.

A bitmap (BMP) or a TIFF file requires additional information, which links the background map image with the model co-ordinate system. The required information is usually provided by the supplier of the digitized image. In case that you do the map scanning on your own, a trial and error method is the only way to make the image fit with your model.

You should provide this information in a special ASCII file (file name the same as the .BMP/TIFF file, default extension .BMW/.TFW).

A .BMW/.TFW file consists of 6 lines, containing the following information:

- number of meters per pixel in X direction of the bitmap image
- 0 (should not be modified!)
- 0 (should not be modified!)
- Number of meters per pixel along Y direction of the bitmap image. The negative sign reflects the default choice of the image origin point at the left upper corner, i.e. the image is drawn along the negative direction of the Y axis
- X co-ordinate of the left upper corner of the bitmap image (should be consistent with your model co-ordinates)
- Y co-ordinate of the left upper corner of the bitmap image (should be consistent with your model co-ordinates).

An example of a .BMW file is given below.

```
4.100000
0
0
-4.100000
5234000
3245000
```

Figure 3.6   Example of BMW file

3.5   Controlling the Axes

The orientation of the co-ordinate axes, the appearance of the Horizontal Plan grid and the axes descriptors can be controlled through the axis view

box. The axis view options menu can be activated under the Horizontal Plan local menu selection "Options | Axis".
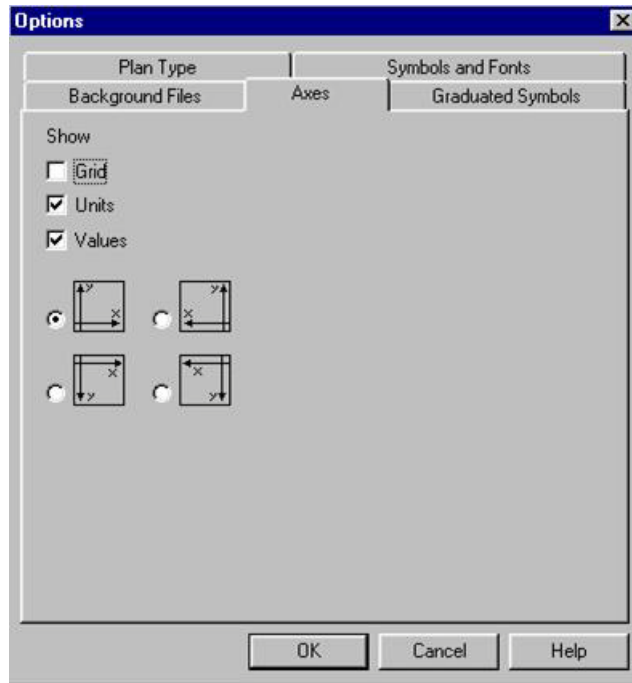
Figure 3.7   The Options View showing Axes settings

The 'Options' selection is accessed through a click on the options icon on the Horizontal Plan tool bar, or the right mouse button.

If the orientation of the axes changes, the background images will automatically follow the change.

3.6   Using Graduated Symbols

Graduated Symbols are a graphical facility for dynamic presentations of the node- and link-related results in the Horizontal Plan window. This is an excellent way to show the spatial distribution and dynamics of combined sewer overflows, flooding, critical pressures in water net etc.
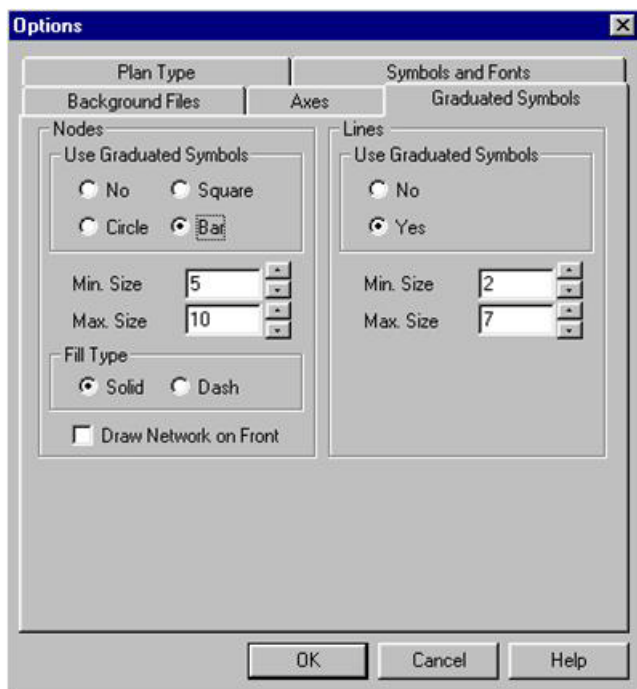
Figure 3.8   The Options View showing Graduated Symbols settings

Under the Horizontal Plan local menu selection "Options | Graduated Symbols", the graduated symbols can be switched 'ON' and 'OFF', toggled through three available shapes (circle, square and bar), the symbols' minimum and maximum size controlled, the type of fill selected, as well as the grid allowed to be plotted on the top of the graduated symbols. Please note that the Graduated Symbols option is not available with MIKE 11 result files.

Graduated symbols can display the value of a selected item by simultaneously sizing the symbols and by applying the palette colors. This display can be simplified, if desired, by removing one of the presentation variables: e.g. either set the symbols min. and max. size to be equal, or adjust the palette so that only one color is displayed for the entire range of the selected item.

3.7   Zooming and panning

A global view of a large network may not display the required level of detail of the model. MIKE View allows you to focus on smaller parts of your network, through the Zoom function. The available 'Zoom' functions are:

- Zoom in;
- Zoom out;
- Zoom to model extent;
- Return to previous zoom frame.

The MIKE View zoom facility is activated by the toolbar zoom icons, or as a selection on the local menu (the right mouse button).

The activation of the 'Zoom In' function, changes the standard cursor shape into a magnifying glass symbol. Zooming can be enabled in two modes. By repeatedly clicking on the left mouse button, the display is enlarged in steps. The location pointed to by the cursor is becomes the center of the next enlarged plot. The 'zoom' mode is terminated by pressing the <Esc> key.

Alternatively, the desired zoom area can be framed by a rectangle with an origin corner located by the position of the center of the cursor. The size and the coverage of the zoom area is determined by dragging over the screen (with left mouse button down!) to the diametrically opposite corner of the desired zoom rectangle. The desired zoom area is 'caught' and displayed by releasing the left mouse button at the required location.

The 'deepest' possible zoom enlarges of the model view down to a 1 meter resolution.

The 'Zoom Out' option works in the opposite manner to the 'Zoom In' facility.

The pan functionality is especially useful when zoomed in as it allows to move around with the chosen zoom. The 'pan' mode is entered by clicking on the symbol (the small hand) found on the toolbar next to the zoom tools. The 'pan' mode is terminated by pressing the <Esc> key.

The Overview plan window can be of great assistance when zooming in. It displays the extent of the zoomed frame on the overview plan and, also allows moving the zoomed frame within the overall model area. To move the zoom area within the overview window, click on the zoomed frame inside the Overview plan and drag it to the desired location.

3.8   Controlling the Palette

The color palette for the Horizontal Plan presentation is displayed in the Palette Window (see Figure 3.9).
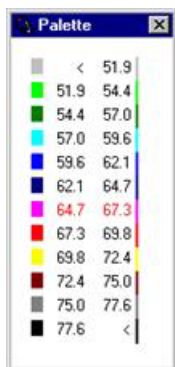
Figure 3.9   The palette window displaying one of the five available palette types

Figure 3.10   The local menu of the palette

The local palette menu allows numerous possibilities to adjust the palette precisely to fit the current needs and preferences. The following palette appearance features can be controlled:

- The graphical palette shape can be selected from five different shape options and the favourite shape set as default
- Palette fonts
- The number of palette intervals. The maximum number is ten, which results in a 12- color palette
- The ranges for individual intervals. Equidistant or customized interval ranges may be selected
- The high and low boundaries of the palette range

The upper and lower limits of the palette range may be also modified under the Horizontal Plan local menu 'Change palette Intervals'.
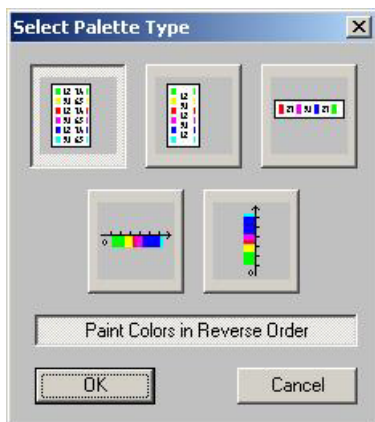
Figure 3.11   The five possible shape options of the color palette

The default palette colors can be customized. When you want to change a color, just point at the color to be changed in the palette and double-click the mouse: This will open a color editor, where the color can be adjusted as required.

Similarly, the upper and lower limits of the palette can be adjusted. A double click on the interval to be modified (i.e. on the highlighted number in the palette window) opens the Edit Intervals panel (see Figure 3.12).
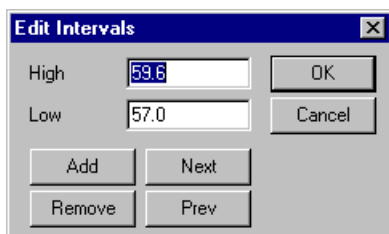
Figure 3.12   Editing the palette intervals

MIKE View does not check the consistency of the custom intervals during the editing process. The consistency of the palette can be confirmed using the 'Check Intervals' facility under the palette's local menu.

A custom palette can be saved into a .PAL file. Gradually, a small library of favourite palette files can be built, which can be reused in any other presentation, simply by loading an appropriate one.

The palette can be removed if required by switching the palette tool bar button to the OFF position. The palette can be restored by switching the palette tool bar button back to the ON position.

4 Viewing Longitudinal Profiles

4.1 Selecting a Profile

Longitudinal profiles of the model network can be selected graphically by pointing and clicking on a Horizontal Plan view. Click on the longitudinal profile icon on the tool bar, or select the "Plot | Profile" on the main menu while in an active Horizontal Plan window to activate the profile selection mode.

The profile icon on the tool bar.

The change of the program mode is indicated by a change of the cursor shape. Selection of a profile is now possible when you approach a valid node to be included in the profile with the cursor. If the model node is valid for inclusion in a profile, the cursor shape will change into an arrow icon.

Status bar in the bottom of the screen informs about the current program mode and suggests the action

When a valid node for the profile is selected it changes to a bold green circle or in the case of MIKE 11 the model reach turns green. Each subsequent click on the mouse button selects the next valid node along the network branch. Selections are painted green. In case of two or more possible node options at a model junction, you should help MIKE View by pointing and clicking on the node in the desired direction of the profile.

It is usually sufficient to click on the first node of the profile, and point and click repeatedly on the last node required in the profile - MIKE View will find the right path to the cursor on its own!

If the last selected node is not required <Shift> + mouse click will cancel the last selection(s), allowing for the redirection of the profile to the desired network branch.

The direction that the profile proceeds from a node can be changed by using the cursor arrow keys. Pressing <Esc> will cancel the entire selection.

A mouse click in a combination with <Ctrl> finishes the profile selection. The program then prompts you to select the type of item to be shown on the profile.

If the profile selection arrives at a dead end in the model network, MIKE View automatically closes the selection and the Longitudinal Profile window appears and becomes active.

A profile can only be selected with displayed nodes.

In the longitudinal profile you can also use the pan funcionality.The 'pan' mode is entered by clicking on the symbol (the small hand) found on the tool-bar next to the zoom tools. The 'pan' mode is terminated by pressing the <Esc> key.

4.2   Working with the Local Menu

Figure 4.1   The local menu of the longitudinal profile (MIKE11/MIKE 1D)

The Longitudinal Profile local menu is accessed by a click on the right mouse button, while in an active Longitudinal Profile window. This menu contains the following:

- the zooming facilities, including 'Advanced zoom' with exact specification of axes ranges
- buttons for exporting the Longitudinal Profile to the Clipboard and to the Metafile (*.EMF)
- the 'Options' that control the layout of the longitudinal profile
- the 'Clear History' facility (only available for MIKE 11), which clears the plot of the profile history if this option has been selected
- the 'Multi-link' facility that makes it possible to choose the how to display links if multiple links exist between two nodes (used in MOUSE and MIKE URBAN)
- the 'Swap Profile' that swaps the direction the profile is plotted (left/right)
- the animation synchronization control
- the 'Save Window Layout' feature
- the 'Save Profile' (*.LPF) feature (not available in MIKE 11)

The zoom is also accessible via the toolbar.

The effects of various presentation options are best understood by trying on a small example (use the tutorial).

4.3   The Options for the Local Menu
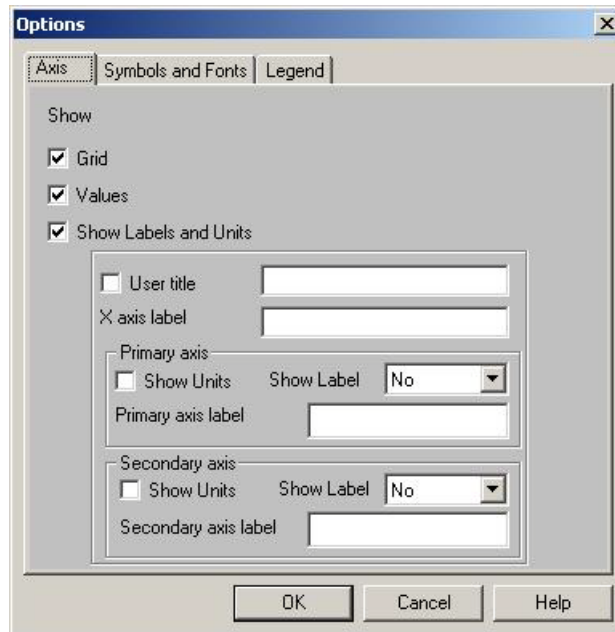
Figure 4.2   The options controlling the layout of the longitudinal profile

Three tabular pages exist in MOUSE/MIKE URBAN:

- Axis
  The grid, values, axis labels and units are controlled from this page.
  The labels can be displayed, placed horizontally or vertically.

- Symbols and Fonts
  The display of node labels, node labels orientation, fonts and certain
  computed values (for MOUSE e.g. the pressure line) are controlled

- Legend
  Controlling which legends are displayed on the screen and/or on prints

Additional tabular pages available for MIKE 11/MIKE 1D files are:

- Graphical Items
  The Graphical Items tab allows you to add additional information to the
  existing profile. The additional information consists of either static or
  dynamic items and can be from added result files.

- Tabular Items
  The Tabular Items tab allows you to add additional tabular information
  to the bottom of the existing profile. The additional information consists

of either static or dynamic items expressed as a numerical value and can be sourced from added result files.

- User defined Indicators
  The User Defined indicators tab allows you to display markers on the longitudinal profile at user defined levels. The markers can be used to indicate critical levels (such as culvert details or road levels) at locations along the profile.

The effects of various presentation options are best understood by trying a small example (use the tutorial).

5 Viewing Time Series

5.1 Selecting a Time Series

Time series information can be associated with nodes or with links. Some result files also contain time series related to the whole system ('system items').

You can select a time series graphically, pointing and clicking on a Horizontal Plan, or by making a selection from a list.

The selection mode is activated while in an active Horizontal Plan window by clicking on the node or link icon on the toolbar, or selecting the "Plot | TS in Nodes", or "Plot | TS in Links" in MOUSE/MIKE URBAN or "Plot | TS in Grid Points in MIKE 11 on the main menu. This will open the time series selection dialog.
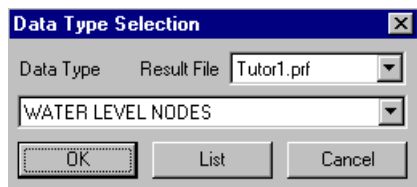
Figure 5.1   Time Series Selection Dialog

The time series selection dialog allows for the specification of the result file name (if several result files have been added) and the time series type. Available types depend on the result file.

The 'OK' button closes the dialog and activates the graphical selection mode. The change of the program mode is indicated by a change of the cursor shape. The selection is possible only when you approach a valid node or link and the cursor shape turns into an arrow.

Clicking the mouse on the required node opens a time series graphical window and makes it active.

You can zoom-in and zoom-out even while in a selection mode.

You can use the pan functionality in the time series. The 'pan' mode is entered by clicking on the symbol (the small hand) found on the toolbar next to the zoom tools. The 'pan' mode is terminated by pressing the <Esc> key.

In MOUSE/MIKE URBAN, when you select a link, you should further specify the location of your choice along the selected link (computational point).
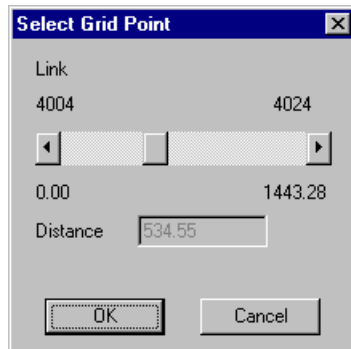
Figure 5.2   Selecting a location along a link in MOUSE/MIKE URBAN

The selection of the 'system items' can only be activated from the main menu "Plot | TS of System data".

The 'List' button in the "Data Type Selection Window" opens the time series list window. The time series list contains the time series identifiers, as well as maximum and minimum values of the item along with the associated times that the maximum and minimum occurred in the time series. This window is equipped with fast search facilities and the group selection/de-selection tools. An individual time series is selected by selecting the check box in front of the time series ID. Multiple time series can be selected for viewing.

The selected time series can be presented as a data list ('Show values') or as a graph ('Draw graph').

Figure 5.3   Example of the time series list window (from MOUSE, water level in nodes)

You can export your time series data to Clipboard by activating a 'Copy Values' option on the local menu in a time series graphical window.

5.2   Working with the Local Menu in the Graphical View

The time series local menu is accessed by a click on the right Mouse button, while in an active time series window.
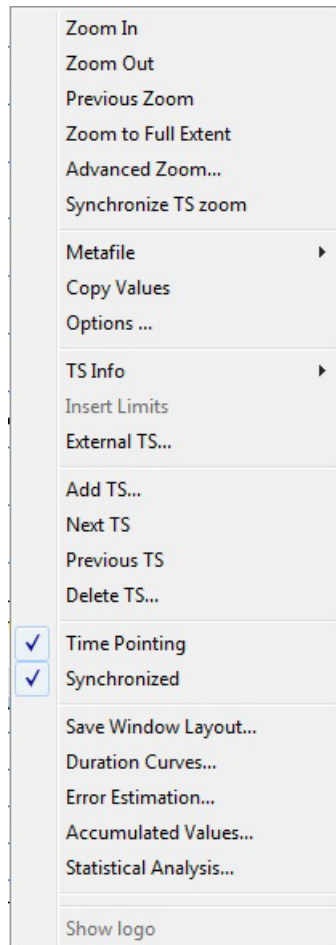
Figure 5.4   Time series local menu

This local menu contains the following facilities:

- the zooming facility, the 'Synchronize TS zoom' will make other open TS windows apply the same zoom as in the active window

- buttons for exporting the time series graphics to the Clipboard and to the metafile (*.EMF). It is possible to copy the values of the TS to the clipboard and then paste them into e.g. Excel
- the options for the graphical TS presentation
- Item info tool, displays information describing the node where the TS is displayed, e.g. min and max values are given for the TS.
- External TS, opens the External Time Series dialog allowing to import and export time series data
- Add/Delete tools, i.e. adding TS by pointing on the horizontal plan, displaying next and previous TS (defined as the order the results are stored in the binary result file), 'Redirect TS' will apply the options set for the active TS window to the next TS chosen to display.
- the animation synchronization control,
- Save window layout.
- Duration Curve tool,
- Error Estimation tool,
- Accumulated values

The Zoom and Add/Delete tools are also accessible via the toolbar.

5.3 Working with the Local Menu

The 'Options' facility allows you to control various items:

- Grids (toggles on/off the display of grid)
- Units (toggles on/off the display of units)
- Legend (toggles on/off the display of legends)
- Decimals (controls the number of decimals (max. 10) to be copied when copying the values of the TS to the clipboard)
- Font (controls the fonts used)
- TS Settings (controls various TS settings, see Figure 5.6)
- Information
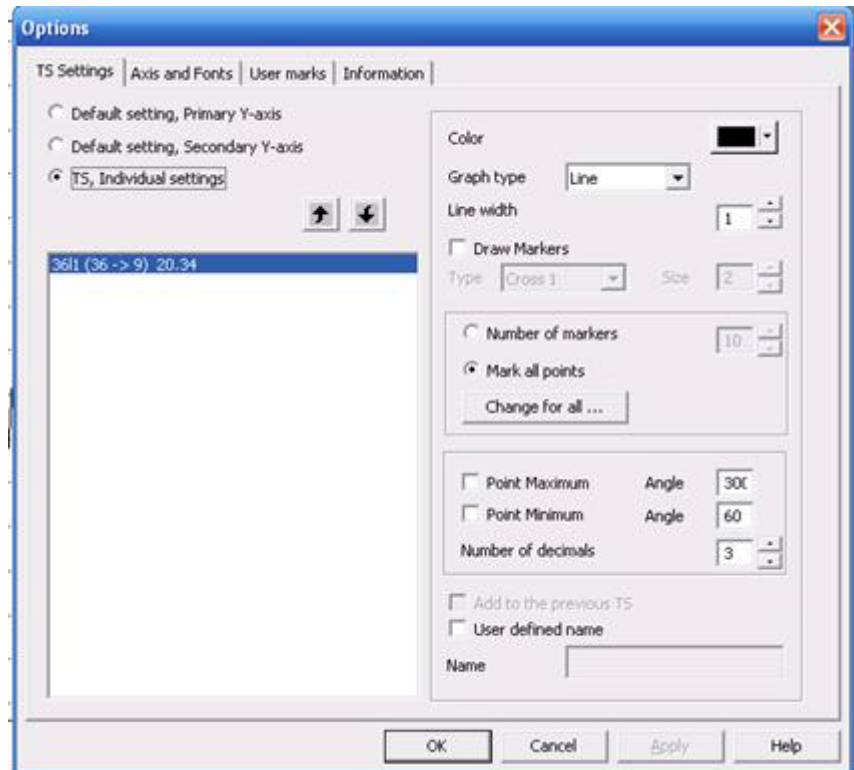
Figure 5.5   The TS Options

On the 'TS Settings' dialog it is possible to change the e.g. the line width, markers to be displayed etc. If more than one TS is displayed it is possible to add a TS to the previous TS on the graph. This is done by using the 'Add to the previous TS' tick box when the TS to be added is highlighted.

There are numerous ways to change the display of the graph to suit.
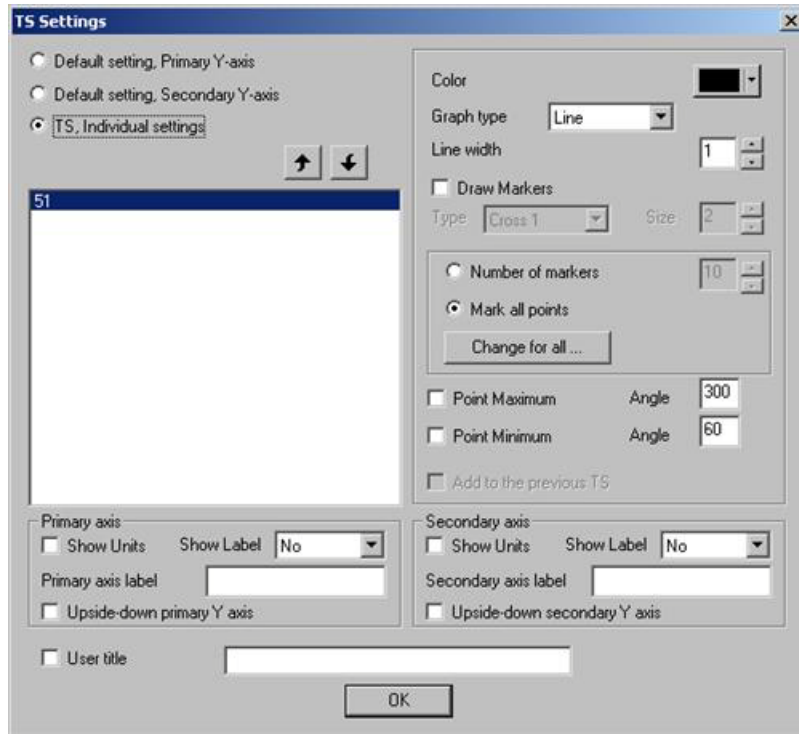
Figure 5.6   TS Settings

5.3.1   Plot – Options – information

For time series plots with discharges the pop-up menu by right-clicking and choosing Options offers a tab page named Information. This enables a label on the plot with the accumulated flow within the displayed flow graph.
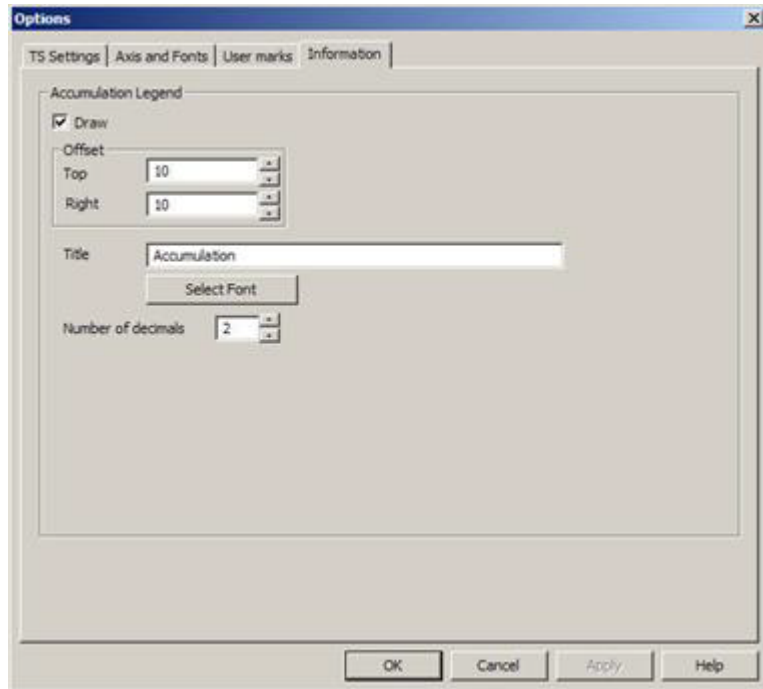
Figure 5.7   Accumulation legend input page

Notice that the accumulated volume is calculated from the part of the flow curve that is visible. The resulting Accumulation legend is shown in Figure 5.8.
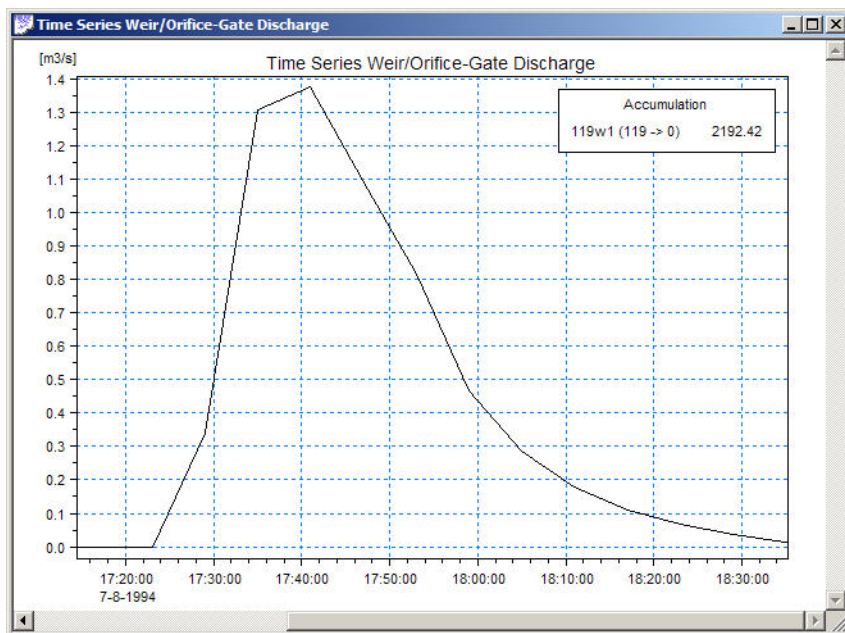
Figure 5.8   Accumulation legend

5.4   Adding Time Series to a Graph

Each time series graph can contain several time series. However, only two different time series types can be presented simultaneously.

The 'Add Item' function of the time series local menu re-activates the time series selection mode. The new item is added to the time series graph upon the selection. If the added item is of a different type, the right (secondary) y-axis will be activated, with the units corresponding to the added item.

If you add too many items to the time series graph you can restore the desired content of the graph using the 'Delete item' function of the local menu.

'Add Item' and 'Delete Item' functions are also available on the tool bar as '+' and '-' icons.

'Next Item' and 'Previous Item' are short-cuts for adding neighboring time series of the same item type to a time series graph. The tape recorder symbols 'forward' and 'backward' on the toolbar activate the same function.

5.5   Getting Detailed Information

Detailed information relating to the selected item is at hand with the 'Info...' function of the local time series menu.

Click the 'Copy All' button, and the time series is put on the clipboard.

5.6   Duration Curves

The Duration Curve function is accessed through the time series local menu. The function allows for the display of several types of duration curves:

- Cumulative time;
- Maximum continuous period;
- Maximum continuous period with specified recovery period.

Before the duration curve is plotted, a dialog for the specification of the relevant parameters for the duration curves is displayed. The parameters are self-explanatory.

Figure 5.9   Specification of the duration curve parameters
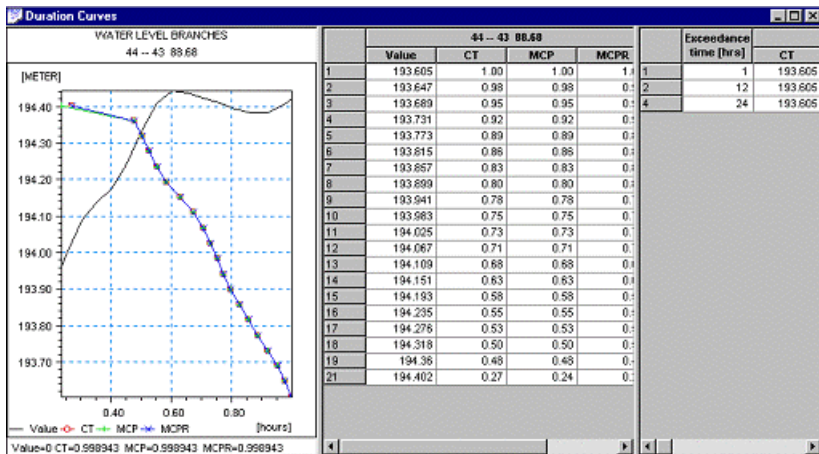
Figure 5.10   Example of the Duration Curve window, with the graphical (left) and numerical section (right)

5.7   Error Estimation

The error estimation function allows for the comparison of simulated and measured data for the statistical evaluation of simulation error.

In order to be able to utilize the function, a Time series plot containing some of the computed variables and a corresponding time series of measured data must be opened. The "Error estimation" function may then be activated from the time series local menu.
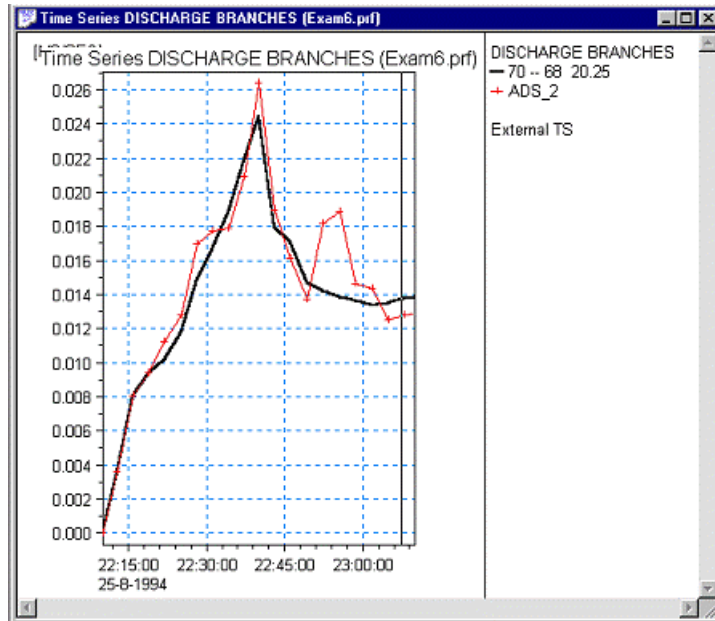
Figure 5.11   Example of a TS plot containing simulated TS and a corresponding time series of measured data
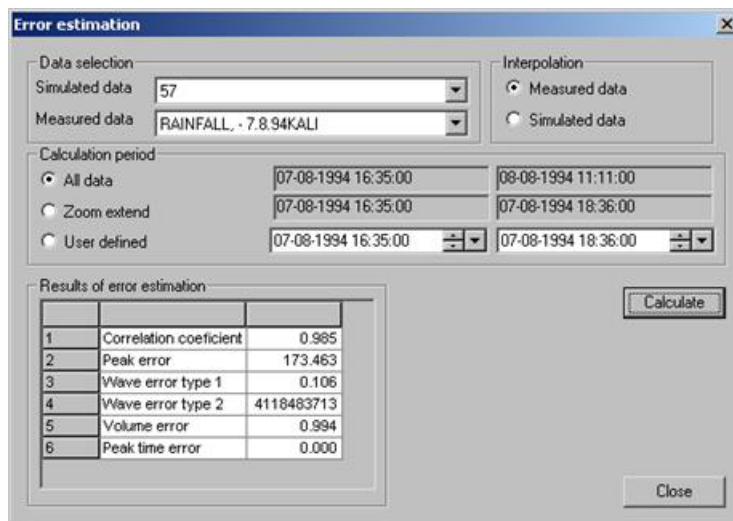
Figure 5.12   An example a corresponding "Error estimation dialog"

Data selection group box: Selection of simulated and measured data takes place using a combo box in which the time series currently shown in the time series view is listed.

Interpolation group box: Some of the error estimations require that the two data sets be defined at the same dates/times. If this is not the case interpola-

tion (linear) in one of the data sets will be activated. The user selects which data set to interpolate.

Calculation period: The user can specify the calculation period in three ways (all data, to the zoom extent, or user defined)

Result of error estimation: Once the Calculate button is pressed, the error estimates are calculated and displayed. For information on how the computation is calculated please refer to the online help for MIKE View.

The "Error estimation" is best used with discharge (flow).

5.8   Accumulated Values

The accumulated values facility can be used for integrating discharge TS either for the entire time period, for zoomed in time period or for a user defined time period.

Figure 5.13   Accumulated values (can only be used with discharge TS)

Note that MIKE View uses the average values from the time series to calculate accumulated values. Accumulated values might therefore differ slightly from accumulated values reported by calculation engines which may use context specific methods for calculating accumulated values. Also, accumulated values in the engines may be calculated based on a different time step than the result files that MIKE View reads.

6 Viewing Q-H Relations

6.1 Selecting the Location

A Q-H graph provides useful information about the flow characteristics of a conduit or river branch at a selected location.

MIKE View creates a Q-H graph by plotting the whole set of computed discharges at the selected location and the water levels at the neighboring H-point (optionally downstream or upstream), within a co-ordinate system defined by the Q and H items on the 'x' and 'y' axes, respectively.

The selection of the Q-H location is similar to the selection of a time series related to a link, i.e. by pointing and clicking on a Horizontal Plan. While in an active Horizontal Plan window, click on the Q-H graph icon on the tool bar, or select the "Plot | Q-H Curve" on the main menu. This will activate the Q-H selection mode. The change of the program mode is indicated by a change of the cursor shape. The selection is possible only when you approach a valid link indicated by the cursor shape changing into an arrow.

Clicking the mouse opens a view, which prompts you to select the Q-point along the link (MOUSE/MIKESWMM), and to switch between the 'upstream' or 'downstream' options for the H-point selection. Upon the confirmation of your choices, a Q-H graph window opens and the active window for further configuration and for viewing. Zooming and panning can be used to look at specific areas of interest.

6.2 Working with the Local Menu
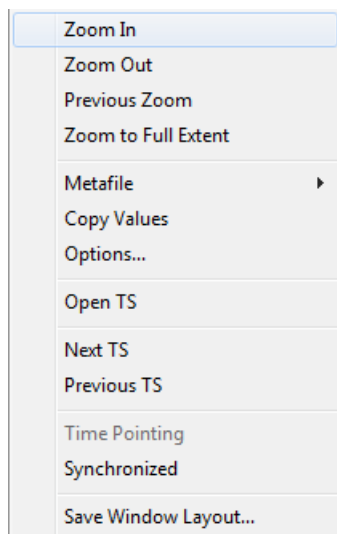
Figure 6.1   The Q-H local menu

The Q-H local menu offers numerous options for fine-tuning your Q-H plot. This menu contains the following:

- the zooming facility,
- buttons for exporting the Q-H graph to the clipboard metafile,
- various presentation options,
- Open Time Series tool,
- Next/Previous Item tool,
- the animation synchronization control,
- tools for exporting time series
- Save Window Layout

Most of these features work similarly to the Longitudinal Profile or the time series graph. The 'Next Item' and 'Previous Item' replace the selected Q-H relation with the neighboring node locations. 'Open TS' automatically opens a time series window with the Q and H time series of the selected Q-H location.

Control of the visual appearance of the Q-H graph is available under "Options" (see Figure 6.2).
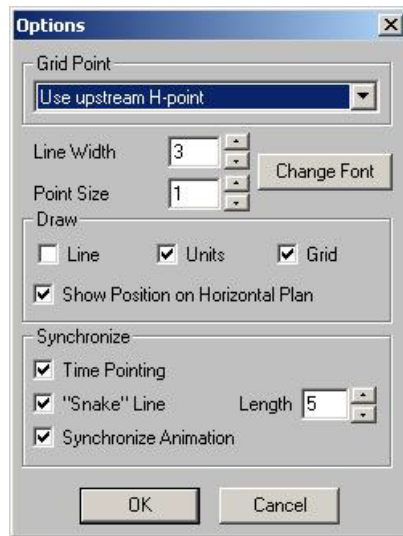
Figure 6.2   Options view for the Q-H graph

The 'snake' will help to follow the dynamics of a presented Q-H relation in case of a scattered Q-H plot (backwater).

7 Operating with Subjects

7.1 Creating Subjects

When working with MIKE View it may become necessary to have several windows opened simultaneously e.g. several longitudinal profiles and several TS windows. As more and more windows are opened it becomes difficult to keep an overview of the available information. The "Subjects" facility aims to overcome this problem by creating several subjects, i.e. several virtual screens. Switching between the different subjects (virtual screens) is accomplished by clicking on different tabular pages in MIKE View. The subjects dialog is activated on the 'View' menu under 'Subjects' (see Figure 7.1)
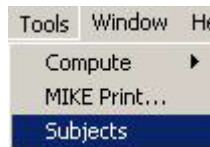
Figure 7.1   The subjects dialog is activated on the Tools menu

In the subjects dialog (see Figure 7.2) it is possible to:

Create, rename and delete subjects

Select/Deselect windows and subjects that the windows should be assigned to

On opening the subjects dialog for the first time, all the open windows at that time will be displayed. If all windows are not subsequently selected to be in a subject group then the all the initially open windows remain on the list of windows in the Subjects dialog. New windows will automatically be made a part of the subject they are opened in. Only one result file may be opened at the time when applying the Subjects facility.
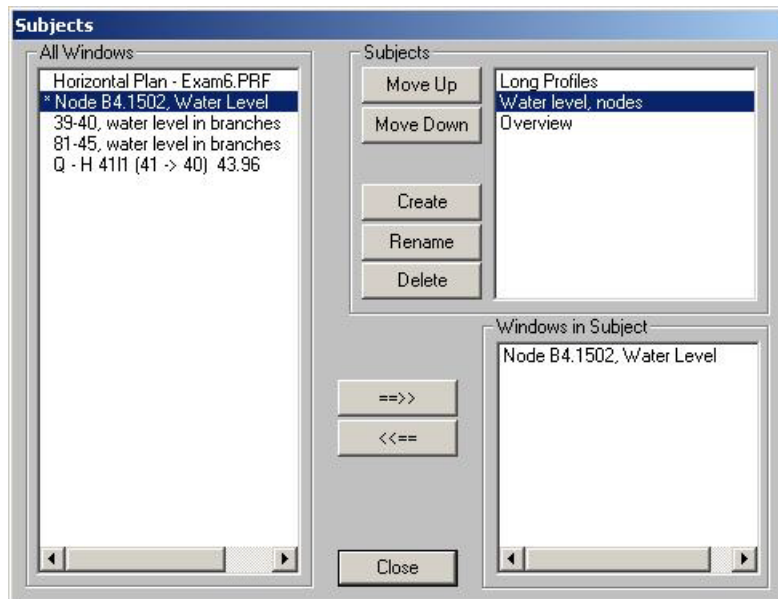
Figure 7.2   Selecting a window and subject to place the window in

In the Subjects dialog pressing the '==>>' button moves the selected window to the selected subject group. Vice versa pressing the '<<==' button will deselect a window from the selected subject. The windows are displayed on the windows list with their title (e.g. if a user defined title has been made this will be displayed). A window can be in several subjects. A star (*) in front of the title indicates when a window is included in one or several subjects. In Figure 7.2 the window with the title 'Node B4.1502, Water Level' is put in the subject 'Water level, nodes'.
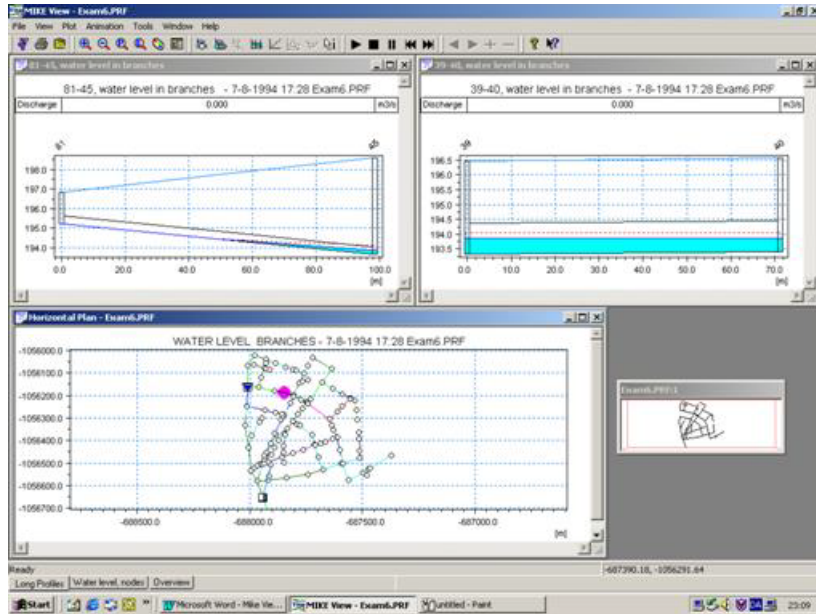
Figure 7.3   Changing between the subjects is done by pressing the different tabular pages (highlighted by the circle on the above figure)

8 Animation of Results

8.1 Animating a Presentation

Time series items can be viewed as animated presentations in Horizontal Plots and in Longitudinal Profiles. Illustration of the system dynamics is created by sequentially displaying time series values within the time interval covered by the loaded data. In a horizontal plot, the items' dynamics is represented by the palette colors or by graduated symbols.

In order to start a horizontal plot animation, the plot type has to be set to 'Animate', and the required item selected. Animation of the Longitudinal Profile is always available i.e. the animation does not need to be pre-set.

Animations are started and controlled by the tape recorder buttons on the toolbar or by the main menu option 'Animation'. Options for starting, stopping with rewind, pausing and stepping forward or backward are accessible in both forward and reverse directions.

Design your custom palette and set Max and Min. values for the range to be displayed in the horizontal plot, for the best fit to your needs.

8.2 Controlling the Animation

The speed of the animation, starting and ending time step, as well as the stepping rate are controlled from the animation 'Advanced Control' dialog box. This dialog can be accessed via the main menu selection "Animation | Advanced".
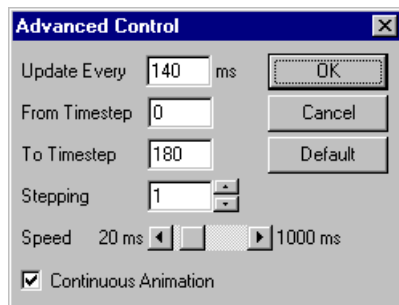
Figure 8.1   Animation Advanced Control dialog

The maximum animation speed and the 'smoothness' of the animation may depend on the power of your computer. A fast machine allows more freedom to adjust the preferred animation speed.

A check in the 'Continuous Animation' box keeps the animation running continuously.

8.3   Synchronizing the Animation

Animations most often run in the currently active Horizontal Plan or Longitudinal Profile window. Optionally, animated presentations in multiple windows can be run simultaneously and fully synchronized.

The main switch for synchronized animation is accessed under the main menu 'Animation' facility. Selecting the 'Synchronize All Plots' or 'Clear All Synchronization' respectively switches 'ON' or 'OFF' the synchronization switches in all opened windows. Additionally, any window can be included or excluded from a synchronized presentation simply by checking the 'Synchronize' box under the 'Options' of a local menu of an active window. In this case the synchronization has to be turned 'ON' in the window from which the animation is started.

The animated presentations of multiple networks (e.g. sewer system and recipient) can be combined in order to visualize the impact. Similarly, the same network can be animated in two windows, displaying the system operation under different conditions.

Note that the time series and Q-H windows can also be included in a synchronized animation. In the time series window, a vertical line moving along the time axis represents the passage of time as the animation progresses. A 'snake' takes the same role on a synchronized Q-H plot.

By dragging the vertical line on a time series plot or the "snake's" head on a Q-H plot, the animation can be rapidly set to a desired time. If the exact required time has not been hit at first attempt, fine-tuning of the time may be accomplished using the 'step forward' and 'step backward' buttons on the tool bar.

9 Exchanging Data

9.1 Exporting Graph Data

MIKE View makes use of the Windows Clipboard for exporting time series data to other applications. The option 'Copy Values' under the local menu is available both in the time series and in the Q-H windows.

All underlying data are copied onto the Clipboard, with the date and time column in the current Windows setting time/date format (Control Panel, Regional Settings), with the short date format applied. The time and date column format can be customized by selecting one of the available Windows options, or by typing your own format.

If you paste the time series data into a spreadsheet, make sure that the local format settings of the current workbook are consistent with the general Windows setting!

9.2 Exporting Windows

The exporting of the MIKE View windows to other applications utilizes the Windows Enhanced Metafile Clipboard. This means that the graphs are copied onto the clipboard as fully vectored graphical files. From the Clipboard, a graph can be pasted into a text processor, or into a compatible drawing program. This opens great possibilities for efficient illustrations of model outputs in a report text. Alternatively, a graph can be saved in a file (*.EMF), for later use.

The 'Metafile' functions are located under the local menu in each MIKE View presentation window.

9.3 Including External Time Series

MIKE View can import and display time series from various external sources. The sources of external time series include:

- ASCII files in specific formats (.TXT)
- MOUSE, MIKE 1D and MIKE 11 time series files
- the Clipboard
- special MIKE View ASCII files (.ETS).

All loaded external time series, independent of the source are placed in the external time series (ETS) RAM buffer, which is then used as a source for the selection of those time series to be presented in the individual MIKE View windows.

The content of the ETS RAM buffer can be saved to a special ASCII file (default extension .ETS) for later reuse. The ETS save option is accessed via the main menu option "Plot | External TS | Save", where a new .ETS file name should be specified before the data are saved.

An already existing .ETS file can be loaded to the ETS RAM buffer. The .ETS file load can either rewrite the existing buffer, or append the data to the current content of the ETS RAM buffer. You pick up the desired option under the main menu selection "Plot | External TS | Load". The successively appended loads accumulate the time series in the ETS buffer.

The 'Edit' option under the main menu selection "Plot | External TS" opens possibilities for loading time series from other listed sources, renaming time series and for getting essential information related to the specific time series (see Figure 9.1).
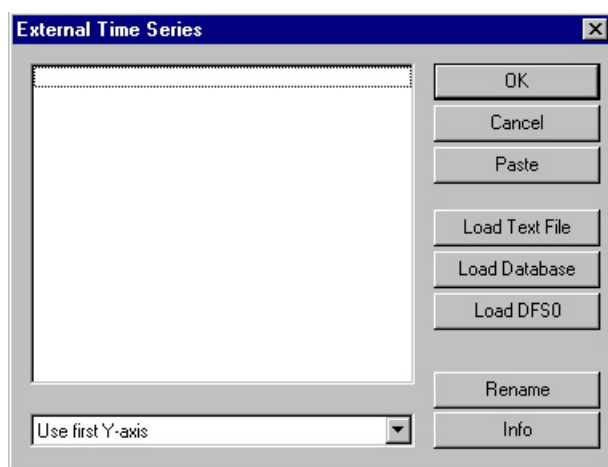
Figure 9.1   External Time Series Buffer 'Edit' menu option

External Time Series 'Edit' is accessible under the time series window local menu option 'External TS'.

Any MOUSE, MIKE 1D or MIKE 11 time series can be loaded into the ETS RAM buffer. The loading function for MOUSE time series is found under the external time series 'Load Database.' option and is composed of two steps. First, the database containing the time series for loading is located in the directory structure, and second, the time series to be loaded are marked.
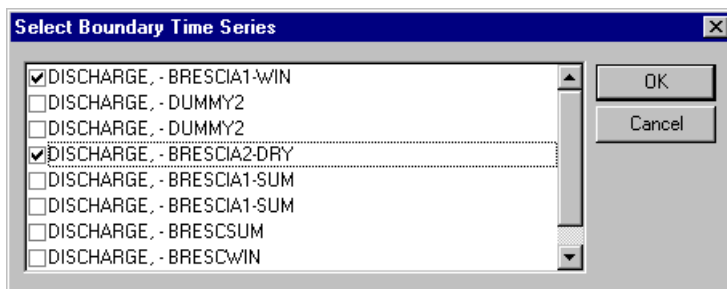
Figure 9.2   Locating and selecting the MOUSE database time series for loading into the ETS buffer

MIKE View can load any type of time series from a MOUSE database.

MIKE View can load time series from ASCII files in a specific format, with a '.TXT' extension. A '.TXT' file can contain only one time series. The '.TXT' format supported for loading into MIKE View is:

YYYY-MM-DD HH:MM:SS <value>

The '.TXT' loading function is found under the external time series 'Load TXT' option.

The 'Paste' function imports the time series data from the Clipboard. Usually, you would copy the measurement data from e.g. spreadsheet program.

MIKE View expects a date and time column format consistent with the current Windows setting, with the date and time written in one column.

A time series loaded from a '.TXT' file or from the Clipboard will be displayed without any scaling, independently of the system of units you use.

10 Tools and Accessories

The menu named Tools provides access to a number of features.
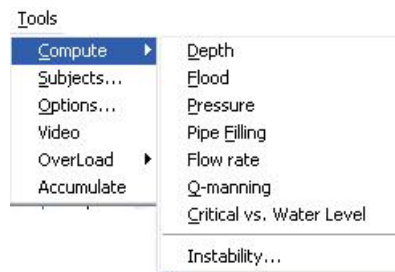
Figure 10.1   The Tools menu

10.1 Computing Derived Items

MIKE View can enhance your presentation by providing you with the additional types of derived output including:

- Depth - computed as a difference of the computed water levels and the bottom levels.
- Flood - computed as a difference of the computed water levels and the ground levels.
- Pressure - computed as a difference of the computed water levels and the conduit ceiling (available for closed conduits only).
- Pipe Filling – computed as the ratio between the water depth and the conduit height (available for closed conduits only).
- Flow rate
- Q-manning - computing the Q as given by the Manning formula for the link (only for prf-files)
- Critical vs. Water level
- Instability – opens up a dialog where the settings for computing the instability is defined

The computed items can be presented in a horizontal plot, on a Longitudinal Profile and as time series graphs. The computed item is automatically added to the list of item types available for the presentation.

Compute your flood, depth, pressure, pipe filling, Q-manning or items by activating the main menu option by choosing the "Tools|Compute|..." facility.
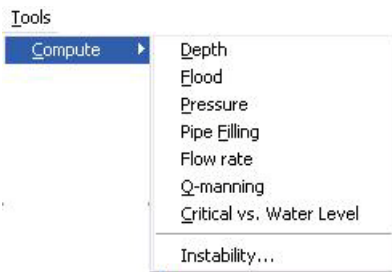
Figure 10.2   The Tools menu allows to computed derived items

10.1.1 Instability

The instability tool identifies time series that fail one of two criteria: either the change in value from one time step to the next exceeds an allowed maximum, or the number of local minimum and maximum within a time window of specified duration exceeds a threshold value. The allowable maximum change in value can be a fraction of the first value (coeff.), an absolute value (maxdiff.), or both. The results from the instability computation may also be written to a text file.
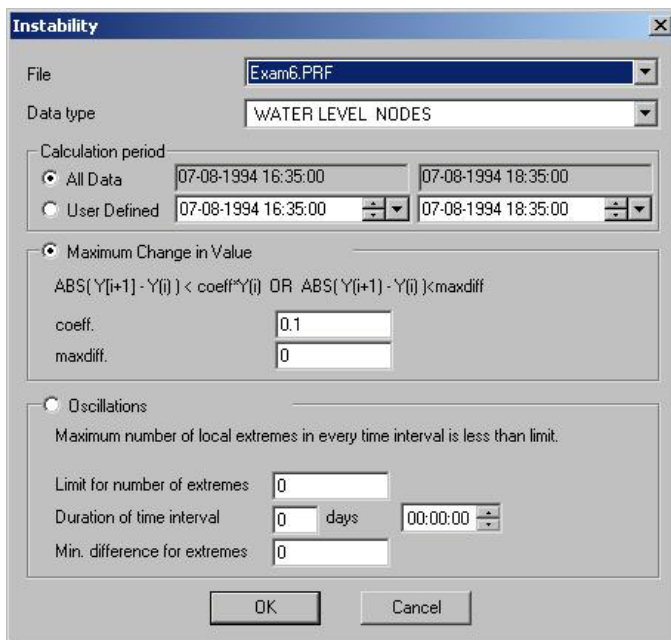
Figure 10.3   The Instability dialog
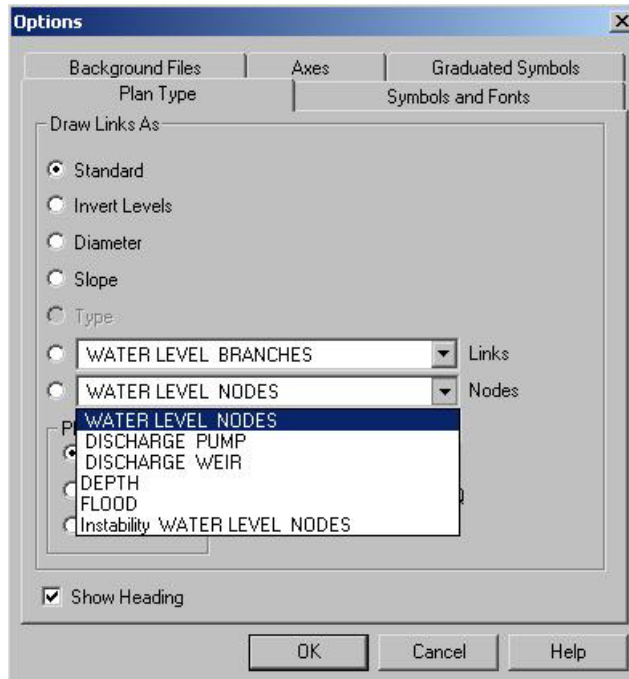
Figure 10.4   The computed derived items can be presented in e.g. a horizontal plot

Oscillations

Figure 10.5   Oscillations in the Instability dialog

Extremes are defined as local maxima or minima.

Limit for number of extremes
This parameter returns a value of True or False when using the tool.

Figure 10.6 shows an example of a time series result file with 3 extremes.

Figure 10.6   Example of results time series

For:

- Limit for number of extremes = 0, Result = True
- Limit for number of extremes = 2, Result = True
- Limit for number of extremes = 3, Result = False

Duration of time interval

Defines the extents of the movable/floating time window. It is moved from one saving time step to the next. Extremes are evaluated/counted for each window.

Time and date format follows the Windows Settings.

If duration of the interval is zero (days + hours), the floating window is the full simulation interval (or user-defined calculation period), and all local extremes are counted.

Min. difference for extremes

This parameter is used by the program for counting extremes.

Very small instabilities (i.e. due to value rounding, etc.) may be filtered using the "Min. difference for extremes" parameter. To illustrate, for the case shown in Figure 10.7, if the minimum difference of extremes = 0.002, only one extreme will be counted. If minimum difference of extremes = 0, 3 extremes will be counted.

Close-up view around a local maximum
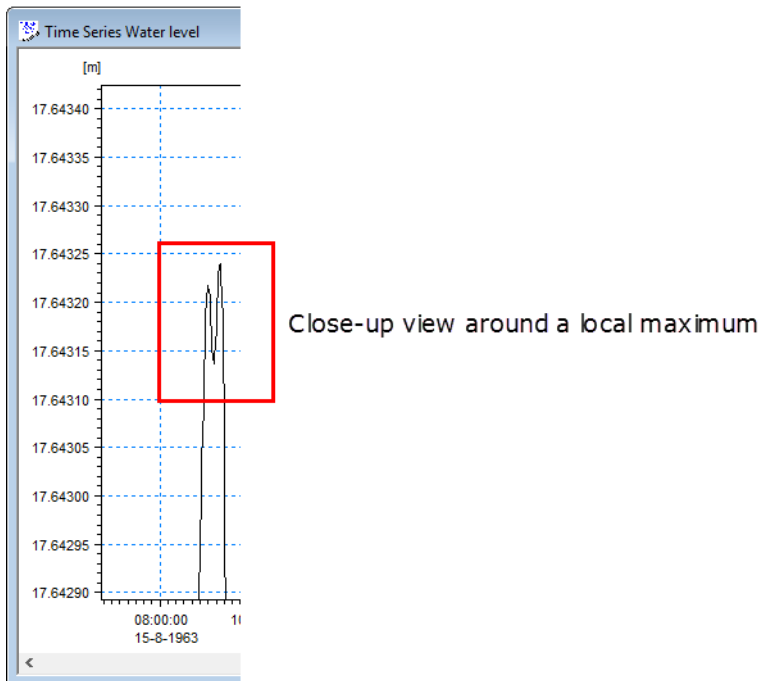
Figure 10.7   Close-up view of a local maximum showing minute instabilities around the maximum value.

Special Notes

If, for example, on closer inspection of the tip/peak of a local extreme, it is as shown in Figure 10.7 wherein the local extreme is "locally constant" - and if the duration of the interval is bigger than length of constant period, only one extreme is counted. The tool does not count the values on the other time steps (i.e. flat/constant period) as extremes as well.
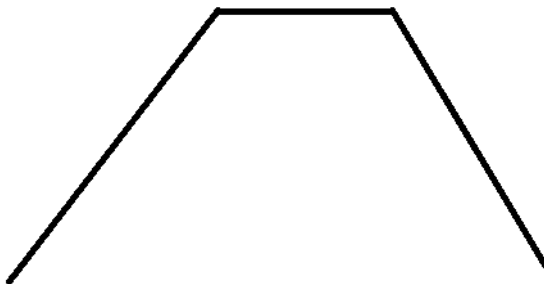
Figure 10.8   Close-up view of a local peak.

10.2   Tools – Options

On the main menu under Tools – Options, the logo tab page provides functionality to display a bitmap (*.bmp) file in the lower left corner of every window displayed in MIKE View (see Figure 10.9 to see the input required). It is typically used to display a company logo and title. The Text fields are displayed right of the logo.
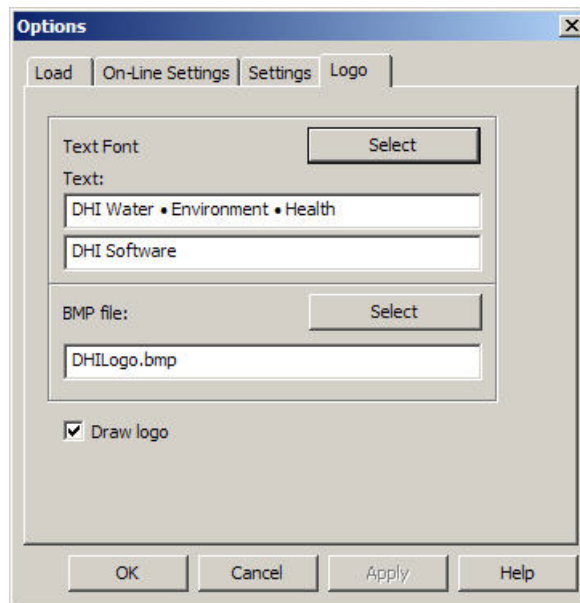
Figure 10.9   The input parameters for the logo

The result is not shown before MIKE View have been closed and started up again. Right clicking on any window gives the option on the pop-up menu to switch on and off the logo. The result of adding a logo to MIKE View can look like shown in Figure 10.10.
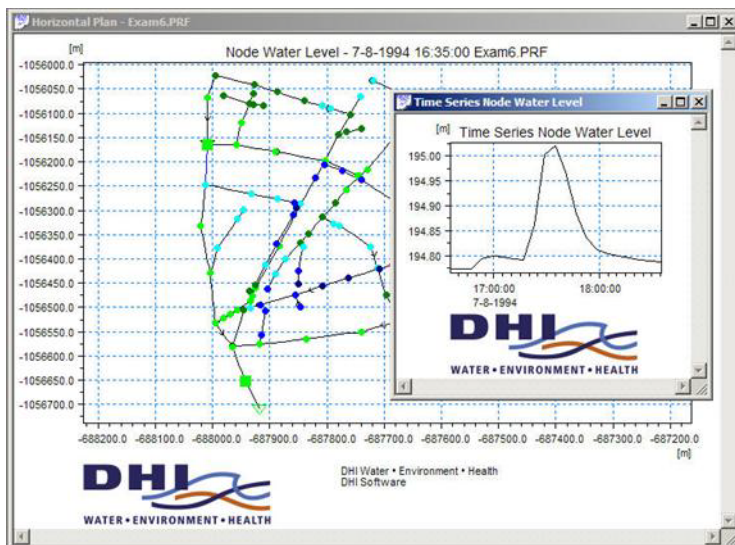
Figure 10.10   Logo included on plots in MIKE View

10.3   Tools – Overload

The Overload functionality on the Tools menu is directed towards showing fixed level lines on various plots. This can be used to e.g. show design levels for acceptable surcharging levels on time series and longitudinal plots. The Save Channels Information saves a text file that can easily be edited. When the file has been saved, the contents looks like shown in Figure 10.11.

Figure 10.11   Channel information file saved from MIKE View

When adding three new columns to the file it is possible to give information on the line number (it is possible to have several lines on a plot), the level at which the line occurs and the legend text for the line. The columns are tab delimited. In the example shown in Figure 10.12 two lines have been added to water level plots of the manhole named "50".

Figure 10.12   Added columns in channel information file
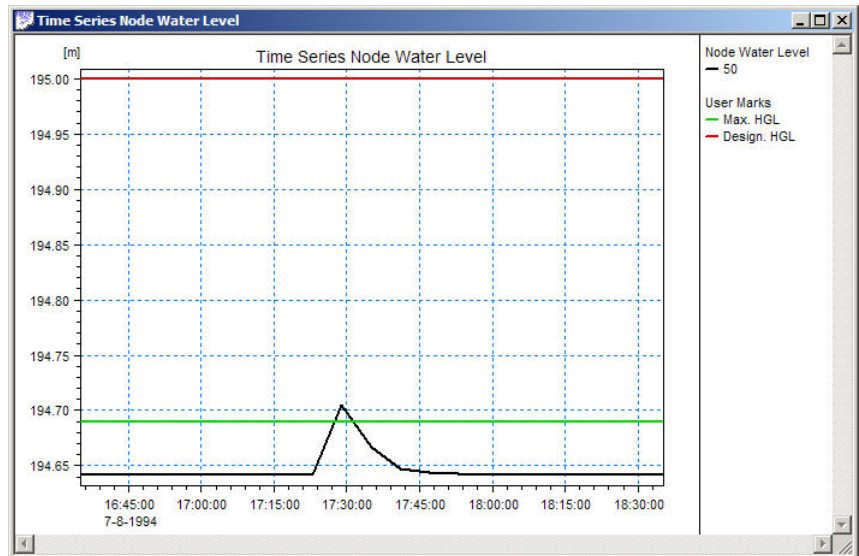
Figure 10.13   User marks shown on a water level plot

After loading the file right click on the plot and click "Insert Limits". The lines appear on time series plots under Options as User marks where colours etc. can be changed. Longitudinal profiles do not allow for adjustment of the lines colours, thickness, etc.

10.4   Tools – Accumulate

The Tools Accumulate functionality (only available for crf- and prf-files) enables aggregation of a result file to accumulated and average values. The tool produces a new entry in the various lists of data. The result appears as if it were a new result file named AccumExam6.prf if the example in Figure 10.14 is used.
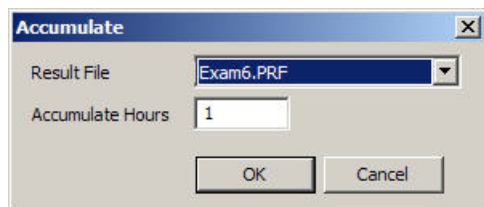
Figure 10.14 Accumulate dialog

The Accumulate hours indicates the time prior to the accumulated value that the value comprises. If e.g. 1 hour is selected results are computed at exactly 1 hour intervals. All water levels are averaged within the chosen period whereas all discharges are accumulated.

10.5   Saving Window Layout

The local menu for each graphical window contains the 'Save Window Layout" option. This facility saves the relevant information about the current active window layout including: data source (result file name, external time series, etc.), the item identifier, the window size and position and graphical layout features in an ASCII file with a default extension (*.WLA).

At any later occasion after the specified result file has been loaded into MIKE View, the window layout can be restored by loading the corresponding *.WLA file. Re-loading of *.WLA files is activated using the Tools main menu option.

10.6   Saving Complete Layout

The local menu for the Horizontal Plot window contains the 'Save Complete Layout" option. This facility allows the relevant information about the current presentation layout such as: data source (result file names, external time series, etc.), the presented item identifiers, window sizes and positions and graphical layout features to be saved in an ASCII file with a default extension (*.CLA).

At any later occasion, the presentation can be automatically restored by loading the corresponding *.CLA file. The *.CLA files can be re-loaded using the Tools main menu option.

A presentation layout can be restored with any result file, which is consistent with the result file network used when the presentation was originally created. This opens the possibility for efficient presentations of successive result files, e.g. during the calibration process.

10.7 Changing Units

MOUSE, MIKE 11 and MIKE 1D computations are based on the Systeme International (SI) units. This means that all input files, including the time series from the boundary database actually store the data in SI units. Similarly, the result files store the results in SI units.

However, both the model input and the result presentation user interfaces may be adapted for to accommodate many alternative systems of units.

10.7.1 Changing the units for MOUSE result files

The information used by MIKE View and for a unit description, and for the scaling of the result data is stored in the ASCII files US-UNITS.US and US-UNITS.SI in the bin directory of the MIKE View installation. To have MIKE View display e.g. US units, the file US-UNITS.US should simply be copied to a file named US-UNITS.TXT (also located in the bin directory of MIKE View). Alternatively, if SI units are to be displayed in stead, the US-UNITS.SI should be copied to the US-UNITS.TXT. The units to be displayed in MIKE View are set as MIKE View is initiated, i.e. the US-UNITS.US/SI should be copied to the US-UNITS.TXT file before starting MIKE View.

MIKE View is delivered with two standard sets of units: the SI (SI-UNITS.UNI) and the American system (US-UNITS.UNI). You can easily adapt MIKE View to show the results in your own system, fully consistent with the usual practice in the country or the region of the application

The unit configuration files contain the unit information relevant for the system geometry and for each of the result item types as identified by the 'M11 record type' integer codes. The unit information is actually specified for a range of item types all sharing the same unit of measure.

The unit information for the geometry includes the X, Y and Z conversion factors relative to the standard SI unit, offset adjustment, the unit identifier, and the number of decimals to be shown.

The information related to the result items includes the specification of the M11 record type range (low and high boundary), the conversion factor relative to the standard SI unit, unit offset, the unit identifier and the number of decimals.

An example of one units parameter file (US-units) is presented in Figure 10.15.

```
[UNIT_DEFINITION]
[TRANSFORMATION] // one set of transformation parameters
MODULE = 'MIKEVIEW' // name of module
UNITS_NAME = 'US - Units'
[GEOMETRY]
```

```
XY = 0.3048,0,'feet',1 // factor,offset,unit name,decimals
Z  = 0.3048,0,'feet',1
EndSect  //GEOMETRY
[M11_ITEMS]
//RecTypeLow, RecTypeHigh,factor,offset,unit name, decimals
INTERVAL = 100, 199,  0.3048,0, 'feet', 1
INTERVAL = 101, 102,   304.8,0, 'feet', 1
INTERVAL = 106, 106,   304.8,0, 'feet', 1
INTERVAL = 200, 249, 0.02832,0, 'cfs', 3
INTERVAL = 250, 252, 0.02832,0, 'ft3', 3
INTERVAL = 300, 301, 0.30480,0, 'ft/s',3
INTERVAL = 302, 302, 7.05555,0, 'in/h',3
INTERVAL = 303, 304, 25.4000,0, 'in/day',3
INTERVAL = 305, 305, 25.4000,0, 'in/h',3
INTERVAL = 306, 307, 7.05555,0, 'in/h',3
INTERVAL = 308, 310, 25.4000,0, 'in/h',3
INTERVAL = 311, 312, 304.800,0, 'ft',3
INTERVAL = 313, 313, 0.02832,0, 'cfs',3
INTERVAL = 450, 453, 0.55590,32.00,'deg. Farenheit',3
INTERVAL = 600, 600,4882.43,0,'lb/ft2',3
INTERVAL = 800, 800,0.09290,0,'ft2',3
INTERVAL = 802, 802,0.09290,0,'ft2',3
INTERVAL = 1000,1000,1.4882,0,'lb/ft',3
INTERVAL = 1401,1401,0.09290,0,'ft2',3
INTERVAL = 1402,1403,0.30480,0,'ft',3
INTERVAL = 1408,1408,0.09290,0,'ft2',3
EndSect  // M11_ITEMS
EndSect  //TRANSFORMATION
EndSect  //UNIT_DEFINITION
```

Figure 10.15   US-UNITS.PAR file: An example of units configuration

10.7.2   Changing the units for MIKE 11 result files

Launch MIKE VIEW and then go to File|Edit Unit Base Groups…
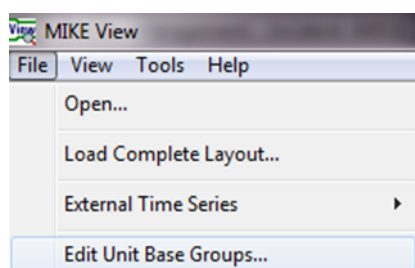
This will bring up the following dialog with a warning letting you know that only some files will be affected by this. Accept by clicking 'OK'.
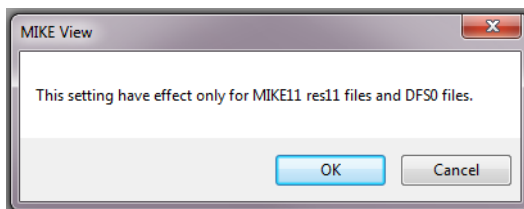
Then the Unit Base Group Editing dialog will appear Figure 10.16. Here you can edit the display for each item or load an existing unit file (*.ubg). After changing the relevant units click the 'Save and Close' button.
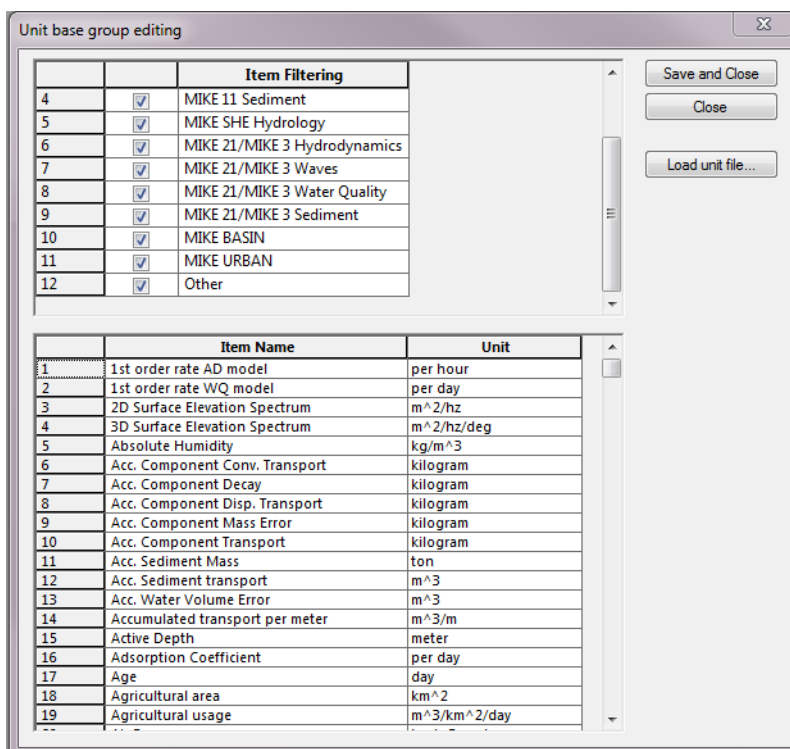
Figure 10.16   Editing the unit for items is done in this dialog

10.8   The MIKEVIEW.INI File

At program start-up, MIKE View initializes information related to various program settings. This information is stored in the ASCII file MIKEVIEW.INI (located in the Windows directory). The file will be created the first time you start MIKE View.

The MIKEVIEW.INI file is updated automatically with the latest program settings each time MIKE View is exited in a regular way. This means that MIKE View 'remembers' the last active settings and that your next MIKE View ses-

sion will be started with the program settings active at the end of your previous session.

The parameters recorded in the MIKEVIEW.INI file include various setting for each of the MIKE View window types, window positions, palette settings, last loaded files, etc.

Some information in the MIKEVIEW.INI file has a static character, i.e. is not subject to the automatic dynamic update. If necessary, such information can be modified to your specific needs by any ASCII editor.

If you change the screen resolution in consecutive MIKE View sessions, you may experience some problems viewing some of your windows from a previous session e.g. some windows may have 'disappeared' from the screen. This typically happens when you switch between the built-in low-resolution screen of your portable PC and a high-resolution external display. If you experience problems of this type, an instant solution would be to simply to delete the MIKEVIEW.INI file to reset the MIKE View settings.

10.9   The Load Geometry Feature

The Load Geometry feature (MOUSE only) found on the main menu under 'File | Load Geometry' can be used for loading in the geometry of a result file e.g. enables a MOUSE Runoff result file to be displayed with the network geographical information. The geometry needs to be saved from within MOUSE before the geometry can be loaded into MIKE View. The geometry is stored in 3 separate files: The node geometry file (*.n_f), the pipe geometry file (*.p_f) and finally the coordinates file (*.xy*). These 3 files must be loaded in separately in MIKE View before the geometry becomes available.

10.10 The Info Mode

The info mode is activated by either clicking the info mode icon on the tool bar or selecting 'View | Info' on the main menu.

The info mode provides instant detailed information of calculated values related to the selected node or link in the Horizontal Plan. You can also move through a time series by clicking on the preview or next time arrow button. The minimum and maximum values for a given item can also be displayed.

Figure 10.17   Example of information given in info mode on a node (MOUSE)

11 Result Verification Tool

11.1 Description of the Tool

The result verification toolbox is designed as a calibration aid and to provide a range of numerical values to quantify the differences between modeled and measured time series. It can also be used to compare options (i.e. between various analyses). Comparisons can be made for the entire set of the time series (TS), a zoomed in portion of the TS or a user-specified time period of the TS. All parameters (locations of the TS, names of the external TS to compare, and the setup of the graphical presentations (colors, types of symbols, fonts etc.)) can be saved in a PFS (Parameter File Standard – DHI standard format for the TXT files) and reused. The tool can be used for any type of the TS included in the result file.

The tool creates up to 6 plots which graphically present different types of analyses:

1. Time Series Plot
Time series plots are useful guides as to the overall 'shape' of the model vs the data. Time series plots are at best a qualitative measure of the goodness of fit, but they are probably the best indication of how the model trends in comparison with the data trends. Time series plots are therefore good for understanding of the process description in the model and how it can represent our concept of the processes in the prototype, as represented by the data. No other plot can demonstrate the time-varying behaviour of the model and data, and how the catchments respond.
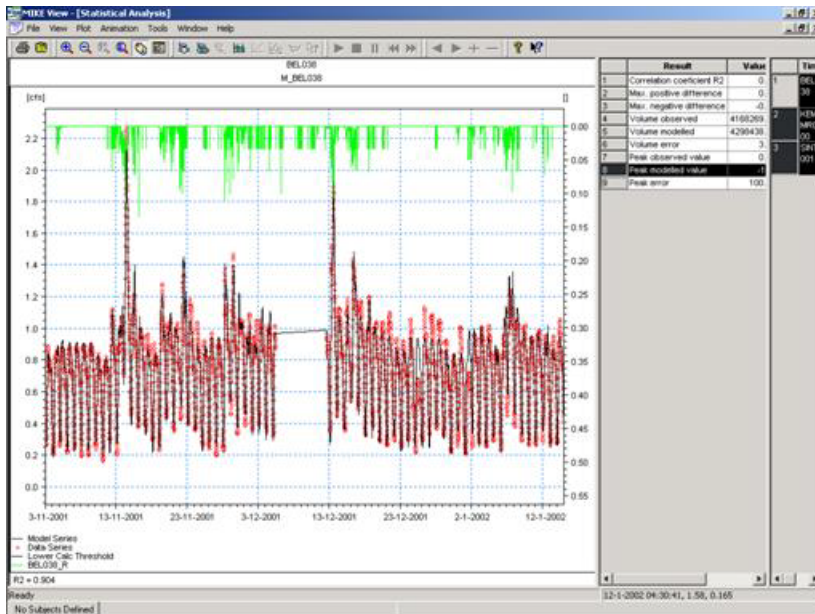
Figure 11.1   The Time Series Plot

2.   Scatter Plot

The scatter graph is a traditional hydrologic analysis plot, where the Vertical Axis is the magnitude of the modeled flow and the horizontal axis is the magnitude of the gauged flow. At the time of each gauged data point, the model result is interpolated and plotted as an X,Y point on the graph. If the model and data are in perfect match, then the point is plotted on a 45 degree line. If the model is low by comparison with the gauge, then the point will be plotted below the 45 degree line.

The red line in the scatter plot, see Figure 11.2, is the line of best fit, and the coefficient of correlation (r2) is calculated. The r2 value is a measure of the degree of intensity of association between the data and the model. A high value (1) indicates that there is a very intense association, while a low value (0) indicates that there is a very low association.

The scatter plot visually shows if there is high or low behaviour in specific flow ranges, and also the width of the scatter gives a qualitative estimate of the amount of variability at a given flow rate or level. The analysis therefore lends much more than just a quantitative r2, it also allows the modeler to observe where the bias occurs in general areas of the modeled behaviour.
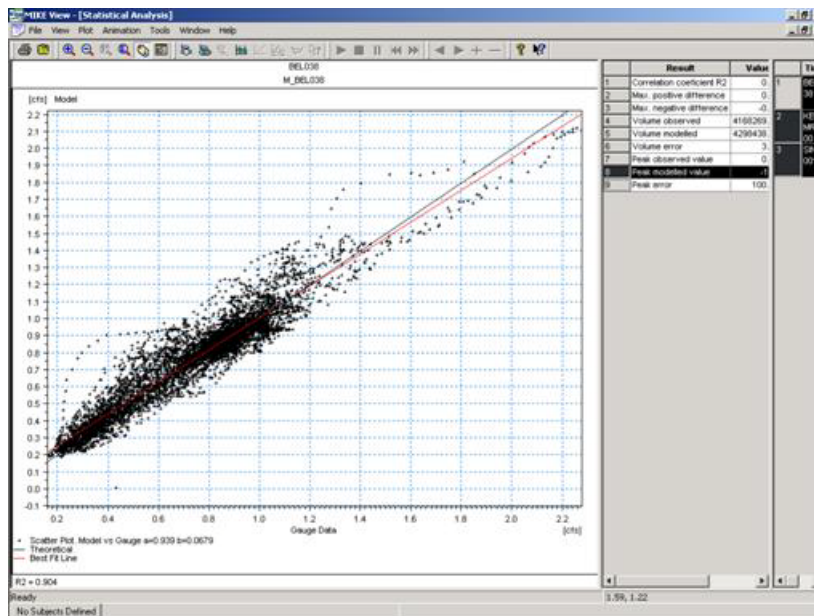
Figure 11.2   The Scatter Plot

3.   Temporal Bias

Residual is defined as the difference between the model and the data at the same point in time, or expressed as (model-data). A plot of the residual against time is shown in Figure 11.3 and is called a temporal bias plot. The vertical axis is the difference between the gauge and the model, and the horizontal axis is time. Bias in the predictions of the model vs the data would be expressed as a trend or shift of the residual which is systematic across the time period of the analysis, hence the plot is called a Temporal Bias Plot. This plot indicates temporal biases for model fitted and/or gauge values of level and flow rate measurements, and it indicates where this temporal bias occurs, if it is a regular or random occurrence. This plot tends to show if there are certain time periods where errors occur. If there are multiple instances of the same behaviour, then it is not due to an isolated event and is something which repeats.

A line of best fit is calculated, and if the slope on the line is zero (i.e. the line of best fit is parallel to the horizontal axis), then there is no trend of bias in the comparison. In the case where the slope is zero but the Y-intercept is non-zero, then there is probably a baseflow missing or else there is too much baseflow.
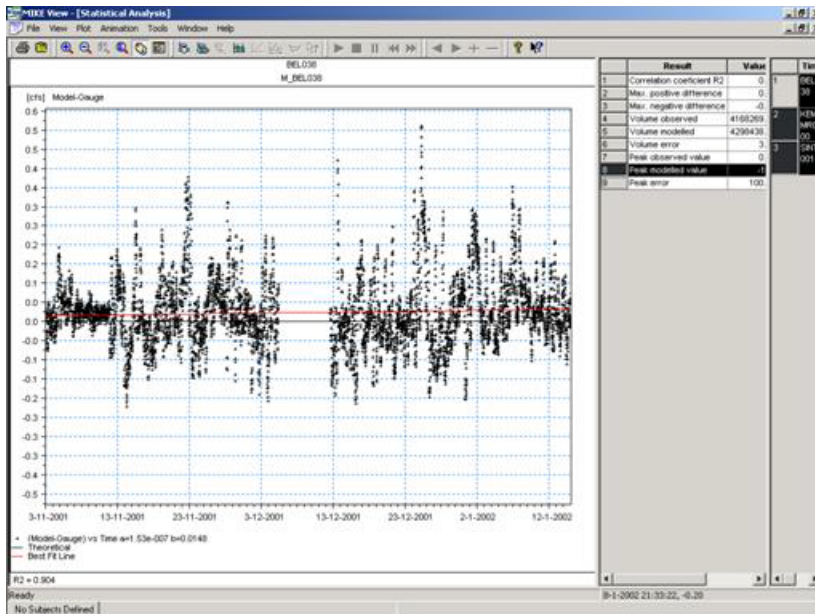
Figure 11.3   The Temporal Residual

4.   Magnitude Bias

The magnitude residual plot is like the scatter graph but normalised to a horizontal axis. By plotting the difference between the model and gauge on the vertical axis and the average of the model and the gauge on the horizontal axis, the line of best fit becomes a horizontal line with Y-Intercept zero. Hence this plot shows more clearly the width of the scatter at certain flow rates or levels, and will signal wide errors at certain hydraulic conditions or when certain thresholds are exceeded.
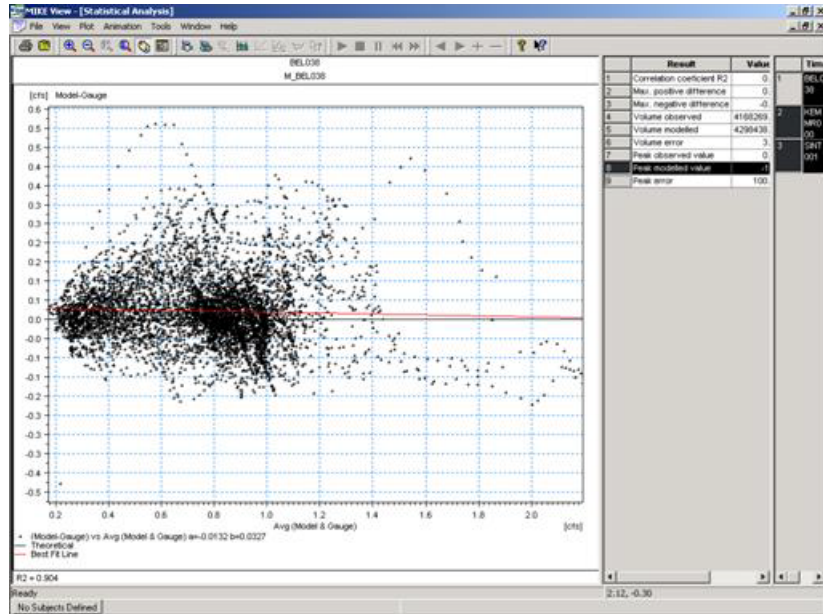
Figure 11.4   The Magnitude Residual Plot

5.   % Error Distribution

The %Error Distribution is used to determine if the difference between the model and the data fall outside a set accuracy of the measurement. This plot does not perform any analysis, but it does give an indication of where the model and data diverge on a percentage basis.

6.   R2 vs Lag Time

This plot is specifically designed to analyse for lags in the data or model which would otherwise provide for a good fit. The analysis is also very useful for determining the travel time of dry weather flows.

11.1.1   Quantitative analyses

A number of parameters are calculated for each plot which provide further quantitative analysis to support the assessment of overall reliability of the model. These are:

- Coefficient of Determination (R2), which is a measure of the goodness of fit
- Maximum Observed Flow over the calculation period
- Maximum Modelled Flow over the calculation period
- Difference in Peak, which is the difference of the above 2 numbers
- Max Positive Difference - the Maximum Positive difference of flows over the period
- Max Negative Difference - Maximum Negative Difference in flows over the period

- Volume Observed, which is the Accumulated volume under observed curve
- Volume Modelled, which is the Accumulated volume under modeled curve
- Volume Error, which is the difference between the above two numbers as a percentage

11.2 Description of the Functionality

In order to be able to utilize the function, a TS plot containing the computed variables and a corresponding time series of measured data must be loaded. If several time series are to be analysed then the plot should be set up so that all the simulated time series are displayed on the same plot. It is not necessary to plot both the measured and modeled time series, but it is necessary to have loaded the measured time series into the 'external' time series, either by loading an *.ETS file or by loading External Time Series. The tool is activated from the Time Series Window using the local menu command Statistical Analysis. Once the tool has been activated, the main dialog appears:
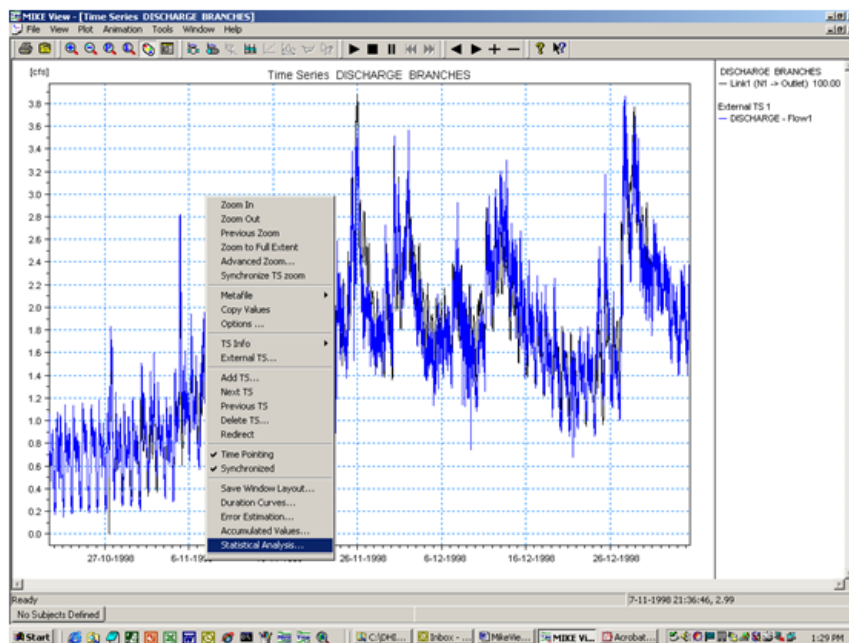
Figure 11.5   The result verification tool is activated by choosing 'Statistical Analysis' on the local menu for time series windows

11.2.1 Main dialog

The following dialog is used for entering the parameters:

Figure 11.6   Main dialog for the Result Verification Tool

There are a number of inputs required for the main dialog:

- Error Estimation Calculation Period: The user can specify the calculation period in three ways (all data, to the zoom extent, or user defined).
- Select Output Graphs: The user can specify the type and number of graphs for the presentation. The number of the output graphs on a page can also be controlled. If the user selects less graphs per page than number of the presented graph, the plots can be scrolled through by pressing the page down and page up Keys once the output graphs and error estimations are calculated and presented.
- Gap: The user can specify if data gaps are removed from the output graphs and the statistical analysis. The user also specifies the length of time between measured data points that is to be considered a data gap. Example: Measured data is 15-minute discharge data. User specifies any data gap over 1 hour is to be removed from the statistical analysis, so if there is a period of missing data for more than 1 hour then the comparison of volume calculations will ignore the 1 hour period where a gap is identified.
- Time Interval for Temporal Bias Plot: The temporal bias plot is divided into a number of time periods, and the values in the time window are used as a population to calculate mean and 5% and 95% confidence intervals. The confidence intervals are based on the assumption that the error is normally distributed.

- R2 vs Lag Time Plot: The user specifies a time period in the menu and the number of lags to be analysed. The tool then shifts the modeled time series both forward and backwards in time compared to the original position, and calculates the R2 for each position. The plot produced is there a plot of number of lags on the horizontal axis (both negative and positive) and the R2 on the vertical axis. The plot can be used to determine if there are fundamental data shifts in the information, or for example it can be useful in dry weather calibration to show the lag in the pipe system.

Plot and analysis characteristics

The user can also specify the number of intervals for the Magnitude Bias and % Error Distribution Plots and whether the gauge uncertainty will be used in the error estimations.

The lower part of the window shows the contents of the analysis. One line is needed for each individual analysis. The following values can be selected:

- Actual Name: Name of the analysis used for navigation and as the Title in presentation.
- TS: The TS name is loaded automatically and cannot be changed. All computed TS that are shown in the Time Series Window are automatically included.
- External TS: User can select any external TS that are currently loaded in MIKEView.
- Presentation: Presentation set containing user specified graphical parameters.
- Use Threshold: User can select if a threshold value is used.
- Threshold Value: User defined threshold value. All values below threshold value in measured TS and in computed TS are eliminated from error estimation.
- Presentation: Presentation set containing user specified graphical parameters.
- Secondary TS Axis: User can specify a secondary TS to be plotted on the time series graph using the primary axis, secondary axis or opposite secondary axis.
- Secondary TS: User can specify a secondary TS located in result file.
- Secondary External TS: User can specify an external secondary TS.
- Uncertainty Type: %Relative is used for discharge, i.e. the gauge is 80% reliable, and Absolute is used for levels, i.e. the measurement is +/- 0.25 meters.
- Uncertainty Value: User can specify the uncertainty value of the gauge.

Buttons in the main dialog have following functionality:

- Load PFS: Loads a saved list of the error estimation and presentation formats from a *.PFS (text) file.
- Save PFS: Saves all error estimation and presentation formats to the *.PFS file.

- Add: Adds all computed TS plotted in the Time Series Window to the list of analysis. The first external TS and default presentation are selected.
- Presentations: Opens editor for creating/editing of the output graph presentations.
- Calculate: Calculates the data for the output graphs and error estimation values for all selected TS.
- Close: Closes the dialog.

11.2.2 Presentation set dialog

The presentation dialog is opened by the button "Presentations" from the main dialog.
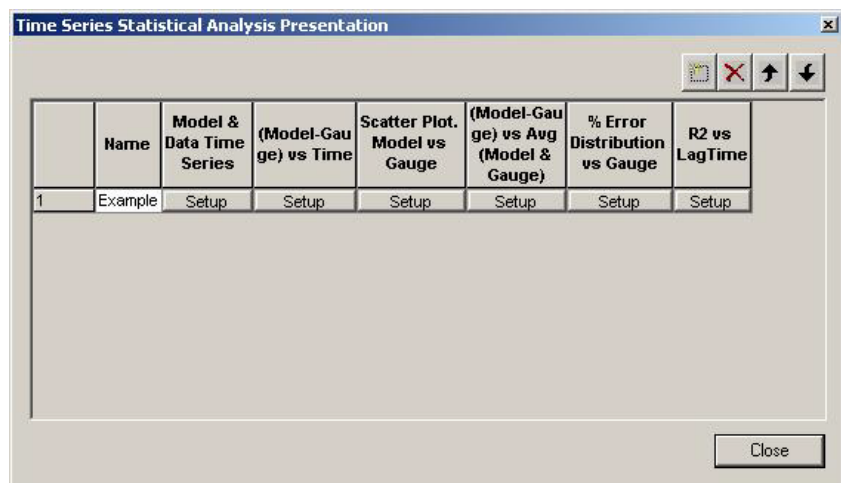
Figure 11.7   The Presentation Dialog

User can add, remove presentations using the standard MIKEView icons. The parameters for the individual presentation areas can be edited after pressing the button "Setup".
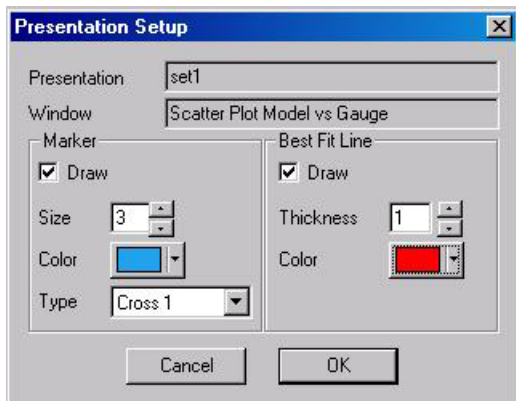
Figure 11.8   The parameters for the individual presentation areas can be edited from the setup dialog

In this dialog user can select the size, type and the color of the markers for the data points and color and thickness of the best fit line.

11.2.3 Presentation of the results

The presentation window contains three parts:

- Graphical presentation window
- Error estimation window
- Navigation window

The graphical presentation contains each of the output graphs selected by the user. The second window contains the error estimations calculated when comparing the computed TS to an external TS. Individual analysis can be selected by selecting the name of the analysis in navigation window.

Graphical presentation window

The six graphical presentations are described below:

1. Model and Data TS vs Time: Computed TS and external TS are plotted on the y-axis and time is plotted on the x-axis. A secondary TS can be plotted on the primary y-axis, a secondary y-axis or on an opposite secondary y-axis.
2. Temporal Bias: The difference between the external TS and the computed TS are plotted on the y-axis and time is plotted on the x-axis. The best-fit line with 5% and 95 % confidence levels are also plotted. The temporal bias plot can be used to determine visually when in time the model and gauge are matching or not matching.
3. Scatter Plot: Computed TS is plotted on the y-axis and the external TS is plotted on the x-axis. The best-fit line for the plotted TS and the "perfect calibration" best-fit line are also plotted in the scatter graph.

4. Magnitude Bias Plot: The difference between the external TS and the computed TS are plotted on the y-axis and the average of the external TS and the computed TS are plotted on the x-axis. The best-fit line with 5% and 95 % confidence levels are also plotted. The magnitude bias plot can be used to determine at what magnitude the model and gauge are matching or not matching.

5. % Error Distribution: The %Error Distribution is plotted on the y-axis and the external TS is plotted on the x-axis. The best-fit line with 5% and 95% confidence levels are also plotted. The %Error distribution shows the external TS data points that lay outside and inside the user specified uncertainty range.

6. R2 vs Lag Time: The lag is plotted on the X axis (not time, the X axis is integer numbers of lags) and the R2 for each lag is plotted on the Y axis. If a threshold is selected for the plot, the lag is undertaken for all points, points above the threshold and points below the threshold. This means that lag analysis can be done for both wet and dry weather points in the same plot.

Error estimation window

This window include following error estimation parameters:

- Correlation coefficient: A measure of the interdependence between the measured data and the model (Reported as R2).
- Max. Positive Difference: Maximum positive difference between computed TS and the external TS.
- Max. Negative Difference: Maximum negative difference between computed TS and external TS.
- Volume Observed: Volume of external TS.
- Volume Modeled: Volume of computed TS.
- Volume Error: Percent error between volume observed and volume modeled.
- Peak Observed Value: Maximum value in external TS.
- Peak Modeled Value: Maximum value in computed TS.
- Peak error: Percent error between peak observed value and peak modeled value.
- Contents of the error estimation window can be saved to a comma separated value (*.csv) file and viewed in Microsoft Excel.

Navigation window

This window is used for the selection of the analysis for the presentation.

- Handling of missing or bad data
  Often data series have dropouts or bad or missing data. These data are normally flagged by a special value, or are simply missing in the time series. A technique for undertaking the analysis to accommodate bad or missing data is to implement a threshold value, and for values below this, the analysis will not be undertaken.

- For the time series residual, scatter plot and magnitude residual plots, model/data pairs where the data is below the threshold are not included in the analysis.
- For the volume checks in the tabular data, model node/data pairs where the data value is below the threshold are ignored. If for example the data value is above the threshold and the model data is below the threshold, then the volume error is included in the calculation.

  However, if the data value is below the threshold then the pair of values is ignored in the assessment, regardless of whether the model is above or below the threshold.

An advantage of the plotting is that errors in the data can be easily identified visually, as data errors often show as a set of strange-looking points in the magnitude residual or in the temporal bias plots. Other issues such as spatial variability of rainfall are also easier to justify when looking at the model and data comparison in a variety of ways.

12 MIKE URBAN: MIKE VIEW TUTORIAL

12.1 What Will I Learn?

This tutorial will guide you step by step through the basic features of MIKE View. The result file included in this tutorial is a result file produced with the MOUSE engine.

When working through the tutorial will help you learn to apply most of the important features of MIKE View. Your understanding of MIKE View can also be enhanced with the help of the MIKE View Manual and the MIKE View online Help.

Since SWMM 5 result files need to be converted into MOUSE result files before being loaded into MIKE View the entire tutorial also applies to features that can be used with SWMM 5 data.

12.2 Before You Begin

You can work through this Tutorial without any prior experience with MIKE View. However, before you begin the tutorial you should ensure that MIKE View is properly installed and that you can start the program.

Even though MIKE View is a user-friendly presentation tool for urban drainage systems, rivers and canal networks, the successful use of the program requires some basic understanding of the respective hydraulic engineering areas.

A copy of the result file used by this tutorial is automatically installed on your computer with the MIKE URBAN installation. It is located in the MIKE URBAN Examples directory under CollectionSystemMOUSE\TUTOR1.

12.3 About MOUSE Result Files

MOUSE produces several types of result files, which have one thing in common - they all contain time series data for various variables in the modeled system.

The different result files types are distinguished by their file name extension. Each result file type contains specific types of time series relevant to the processes simulated by the model (e.g. runoff, sewer network flow, river flow, pollution transport, etc).

In MIKE View, the result files are hierarchically divided in two groups:

Files including the network information, i.e. the MOUSE HD result file, *.PRF

Files without the network geographical files, e.g. the MOUSE Runoff result, *.CRF, and the Advection-Dispersion, *.TRF

As MIKE View is a map-based presentation tool, files from the first group establish a platform for result viewing in MIKE View. These files can be loaded into MIKE View through the <Open> function. The files from the second group can be loaded into MIKE View with the <Add> function as they need to be associated with the relevant (already loaded) file from the first group. The 'added' file must be fully consistent with its associated MOUSE file.

Alternatively, it is also possible to save the geometry of a model from within the MOUSE software package and then re-use this information in MIKE View with e.g. the Runoff result file (*.CRF). The model geometry is stored in 3 separate files:

1. the node geometry file (*.n_f)
2. the pipe geometry file (*.p_f)
3. the coordinates file (*.xy*)

An example of when this approach is relevant is, for example, when you want to open a MOUSE Runoff result file. The geometry of the model network is not automatically attached to this type of file and will be missing. The geometry therefore needs to be loaded in separately via 'File | Load Geometry' on the main menu.

12.4   What Are We Going to View

The urban drainage network related to the result file in this tutorial consists of the following elements:

- The urban catchment area: The area is divided into 8 urban sub-catchments. The storm surface runoff from these areas is loaded into the sewer system during rain events
- 14 network nodes: The network consists of 3 different node types: manhole, basin and outlet
- A pump station: The pump station in this example consist of two separate Q-H snail pumps with a total capacity of up to 400 l/s
- 13 network links (conduits, pipes): The network includes three pipe types: circular pipe, egg-shaped pipe and non-standard pipe
- A weir: In this case, the weir diverts some of the flow away from the treatment plant during heavy rainfall events. The excessive water is discharged untreated to the receiving body due to the limited capacity of the treatment plant

We are going to examine the details of the network layout throughout the tutorial exercise.

12.5   Loading Results

Start MIKE View by choosing 'MIKE View' in the 'MOUSE' program group.

If you intend to use MIKE View frequently, make a short-cut and place it on the desktop!

At start-up, MIKE View opens the File Open view. This facility makes it possible to navigate through the accessible drives, making it easy to find the data directory with your result file.

Examine the possible choices of result file types by clicking on the 'Files of Type' field. Select the 'PRF' option, i.e. MOUSE HD result file.

All PRF files available in the TUTOR1 directory are listed. Select the TUTOR1.PRF file.
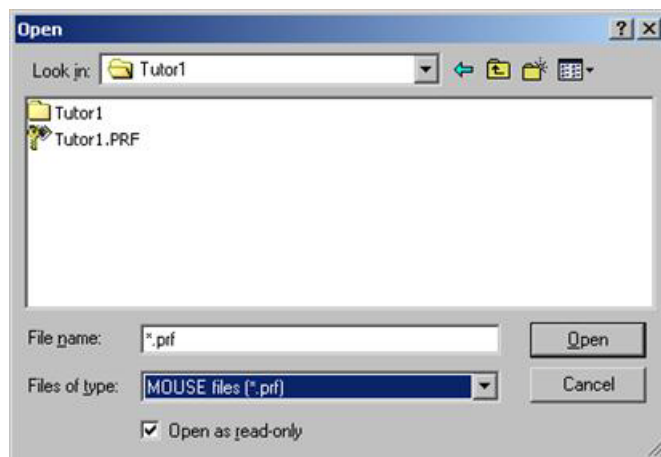
Figure 12.1   Opening the MOUSE result file

Some result files can be rather large, making them difficult to work with even with a fast PC. It is often wise to load the data from a result file selectively, so that your hardware is used in a more efficient way. MIKE View gives you the opportunity to load up only the data types and simulation periods immediately relevant to you, or to reduce the level of time resolution of the displayed data to a more manageable level. You simply switch ON or OFF certain data types, redefine the time interval for loading, and select the appropriate step loading factor to your requirements.
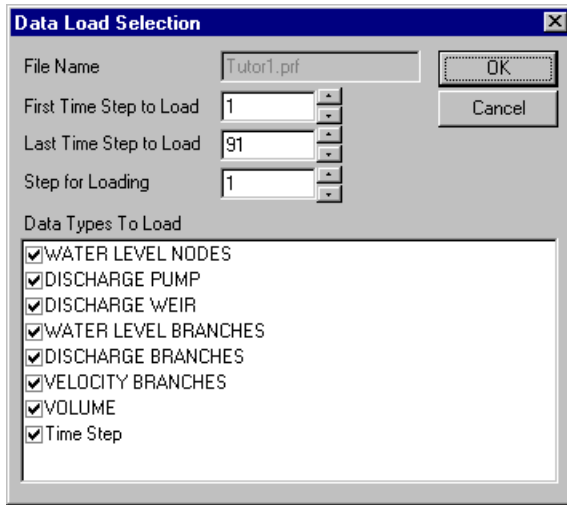
Figure 12.2   The Data Load Selection View. As default, MIKE View proposes the complete result file to be loaded

The tutorial result file is small, and you should simply confirm the default selection by clicking on the <OK> button. After a short while, the file is loaded and MIKE View opens two new windows.

12.6   Exploring the MIKE View Screen

MIKE View has now opened two windows:

- Horizontal Plan window
- Plan Overview window

If your MIKE View 'remembers' some other settings from the previous session, the Palette window may also be on the screen, and the Horizontal Plan will be colored.

The Horizontal Plan window is the central feature of MIKE View. It displays the layout of the sewer network. If you select the Horizontal Plan window it becomes the active window, and the Horizontal Plan toolbar appears under the main menu.

When you move the cursor within the Horizontal Plan window, the coordinates of the current position are displayed in the status bar, located in the bottom left corner. The status bar also provides useful information on the program mode, help text, etc.

The Overview Plan window contains an outline of the network Horizontal Plan. It makes it easier to see which part of the network you are viewing when zooming. Try to arrange the size and position of the MIKE View windows until you get them in a desired layout.

12.7   Viewing System Data

The best way to get a better appreciation of the Horizontal Plan functions is through exploring the various available options.

The Horizontal Plan Options View is activated by clicking on the Options tool in the toolbar, or by activating the Horizontal Plan local menu (press the right mouse button, while in the Horizontal Plan window).
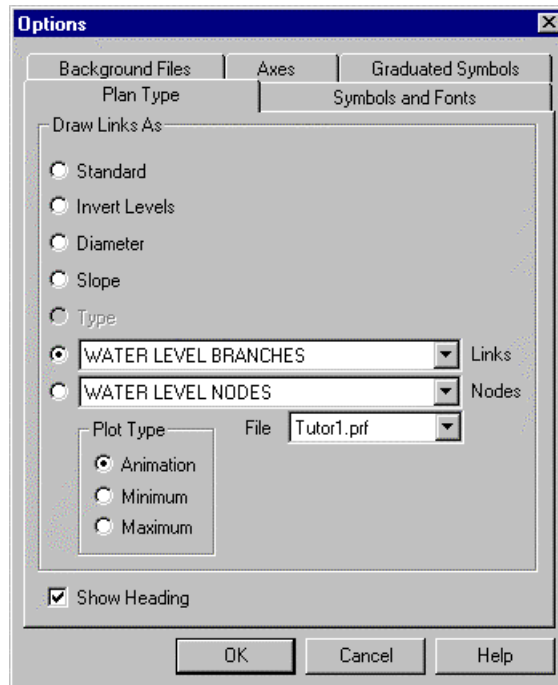
Figure 12.3   Horizontal Plan Options View

The various options available in MIKE View have been designed to help you understand the flow behaviour of the modeled network.

Try the different options from the Plan Type group. Select one of the drawing modes featuring the system information, e.g. select 'Slope'. The display now gives you a first impression of the topographical layout of the network: a palette of colors will describe the slopes of the links. Furthermore, you may select 'Slope' under the 'Fonts and Symbols' group, which will add arrows to the links, indicating the direction of the slopes (works only if 'Slope' has been selected under 'Plan Type'). Similarly, if you select 'Diameter', the color palette will illustrate the pipe dimensions.

By switching various symbols under the 'Symbols and Fonts' group 'on' and 'off', you may improve your knowledge of the network layout. If you switch

'Nodes', 'Links', 'Labels' and 'Pumps' 'ON', you can see that there is a pump located in node B4.1510, pumping water to node B4.1500.

The triangular symbol at node A0.0327 indicates that this is an outlet. The rectangular symbols in nodes B4.1510 and B4.1480 represent the basins, while the circles in all other nodes symbolize ordinary circular manholes. By switching 'Weirs' 'ON', you can identify the existence of one overflow weir in basin B4.1480.

You can adjust the size of symbols and the thickness of links and select your preferred font type and size under the 'Symbols and Fonts' group.

The system can be explored further by selecting and viewing the longitudinal profiles along various selected paths.
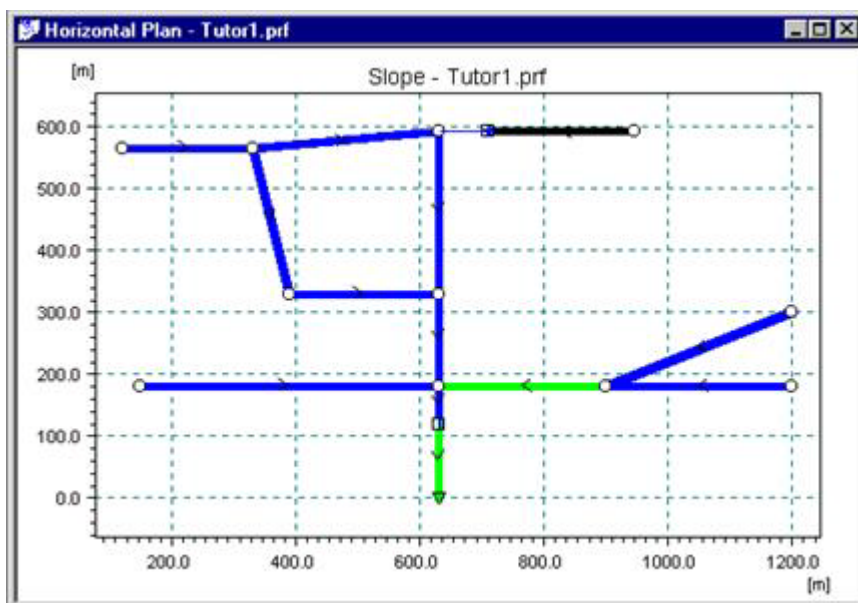
Figure 12.4   The network plan plot with the system elements displayed and with slopes selected as the drawing mode

12.7.1 Zooming

The zoom function is available in all of the MIKE View graphical presentation windows. It is activated by choosing the various zoom tools in the toolbar, or by selecting <Zoom In>, <Zoom Out> or <Zoom Previous> in the local menu.

Activate the local menu by clicking on the right mouse button.

If you choose 'Zoom In' the cursor changes to a magnifying glass symbol. Move it to the location on the plan plot that you would like to be one of the corners of the zoomed-in frame. Then press the left mouse button and drag

the cursor across the Horizontal Plan. The cursor has again changed shape. The zoom frame rectangle appears and indicates the area, which will be included in the zoomed Horizontal Plan window. Continue dragging the cursor until you are satisfied with the area included. Release the mouse button, and the displayed part of the network Horizontal Plan reduces to the framed area.

Note that the zoomed frame is indicated in the Overview as a rectangle.

The scroll bars of the Horizontal Plan window make it possible to 'pan' the zoomed frame over the network area. Also, you can drag the zoom frame rectangle in the Overview window over the network area to the desired position.

If you would like to see the whole network again, use <Zoom Out> or <Zoom Previous>

12.8 Viewing Results

12.8.1 Selecting result variable and plot type

Under the Horizontal Plan <Options><Plan Type> select one of the result variables. You will notice that the result variables are divided into two groups: variables attached to network nodes, and variables attached to links.

If you have more than one result file loaded into MIKE View, you should first select the desired result file. Then select a desired variable from the list of the available variables.

Finally, select a mode for the presentation. The results may be viewed as an animation, as minimum results or as maximum results. Selection of 'Min' or 'Max' results in the lower or upper envelope for the selected variable being displayed as a static Horizontal Plan plot. If you select Animation, then the selected variable will be displayed dynamically as a replay of the model simulation through time. We will choose the animation.

The only file available (i.e. loaded in MIKE View in this exercise) is the TUTOR1.PRF. Choose the group of variables related to links, and select 'Discharge Branches' (i.e. Links). Check to be sure 'Animation' is selected as the plot type.

When you are happy with your selections, click <OK>. MIKE View now displays the discharges in the Horizontal Plan, corresponding to the start time of the simulation.

Note the date and time indicator in the upper part of the Horizontal Plan (the 'Clock'). The 'tape recorder' buttons in the toolbar have been activated, too.

12.8.2   Animation of the horizontal plan

If you click on the 'run' button, the animation will start. You will notice that the time elapses in the 'Clock', and that the colors on the plan plot are changing, following the modeled discharge dynamics.

Try the other animation functions: Pause, Stop, Step Forward and Step Backward. The functionality is the same as controlling a tape recorder.

Under the Main menu <Animation><Advanced>, you can adjust the speed of the animation and some other parameters. Try changing these parameters and note the difference to the display.

12.8.3   Graduated symbols

The result variables associated with the network nodes can be animated using graduated symbols. Under <Options><Graduated Symbols> you can activate and control this type of presentation in the Horizontal Plan.

Change the selection from 'results in links' to 'results in nodes' and choose 'Discharge Weirs' among the available choices. Activate the <Graduated Symbols> group, and select 'Circle' as the graduated symbol shape. Set the 'Min. Size' to 5 and 'Max. Size' to 50. The 'Fill Type' should be set to 'Solid'. Confirm your settings and start the animation.
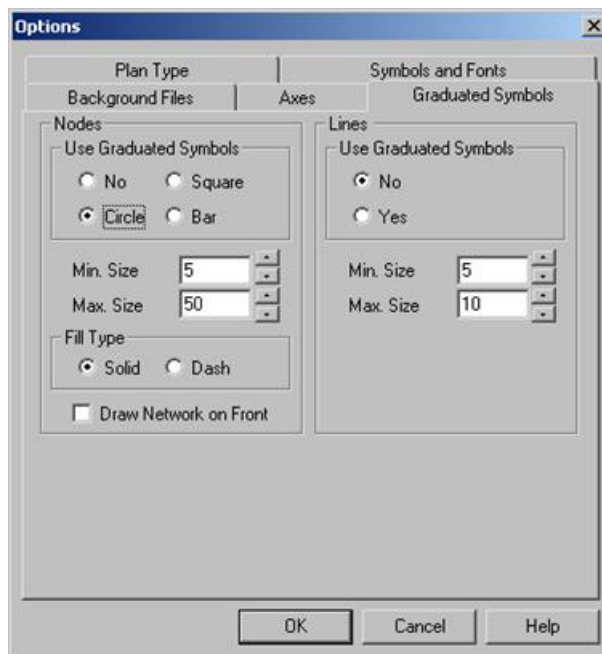

Figure 12.5   The graduated symbols dialog

A graduated symbol appears only at the weir location. Changes to the overflow discharge through time are represented by the size and the color of the circle.

Try 'Graduated Symbols' to show the pump station activity. Use 'Bar' as the symbol shape.

12.8.4 Flooding

MOUSE computes absolute water levels. However, it may be of a particular interest to locate areas where flooding (water level above ground level) occurs in the system. MIKE View can compute flooding as a new variable, which you can view as any other result variable.

In the Main menu choose <Tools><Compute><Flood>. Flooding is now computed for the whole system and added on the list of the available variables (in the Options View).

Go back to the Horizontal Plan window and select 'Flood' in links under <Options> of the local menu. Change the plan type to 'Max' and click <OK>.
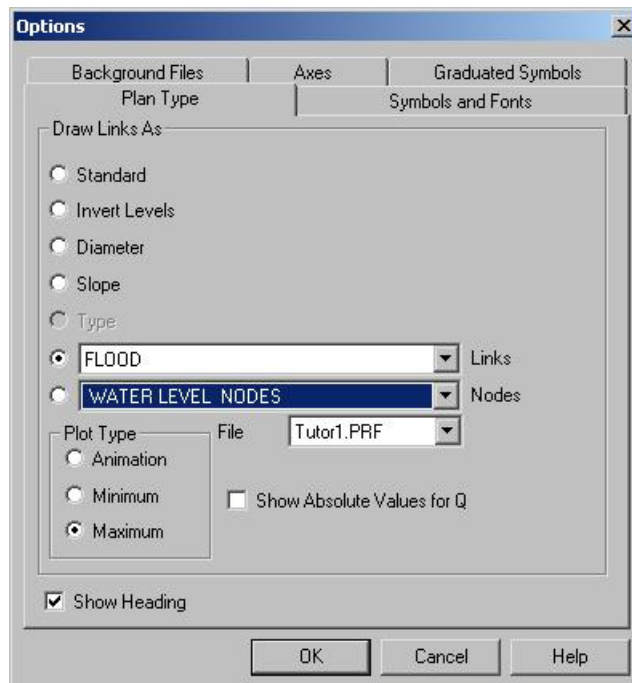

Figure 12.6   Setting the horizontal plan to display the extent of the maximum flood

Your Horizontal Plan now displays the upper envelope of flooding that occurred during the simulated event. You can also view the dynamics of the

flooding by simply switching to the plot type 'Animation' and pressing the 'run' tool.
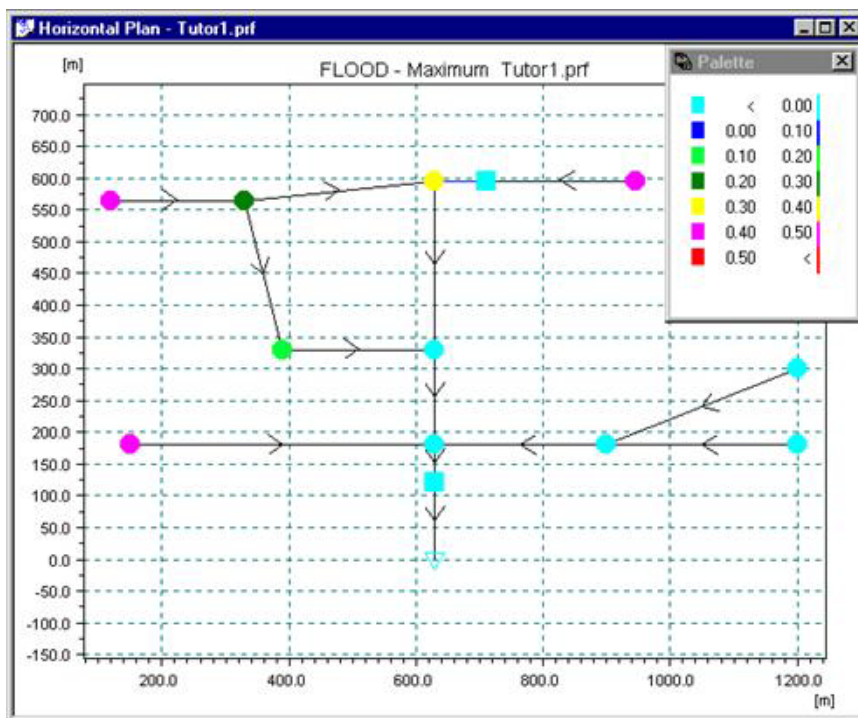
Figure 12.7   Maximum extent of flooding for the simulated event

Compute pressure and depth. Show these variables as 'Max' and as 'Animation'.

12.8.5 Adjusting min/max range and color palette

In reality, you are not interested in seeing the 'negative' flooding displayed, i.e. the situation where the water levels are below the surface levels as specified in the MOUSE model. You can easily avoid plotting negative flooding values by adjusting the 'Min' and 'Max' of the displayed range.

Figure 12.8   Accessing the min/max range of the color palette

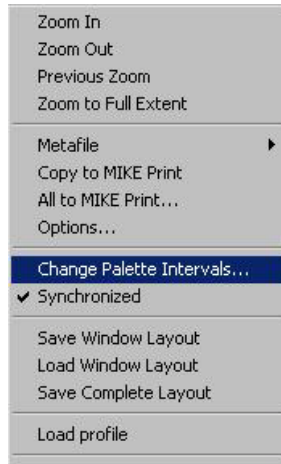Activate the <Change Palette Intervals> function in the Horizontal Plan local menu, and change the 'Min' to zero (if it is not already zero as on Figure 12.9). You may also change the 'Max' value to be a round number. Confirm your settings and have a look at the plan plot. Initially, the whole network is gray, indicating that there is no flooding. When you start the simulation only those parts of the system where the flooding actually occurs will change color, according to the palette.
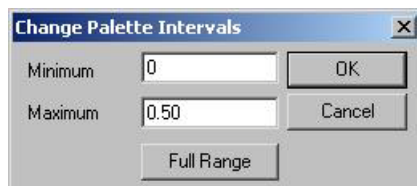

Figure 12.9   Changing the min/max range of the color palette

The palette may be controlled in many ways. Click the right mouse button while in the palette window, and examine the possibilities.

If you do not like the colors on the palette, you may change them with the palette editor. Just double-click on the color you want to change, and the palette editor opens. Similarly, if the range intervals are not suitable for your presentation, double click on the value that you want to edit.

Spend some time exploring the palette options. If you create a custom palette of general applicability, or if you want to be able to recall the same customized palette the next time you work with MIKE View you can save the palette settings into a PAL file. Next time you are using MIKE View you will be able to reload the same PAL file for use with your new result files.

12.8.6 Exporting the horizontal plan

You will often be in a situation where you want to include some of the MIKE View graphics into your text report. This is a simple task.

Your word processor should be a true Windows application with Copy and Paste facilities, e.g. Word. Start the word processor, and open your report document (or create a new one for the exercise).

Return to MIKE View, adjust the Horizontal Plan to fit your needs in the report, and activate the local menu. Then simply click on the <Make Metafile> option. The content of your Horizontal Plan will be copied to the Windows Clipboard.

Switch again to the text editor, position the cursor at the desired location, and activate the standard <Edit><Paste> function. The MIKE View plot is pasted directly into your document as a fully vectored graphic image that may be edited or resized.

This facility works for any graphical window in MIKE View.

12.9 Viewing Results in a Longitudinal Profile

The Horizontal Plan window is the working area from which items may be selected for all the other presentation modes e.g. longitudinal profiles, time series and Q-H relations.

12.9.1 Selecting a longitudinal profile

Let's assume that we want to have a look at the longitudinal profile starting from node B4.1520, through the pumping station, and then down to outlet A0.0327.

Click on the Longitudinal Profile tool in the toolbar and point with the cursor in the vicinity of node B4.1520. When the cursor changes to an arrow, it means that you are near a valid node and can start the selection of the longitudinal profile.

Click on the mouse, and node B4.1520 changes into a large green circle. Continue the selection by clicking at each node (or in the direction) along the desired path. MIKE View paints your selections bold green. When you arrive to the outlet, MIKE View recognizes the end of the path and proposes that the selection be closed. Confirm the profile selection and choose the type of variable that you would like to show in the longitudinal profile. The default selection is 'Water Levels'.
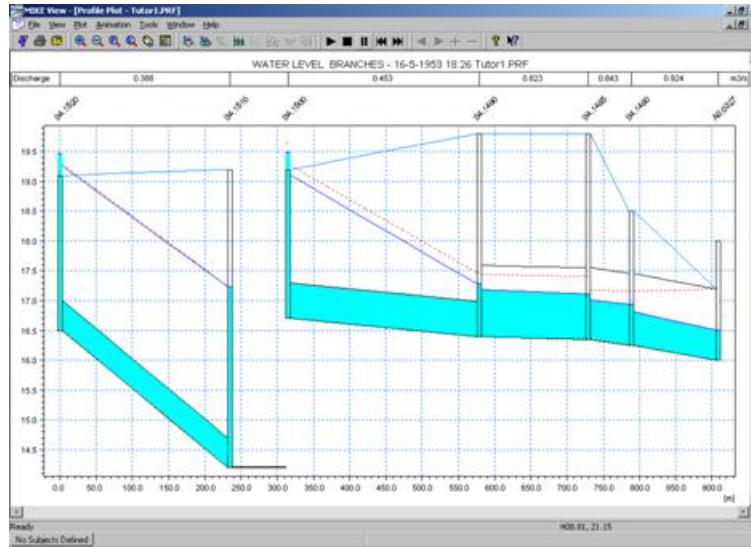
Figure 12.10   The longitudinal profile from B4.1520 to A0.0327

If you have selected an incorrect node, 'deselect' the last selected node by pressing the Shift key and clicking on the mouse button at the same time (or simply press Backspace).

12.9.2   Animating a longitudinal profile

First resize and reposition the Longitudinal Profile window, until you are satisfied with its appearance. You can also, zoom-in, zoom-out and control various display options by opening the Options View from the local menu similar to the Horizontal Plan window.

The animation is started in exactly the same way as in the Horizontal Plan - : just click on the 'Run' tool in the toolbar.

12.9.3   Synchronizing Animations

Should you desire, the animation in two or more windows can be fully synchronized. This means that the animation in each of the open Horizontal Plan and Longitudinal Profile windows will elapse synchronously and that you can follow the system dynamics from several 'angles' simultaneously.

You can only have one Horizontal Plan window opened with each of the network files loaded. But, if you load more network files (e.g. one from MOUSE and the other from MIKE 11), a synchronized animation can also be made for two Horizontal Plan windows.

To make sure that all your windows are 'synchronized' select <Animation><Synchronize All Plots>. Next time you start the animation, all windows will be in motion.

Each of the windows can individually be excluded from the synchronized animation. This switch is found under the local menu of the window.

12.10 Viewing Time Series

Time series graphs are usually the most relevant graphs for the system analyses. MIKE View allows you to plot any of the existing time series from the loaded result files, view them in combination with the measured data, and create time series graphs with all time series combinations.

12.10.1 Adding a result file

Earlier it was mentioned that some result file types can be added to an existing presentation. In our case, it is relevant to combine the view of some flow variables in the network with the runoff hydrographs from the hydrological model.

Select <File><Add> in the main menu, and choose CRF as the file type to be loaded. Then locate the file TUTOR1.CRF in the directory of the TUTOR1 example, and load it into MIKE View.
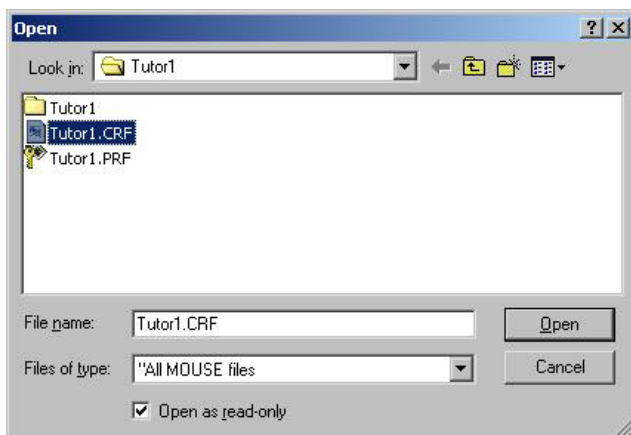

Figure 12.11   Adding the runoff results

View the hydrographs first in the Horizontal Plan. Select the TUTOR1.CRF file for the presentation (Horizontal Plan, local menu: <Options><Plan Type>), and set the runoff as the variable to be presented. Switch on the graduated symbols (circle). Start the animation, and watch the spatial and temporal distribution of the storm runoff load.

12.10.2 Selecting a time series

Let's assume that the overflow hydrograph is of our primary interest. Click the 'TS in Nodes' tool in the toolbar and select 'Discharge Weir' as the variable.
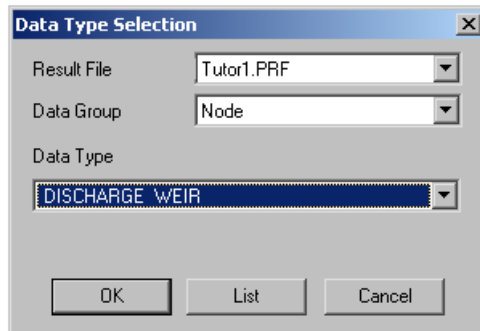

Figure 12.12   Choosing the discharge of the weir as the variable

Select the PRF file and point the cursor to node B4.1480, where the overflow weir is located. At this location the cursor changes to an arrow, indicating that the weir data are available. After clicking with the mouse, MIKE View opens a window with a graph showing the weir flow.

You can control the appearance of this window in many ways with the local menu, in a similar manner as you would with the longitudinal profile window.

Try and select a runoff time series at node B4.1520.

12.10.3 Adding items to a time series graph

You can complement any time series graph with as many other time series as you wish. These may come from the same result file, from a second loaded result file, from a MOUSE/MIKE 11 time series database, from the Clipboard or from a text file.

For purposes of this exercise, let's view the weir flow in combination with a rainfall time series.

Activate the time series window and open the local menu. Then choose <External TS> in order to open the External Time Series View.
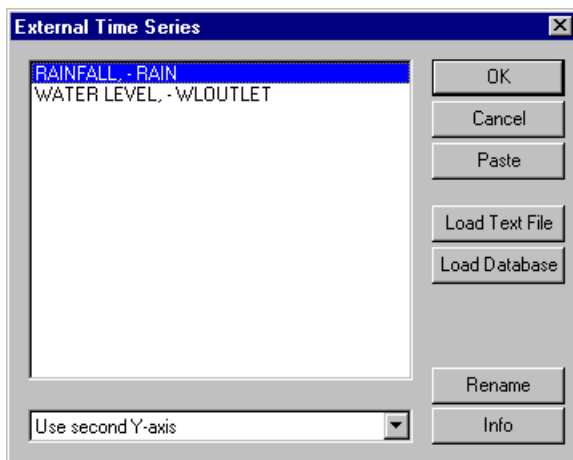
Figure 12.13 The External Time Series View

Click on the <Load Database> button and find the TUTOR1 database (it is actually a subdirectory under the working directory). MIKE View will now display a list of the time series contained in the database. Click on the rain time series and confirm your choice. By this process you have actually loaded the rain time series into MIKE View.

Select the time series to be displayed on the time series graph by clicking with the mouse on the time series identifier. Before confirming your selection with <OK>, select 'Use second y-axis' from the list. This will cause the weir discharge and the rainfall to be presented at different scales on separate y-axes.

13 MIKE 11: MIKE VIEW TUTORIAL

13.1 What Will I Learn?

This tutorial will guide you step by step through the basic features of MIKE View with a focus on viewing MIKE 11 simulation output.

When working through the tutorial, you will get an overview of the operation and application of most of the important features of MIKE View. With the help of the MIKE View Manual and the MIKE View Online Help you will learn how to obtain optimal results from your work with MIKE View.

13.2 Before You Begin

You can work through this Tutorial without any prior experience with MIKE View. Before you begin you should ensure that the MIKE View software is properly installed and you can start the program, otherwise refer to the MIKE View User Manual for installation guidelines.

This session follows naturally after having developed and simulated a MIKE 11 model. You've simulated a MIKE 11 model - now you want to have a look at the model results.

Even though MIKE View is a user-friendly presentation tool for urban drainage systems, rivers and canal networks, the successful usage of the program requires some basic understanding of the respective hydraulic engineering areas.

A copy of the result file used by this tutorial is automatically installed on your computer. It is located in the \MikeZero\Examples\MIKE_11\Vida directory.

13.3 About MIKE 11 Result Files

MIKE 11 produces several different types of result files which all have one thing in common - they contain time series results for various variables in the modeled system. The files all have the extension *.RES11 or *.RES1d. Result files containing additional output parameters as selected in the MIKE 11 HD11 parameter file are found in a result file with the naming convention <prefix>ADD.RES11. These files can all be loaded into MIKE View through the <Open> function. When loaded, the files establish a map-based platform for the result viewing in MIKE View.

From the MIKE View point of view, the result files are hierarchically divided in two groups:

- Files including the MIKE 11 network geographical information and,

- Files with information that can be associated to the geo-graphically referenced cross sections.

In this tutorial we will deal specifically with a MIKE 11 result file for a hydrodynamic (HD) model simulation.

13.4   Starting MIKE View

Start MIKE View by choosing 'MIKE View' in the 'MIKE 11' program group or if you have a short-cut on your desktop by double clicking on the icon.

If you intend to use MIKE View frequently, make a short-cut and place it on the desktop!

13.5   Selecting Data for the Presentation

When the MIKE VIEW program is started, an 'Open File' view opens up. MIKE 11 generates result file types that can be opened from the 'Open File' view with file extension ''RES11'' or ''RES1d''.

By default, the complete information from a result file is selected for loading. This is usually OK, but some large result files may require a selective approach. You can reduce the amount of data to be loaded by excluding some of the available data types, or by truncating the time period to be presented (choosing the appropriate FIRST and LAST time step), or by specifying a load step factor larger than one.

The Data Load Selection view opens up automatically after you choose the right file for loading.
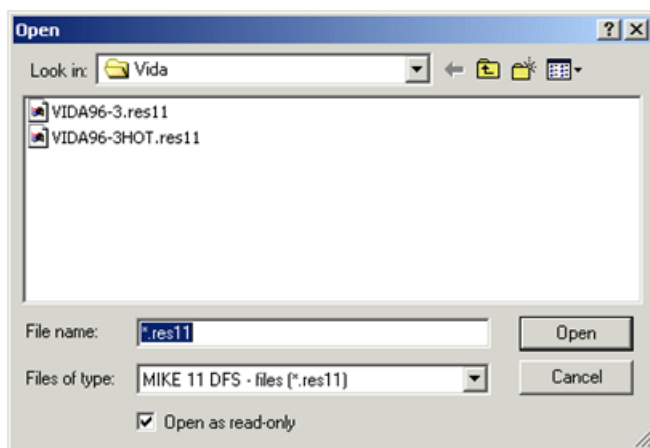

Figure 13.1   Open File Menu

1. At start-up, MIKE View opens the File Open view. This facility makes it possible to navigate through the accessible drives, and you can easily find the data directory with your result file.
2. Examine the possible choices of result file types by clicking on the 'Files of Type' field. Select the 'MIKE 11 files (*.RES11)'or 'MIKE 1D files (*.RES1d) option.
3. All RES11 files available in the Tutorial directory (\MikeZero\Examples\MIKE_11\Vida) are listed. Select a file and select<Open>.
4. This opens the "Data Load Selection" dialog. In this dialog you can de-select unnecessary data types or unwanted simulation periods, or change the time resolution of the displayed data. You simply switch ON or OFF certain data types, redefine the time interval for loading, and select the appropriate step for loading factor.
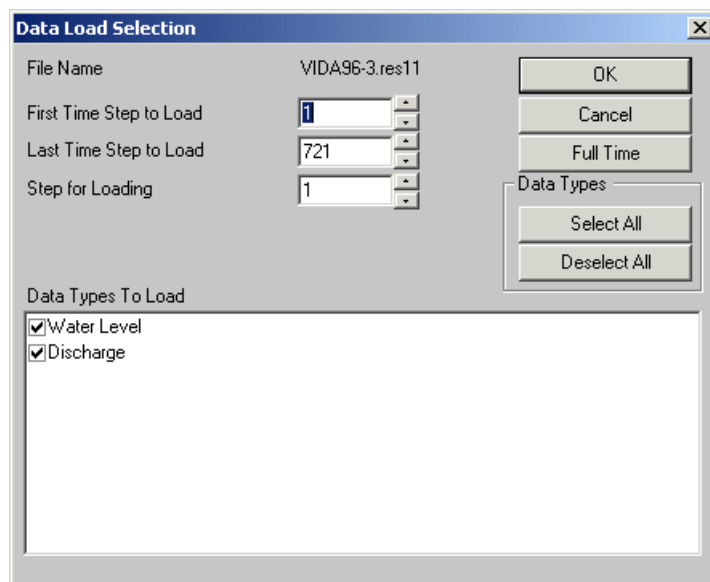

Figure 13.2   Data Selection View

5. The result file is then loaded by selecting the <OK> button. After a short while, the file is loaded and MIKE View opens two new windows of the Horizontal Plan View showing the loaded network.

13.6   Horizontal Plan Options

You will now have two windows opened in MIKE VIEW:

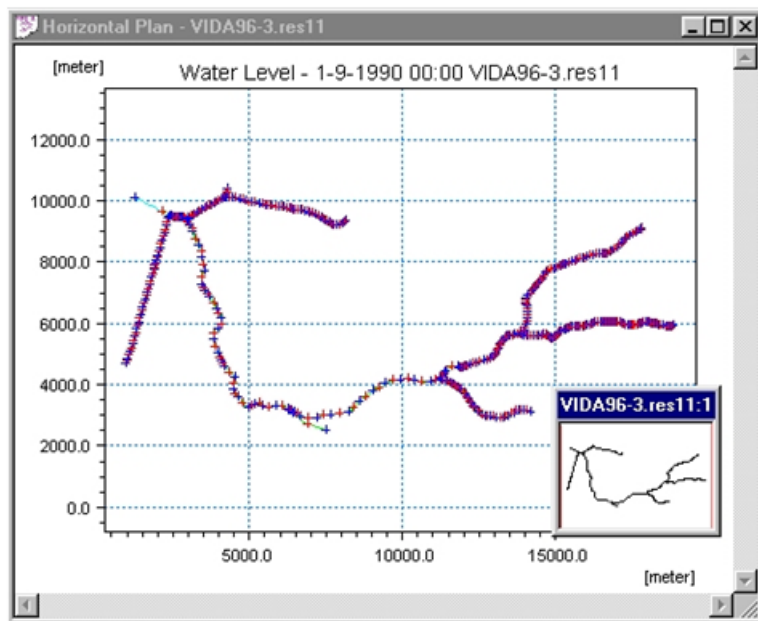•   Horizontal Plan window (HPW)
•   Plan Overview window (POW)

Figure 13.3   Typical Horizontal Plan Window and Plan Overview Window

If your MIKE View 'remembers' some settings from the previous session, the Palette window may also be on the screen, and the Horizontal Plan will be colored.

The HPW is the main window in MIKE View. It displays the layout of the channel network. If you select the HPW it becomes the active window, and the Horizontal Plan toolbar appears under the main menu.

When you move the cursor within the HPW, the co-ordinates of the current position are displayed in the status bar, located in the bottom left corner of MIKE View. The status bar also provides useful information on the program mode, help text, etc.

13.6.1 Horizontal Plan Window options

Plan Type

MIKE View can display various static or dynamic items from the loaded (or added) result files in the HPW. However, only one item can be shown at a time on the HPW. The selection of the presented item and the control of the HPW type options take place in the HPW local menu.

1. Select the HPW local menu Options tool by selecting the icon on the tool-bar, or by pressing the right mouse button, while in the HPW. This opens the HPW Options dialog.
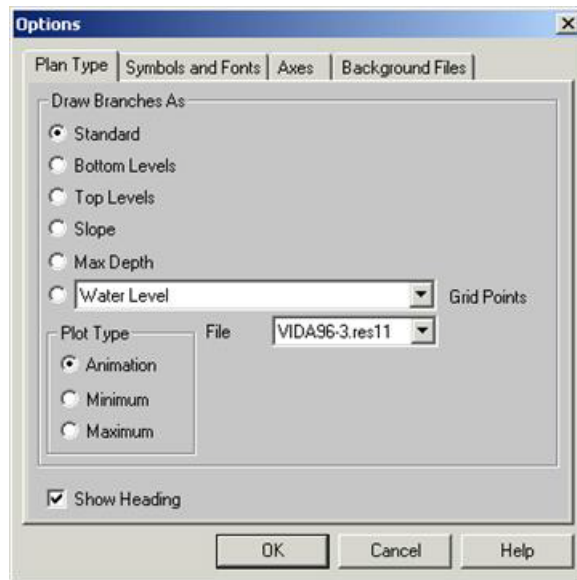

Figure 13.4   Horizontal Plan Window Options – Plan Type Selection

2. Initially the "Plan Type" tab group is active. Check the "Draw Branches As" modes one by one (press OK in between) and observe how this is displayed in the HPW. Use this feature to give you a first impression of the topographical layout of the network: the slopes of the links will be displayed as a palette of colors (we'll work with the palette in later exercise).
3. Open the drop down list in "Grid Points" selection to choose between available data types that can be displayed. These are dynamic data types (time series) and can be animated.

Symbols and Fonts

1. Open the HPW options dialog and select the "Symbols and Fonts" tab group.
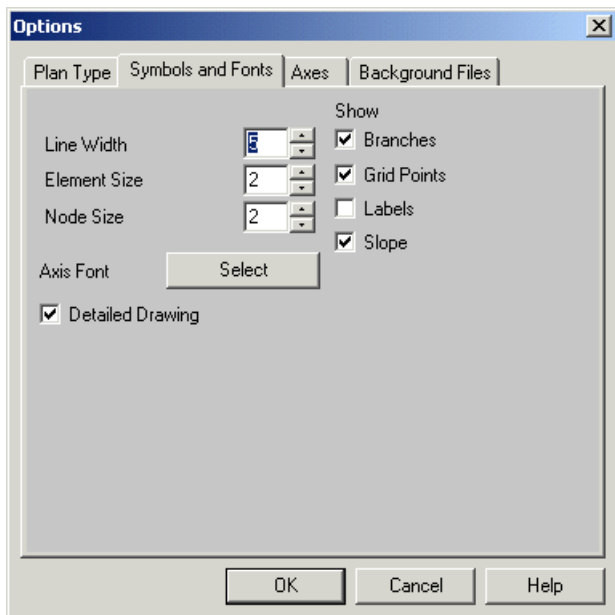

Figure 13.5   Horizontal Plan Window Options – Symbols and Fonts

2. Check display of 'Branches', 'Grid Points' and 'Labels' ON, and press OK. The HPW now displays network and grid points with labels.
3. Select the "Plan Type" tab group and select Draw Links As Slope, Press OK. The HPW now indicates the branch slope defined by color coding of the palette.

Axes

1. Open the HPW options dialog and select the "Axes" tab group. Here you can control display of Grid Lines, Axis Units and Axis values.
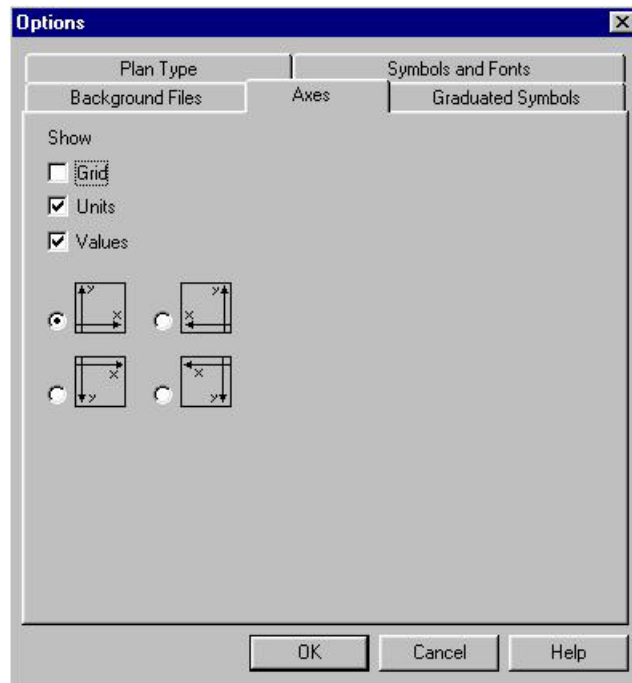

Figure 13.6   Horizontal Plan Window Options – Axes Selection

2. Check the Axis direction box so that you display the X-Axis as negative. Press OK to view this in the HPW. Go back and check positive X-direction.
3. The remaining Tab group "Background Files" will be addressed in the following exercises.

13.6.2 Background images

Working with the Horizontal Plan can be easier if you import a background geographical map. MIKE View supports two standard graphical file formats (DXF and BMP) for this feature.

Loading DXF files is straightforward, assuming full consistency between the co-ordinate systems used in the model and in the DXF file.

If your DXF file is based on a co-ordinate system different from your network model and/or if it is based on other units than SI, you should manually edit the

MIKEVIEW.INI file (located in the Windows directory), in order to compensate for the co-ordinate offset, and/or for the scaling in relation to SI.

Background image

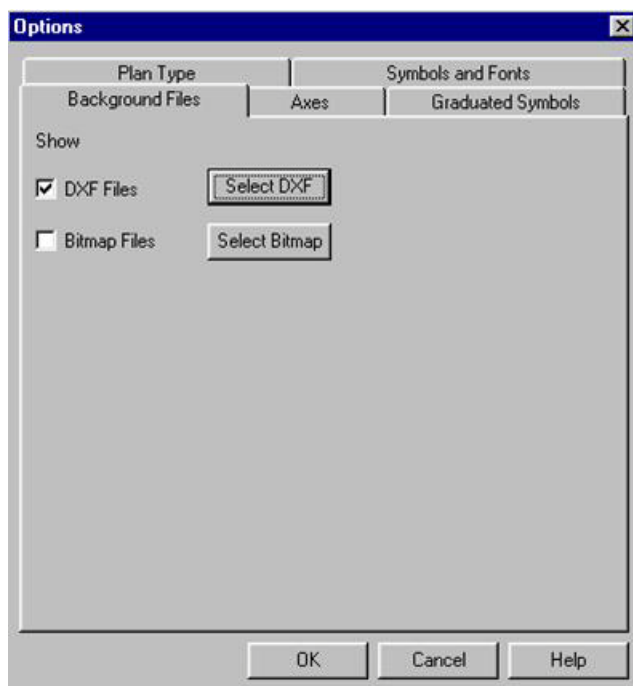1. In the HPW click the right mouse button to open the local menu, select "options"

Figure 13.7   Horizontal Plan View Options – Background Images

2. On the "Options" dialog select the "Background Files" Tab
3. Select the button "Select Bitmap" this opens a Bitmap Selection Window. Select the ''Add'' button to create an entry in the Bitmap list.
4. Select the browse [icon] button to select the required bitmap. The ".bmw" file contains the geo-references for the bitmap image file. Click the <Open> button.
5. Select the Full Visibility option and click <OK> to display the image in the HPW. Try viewing the image with the various visibility Options
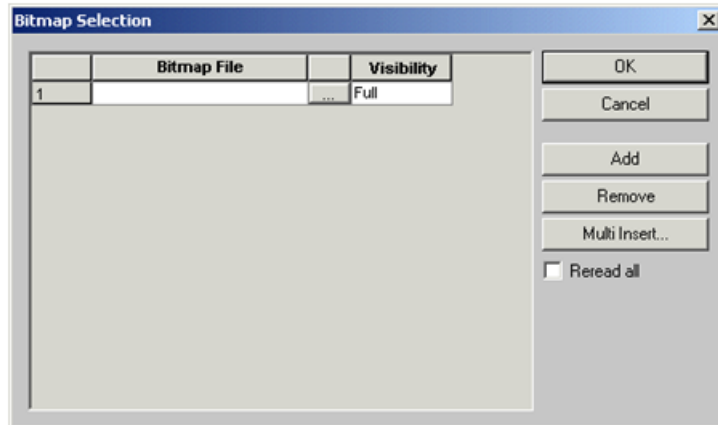
Figure 13.8   Bitmap Selection Window

13.7   Viewing Time Series in Grid Points

MIKE 11 result files include variable series such as water levels, flows, concentrations etc. as a function of time. The time series graphs are displayed in graphical Time Series Windows (TSW).

You can select a time series either graphically by pointing and clicking in the HPW, or by making a selection from a list.

13.7.1   Selecting and displaying a time series from a grid point in the Horizontal Plan Window

1. To select a time series for a specific grid point use either the "grid point selection" button on the tool bar or select the menu entry "Plot | TS in Grid Points…" from the menu bar.
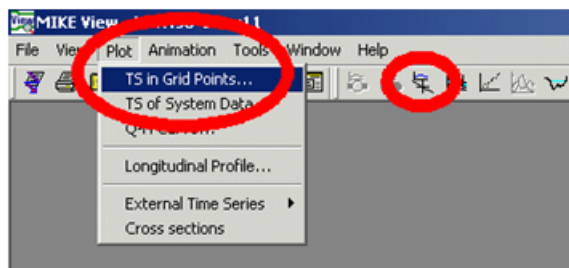

Figure 13.9   Plot Times Series in Grid Points Function

2. A "Data Type Selection" dialog box will open from which you select the specific result file you want to view and data type that you require. From the data type list select, ''WATER LEVEL or DISCHARGE"
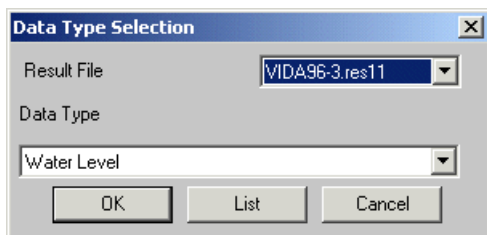
Figure 13.10 Data Type Selection for Time Series Plot

3.   Select <OK> to return to the HPW, notice that the cursor will change
      shape. As you move the cursor over a grid point in the HPW it will
      change shape to indicate it has found a valid grid point with the data type
      that corresponds to your selection. You may select this particular grid
      point to display the TSW for the selected data type.
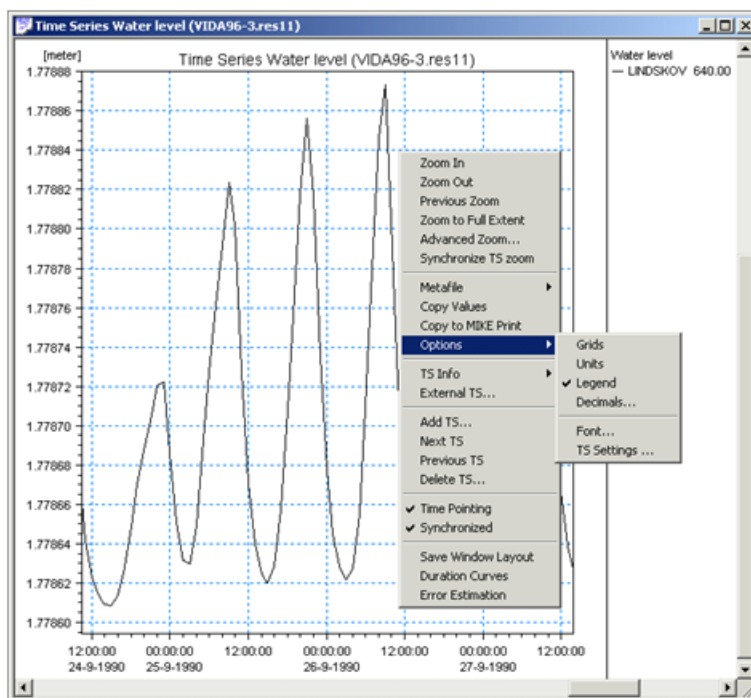

Figure 13.11   Typical Time Series Plot Showing Local Menu Options

4.   On the TSW right mouse click and explore the various options available
      on the pop up dialog. The legend displayed to the right can be modified
      using the options menu. You can modify the window layout/display from
      options menu.

13.7.2 Selecting and displaying a water level time series from a list

1. Repeat the grid point selection process (Steps 1 and 2 in Section 13.7.1) until you have again activated the ''Data Type Selection'' window.
2. From the "Data Type Selection" dialog select the Data Type (water level or discharge), then select the <List> button. A complete list of all available grid points containing the selected data type will be displayed.

Figure 13.12   Typical List of Available Grid Points

3. The first column contains an index or the model grid points with a check box to the left-hand side where you can select the required grid points for display in a TSW. The remaining columns contain summary information on the minimum and maximum water levels or discharge and the time step at which these values occurred.
4. Check box's for the various grid points you want to view and select the <Draw Graph> button to open a new TSW with the checked grid point information.

13.7.3 Adding time series to the TSW

1. If you wish to Add additional time series to an existing TSW you cam make your selection using the local TS entry "Add TS…" from the pop up dialog in the TSW (right mouse click). A data type selection window will pop up allowing you to display any of the available data types. Once you have made your data selection you should select the required gridpoint from the HPW.
2. You can also add a time series from a neighboring grid point by using the TS button selection tools on the tool bar   Time Series Node .
3. There is no limit to the number of time series that you can display in the same TSW. If you display more than one data type (e.g. water level and

discharge) in the TSW a secondary Y-axis will become active on the right hand side of the TSW. Try displaying two data types in the one TSW.

13.7.4 Deleting time series in the TSW

Delete time series to from existing TSW by selecting the local TS entry "Delete TS" or by selecting the "-" button located on the tool bar. This opens the Delete TS dialog prompting you to select a TS to remove from the TSW. Explore the Delete time series options using the tool bars.
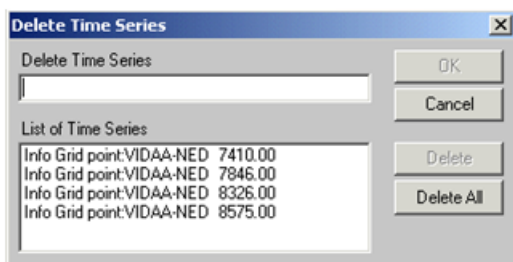

Figure 13.13   Delete Time Series Option

13.7.5 Time series information

1. Make the TSW active and click the right mouse key. Select the entry "TS Info" to open the local Info Node dialog. This dialog displays detailed information about the current grid point.
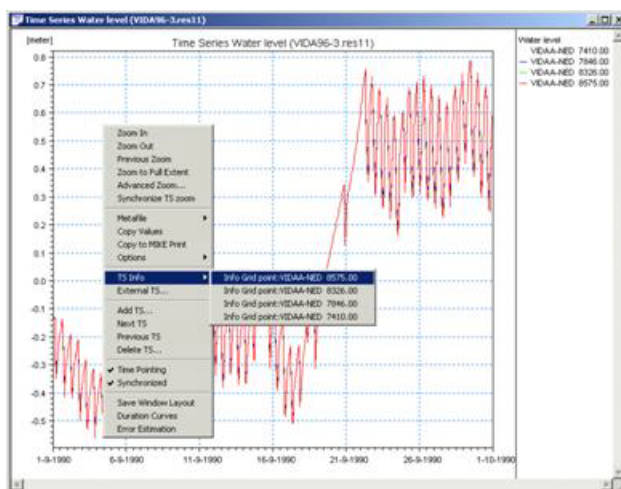

Figure 13.14   Time Series Info Selection from the Local Menu

2. Select the <Info> button to open the dialog showing the actual time series values.
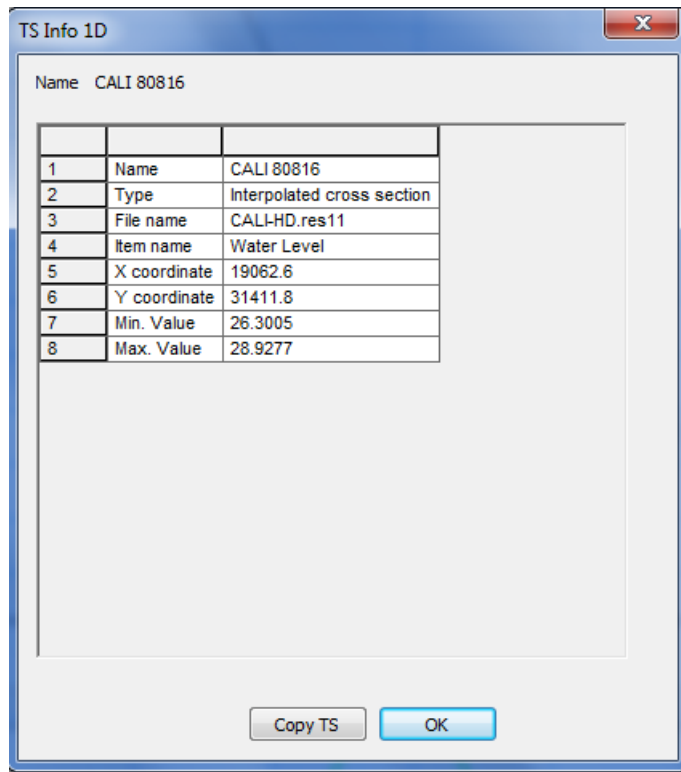
Figure 13.15 Typical Grid Point Info

3. Select the <Copy TS> button to copy the entire time series to the clipboard. You can now paste the time series into any other windows application e.g. EXCEL or Windows Notepad. Go to the Windows Start button, select programs | Accessories | Notepad. Once in Notepad select "Edit | Paste" to paste the time series values.
4. The local entry "Copy Values" has the same functionality

13.7.6 Copy graphics

1. Make the TSW active and click the right mouse key. Select the entry "Copy Graphics". This action copies the TSW as vector graphics to the clipboard. You can now paste the TSW graphics into another windows word processing application e.g. WORD or a similar. The graphic is a true vector based picture that you can edit and e.g. add your own annotation.
2. You can copy all graphical windows in MIKE View to the clipboard.

13.8   Viewing Water Levels In Cross Sections

You can display the water levels as an animation on a schematic plot of the corresponding cross sections in the model. The cross section windows are linked to the TSW and will show the corresponding water levels for the time shown in the TSW.

13.8.1   Selecting and displaying a time series from a link

1. Make the HPW active and select the cross section tool from the tool bar and then move the cursor over the HPW

Figure 13.16   Selection of the Cross Section Presentation Tool

2. Notice that the cursor will change shape as you move the cursor over a valid cross section grid point. Select a grid point to display the Cross Section Animation window.
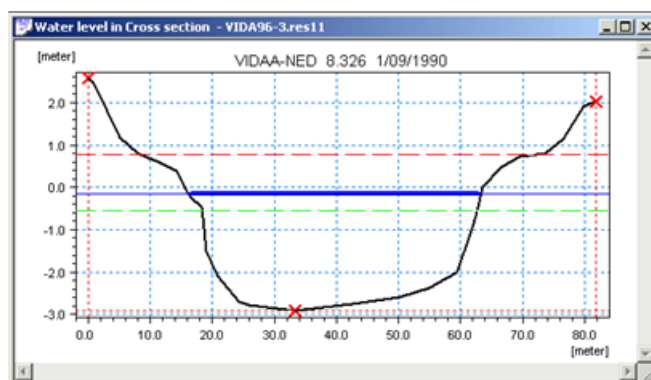
Figure 13.17   Typical Cross Section Plot

3. Using the methods described in the previous exercises select a TSW for the same grid point and display the corresponding water level time series.
4. Use the left mouse button to select any point in time in the TSW and note how the water level in the cross section animation window updates to reflect the water level at the selected time period. Note; how the red and green envelope lines reflect the maximum and minimum water levels that occurred at the selected cross section.

13.9 Viewing Q-H (Discharge-Stage) Curve

A Q-H Curve facility provides useful information about the flow characteristic of a river branch at a selected location.

MIKE View creates a Q-H Curve by graphing the whole set of computed discharges at the selected location and the water levels at the neighboring downstream H-point, within a co-ordinate system defined by the Q and H items on the x and y axes, respectively.

13.9.1 Selecting and displaying a Q-H (discharge-stage) curve

1. To select a grid point for which to display a Q/H Curve select either the "Q/H" button on the toolbar or select the menu entry "Plot | Q-H Curve…".
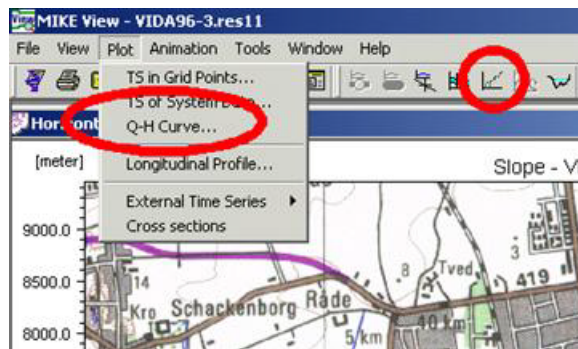

Figure 13.18   Selection of the Q-H Curve Plot Tool

2. Return to the HPW, notice that the cursor has changed shape. Move the cursor in the HPW and notice how the shape changes when you move close to a valid grid point. This indicates that you may select this particular grid point to display a Q/H Curve.
3. Select a grid point to open the Q/H Curve window.
4. Move the cursor to the TSW and click the right mouse key, this opens the local menu. Select the entry "Open TS", this opens a new TSW displaying the Q and H time series the current Q/H Curve has been computed from.
5. Move the cursor to the Q/H TSW and click the right mouse key, this opens the local menu. Select the entry "Options", to open a dialog with options for customizing the Q/H Curve display.
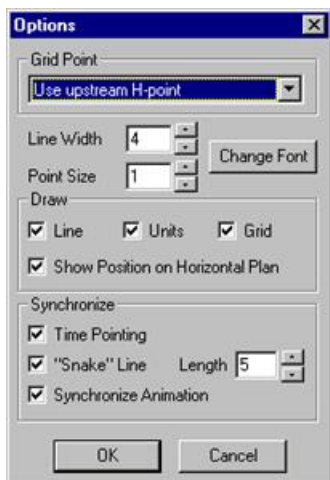
Figure 13.19 Q-H Plot Options

6. Select the local menu entry "Copy Values" to copy the Q and H values from the two time series to the clipboard. Open Notepad and paste the time series.

13.10 Viewing Longitudinal Profiles

A longitudinal profile displays the spatial and temporal variation of the Simulated Hydraulic Grade Line (HGL) or Discharge along user selected segments. The user selects the profile path interactively directly from the HPW.
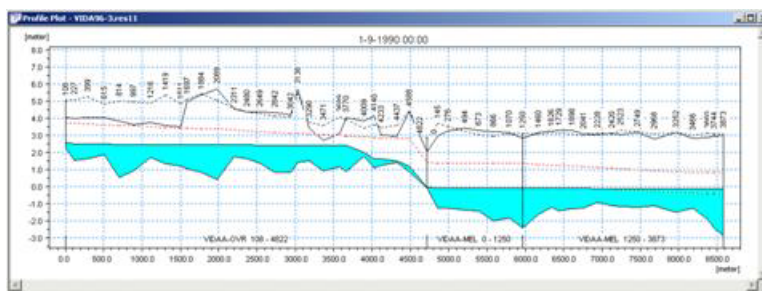
Figure 13.20 Typical Longitudinal Profile Plot

13.10.1 Selecting and displaying the Longitudinal Profile

1. Select the Longitudinal Profile option from the Plot Menu or the Profile tool from the toolbar.
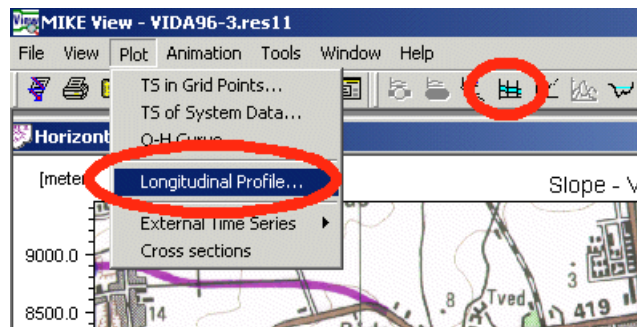
Figure 13.21   Activating the Longitudinal Profile Plot Tool

2. Select the first node you want to include in the profile (make sure that you have set display of node labels to 'ON'. Do this in HPW | local menu | Options | Symbols and Fonts | check the Labels box) continue the selection by moving the cursor in the desired direction and clicking the left mouse key for each additional segment you want included in the profile. The selection will close automatically if an end node (i.e. a node with only one connected link) is selected. The selection can be "toggled" between multiple flow branches at a node by using the arrow cursor keys.

It is usually sufficient to click on the first node of the profile, and point and click on the last node you want included in the profile - MIKE View will find the right path to the cursor on its own!

3. End the selection, by pressing the {Ctrl} key and clicking the left mouse button at the same time. It is possible to 'step back' or undo a selection by pressing the {Shift} key and clicking the left mouse button at the same time.

Closing the profile selection will activate the data type selection dialog box. Select the "Water Level" option and select <OK>. This opens the Longitudinal Profile Window (LPW) in a separate graphical Window.

Figure 13.22   Data Type Selection for the Longitudinal Profile View

13.10.2 Longitudinal Profile options

1.   Make sure that the LPW is active then click the right mouse button to open the local menu. Select the entry "options''. The options dialog has four tabs the first "Axes" controls the grid layout while the second "Symbols and Fonts" controls the display layout. Try changing the labels for the longitudinal profile.
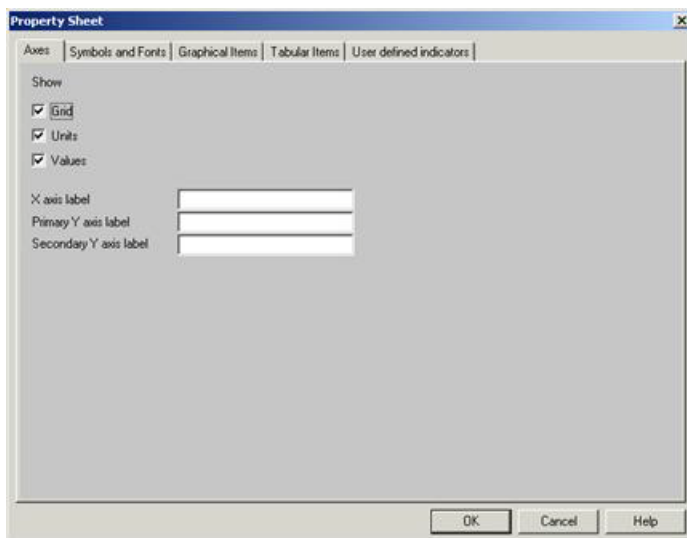
Figure 13.23   Longitudinal Profile Options Menu - Axes

2.   In addition to changing the Symbols and Fonts you can alter the display of the profile and gridlines, labels and maximum and minimum indicators. Try changing the display of the profile using the options available on the Symbols and Fonts tab

Figure 13.24   Longitudinal Profile Options Menu – Symbols and Fonts

3.   The Graphical Items tab allows you to add additional information to the existing profile. The additional information consists of either static or dynamic items. Select the ''Insert'' tool (circled below) and explore the dynamic and static options available to you. Try displaying various items and observing how the LPW is updated.
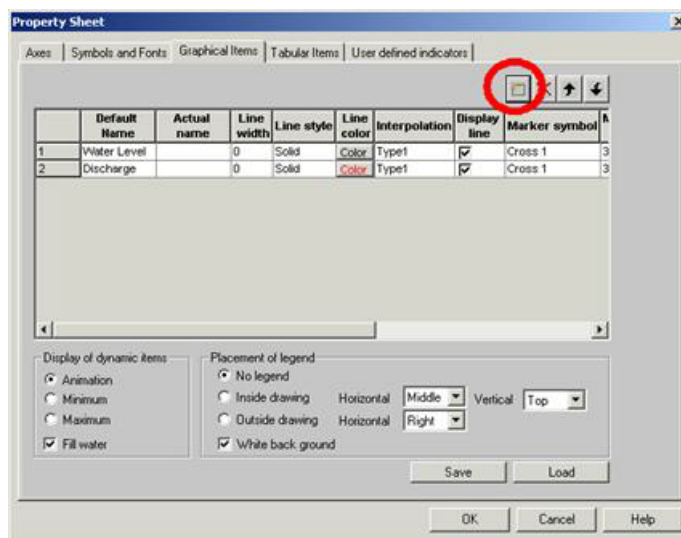
Figure 13.25   Longitudinal Profile Options Menu – Symbols and Fonts

4. The Graphical Items tab allows you to add tabular information to the longitudinal profile. The additional information consists of either static or dynamic items.
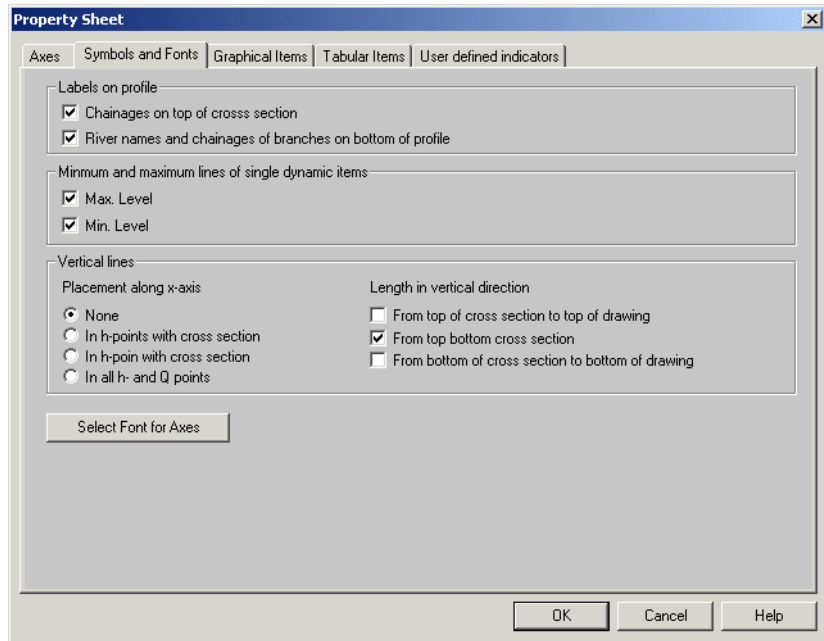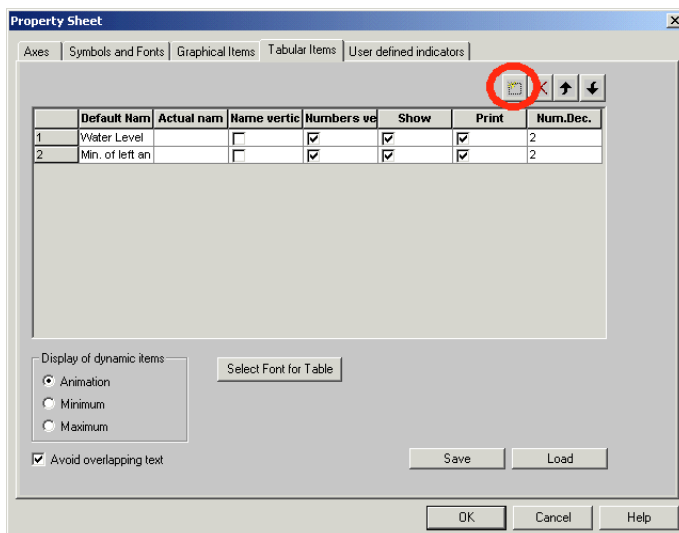

Figure 13.26 Longitudinal Profile Options Menu –Tabular Items

5. Select the ''Insert'' tool (circled in Figure 13.26) and explore the dynamic and static options available to you. Try displaying various items and observing how the LPW is updated.

Figure 13.27 Item Selection Menu

6. The User Defined indicators tab allows you to display markers on the longitudinal profile at user defined levels. The markers can be used to indicate critical levels (such as culvert details or road levels) at locations along the profile. Select the ''Insert'' tool and explore the dynamic and static options available to you. Try displaying various items and observ-

ing how the LPW is updated. Note that you must check the ''Show indica-
tors check box at the bottom of the window to display the markers in the
LPW.
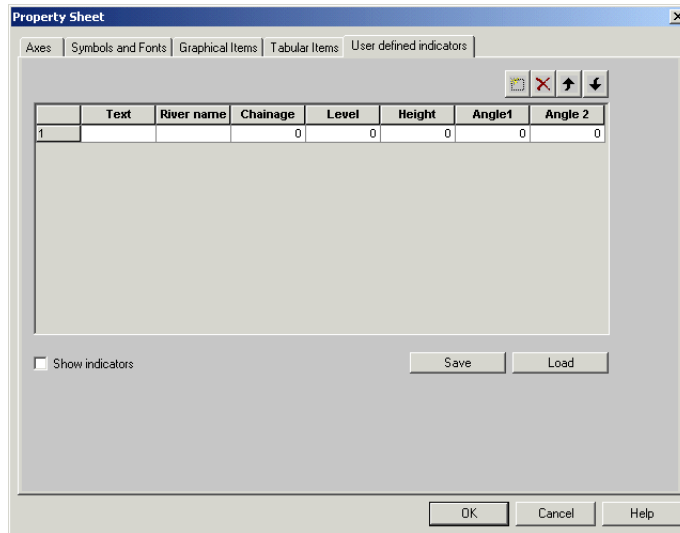
Figure 13.28   Longitudinal Profile Options Menu –User Defined Indicators

13.11 Zooming In The Graphical Windows

It is often difficult to select precise grid points from a large network in the
HPW. In MIKE View you can focus on smaller parts of your network, using the
Zoom functions. You can:

- Zoom in
- Zoom out
- Return to previous zoom frame
- Zoom to Full Extent.

The Zoom function is also accessible from the TSW and LPW to display
details from the results.

13.11.1 Zooming in the Horizontal Profile Window

1. Make the Horizontal Profile Window active
2. The MIKE View zoom facility is activated by the tool bar zoom icons, or
   as a selection on the local menu (the right mouse button!).

Figure 13.29   Activation of Zoom Tools

3.   Select "Zoom In" observe how the cursor changes shape into a magnify-
      ing glass symbol.
4.   Position the cursor at a one of the corners of your zoomed-in frame, click
      the left mouse key (The desired zoom area is framed by a rectangle with
      an origin corner located by the position of the center of the cursor). Keep
      the left mouse button pushed down and drag to the diagonal corner of a
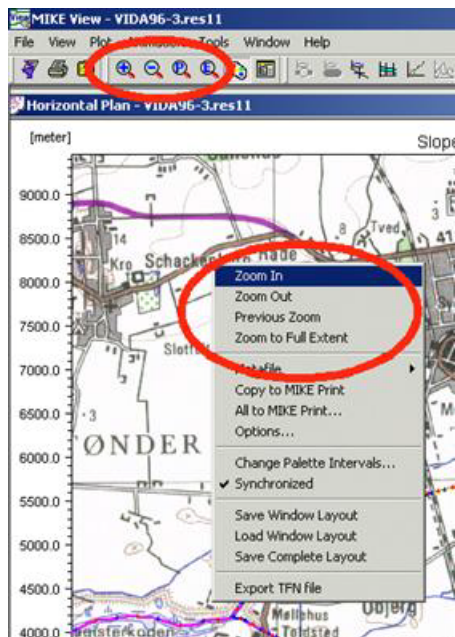   rectangle over the area of the screen you want to zoom to. The size and
      the coverage of the zoom area, is determined by when you release the
      button and - you are "zoomed in"
5.   You can zoom in as many times as you want.
6.   The overview plan window can be of great assistance when zooming in.
      It displays the extension of the zoomed frame on the overview plan as a
      rectangle and, moreover, allows you to move the zoomed frame inside
      the whole model area. Make the overview window active select the
      zoomed frame and drag it to the desired location, observe how the HPW
      pans.
7.   Use the scroll bars in the HPW to 'pan' the zoomed frame over the net-
      work area.
8.   Select the option "Zoom Out" to return to full view.
9.   The option "Zoom Previous" returns the zoom frame to the previous
      zoom extent.

13.11.2 Zooming in the Time Series Window and the Longitudinal Profile Window

1. Make a TSW active, click the right mouse button and select "Zoom In". Drag and drop the zoom frame.
2. Click the right mouse button and select the entry "Advanced Zoom", here you can control minimum and maximum for the Primary and Secondary axes.
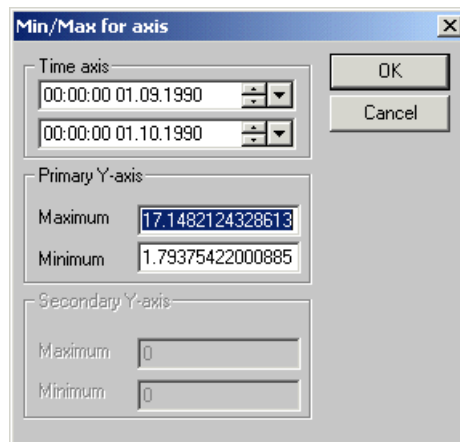

Figure 13.30   Advanced Zoom Options

3. You can return to the full view at any time by selecting "Zoom Out"

13.12 Animating Results in the Longitudinal Profile Window

The basic result file time series items (Water Level and Discharge) can be viewed as animated presentations in the longitudinal profile (LPW). The dynamics of the model network are illustrated by displaying sequentially swapped time series values within the time interval covered by the loaded data.

13.12.1 Animating results in Time Series Window and Longitudinal Profile Window

1. Your View should, by now, have the HPW, a TSW and a LPW open.
2. Make the LPW active, then select the run button or "Animation | Run" from the main menu bar (Circled to the below). Observe how the time indicator in the LPW changes by continuously updating the current time step. The animation will continue to loop until you stop it. Observe the TSW where the time passage is presented by a vertical line moving along the time axis as the animation progresses.
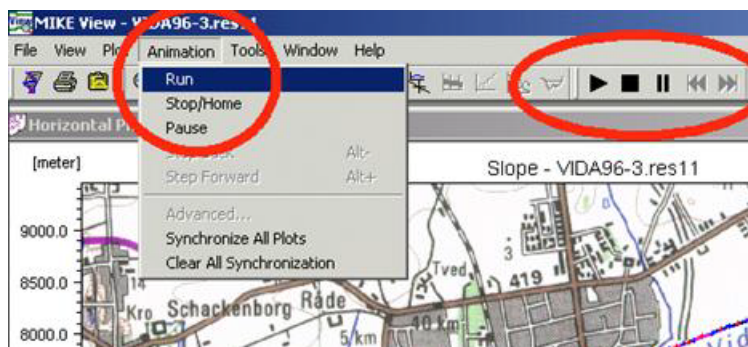
Figure 13.31   Animation of the Longitudinal Profile

3. Pause the animation by selecting the "pause|" button. You can step forwards and backwards through the result file by selecting the "forward step and backward step buttons (circled above). Finally stop the animation by selecting the stop button.
4. "Synchronize" results by selecting "Animation | Synchronize All Plots". All windows will now display results from exactly the same time step. You can remove the synchronized option for all windows "Animation | Clear All Synchronization" at the same time or each window individually by selecting the windows "local menu | Clear Synchronization"
5. Make the LPW active and select the entry "Animation | Advanced", this opens a dialog from which you can control the animation. Set the parameters as indicated in the dialog, press <OK> and start the animation again.
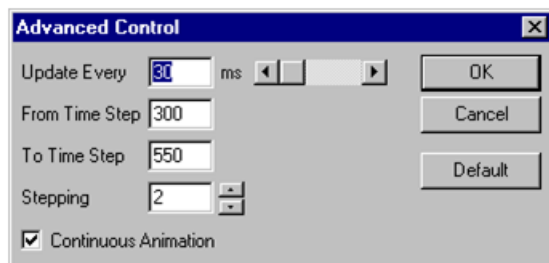

Figure 13.32   Advanced Animation Control

6. Select a Q/H Curve and in the Q/H curve window select the "local menu | Options" and check all options. Press <OK>.
7. Click anywhere on the Q/H Curve or in the TSW and observe how all windows display results from exactly the same time step. You can also click when the animation is running thereby replaying a certain section over and over again.

13.13 Animating Results in the HPW

In addition to animating result file time series items in the longitudinal profile (LPW) the same items can be animated in the HPW. The item dynamics are represented by the palette colors and/or by graduated symbols.

13.13.1 Animating results in Horizontal Profile Window, Time Series Window and Longitudinal Profile Window

1. Your View should by now have the HPW, a TSW, a QHW and LPW open.
2. Make the HPW active and select the "local menu | Options", check the box "Draw Branches As | Water Level". Select <OK>. The links now appear colored and a Palette Window (PW) is opened. The number of classes, range and color ramp is determined by MIKE VIEW but can be modified by the user.

Figure 13.33 Palette Options
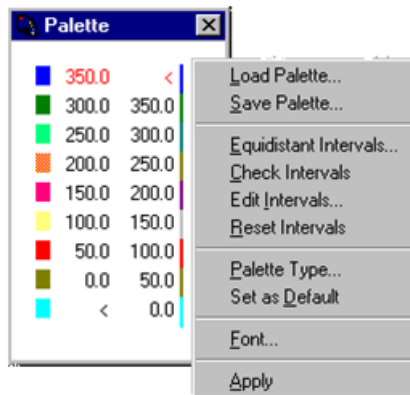
3. Make the PW active and click the right mouse button to access the local menu. Select the entry "Equidistant Intervals." to open the dialog, use setting as indicated. Next, double-click on each interval group to define appropriate range intervals, following this double-click on the color square to define appropriate color ramp for display.
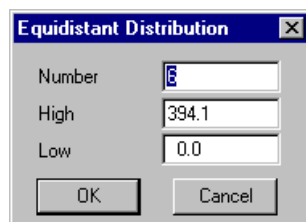

Figure 13.34 Adjustment of Palette Intervals

4. You can control the palette in many ways and also save specific palette setting. It will pay-off when you can efficiently apply your new experience in future work. Moreover, if you create a custom palette of a more general applicability, or if you want to have it at hand next time you work with MIKE View, just save the palette settings into a ".PAL" file. Next time, load it and your own palette will be there again for your animation.

5. Start the animation by making the HPW active, then select the run button ">" or "Animation | Run" from the main menu bar. Observe how the colors of the Links change with time continuously reflecting the simulated water level at the current time step. The animation will continue to loop until you stop it. Observe the TSW where the time passage is presented by a vertical line moving along the time axis as the animation progresses.

6. Pause the animation by selecting the "||" button. You can step forwards and backwards through the result file by selecting the "|<<" and ">>|" buttons. Finally stop the animation by selecting the stop button.

7. While replaying the animation you can switch to make any of the other windows active. In these windows you can zoom in, move the time cursor in the TSW, etc. You should note that the animation only can be paused and stopped from the window it was started from.

8. Stop the animation and display an animation showing Discharges in Branches.

13.14 Viewing Derived Results

MIKE View computes depths and flood levels by combining the model results with the geometrical cross section information contained in the result files. The following derived results can be computed in MIKE VIEW:

Depth   (water level elevation) – (invert level (bed) elevation)

Flood   (water level elevation) – (ground level elevation)

13.14.1 Computing and viewing derived results in TSW and LPW

1. Select the main menu entry "Tools | Compute | Depth". When the computation is completed a message info dialog appears, press <OK>.
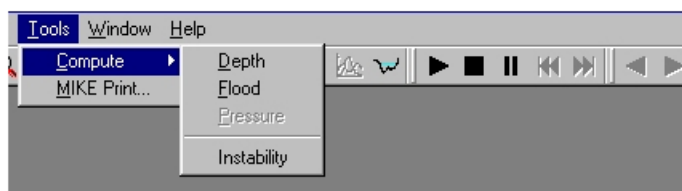

Figure 13.35   Computation of Derived Results

2. Choose the "Select Grid Point" button to select a location for viewing results. This opens the Data Type Selection dialog, in the drop down list select the item "Depth", select <OK>

13.14.2 Viewing derived results in HPW

1. Make the HPW active. Select the "options" entry from the local menu.
2. In the local menu dialog select "Depth" from the "Draw Branches as" drop down list. Select <OK> to close the dialog and animate Depth as the variable in the HPW. Observe that the Palette automatically changes range and number of classes. Customize the palette to by preparing a suitable color ramp.
3. Select the ">" replay animation button to start the animation.

13.15 Save/Load Window Layout

The local menu for each graphical window (TSW, LPW, QHW) contains the 'Save Window Layout" option. This allows the relevant information about the current active window layout to be saved. Information such as the data source (result file name, external time series, etc.), the item identifier, the window size and position and graphical layout features are saved in an ASCII file with a default extension (*.WLA).

At any later occasion when the specified data source is already loaded into MIKE View, the window can be restored by loading the corresponding *.WLA file.

13.15.1 Save Time Series Window layout

1. Make the TSW active
2. Select the local menu option "Save Window Layout"
3. In the file dialog specify the file name "exampleTSW" using the extension ".wla", select <OK>
4. Close the TSW

13.15.2 Load Time Series Window layout

1. Make the HPW active
2. Select the local menu option "Load Window Layout"
3. In the file dialog select the file "exampleTSW.wla", select <OK>, this automatically opens a TSW.

13.16 Save/Load Complete Layout

The local menu for the Horizontal Plot window contains the 'Save Complete Layout" option. This allows the relevant information about the current pres-

entation layout to be saved. Information such as the data source (result file names, external time series, etc.), the presented item identifiers, window sizes and positions and graphical layout features are saved in an ASCII file with a default extension (*.CLA).

At any later occasion, the entire layout (HPW, TSW, QHW and LPW) can be restored by loading the corresponding *.CLA file into MIKE View.

13.16.1 Save complete layout

1. Make the HPW active.
2. Select the local menu option "Save Complete Layout".
3. In the file dialog specify the file name "example" using the extension ".cla", select <OK>.
4. Close the HPW, this closes all other windows.

13.16.2 Load complete layout

1. Select the main menu entry "Tools | Load Complete Layout", in the file dialog select the file "example.cla".
2. Select <Yes> to use the result file as primary result file
3. In the "data load selection" dialog select <OK>, this opens the HPW and the relevant TSW.

13.17 Opening Additional Result Files

Several result files may be opened simultaneously in the same MIKE VIEW session.

Opening multiple result files in a single session allows the user to compare model results from different simulations.

To open two result files in a single session it is imperative that the two result files contain the same branch layout with the same grid point names.

13.17.1 Opening a Second MIKE 11 Result Files

1. You must first have two model simulations from the same network in your working directory. If you do not have two model simulation result files then try running a separate MIKE 11 simulation using a higher roughness.
2. Select the main menu entry "File | Open…" in the file selection dialog select the MIKE 11 result file for the first model simulation. Load all time steps. This opens a new HPW, with a network
3. Select the main menu entry "File | Add…" in the file selection dialog select the second MIKE 11 result file. Load all time steps. The second network results are loaded however the HPW window does not change.
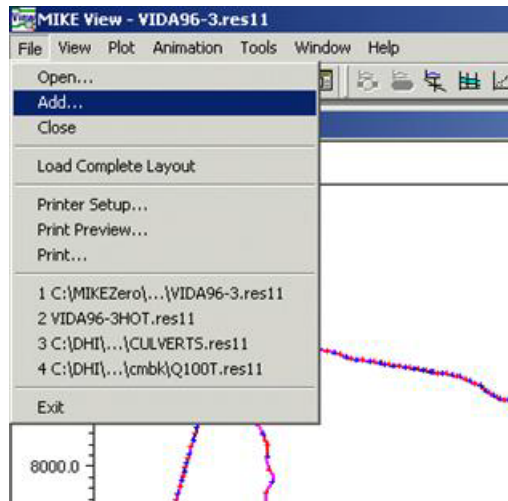
Figure 13.36   Adding Multiple Result Files to MIKE View

4.   Make the HPW active and then select the "Select grid Point" button and in the Data Selection Dialog select "Water level''. This will display the TSW for a grid point. From the local menu in the TSW select the ''Add TS…'' option to display the Data Type Selection Window. From the Result File drop down dialog, select the second result file name and pre < OK >. This will return you to the HPW where you should again select the grid point that you are displaying. A second TS from the second result file will be added to your TSW.
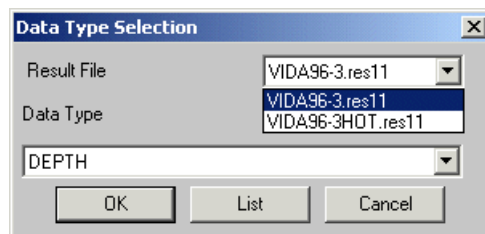

Figure 13.37   Selecting the Result File to be Viewed

13.18 External Time Series

In MIKE View you can also import and display time series from various external sources. The relevant sources of for external time series (ETS) are:

- ASCII files in specific formats (.TXT)
- the Clipboard
- special MIKE View ASCII files (.ETS)
- MOUSE/MIKE 11 time series database files (DFS0)

All loaded external time series, independently of the origin are placed in the external time series (ETS) RAM buffer, which is then used as a source for the selection of those time series to be presented in the individual MIKE View TSW.

The content of the ETS RAM buffer can be saved to a special ASCII file (default extension .ETS) for later reuse.

13.18.1 Copying Discharges to ETS

1. Make the TSW active. Select the local menu option ''Copy Values''. The time series values are now copied to the Clipboard.
2. Open Wordpad or Notepad and paste the clipboard contents to view the copied data.
3. Make the TSW active. Select the local menu option "External TS… ". In the ETS dialog select the <Paste> button. This opens the dialog for specifying a file name for the time series that we want to paste from the Clipboard. Specify a file name and select <OK> (see Figure 13.38).
4. Select the new item in the ETS list (its highlighted) and select <OK>.
5. The TSW now displays the results from the MIKE 11 result file along with results from another external data source.
6. Select main menu entry "Plot | External TS | Save", to save the entire ETS database to an ASCII file in the current work directory. In the file dialog specify the new .ETS file name.
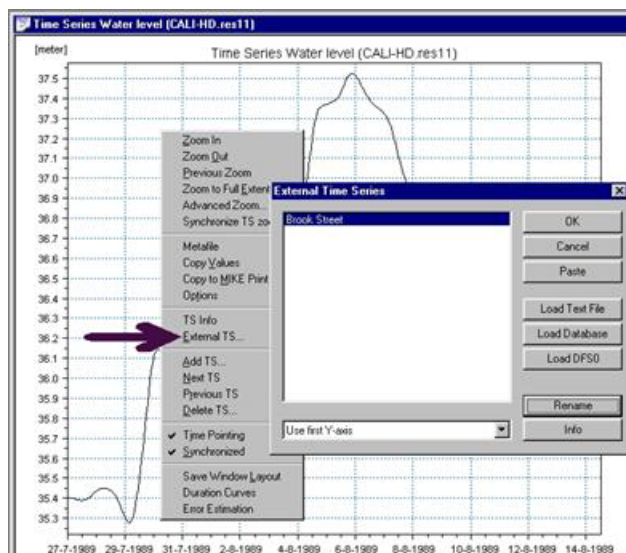

Figure 13.38   Adding an External Time Series via the Local Menu

13.18.2 Working with ETS

MIKE View can load time series from ASCII files a specific format, with a .TXT extension. A .TXT file can contain only one time series. The .TXT format supported for loading into MIKE View:

YYYY-MM-DD HH:MM:SS <value>

1. The "Load Text File" and "Paste" are used to import gauge data from e.g. ASCII spreadsheets or text files. Graphical comparison between simulation and gauge data is easy.
2. Remember to save the ETS when you have imported several gauge time series.
3. You can also save a window layout with ETS displayed and bring it up again using the "Load Window Layout".
4. Use the main menu entry "Plot | External TS | Load" to load an already existing .ETS file to the ETS RAM buffer. The .ETS file load can either rewrite the existing buffer, or append the data to the current content of the ETS RAM buffer.