

MIKE HYDRO

River

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 A/S (hereinafter referred to as "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.





CONTENTS

MIKE HYDRO River User Guide	11
1 Introduction	13
2 Working With MIKE HYDRO	17
2.1 How to Start MIKE HYDRO	17
2.2 The MIKE HYDRO User Interface	18
2.2.1 Tree view	18
2.2.2 Map view and tabular view	22
2.2.3 Property view	24
2.2.4 Output windows	25
2.2.5 Ribbon	28
2.2.6 Menu bar	31
2.3 Layout	31
2.4 MIKE Zero Menu Bar	32
2.4.1 File menu	33
2.4.2 Edit menu	34
2.4.3 View menu	34
2.4.4 Run	36
2.4.5 Tools	36
2.5 Creating Input Files	36
2.5.1 Create a new dfs0 file	37
2.6 Viewing Results	39
2.6.1 River result files	40
2.6.2 Dfs0 files	40
2.7 Getting Help	41
2.8 Examples	41
2.8.1 River module examples	41
3 Tutorial: Setting Up a Simple Model	43
3.1 Setting up a Simple River Model	43
3.1.1 Create a new document	43
3.1.2 Defining branches	44
3.1.3 Defining cross sections	47
3.1.4 Defining boundary conditions	48
3.1.5 Define initial conditions	51
3.1.6 Running a simulation	51
3.1.7 Viewing results	52



4	Simulation Specifications	53
4.1	Modes	53
4.1.1	River modules	54
4.2	Description	56
4.3	Simulation Period	56
4.4	Time Step Control	57
4.5	Computational Control Parameters	60
5	Map Configurations	65
5.1	Coordinate System	65
5.2	Background Map	66
5.3	Background Layers	66
5.4	Digital Elevation Model (DEM)	67
5.4.1	Cross section creation	67
5.4.2	River tracing and catchment delineation	68
5.5	Working Area	69
6	River Network	71
6.1	Creating a River Network	72
6.1.1	Method 1: Digitising a River network	72
6.1.2	Method 2: Importing a River network	73
6.1.3	Method 3: Derive the River network from a DEM	73
6.2	Branches	73
6.2.1	General	74
6.2.2	Link Channels	76
6.2.3	User defined chainage points	79
6.3	Routing	80
6.3.1	Routing Method	81
6.3.2	Water level Calculation	82
6.4	Alignment lines	83
6.5	Cross Sections	84
6.5.1	Cross section in tabular view	86
6.6	Structures	99
6.6.1	Structure details	100
6.6.2	Weirs	107
6.6.3	Culverts	112
6.6.4	Bridges	115
6.6.5	Pump	130
6.6.6	Gates	132
6.6.7	Direct discharges	136
6.6.8	Dambreak	137
6.6.9	Energy losses	143
6.6.10	Tabulated	145
6.7	Control	146
6.7.1	Sensors	146
6.7.2	Variables	148
6.7.3	PID parameters	149



6.7.4	Control rules	149
6.8	Storages	155
6.8.1	Inserting a storage	156
6.8.2	Connecting a storage	156
6.8.3	Storage properties	157
6.9	Decoupled simulation	158
7	Catchments	159
7.1	Catchment definitions	162
7.1.1	General	162
7.1.2	Rainfall-Runoff	163
7.1.3	NAM	163
7.1.4	UHM	176
7.1.5	Time-area	179
7.1.6	Kinematic wave	184
7.2	Combined Catchments	196
7.3	River links	196
7.4	Decoupled simulation	198
7.5	Hotstart files	198
8	Hydrodynamic Parameters	201
8.1	Bed Resistance	201
9	Advection-Dispersion Parameters	205
9.1	The Advection-Dispersion Equation	205
9.2	Components	206
9.3	Dispersion	207
9.4	Decay	209
10	Water Quality	211
10.1	Notes on MIKE ECO Lab	211
10.2	Water quality definitions	212
10.2.1	The 'Edit Filter' option	218
11	Sediment Transport Parameters	221
11.1	General parameters	221
11.2	Sediment fractions definitions	222
11.2.1	Properties	222
11.2.2	Transport theory	223
11.3	Bed parameters	225
11.4	Passive branches	227
11.5	Non-scouring bed level	227
12	Boundary Conditions	229
12.1	Standard boundaries	230
12.1.1	Hydrodynamic	230
12.1.2	AD / WQ boundary	235
12.1.3	ST Boundary	238
12.2	Wind Friction	240



12.3	Groundwater leakage	242
12.4	MIKE SHE Couplings	243
13	Initial Conditions	251
13.1	State File	251
13.2	HD Initial Conditions	252
13.3	AD Initial Conditions	254
13.4	ST Initial Conditions	255
	13.4.1 Concentration	255
	13.4.2 Thickness and distribution	255
14	Data Assimilation	257
14.1	General parameters	257
14.2	Observations definitions	258
	14.2.1 Update parameters	258
	14.2.2 Equation	262
14.3	Input perturbations	264
	14.3.1 Standard deviation	265
	14.3.2 Perturbations parameters	266
14.4	HD additional results	268
14.5	AD additional results	268
15	Result Specifications	271
15.1	Standard Results	271
15.2	HD Additional Results	273
15.3	AD Additional Results	276
15.4	ST Additional Results	276
15.5	State Files	278
15.6	Decoupling	279
15.7	Maps	280
15.8	Time series	284
16	Main menu bar features	289
16.1	File: Import and Export of MIKE HYDRO data	289
	16.1.1 Import	289
	16.1.2 Export	294
16.2	Tools	297
	16.2.1 Auto Generate Cross Sections	298
	16.2.2 Compute storage from DEM	301
	16.2.3 Trim or Extend Cross Sections	303
	16.2.4 Edit multiple cross sections	307
	16.2.5 Auto update cross sections	309
	16.2.6 Resistance values interpolation	310
	16.2.7 Export embankment lines	312
	16.2.8 Importing cross sections from text file	314
	16.2.9 Exporting cross sections to text file	318



Appendix A	321
A.1 CONTROL FUNCTIONS	323
Index	333





MIKE HYDRO RIVER USER GUIDE



1 Introduction

MIKE HYDRO is the common Graphical User Interface framework some of the Water Resources products of MIKE Powered by DHI.

Embedded in one setup editor, MIKE HYDRO offers a state-of-art, map-centric user interface for intuitive model build, parameter definition and results presentation for water resources related applications.

MIKE HYDRO includes the following modules:

- Basin module
- River module

Basin module

MIKE HYDRO Basin is a versatile and highly flexible model framework for a large variety of applications concerning Management and Planning aspects of water resources within a river basin.



Typical basin module applications

- Integrated Water Resources Management (IWRM) studies
- Multi-sector solution alternatives to water allocation and water shortage problems
- Reservoir and hydropower operation optimisation
- Exploration of conjunctive use of groundwater and surface water
- Irrigation scheme performance improvements

Features

MIKE HYDRO Basin models utilise a river network and catchments within the specific river basin as the basic model-data. On top of this, a number of features can be applied depending on the type of application. Features include; River routing, Water users (regular as well as Irrigation users), Hydro power and Reservoirs, Hydrology (rainfall-runoff simulations), Groundwater, Global ranking of water users, Reservoir sedimentation and Water quality options using MIKE ECO Lab.

MIKE HYDRO Basin is the successor of DHI's former product for Integrated water resources management and planning; 'MIKE BASIN'.

For more information see MIKE HYDRO Basin User Guide.

River module

MIKE HYDRO River is the modelling framework for defining and executing one-dimensional river models to a large variety of river related project applications.



River module applications

Typical application areas with river models include:

- River hydraulics application
- Flood analysis and flood alleviation design studies
- Real time flood or drought forecasting
- Dam break analysis
- Optimisation of reservoir and gate operations
- Ecology and water quality assessments in rivers and wetlands



- Water quality forecasting
- Sediment transport and long term assessment of river morphology changes
- Wetland restoration studies

MIKE HYDRO River is the new generation of Graphical User Interface for MIKE by DHI River modelling applications, and MIKE HYDRO River is hence the successor for the existing DHI river modelling package; MIKE 11.





2 Working With MIKE HYDRO

This section includes an overview of the different components in the MIKE HYDRO graphical user interface and how to use them.

The general functionality of the individual windows is described in this section, whereas any specific functionality only available in special cases will be described later.

2.1 How to Start MIKE HYDRO

During the installation of MIKE HYDRO a shortcut for '*MIKE Zero*' is placed in the Start menu. When MIKE Zero is started the MIKE Zero shell is displayed.

To create a new MIKE HYDRO setup go to the *File* menu and choose *New* and *File*, which will open the dialogue window shown in Figure 2.1. Select '*MIKE HYDRO Model*' from the MIKE HYDRO product type group and press the 'OK' button or double-click the icon to create a new MIKE HYDRO document.

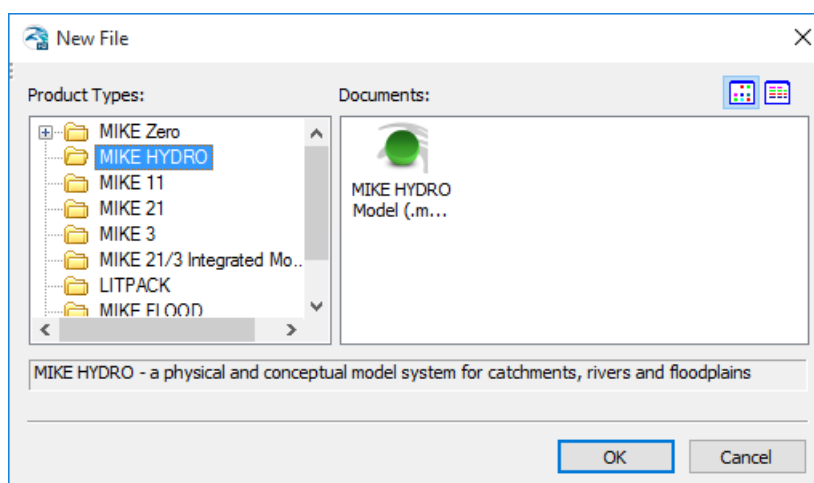


Figure 2.1 Create new MIKE HYDRO document

Creating a new MIKE HYDRO document automatically starts a MIKE HYDRO setup wizard.

The wizard will guide you through defining key settings for the framework of your project. Pressing *Cancel* in the wizard will use default parameters.



Note that any settings specified in the setup wizard may later be changed from the individual dialogue boxes.

If the setup wizard will not be used in future projects, uncheck the checkmark on the first page of the wizard or change the settings in the *User Settings* see section User settings (p. 34).

2.2 The MIKE HYDRO User Interface

The MIKE HYDRO User Interface consists of six main parts (see Figure 2.2):

- Tree view
- Map view and tabular view
- Property view
- Output windows
- Ribbon
- Menu bar

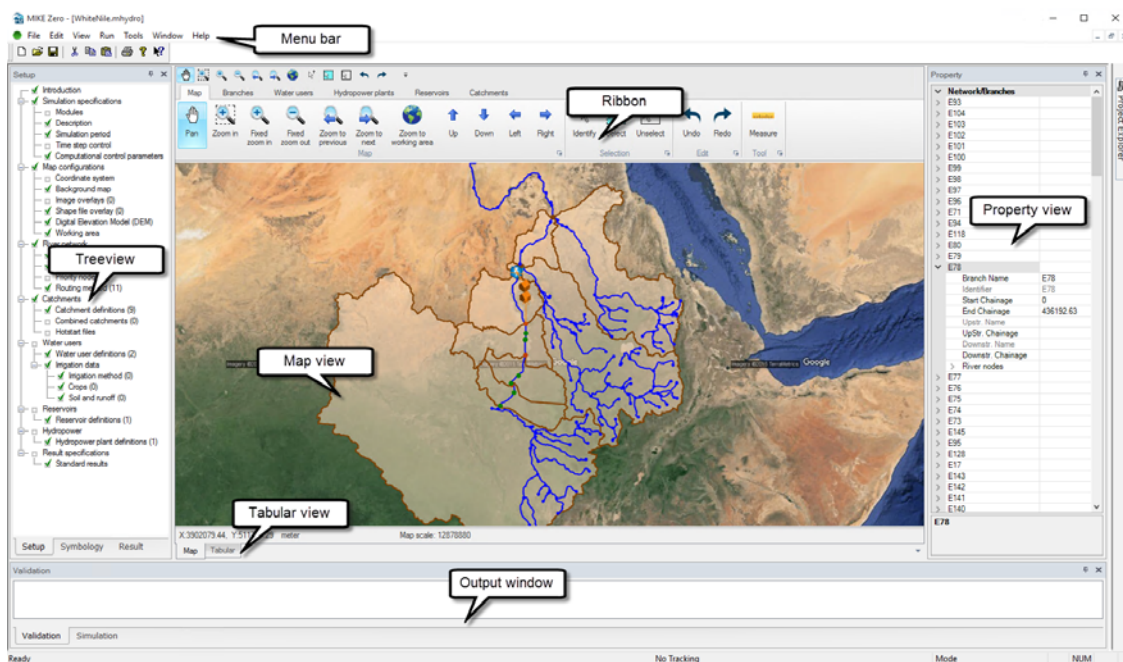


Figure 2.2 MIKE HYDRO User Interface

2.2.1 Tree view

The tree view contains three tabs:

- - control of model parameters
- - control of symbology and labels



- - view simulation results

Setup tree view

The *Setup tree view* allows you to navigate easily between the different sets of model setup parameters. Each node in the tree corresponds to a particular set of parameters, and clicking the node will navigate to the particular set of parameters which will be displayed in the *Property view* as well as in the *Tabular view*.

The tree depends on the settings in the *Modules* page (see User Guide 4.1 Modes (p. 53)) and will only show relevant nodes.

The *Setup tree view* also displays the validation status of the parameters – if a green tick mark is displayed on the tree node, this group of setup parameters has been validated without errors. In case of warnings or errors, a red cross is displayed in the tree node and the messages are displayed in the *Validation* window.



Note: A green validation tick mark is displayed if the model setup parameter node has been opened. It is therefore possible to execute simulations without green tick marks in all tree nodes.

Symbology tree view

The *Symbology tree view* allows you to select the objects for which you want to edit the appearance in the *Map view*. The tree displayed under the *Symbolology* tab matches the tree displayed in the *Setup* tab for all objects shown on the map.

The symbology of the *Map view* shown under the *Result* tab is specified under the 'Results' node in the tree view.

Symbology

For each object type it is possible to edit the symbology on *Symbol* tab. The symbology can be shown or hidden on the map using the setting 'Visible'.



Note that the 'Visible' setting can also be changed by double-clicking on the symbol in the tree view.

Changes in the symbology is reflected on the map when pressing 'Apply to map' at the bottom of the tabular view. The symbology is automatically applied when changing to the *Setup* or *Result* tree view.



Note that by right-clicking on an object in the tree view you can choose 'Reset symbology' and this will set the symbology for the selected object back to the default symbology.

Symbology settings are saved with the project.



The symbology settings will differ based on the object type e.g. line, point, symbol or polygon.

In general all elements will have 4 symbology types:

- Single symbol
- Graduated size
- Graduated color
- Unique values

Single symbol

Using the single symbol will display all elements on the map with a common symbology.

For point objects it is possible to define the symbol style as either 'Simple symbol' defining colour and size or as 'Picture symbol' using predefined pictures.

Graduated size

Using the graduated size will display the elements with different size or width depending on a user-specified parameter.

The general settings for the symbology is defined as for single symbols with the exception of the size which is defined with a minimum and maximum size. The parameter used for differentiating the size is defined in the 'Field' and can only be numeric values.

A grid is defined at the bottom for specifying the intervals used for different sizes. The number of intervals are changed by using the 'Classify' button. This will open a dialogue for defining the number of classes and the corresponding break values.



Note that the button 'Distribute equidistant' will set the break values according to the current values used in the setup.

Break values can be changed either in the 'Classify' dialogue or directly in the grid for existing classes. In the case where the break values does not cover the entire range of values in the setup a default size may be applied by setting the parameter 'Use default symbology'. If this is not specified elements not falling within the specified interval will be hidden on the map.

Graduated size is not available for polygons.

Graduated color

Using the graduated color will display the elements with different color depending on a user-specified parameter.



The general settings for the symbology is defined as for single symbols with the exception of the colour which is defined as a colour ramp. The parameter used for differentiating the size is defined in the 'Field'.

Colour ramps can be modified using the color palette wizard. Documentation of the wizard is available in the common MIKE Zero documentation installed with the standard MIKE HYDRO Basin installation: "The Common DHI User Interface for Project Oriented Water Modelling": User Guide (...MIKE Zero\Manuals\MIKE_ZERO\MIKEZero.pdf).

A grid is defined at the bottom for specifying the intervals used for different colours. The number of intervals are changed by using the 'Classify' button. This will open a dialogue for defining the number of classes and the corresponding break values.



Note that the button 'Distribute equidistant' will set the break values according to the current values used in the setup.

Break values can be changed either in the 'Classify' dialogue or directly in the grid for existing classes. In the case where the break values does not cover the entire range of values in the setup a default colour may be applied by setting the parameter 'Use default symbology'. If this is not specified elements not falling within the specified interval will be hidden on the map.

Unique Values

Using the unique values will display individual symbology depending on a user-specified parameter. The parameter used for differentiating the symbol is defined in the 'Field'.

A grid is defined at the bottom for specifying the relation between symbology and value. New entries in the grid is added using the either 'Add' for one value or 'Add all' for all values currently in the setup. The symbol for each value in the grid is changed by double-clicking on the symbol in the grid.

In the case where a value is not specified in the grid a default symbol may be applied by setting the parameter 'Use default symbology'. If this is not specified elements not falling within a specified value will be hidden on the map.

Labels

For each object type it is possible to edit the labels on *Label* tab. The label can be shown or hidden on the map using the setting 'Visible'.

Label settings are saved with the project.

Map Properties

Map properties are specified under 'Map configuration' in the tree view. Here options are provided for displaying a legend, scalebar and north arrow on the map.

The legend will only display elements shown in the map.



Results tree view

The *Results tree view* allows you to navigate easily between the different simulation results (model outputs). Each node in the tree corresponds to a particular set of simulation results, and clicking the node will navigate to that set of simulation results.

For more information on how to view results see section 2.6 Viewing Results (p. 39).

2.2.2 Map view and tabular view

The central part of the user interface has the following two views:

- Map view
- Tabular view

Map view

The *Map view* is the key element of MIKE HYDRO. Map view provides a graphical – or map – view of the model setup. In *Map view*, the different model objects (e.g. Branches) can be added and edited by selecting the appropriate model object tab in the ribbon.

Right-clicking on the map will provide quick access to zoom functions and exporting the map either to the clipboard or a file.

Selection tools

A set of useful tools is available to identify or select specific features defined in the map view. Tools are located in the '*Selection*' ribbon bar group with the following features:

- **Identity:** Identify button is used to identify and highlight specific model objects. Identify has two ways of operation; when activated at a location where only one single object is defined, it will automatically activate the specific object both in the Property view and activate the Tabular view for the specific object. If Identify is activated at a location where multiple objects are located, a list of objects are presented and the user can select which object should be activated.
- **Select:** Enable selection of model objects. Selections can be made through single click on the map or by drawing a rectangle. All objects positioned at the specified location will be selected and highlighted.
- **Unselect:** Removes all selections within the map
- **Delete selection:** Deletes all the selected objects
- **Zoom to selection:** Zooms to all selected objects on the map



Undo and redo

MIKE HYDRO supports undo and redo on operations on the map and changes in the tabular view or property view.

If the cursor is placed over either undo or redo buttons the hint will list the change that will be either re- or undone.

In general all type of data for the network is covered by undo and redo functionality where as changes to the layout and display of data is not.

The following is not covered by undo and redo functionality:

- Symbology and labels
- Result presentations
- Windows layout
- Selections
- Changes to water quality definitions
- Pop-up windows

Measure tool

The measure tool allows to measure distances between points in the Map or along a path. This tool is located in the ribbon in the upper part of the Map view page.

When pressing the Measure icon, a dialogue pops up. In this dialogue it is possible to choose between two options:

- **Line.** When this option is selected, the distance is calculated along a straight line. The first click on the map locks the start point of the line. While the cursor moves on the map it is possible to see the line along which the distance is measured. At the same time, the length of the line is displayed in the dialogue.
- **Path.** This option must be selected to measure length along paths that are not straight. The first click on the map select the start point of the path. When the cursor moves further on the map and additional points are selected with a click, the distance from the start point are added to the measured length and displayed in the measure tool window. The specified path it is not visible on the map.

In the lower right corner of the Measure tool dialogue, the following two buttons are shown:

Clear. Click on this button to reset to 0 the distance displayed in the Measure tool dialogue.

Close. Click on this button to close the Measure tool dialogue.

Tabular view

The *Tabular view* provides a dialogue-style interface to the model parameter group selected in the tree view. In the *Tabular view* the properties of the model parameter groups are specified and edited. In some cases the *Tabular view* consists of one or more pages. Multiple pages will be shown as tabs which may be shown or hidden based on the individual parameter settings.

In general the tabular view is divided into an upper part with fields and a lower part with an overview table. When changing between records in the overview table the upper part will be updated with parameters for the active record.

Overview table

The overview table contains all attributes for the object group. The records can be sorted by clicking on the header.

One or more cells can be copied to the clipboard by selecting the relevant cells (they will turn blue) and either right-clicking and choose 'Copy' or use Ctrl+C.

One or more cells can be pasted to the table by standing in the top left cell of the group of cells to be overwritten and either right-clicking and choose 'Paste' or use Ctrl+V. Any value pasted to a read-only cell will be ignored.



Note that in tables related to some objects displayed on the map e.g. branches and catchments it is not possible to add new records - only existing records will be overwritten. For other tables new records are created.

In order to delete several objects in the table the entire row must be selected by clicking on the very left column in the grid (row will turn blue) and clicking 'Delete' on the keyboard.

Right-clicking in a table related to objects displayed on the map will give the option 'Zoom to active'. This will zoom the *Map view* to the active record.

For tables related to objects displayed on the map the first column is a checkmark. The checkmark shows if the element is selected on the map. The selection status can be changed either in from the map or the overview table.

Selected attributes in the *Tabular view* has a corresponding line in the *Property window*. The *Tabular view* and the *Property window* are synchronised – a change in either will be reflected in the other. Thus, the same parameters can be edited in either of the two views.

2.2.3 Property view

The *Property view* provides another way of displaying and editing the properties of model parameter groups – including the model objects. The same properties can be edited through the *Tabular view*. The two places are syn-



chronised – any change made in the *Tabular view* is reflected in the *Property view* and vice versa.

The *Property view* will show properties for the active model object group. The data is grouped by a unique identifier (ID) and the ID is highlighted in dark grey for the active record.

The *Property window* may be hidden by unselecting *Property view* in the 'View' drop-down menu in the MIKE Zero menu bar.

The content of *Property window* may be defined to either show all model group objects or selected objects only by setting the '*Property view: Showselected*' in the 'View' drop-down menu in the MIKE Zero menu bar.

2.2.4 Output windows

The *Output windows* can display a collection of different windows depending on the situation. Per default the windows are shown as tabs. The individual windows are described below.

All type of output windows may be shown or hidden by unselecting the output window in the 'View' drop-down menu in the MIKE Zero menu bar. If new information is shown in a hidden window the window will automatically be shown.

Validation

The *Validation* window shows validation status messages and error messages. Each time a model object is edited e.g. changing data in the *Tabular view* or the *Property view* the data is validated and any errors are shown in the *Validation* window.

Some extended validation checks are not performed on the fly, but can manually be run by selecting '*Extended validation*' from the 'Run' drop-down menu in the MIKE Zero menu bar. Running the *Extended validation* will perform both the standard and extended validation checks.



Note that error messages from the extended validation checks are only updated when a new extended validation is performed. Hence extended validation messages may still appear even if the error has been corrected.

Running a simulation will automatically perform an extended validation.



Simulation

The *Simulation* window shows progress and output from the computational engine.



Time series

The *Time series* window is automatically shown when plotting either input or output times series.

To add an input time series to the window, click on the  button next to the time series in *Tabular view* and select 'Add to new plot' or 'Add to current plot'. An input time series may also be added to the window by clicking on the input time series grid cell in the *Property view*. A  button will appear in the right hand side of the cell and clicking on that button gives the options of 'Add to new plot' or 'Add to current plot'.

To add an output time series to the window, navigate to the relevant model parameter group in *Result tree view*, right click on the relevant model object in the *Property view* and select the time series to be added to a new/current plot in the *Time Series* window. See section 2.6 Viewing Results (p. 39) for more details.

The *Time Series* window can contain one or more plots which are shown as tabs. A plot can be renamed by right-clicking on the tab and selecting 'Rename'.

A plot can contain one or more time series. The layout of a plot has 3 controls. The first control shown on the left contains a list of each time series in the plot grouped by file name and time series type. Time series can be removed from the plot by right-clicking and choosing 'Remove' on either an individual time series or a group of time series.

The middle control shows the data of the time series. Depending on the focus in the left control either one or a group of time series are shown in the middle control. Data can be copied to the clipboard by selecting one or more cells and either right-clicking and choosing 'Copy' or using 'Ctrl+C'.

The right control shows all time series added to the plot. Right-clicking on the plot provides a sub-menu for zooming, printing and user settings such as the use of secondary axis, position of legends, etc.

The *Symbology* dialogue from the sub-menu gives access to full control for specifying symbology, titles etc. for the plot.



Note on zoom in the window: Drawing a rectangle with the mouse from top left to bottom right will zoom in and drawing a rectangle from bottom right to top left will zoom out in the *Time series* window.

The configuration of plots is saved with the setup and will automatically appear when the setup is reopened. Any time series not available (either because they come from files that are not reloaded or because the files no longer contain the specified time series) are automatically removed from the plots.



Cross section

The *Cross section* window gives an option to graphically present cross sections in the river network.

Plotting of cross sections can be enabled by clicking the 'Plot' button under the *Cross sections* ribbon. With the 'Plot' feature active, cross sections can be plotted by clicking at cross section within the *Map view*.

The *Cross section* window presents individual cross sections and thereby, enables a visual analysis of any cross section within in the river model.



Drawing a rectangle with the mouse from top left to bottom right will zoom in and drawing a rectangle from bottom right to top left will zoom out in the *Cross section* window.

For detailed analysis of raw or processed data or plotting of multiple sections in same plot, it is required to open the Cross sections node in the tree view.

Structure

In the *Structure* window it is possible to plot a hydraulic structure together with the nearest upstream and downstream cross sections respectively. This is an effective way of checking the validity of the structure geometry. Structures introduce a head loss at the structure location through a contraction and expansion of the flow area. The structure geometry must therefore be smaller than the cross sections at any level within the structure.

To plot a structure, navigate to the structure *Tabular view* by selecting the structure in the ribbon and then changing to *Tabular view* or by activating the required structure type in the *Tree view* and opening the *Tabular view*. Plotting of the structure is made by pressing the 'Plot/Refresh' button in the *Graphics* section of the dialogue window.

It is possible to plot Culverts, Weirs, Bridges and Gates.



Drawing a rectangle with the mouse from top left to bottom right will zoom in and drawing a rectangle from bottom right to top left will zoom out in the *Structure* window.

Longitudinal profile

The *Longitudinal profile* window gives an option to show longitudinal profiles along the river network. This view can display the following profiles:

- **Thalweg:** This shows the bottom level of the river bed along the river network. The bottom level is the lowest level in cross sections, between marker 1 and marker 3 of the cross sections.
- **Left extent:** This line shows the elevation of the left end of the cross sections, obtained from marker 1 in each cross section.



- **Left bank:** This line shows the elevation of the left bank of the cross sections, obtained from marker 4.
- **Right bank:** This line shows the elevation of the right bank of the cross sections, obtained from marker 5.
- **Right extent:** This line shows the elevation of the right end of the cross sections, obtained from marker 3.

It also shows vertical lines indicating the locations of cross sections, as well as polygons indicating the locations of structures.

The longitudinal profile view can be enabled by clicking the '*Trace route*' button under the *Map* ribbon. With the '*Trace route*' button active, it is then necessary to trace the route on the *Map view* defining the location of the longitudinal profile. A first click on a branch will start the route tracing, additional single clicks will add intermediate locations to define the route to follow, whereas a double-click will stop the route tracing. The longitudinal profile view will then show up. For looped river networks, the selected route is always the shortest path between two points.

It is then possible to edit the route, by dragging any intermediate point used to trace the route to another location. Double-clicking further downstream the route will extend the route down to the new selected location. Whenever the route is modified, the longitudinal profile is updated simultaneously.

Right-clicking on the graphical view provides a sub-menu for zooming and for copying or saving as image file. The *Symbology dialogue* from the sub-menu gives access to display options, including:

- Control of the symbology for the various profiles (controlling line type, color, etc.)
- Show or hide locations and labels of structures
- Show or hide locations and labels of cross sections
- Control the visibility, location and title of the legend
- Show or hide the branch names and their chainage range above the horizontal axis
- Reverse the direction of the profiles: when the *Reverse profiles* option is selected, the start point of the route is shown on the right-hand side of the plot and the end point on the left-hand side.

2.2.5 Ribbon

The ribbon is located above the *Map view* and gives access to graphically editing of the geographically positioned objects in the model. Each model object will have a corresponding set of tools. The most common are described below.



Select

Using the function *Select* will select the model object on the map. Choose *Select* and then click on the element on the map. When an element is selected it will change colour to light blue. The selected elements will also be highlighted in the *Property view*.

For some model objects there are multiple types controlled by the same ribbon e.g. Structures. For these model objects the *Select* function will only select objects of the active type (determined by the combobox *Type* in the same ribbon), whereas the *Select any* function will select any object of the active group.

Add

Using the function *Add* will add the model object on the map. Choose *Add* and then click on the location on the map where the object should be inserted. Some model objects can only be inserted on a branch e.g. river nodes or structures. The cursor will change to a cross when the cursor is located at a valid place on the map.

For some model objects it is possible to use the function *Add X, Y* which will insert the object based on a user defined coordinate. If the given coordinate is not valid the object will be inserted at the closest valid location.

Branches and *Catchments* have multiple options for Adding data:


- **Branches:** 'Add' creates a branch polyline from user-defined digitisation on the map. 'Add trace' will add a delineated (traced) river branch when defining river branches through *River tracing and Catchment delineation* using DEM
- **Catchments:** 'Add digitise' creates a catchment polygon from user-defined digitisation on the map. 'Add delineate' add a delineated catchment when defining catchments through the *River tracing and Catchment delineation* using DEM



Note: the above functionality will add model objects one by one on the map. It is also possible to import multiple objects at once, from a shape file, in which case the location of objects is directly read from this shape file. To achieve this, please see section Import from Shapefile.

Move

Using the function *Move* will move a model object on the map. *Move* option is present for point data objects (e.g. structures, storages, initial conditionsetc.).

Choose *Move* and then click on the object that should be moved. The selected object will change colour to light blue and a small symbol  is




shown. When clicking on the symbol the model object can be dragged to a new location.

For some model objects there are multiple types controlled by the same ribbon e.g. Structures. For these model objects the *Move* function will only move objects of the active type (determined by the combobox *Type* in the same ribbon), whereas the *Move any* function will move any object of the active group.

Edit

Using the function *Edit* will move a model object on the map. *Edit* option is present for polyline or polygon data objects (e.g. branches, catchments and alignment lines). *Edit* can be used to either edit the shape of a selected object or to move the entire object.


Choose *Edit* and then click on the object that should be edited. The selected object will change colour to light blue and a small symbol  is shown. When clicking on the symbol the entire model object can be dragged to a new location.

Edit polyline or polygon features has the following options for vertices (or points) in MIKE HYDRO:

- **Move vertices:** Individual vertices can be moved by click and drag when positioning the mouse cursor on top of an existing vertice.
- **Delete vertices:** Individual vertices can be deleted by double-click when positioning the mouse cursor on top of the point to be deleted
- **Add vertices:** Additional vertices can be added with a single mouse click by pointing to a location where the new vertice must be located.
Note: It can be advantageous to click and hold the mouse button down for a second or so to insert a new point. This will avoid that a 'fast single-click' unselects the actual feature.

When editing a branch segment feature e.g boundaries or MIKE SHE Coupling only the end points of the branch segment can be moved. The line between the two points will follow the branch.

Delete

Using the function *Delete* will delete a model object on the map. Choose *Delete* and then click on the object that should be deleted. When the cursor is close to an object that can be deleted it will change symbol to an arrow . When clicking on the map the object is deleted.

For some model objects there are multiple types controlled by the same ribbon e.g. Structures. For these model objects the *Delete* function will only delete objects of the active type (determined by the combobox *Type* in the



same ribbon), whereas the *Delete any* function will delete any object of the active group.

2.2.6 Menu bar

The menu bar contains functionality for viewing and working with your setup. For details about the individual functionality see section 2.4 MIKE Zero Menu Bar (p. 32).

2.3 Layout

MIKE HYDRO has an IDE-style user interface (IDE=Integrated Development Environment) where all windows reside under a single parent window, referred to as the MIKE Zero shell.

The shell contains dockable and collapsible child windows, tabbed windows and splitters for resizing of child windows.

The default windows docking layout is shown in Figure 2.2, but the user may change this.

A window is undocked by either right-clicking in the heading and selecting '*Float*' or by dragging the heading away from its current position. If only a tab in a parent window containing multiple tabs should be undocked then you can either right-click or drag the individual tab.

A window is docked again by either right-clicking in the heading and selecting '*Dock*' or by double-clicking the heading. In both cases the window will be docked again at the last location it was docked.

A window can also be docked by manually placing the window at a new location. Select the heading of the window that should be docked and start dragging the window and a docking guide will appear (see Figure 2.3). While still dragging the window move the cursor to the new location using the docking guide, when the cursor is placed on the docking guide the location will be highlighted in blue. Once the correct location is highlighted the left mouse button is released and the window will be docked.

The docking guide will always relate to the active window, where the active window is the window located below the position of the cursor.

The docking guide will display small symbols where a docking location is possible. In the middle are 9 locations relative to the active window, where the new window can either be placed as a tab to the active window (middle symbol) or left/right/below/above the active window (symbols around the middle). On the edge of the shell are 4 other locations relative to the shell, where the new window can be placed to the left/right/top/bottom of the shell.

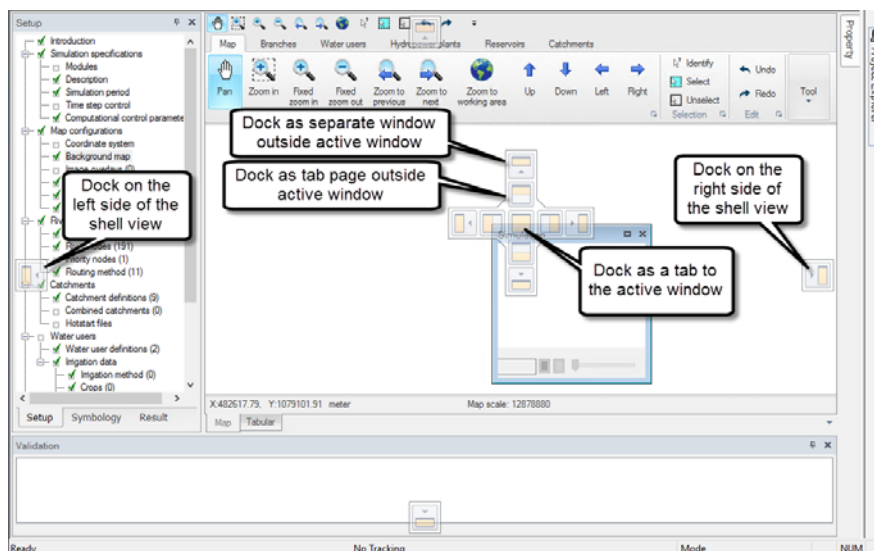


Figure 2.3 Docking guide

Right-clicking on a docked window gives the option of 'Auto Hide'. This will automatically hide the window when it is not in focus and only display a tab. Clicking the tab will show the full window again.

MIKE HYDRO has three type of windows available: *Map view*, *Tabular view* and standard views.

The *Map view* is fixed to the middle of the shell. It cannot be undocked, moved or resized. The size of the map will automatically fit any available space in the shell not used by other windows. Hence indirectly the map can be resized by resizing the other windows.

The *Tabular view* can be both undocked and docked in any location within the shell, however it needs to be docked with another window. The *Tabular view* differs from standard windows because it cannot be closed.

The standard windows which includes the *Tree view*, *Property view* and *Output windows* can be hidden, undocked or docked in any location within the shell.

Any changes made to the layout are automatically saved with the setup. It is further possible to save a specific layout to a file and re-use in other project. The layout can easily be reset to the default configuration through the *View* menu. See section 2.4.3 View menu (p. 34) for further details.

2.4 MIKE Zero Menu Bar

The MIKE Zero menu bar contains some menus that are generic to the common MIKE Zero framework and some that are specific to working with MIKE



HYDRO. For details about the common MIKE Zero framework see '*MIKE Zero, The Common DHI User Interface for Project Oriented Water Modelling, User Guide*' referred to as the 'MIKE Zero User Guide' below.

2.4.1 File menu

The menus *New*, *Open*, *Close*, *Close Project*, *Save*, *Save All*, *Save As*, *Save Project As Template*, *VCS Control*, *Print Setup*, *Print Preview*, *Print* and *Options* are all part of the MIKE Zero framework see 'MIKE Zero User Guide' for more details.

Import

Import from shape file

A comprehensive shape file import (and export) feature is available, enabling model components to be imported through shape files.

More details on the import from shape file option, see "Import from Shapefile" on page 289

Import from MIKE 11

An existing MIKE 11 setup can be imported by selecting '*Import*' in the '*File*' drop-down menu in the MIKE Zero menu bar and choosing '*Import from MIKE 11*'.

For more details on importing from MIKE 11 files, see "Import from MIKE 11" on page 291

Import from ISIS

The main features of an ISIS set-up can be imported by selecting '*Import*' in the '*File*' drop-down menu in the MIKE Zero menu bar and choosing '*Import from ISIS*'.

A more detailed description of this functionality is given in section Import from ISIS (p. 293).

Export

Export to shape files

Model objects can be exported by selecting '*Export*' in the '*File*' drop-down menu in the MIKE Zero menu bar and choosing '*Export to shape files*'.

All shape files are exported in the coordinate system for features (see 5.1 Coordinate System (p. 65))

For additional details on export to shape files, see "Export to Shapefile" on page 294



Export to MIKE 11 files

The setup can be exported to MIKE 11 setup by selecting 'Export' in the 'File' drop-down menu in the MIKE Zero menu bar and choosing 'Export to MIKE 11 files'.

For additional details on export of data to MIKE 11, see "Export to MIKE 11" on page 296



Print

All print functionality is related to the *Map View*.

Options

Install examples

Several examples are included in the MIKE HYDRO installation. Select MIKE_HYDRO in the pop-up menu and click 'Install'. The examples may then be opened from the folder named '...\MIKE Zero Projects\MIKE_HYDRO'.

User settings

The 'User Settings' dialogue contains general settings for MIKE Zero. For more details see 'MIKE Zero User Guide'.

The tab *MIKE HYDRO* relates specifically to MIKE HYDRO and here it is possible to control if the setup wizard should be displayed when creating a new setup see section 2.1 How to Start MIKE HYDRO (p. 17).

The *MIKE HYDRO* tab also controls the language used in the dialogue windows.

2.4.2 Edit menu

For Undo and Redo options, see .

The other options are part of the MIKE Zero framework see 'MIKE Zero User Guide' for more details.

2.4.3 View menu

The menus *Project Explorer*, *Project Map*, *Start Page*, *Simulation History* and *Toolbar* are all part of the MIKE Zero framework see 'MIKE Zero User Guide' for more details.

Load layout

The layout controls the position of all windows. Here it is possible to load a MIKE HYDRO layout saved from another project.



Save layout

The layout controls the position of all windows in MIKE HYDRO. The layout is automatically saved with the current setup, however here it is possible to save the layout to be used in another project.

Set as default layout

Use this option to define a personal default layout for MIKE HYDRO.

Restore layout

This menu will reset the position of all windows to the default layout.

Load symbology

Load symbology settings from a symbology file (*.mhsym).

Save symbology

Save the current symbology settings to a symbology file (*.mhsym).

Reset symbology

Resets symbology settings to the default settings.

Set as default symbology

Option for creating user-defined default symbology settings.

Reset default symbology

Restore the symbology settings to the original as provided in the MIKE HYDRO installation

Export graphics

Option for saving MIKE HYDRO graphics to either Clipboard or graphics file.

Property view: show selected

Specify whether the property view should present all data of the selected data object or only selected data. A checkmark in front of this item states that only Selected objects are presented.

Show or hide features and windows

The visualisation of some of the views and menu bars can be switched on or off in the list of items presented below the 'Toolbar...'. Click on the different



items to show or hide the specific item in MIKE HYDRO. If the item is visible a Checkmark is presented in front of the item on the View list.

2.4.4 Run

The *Run* menu contains options for calculation, validation and simulation of model components.

Recalculate flow conditions

The Q/h relations of structures (weirs and culverts) must be recalculated if any changes are made to the structures and/or the up/down-stream cross sections.

Only structures where the flag '*Allow for recalculation*' has been enabled will be recalculated.

Extended validation

Before running the model, it is recommended to run an extended validation of the model setup. Any validation issues will be shown in the *Validation* window see section Validation (p. 25).

Simulation

To execute the model engine, select '*Simulation*'. Output from the computational engine (status and error messages) are shown in the *Simulation* window see section Simulation (p. 25).

2.4.5 Tools

The *Tools* menu contains different tools for creating and editing the setup objects. For details on the individually tools see section 16 Main menu bar features (p. 289).

2.5 Creating Input Files

Most data is saved within the MIKE HYDRO file, but some input data is stored in external files. Input files will typical consist of time series saved in dfs0 files see section 2.5.1 Create a new dfs0 file (p. 37) for more details.

Where a link to an external file must be created a button with 3 dots is shown. Clicking on the button will show a sub-menu see Figure 2.4.

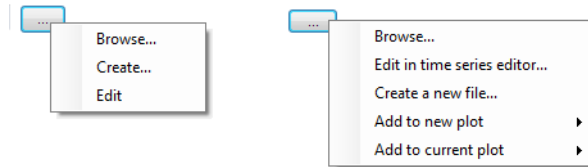


Figure 2.4 Buttons used for input files

The general functionality of the external file buttons has the options:

- **Browse:** Allows you to browse for an existing file.
- **Create:** Will create a file with default settings and automatically open the file for editing with the corresponding editor.
- **Edit:** Will open the file for editing with the corresponding editor.

For files that can be shown on a graph it will also be possible to plot the input file to the *Time Series* window.

2.5.1 Create a new dfs0 file

When creating a new dfs0 file using the button for external files, a dialogue window will be opened. This dialogue is used in different situations for creating a time series (dfs0) file or a table file. The model automatically detects the type of file needed and adjusts the dialogue accordingly.

Create a new time series file

To create a new time series file, the following properties must be specified:

- **Filename:** You may either browse for or manually type the name and location of the time series file.
- **Period:** You may choose the start and end date of the time series using the date-time pickers. The default values will be the simulation period for your setup.
- **Time axis type:** You may choose among non-equidistant calendar axis type and equidistant calendar axis type. For the non-equidistant type, the interval between data values may vary. For the equidistant type, equal time spacing between each time step will be enforced.
- **Time series interval:** The time series interval is the size of the time step.

Example: Specifying the following:

- Start date: 01-01-1980
- End date: 31-12-1981
- Time series interval: 1 month



will cause a time series file with 12 time steps to be created.

- **Items:** Depending on the context in which the file is being created, the file will be created with one or more data items.
- **Item name:** The name of the data item
- **Uniform value:** All the time series files which are created with this dialogue will contain the value specified here in all time steps (uniform value). To create files with values varying over time (non-uniform), first create a uniform value time series and then edit it in the time series editor.
- **Time series type:** Contains information on how the data values are interpreted. The possible types are:
 - **Instantaneous:** Means that the values are representative at one precise instant. For example, a measured water level is an instantaneous value. Linear variation between time steps is assumed.
 - **Accumulated:** Means that the values are representative of one successive accumulation over time from the start of the event (start time specified) to the current time step (.i.e. accumulated over the entire series). For example, the rainfall is accumulated over the year if we have recorded monthly rainfall values. Linear variation between time steps is assumed.
 - **Step Accumulated:** Means values are representative of an accumulation over a time step. For example, rainfall is a step-accumulated value if the rain depth is recorded at different intervals. Values represent the time span between the previous time step and the current time step.

Example: *Rainfall measurement is started at 10:00:00. At 11:00:00 someone picks the recipient, registers the value of 10 mm and empties the recipient. At 12:00:00 the same process is repeated but with a value of 15 mm and so on. Thus, the time series will have the value of 10 mm at time step 11:00:00 and the value 15 mm at 12:00:00 etc.*
 - **Mean step accumulated:** Means that values are representative of an average accumulation per time step. For example, the average rain rate is a mean step accumulated value. Values represent the time span between the previous time step and the current time step.
 - **Reverse mean step accumulated:** Is equal to *Mean Step Accumulated* type, but values represent the time span between the current time step and the next time step. Used for forecasting purposes.



- **Item type:** The type of data item. The drop-down list can be used to choose one of the possible item types. The choices available will depend on the context.
- **Item Unit:** The unit type of the data item. The drop-down list can be used to choose one of the possible unit types. The choices available will depend on the item type.



Note: For more information on time series types, item types and units see the '*DFS File User Guide*' in the MIKE Zero Documentation.

Create a new table file

To create a new table file the following properties must be specified:

- **Filename:** You may either browse for or manually type the name and location of the time series file.
- **No. of timesteps:** This defines the number of rows in the table (or the number of x-axis points on a curve).
- **X-axis unit:** The x-axis unit can be selected from the drop-down menu.
- **Items:** Depending on the context in which the file is being created, the file will be created with one or more data items.
- **Item name:** The name of the data item.
- **Values:** Here the pair-wise values of the x-axis and the item are entered. The table file can consecutively be edited in the time series editor.
- **Value series type:** Contains information on how the data values are interpreted. The possible types are the same as for a dfs0 file see section Create a new time series file (p. 37).
- **Item type:** The type of data item. The drop-down list can be used to choose one of the possible item types. The choices available will depend on the context.
- **Item Unit:** The unit type of the data item. The drop-down list can be used to choose one of the possible unit types. The choices available will depend on the item type.



Note: For more information on time series types, item types and units see the '*DFS File User Guide*' in the MIKE Zero Documentation.

2.6 Viewing Results

Results are viewed from the *Result* tab in the *Tree view*. It is possible to select results either from a list in the *Property view* or from the *Map view*. The results are viewed in the *Time Series* window see section Time series (p. 26).

Result files and other time series are controlled from the *Tree view*. After a simulation the results are automatically added to the *Tree view*. Results from



previous simulations, input files or measured data (dfs0 files) can be loaded through the 'Add file...' button for comparison or calibration.

Files shown in the tree are automatically linked to the setup e.g. when re-opening the setup the same files will be available in the *Result* tree. If a setup contains links to files that are deleted or moved the link will automatically be removed. It is possible to manually remove the link to a file by right-clicking on the file name and choose 'Remove'.

2.6.1 River result files

The output from a river simulation is automatically loaded to the result view. However if results from a previous simulation should be loaded for comparison the corresponding res1d file should be loaded.

At the top of the *Property view* the available data types are shown in a combobox. When choosing a data type the available branches are shown below. When choosing a branch the available grid points are displayed below.

Right-clicking in the *Property view* shows a menu for either adding the result to a new plot or the current plot. Double-clicking on a point will automatically add the result to the current plot.

Selecting a point in the *Property view* highlights both the point and the related branch on the map.



Note: To find the selected branch the button 'Zoom to selected branch' can be used.

Selecting a branch or point in the *Map view* automatically selects the related time series in the *Property view*. Right-clicking on a point gives access to the menu 'Result to plot' where results can be plotted either on a new plot or the current plot.



Note: It is only possible to select results in the Map view, if the *Map* ribbon is active and set to 'Pan'.

2.6.2 Dfs0 files

Dfs0 files can be used for input, output and measured time series. Dfs0 files does not contain information about the network and is therefore not connected to the *Map view*.

When adding a dfs0 file the *Property view* is automatically filled with a list of all time series available in the dfs0 file. A filter based on the item type is shown at the top of the *Property view*.



2.7 Getting Help

Online help is available in all dialogues of MIKE HYDRO. The online help system utilises the Microsoft help technology known as HTML Help.

The online help can be accessed in two ways:

- Through the menu *Help* and choosing '*Help Topics*'
- By pressing F1 in any dialogue window, which automatically will direct you to the relevant section of the online help.



Note: In some cases the dialogue box/window must be activated (e.g. by clicking in a cell) to ensure correct links to the online help.

2.8 Examples

A useful way to quickly get started with MIKE HYDRO is to open the examples that are included in the installation.

To install the examples go to the *File* drop-down menu, select '*Options*' and click '*Install Examples...*'. In the pop-up menu, select MIKE_HYDRO and click '*Install*'. The examples may then be opened from the folder named '*...\MIKE Zero Projects\MIKE_HYDRO*'

2.8.1 River module examples

This section includes a brief overview of the River Modules examples included in the MIKE HYDRO Installation.

Demo

The enclosed Demo example comes from a setup of Cali River which has been modified to reduce the original number of input elements (cross-sections and river branches) to keep within the limitations of MIKE HYDRO Demo version.

The setup comprises 3 branches and 10 cross-sections. The boundaries consist of a single recorded upstream inflow and two downstream water levels conditions.

Using the predefined settings for input files, the simulation period and time step, it is possible to perform a simulation and view the results.



Vida

This example is a setup from a stream (small river) in Denmark, named 'Vid-Å'. The setup was developed by DHI for a project conducted in 1997.

The Vida setup comprises a main river branch with several smaller tributaries feeding into the main river. Boundary conditions are defined as inflow hydrographs on all upstream boundaries and a downstream tidal boundary at the sea. The downstream boundary is defined by applying measured water levels covering a large number of tidal periods.

All input files required to perform the hydrodynamic computation are present.



3 Tutorial: Setting Up a Simple Model

The current chapter includes a short description on how to operate MIKE HYDRO to set up simple examples of the River Module.

Setting up a Simple River Model

3.1 Setting up a Simple River Model

This short tutorial summarises the steps needed to set up a simple river model.

3.1.1 Create a new document

To create a new document, start MIKE Zero and choose the MIKE HYDRO model document type from the MIKE HYDRO group.

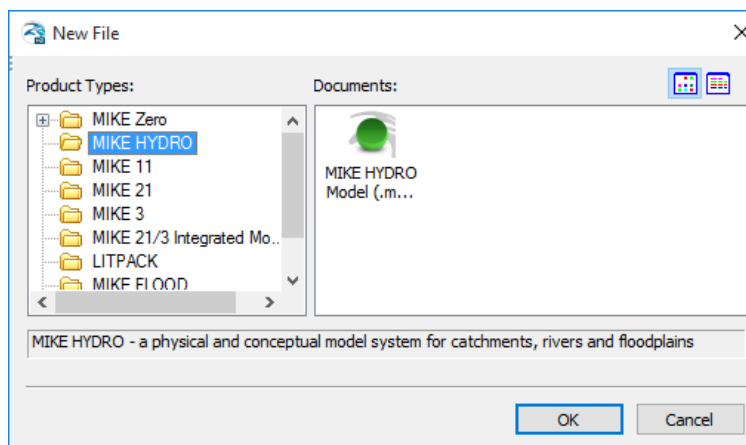


Figure 3.1 Create new MIKE HYDRO document

Press the 'OK' button and an empty document will be created and the setup wizard will automatically open.

The wizard will guide you through setting the framework of your project:

- On the '*Modules*' page define the model type to '*River*'.
- On the '*Description*' page define the title to '*River Tutorial*'. The title will set the default name of your result files.

- On the '*Coordinate system*' page keep both the map and feature coordinate system to Sphere Mercator (default).
- On the '*Background map*' page set the background map to Google map.
- On the '*Background layers*' and '*Working area*' pages press next without specifying anything.

The framework is now in place, so next save the setup to a MIKE HYDRO file.

3.1.2 Defining branches

Zoom to the area of Vid-Aa in Denmark see green box in Figure 3.2. Note the location is not important for the exercise, hence any position can be used.

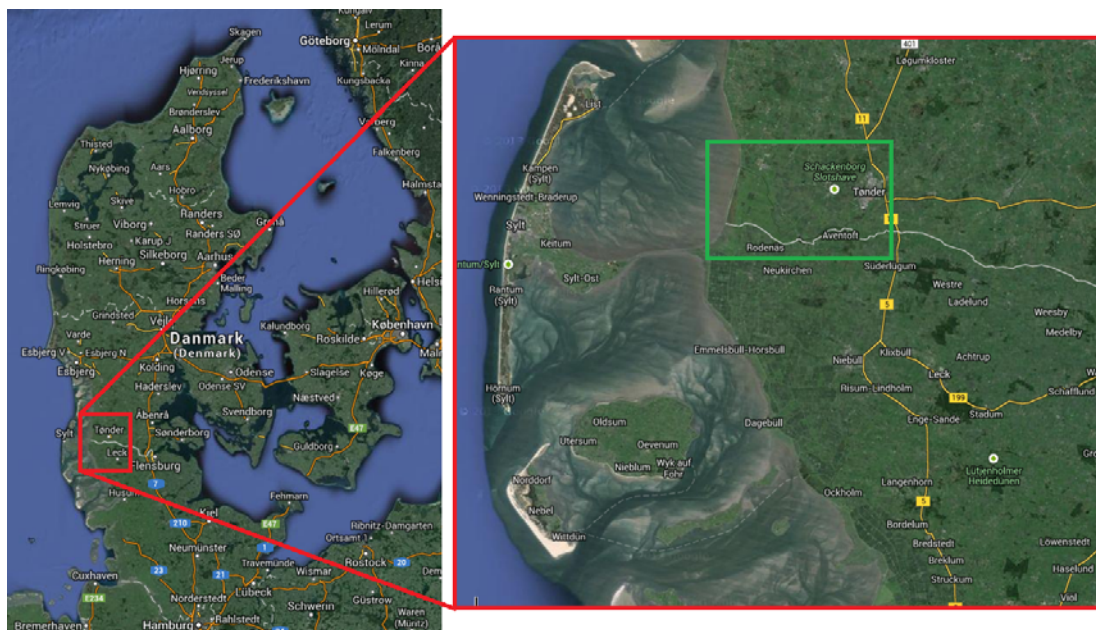


Figure 3.2 Vid-Aa location

Define a branch using the '*Branches*' ribbon in the *Map view*. Select 'Add' and digitise the branch on the map by clicking on the most upstream location and then clicking once on the map for each bend until the most downstream location. On the last point finish the branch with a double-click.

In the *Tabular view* of the branches define the branch name to '*Main*' and set the Topo ID to 2014.

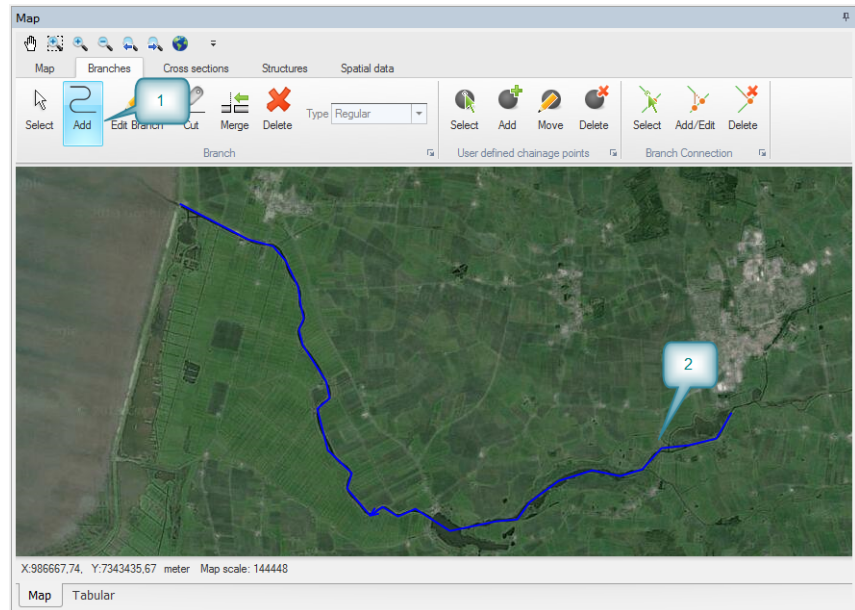


Figure 3.3 Define a branch

Define chainages along branch

Once a branch is defined the chainages along the branch is automatically calculated based on the distance from the first point. The default chainage of the first point in a branch is zero. The calculated chainages may not be optimal and the user may wish to set the chainage manually. This is done by inserted user defined chainage points.

Insert a user defined chainage point at the first and last end by using the '*Branches*' ribbon and '*Add*' for a user defined chainage point. If the point is not placed at the end the point can later be moved by using '*Move*' and dragging the point to the end of the branch.

Go to the Branches page in *Tabular view* and the *User defined chainage points* tab to defined the chainages. Here we define the first point to have chainage 0 and the last to chainage 40000.

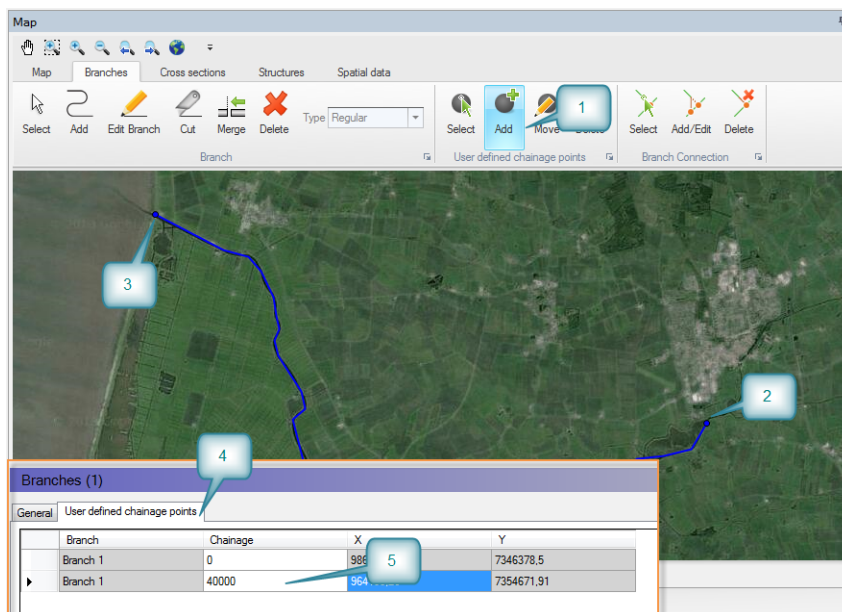


Figure 3.4 Define user defined chainage points

Define connecting branches

Branches can be linked together through branch connections. Define a tributary to your main branch in the same way as before - again starting with the upstream point and ending with the downstream point.

Set the branch name to '*Tributary*' and the Topo ID to 2014.

If the last point is located very close to the main branch it may automatically connect, but if not automatically connected the branches should be manually connected. Select the '*Branches*' ribbon and '*Add/Edit*' branch connections. First click on the last point in the tributary and drag the cursor to the point on the main branch where the tributary should connect. When a connection is possible the cursor turns to a black circle and the branch is selected. Releasing the mouse button will create the connection which is shown with a connection line.

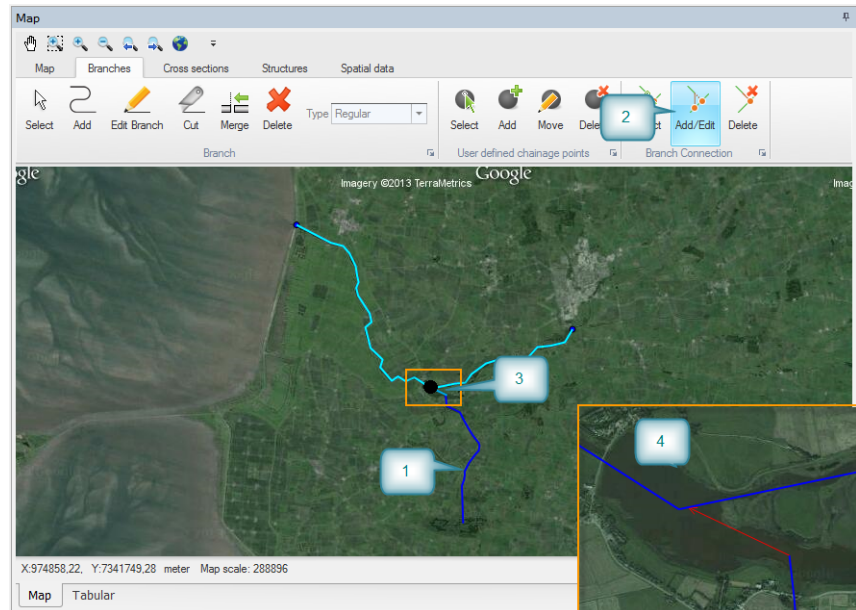


Figure 3.5 Define branch connections

3.1.3 Defining cross sections

Cross sections can be drawn directly on the map. It is required as a minimum to have a cross section at each end of all branches. Insert a cross section by using the 'Cross sections' ribbon. Select 'Add' a cross section and digitise the location on the map. Click on the embankment on one side, drag the cursor and release at the other embankment.

Insert cross sections at the upstream and downstream end of both the main branch and the tributary. Note that the new cross section is shown in the 'Cross section' window at the bottom.

When saving the setup you will be prompted for the name of a cross section file. Place the file next to your setup.

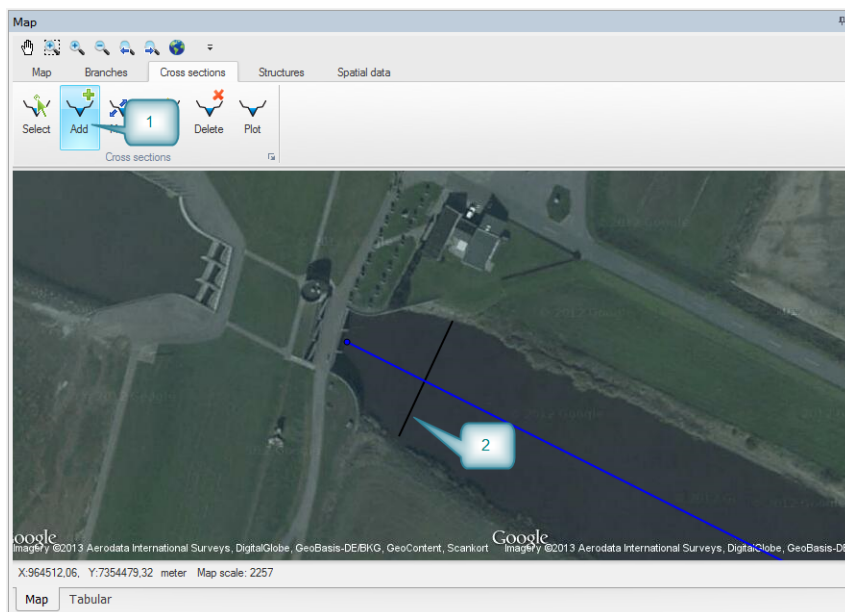


Figure 3.6 Define cross sections

3.1.4 Defining boundary conditions

Every open end of a river branch (not connected to other branches) must have a boundary defined. Hence, in this example we must define boundaries for the upstream and downstream end of *'Main'* and for the upstream end of *'Tributary'*. Here we will define the boundary condition at the upstream ends of both branches as closed (i.e. zero discharge) and define a varying water level between 5 and 6 meters at the downstream end.

From the Boundary ribbon in Map view, select 'Add' and create new boundary conditions at the upstream and downstream point of *'Main'* and upstream end of *'Tributary'* by locating the cursor on these locations and make a single mouse click. Change to Tabular view for additional boundaries details specification.

First specify the upstream boundary of the main branch. Highlight this boundary in the overview grid and set the *Type* to *'Closed'*, the branch name is *'Main'* and adjust the chainage to 0.00 if a different value is present from the map definition. Now specify the upstream boundary of the tributary the same way as for main.

Finally, you need to specify the varying water level boundary with a time series file. As part of the boundary definition is to create a timeseries file used at this downstream boundary. In the *HD Boundary* select the *Input Type* to be *'Time varying'*. Press the file action button *'...'* and select *Create a new file* and specify the time series properties in the opening page. Press *Create file* to create the empty timeseries file and make a reference to it in the *File* field



for the boundary. Secondly, use the file action button '...' and select *Edit in Timeseries Editor* where timeseries values and dates can then be inserted.

Alternatively, it is possible to make a new Timeseries file through the MIKE Zero menu bar and choose 'File' - 'New' - 'File'. Select the MIKE Zero product type and the document 'Time Series (.dfs0)'. In the first shown dialogue window select to create a blank time series and in the following specify the data as shown in Figure 3.7.

File Properties

General Information

Title: Tutorial

Axis Information

Axis Type: Equidistant Calendar Axis

Start Time: 01-01-2014 00:00:00

Time Step: 0 [days]
06:00:00 [hour:min:sec]
0.000 [fraction of sec.]

No. of Timesteps: 13 Axis Units: undefined

Item Information

	Name	Type	Unit
1	WL	Water Level	meter

Insert Append Delete Item Filtering...

Figure 3.7 Define new dfs0 file

Once the 'OK' button is pressed the time variation of the water level can be entered as shown in Figure 3.8.

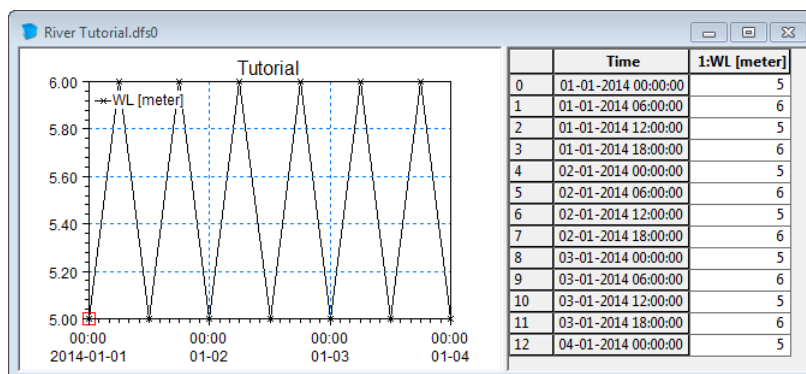


Figure 3.8 Define time series data

Save the file together with your setup.

Close the time series window and put focus on the boundary editor again. Create a new line in the upper grid and set the boundary description to 'Open', the boundary type to 'Water Level', branch name to 'Main' and the chainage to 40000. In the lower part set the TS Type to TS File and click the '...' button to choose the dfs0 file created earlier.

The figure shows a software window titled 'Standard boundaries (3)'. It contains configuration fields for a boundary and a table of defined boundaries. The configuration fields include 'Boundary' (ID: Main outflow, Type: Water level, Location type: Open), 'Location' (Branch name: Main, Chainage: 40000, Dwnstr. chainage: 40000), and 'HD boundary' (Include boundary: checked, Input type: Time varying, Scale: 1, File: Zero Projects\MIKE_HYDRO\River\Demo\HD time series.dfs0, Item: HD boundary ts). The table at the bottom lists the boundaries.

	ID	Type	Location type	Branch name	Upstream chainage	Downstream chainage	Catchment name	Runoff flow type
<input type="checkbox"/>	Main inflow	Closed	Open	Main	0	0		Total runoff
<input type="checkbox"/>	Trib inflow	Closed	Open	Tributary	0	0		Total runoff
<input checked="" type="checkbox"/>	Main outflow	Water level	Open	Main	40000	40000		Total runoff

Figure 3.9 Define boundaries



3.1.5 Define initial conditions

The initial conditions in the network should be set to correspond to the boundary conditions at start time of simulation. Go to the '*HD Initial conditions*' in the *Tree view* and switch to *Tabular view*. Under the '*Global values*' set the water level to 5 and the discharge to natural flow - this will apply to the entire network.

HD initial conditions (0)

Global values

Type: Level type: Discharge type:

☐ Local values

	Branch name	Chainage	Level type	Level / depth	Discharge type	Discharge

☐ Hotstart files

	Hotstart file	Use simulation start time	Date and time
	C:\Users\inkedmen\Documents\MIKE Zero Projects\MIK...	<input type="checkbox"/>	1989-07-27 00:00:00

Figure 3.10 Define initial conditions

3.1.6 Running a simulation

First set the simulation period by going to the '*Simulation period*' in the *Tree view*. Click the '*Apply default*' button to search the boundary conditions and apply the corresponding simulation period. In this setup the only restriction on the simulation period is the water level boundary defined earlier and the simulation period is automatically set to the extent of the time series file.

Secondly set the time step by going to the '*Time step control*' in the *Tree view*. Set a fixed time step of 30 minutes.

Now we are ready to run a simulation, but before executing a simulation you should validate the setup to ensure everything is specified correctly. In the MIKE Zero menu bar go to the menu 'Run' - 'Extended validation'. Any warnings or error message will appear in the '*Validation*' window.

If there are no errors continue to run the simulation by going to the menu 'Run' - 'Simulation'. Progress and messages are shown in the *Simulation* window.



3.1.7 Viewing results

When the simulation has finished the results are automatically loaded and the focus is shifted to the 'Result' tab in the *Tree view*.

On the right side choose to plot water level in the main branch by setting the drop-down menus in the *Property view*. Double-click in the point list on the first and last chainage and they are added to a plot in the *Time series* window.

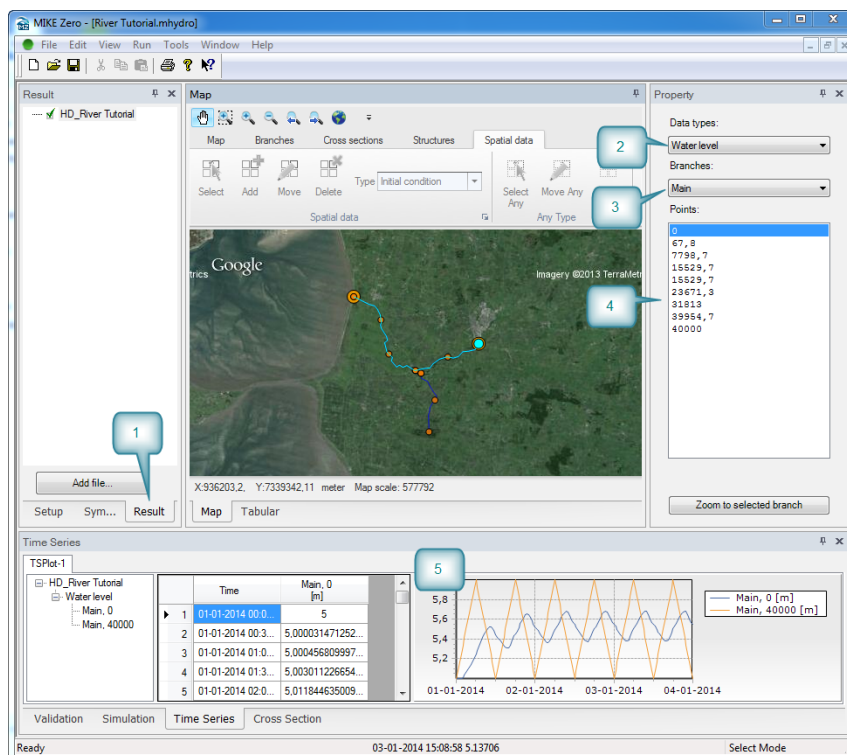


Figure 3.11 Viewing results



4 Simulation Specifications

The Simulation specifications includes the more overall control settings for model simulations such as; which sub-modules are included in the simulation, which simulation period and time step are to be applied and additionally, overall control parameters for the simulation performance are included.

The following pages are included for defining Simulation specification for each model type:

4.1 Modes

The first action to take in a MIKE HYDRO project definition is to specify whether the actual project is a Basin model or a River model. This selection is made in the upper drop down selection box. From this box, choose either 'Basin' or 'River' to activate one of these model types.

The content of the tree view in the GUI depends on the overall module selection as different features are available for the Basin and the River models respectively. Additionally, the content and appearance of the Modules page depends on the choice of 'Model type'.

Following the selection of 'Model type' there are a number of options for selecting additional features through activation of Model specific modules and global settings for the specific project (see Figure 4.1).

Modules	
Model type	
River	
River modules	
Data	Sim.
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/> Rainfall runoff
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/> Hydrodynamic
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/> Advection-Dispersion
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/> - Water quality
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/> Data assimilation

Figure 4.1 Model type and Module selection page for MIKE HYDRO River.

The 'Modules' section of this page contains two rows of checkboxes. These can be activated and deactivated by the user depending on the preferred GUI appearance and requirement for the project simulations.



The checkbox functionality are as follows:

- **'Data' :**
Controls whether the data related to the specific module or feature are visible in the tree view.
Enable the specific '*Data*' checkbox for all features that you want be visible and editable from the GUI tree view.
Important Notice: The Data checkbox activation does not enable a feature for simulation.
- **'Sim.'** :Control
Controls whether the specific module is active in the simulation or not.
'*Sim.*' checkbox must be enabled to include the feature in the simulation.
Note: It is not possible to activate a 'Sim' checkbox if the equivalent 'Data' checkbox is no active as well.
All '*Sim*' checkboxes are per default active

Module options are described in details in following sections.

4.1.1 River modules

The River modules includes specification of River specific features for the project. Options are:

- River modules
 - Rainfall runoff
 - Hydrodynamic
 - Advection-Dispersion
 - Water quality
 - Sediment transport
 - Data assimilation

Rainfall runoff

Rainfall-runoff modelling can be included in the River model.

Inclusion of Rainfall runoff in a simulation requires that one or more Catchments are defined in the River model. Runoff from catchments is calculated from user-defined Catchment characteristics and meteorological data.

Following rainfall-runoff models are available:

- **NAM:**
A lumped, conceptual rainfall-runoff model, simulating the overland flow, interflow, and baseflow components as a function of the moisture contents in four storages.



- **UHM:**
The Unit Hydrograph Model includes different loss models (constant, proportional) and the SCS method for estimating storm runoff.
- **Time Area:**
Runoff simulation based on the Time area method considering losses , size of contributing area as well as concentration time as function of catchment shape. The Time-area model is also known as 'Urban Runoff Model A'.
- **Kinematic Wave:**
The Kinematic wave runoff model calculates surface runoff as flow in open channels considering gravitational and friction forces only including various hydrological losses. The Kinematic wave model is also known as 'Urban Runoff Model B'.

The type of rainfall runoff model and associated, required input data are specified in the Catchment definitions page (cf. Section 7.1.2).

Rainfall runoff results can be used as input for a hydrodynamic simulation. It is possible to either run the two modules simultaneously, or to use a Rainfall runoff result file as input for the hydrodynamic simulation. The latter will reduce the simulation time, and is achieved by enabling the 'Data' but not the 'Simulation' for rainfall runoff.

Hydrodynamic

The *Hydrodynamic* checkboxes enable editing of River model data and activates the hydrodynamic river model simulations. Hydrodynamic river simulations in MIKE HYDRO River uses the MIKE 1D simulation engine (for details on the MIKE 1D simulation engine, please refer to the MIKE 1D reference manual).

Link to the MIKE 1D manual can be found in the MIKE HYDRO Documentation index).

Advection-Dispersion

Advection-Dispersion modelling include simulation of transport and spreading of pollutants as function of the hydrodynamic conditions in the rivers. The Advection-Dispersion module includes a simple decay function but no other water quality processes.

Advection-Dispersion and Hydrodynamic simulations can be performed simultaneously. However, it is also possible to decouple them, by using a Hydrodynamic result file as input for the Advection-Dispersion simulation. This will significantly reduce the simulation time, and is achieved by enabling the 'Data' but not the 'Simulation' for the Hydrodynamic module, while enabling both for the Advection-Dispersion module.



Water quality

Water quality processes in MIKE HYDRO River is included by use of MIKE ECO Lab, which is a highly flexible framework for defining water quality models. MIKE ECO Lab utilise a concept of templates where water quality models are defined with equations and variables, and the water quality option therefore, allows the usage of different water quality models/templates depending on the issue of concern. MIKE HYDRO Basin installation includes two predefined water quality templates that can be applied as is - or can be adjusted by the user to conform with the specific project requirements.

Sediment transport

Sediment transport can be described based on various model types (e.g. van Rijn, Meyer-Peter & Muller, Engelund-Hansen, Engelund-Fredsoe, Yang, or user-defined empirical formulas). Graded or mixed sediment descriptions can be applied by defining a number of different sediment fractions, which are treated separately by the sediment transport module. Sediment transport is computed from hydrodynamic conditions, and dynamic changes in the river morphology can in return affect the hydrodynamic conditions.

Data assimilation

Data assimilation module include an updating feature for continuous processing and updating of model states during simulations. Data assimilation is used for many types of applications, but in particular for flood forecasting (or other types of forecasting) where the forecasted values of river state conditions (water levels, flows etc.) is much more accurate and reliable when updating and error estimation is included as part of the forecast simulations.

4.2 Description

The *Description* dialogue includes the option for supplying a user specified Title and a Description for the simulation and the project in general.

Every simulation should be given a Title. Except from helping to describe the actual simulation performed the title will also be included as part of the default name for result files defined by MIKE HYDRO (see section 15.1 Standard Results (p. 271)).




Note: Simulation results with a title already used will overwrite the existing results unless a new result folder/file has been specified.

4.3 Simulation Period

The simulation period is generally defined by specifying the required Simulation start date and time and Simulation end date and time.



Date and time can be edited manually by clicking the start date or end date field and change the date (Year+Month+Day) and time (hour+minutes+seconds). In this case it is possible to use the right and left arrow keys to navigate between the date and time variables.

Alternatively, dates can be selected using the calendar icon; , which will open a calendar that allows to browse for the required date (see Figure 4.2). Note, that the simulation time will still have to entered manually after choosing a date from the calendar date selection.

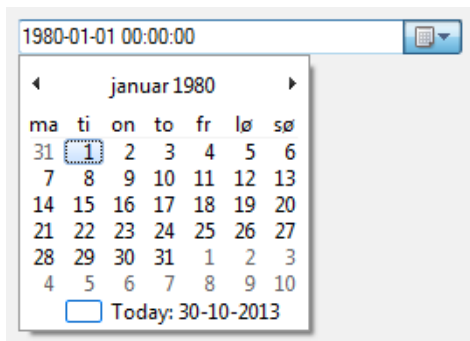


Figure 4.2 Calendar date selection option for simulation periods

Note that it is possible to use input time series for model input parameters, which have different length than the simulation period, and different temporal resolution than the simulation time step.

The *'Apply default'* button can be used to extract the possible simulation start-time and end-time. Once the button is activated a search of time-intervals in all time series files included in the actual simulation takes place and the earliest possible start-date and latest possible end-date is automatically transferred to the date-fields.



Note: If no dates are proposed from activating the button the most likely reason will be that there are no overlapping time series in the setup.

4.4 Time Step Control

Time step control features includes options for defining simulation period and time step amongst other parameters for the different modules.

Time step type

The drop-down menu includes three time step types:

- Fixed time step
- Tabulated time step
- Adaptive time step



The required definition of time step depends on the selected time step type. The time step type dependent specification is described below.

Time step definition

Rainfall runoff time step multiplier

If the simulation of rainfall runoff (RR) module has been enabled together with the hydrodynamic module (HD) in the Modules page, the simulation will run the two processes in parallel (HD and RR). RR- and HD-modules will typically operate on different time-scales where RR typically require larger time steps than what is required for the HD-module. The RR time step multiplier is therefore available to define the time step to use for the RR-module in case the two modules are simulated in parallel. RR time step will be the multiplum of the time step defined for the HD-model.

Fixed time step

Simulation uses a fixed time step throughout the entire simulation. The fixed time step must be entered as a number and the time unit selected from the drop-down menu (seconds, minutes, hours or days).

Tabulated time step

The tabulated time step is an option for having a temporal varying time step throughout the simulation. Tabulated time step is defined using a time series from a time series file (dsf0). Click on the Browse button '...' to select an existing time series file or to create a new file.



Note: Time series must have the item type definition defined as 'Time Step' to be used as a Tabulated time step in a River simulation.

Adaptive time step

If Adaptive time step is selected, the simulation automatically maximises the time step while ensuring that all user-defined criteria are met. The mandatory criteria includes:

- **Minimum/maximum/unit:** Defines the limits for the adaptation of the time step.
- **Maximum increase factor:** The maximum allowable increase in time step, defined as a factor/ratio (e.g. 10% increase = 1.10)

The following criteria may be enabled and defined:



- **|resid(BC)/BC|** is a measure for the largest acceptable error introduced at the boundaries. The simulation interpolates the boundary values between t and $t+dt$ using linear interpolation. In case the boundary values has a resolution finer than dt this may introduce unfavourable behaviour where details are neglected. The term resid(BC) describes the residual between the actual value in the time series boundary condition (BC) file and the value found using linear interpolation between t and $t+dt$.
- **|delQ|** is a measure for the largest acceptable discharge change anywhere in the grid within a time step. The criteria helps to lower the time step when sudden changes appears in the discharge. The changes may be either physical changes due to a sudden increase in inflow for instance or mathematical changes due to numerical instability. In either case a decrease in time step may well be desirable.
- **|delQ/Q|** is a measure for the largest acceptable relative discharge change anywhere in the grid within a time step. The criteria helps to lower the time step when sudden changes appears in the discharge. The changes may be either physical changes due to a sudden increase in inflow for instance or mathematical changes due to numerical instability. In either case a decrease in time step may well be desirable. The criteria is well suited for dam break studies where it can be used for refining the time step in the period after the break.
- **|delh|** is measure for the largest acceptable water level change anywhere in the grid. As for the $|\text{delQ}|$ and $|\text{delQ/Q}|$ criteria this criteria helps lowering the time step when large changes appears in the water level due to either physical changes or mathematical instabilities.
- **|delh/D|** is measure for the largest relative acceptable water level change anywhere in the grid. As for the $|\text{delQ}|$, $|\text{delQ/Q}|$ and $|\text{delh}|$ criteria this criteria helps lowering the time step when large changes appears in the water level due to either physical changes or mathematical instabilities.
- **[Courant] (HD)** specifies the maximal allowed courant number within the grid and time step. The courant number, defined above, expresses the length in terms of grid cells that information travels within a time step. The HD Courant number refers to the momentum equation and Dx is hence in this context the distance between two h-points. The simulation applies a 6-point Abbott scheme for solving the equation which does not have the typical Courant number below one demand. Good results are obtained up to Courant numbers as high as 10-20.



- **[Courant] (AD)** specifies the maximal allowed courant number for the advection dispersion calculation. The AD Courant number, defined above, is a measure for the length in terms of grid cells that the species are convected within a time step. The applied computational scheme is stable for AD Courant numbers less than 1. The AD solver includes both h- and Q- points as species grid points, hence the Dx is half the distance between two h-points when calculating the AD Courant number. The criteria is well suited for ensuring stability of AD calculations by lowering the time step when the flow velocity increases and increasing the time step when the flow velocity decreases.

4.5 Computational Control Parameters

Computational Control Parameters includes options for specifying parameter values for variables used to control the behaviour of the computational engine within MIKE HYDRO.

The River module parameters includes entries for refining the numerical solution scheme in model simulations as well as specific settings for numerical methods applied engine specific control parameters.

Grid spacing

Grid spacing (or *Maximum dx*) is the maximum grid-spacing allowed between two adjacent h-points (water level calculation points) in a the numerical scheme of a River simulation.

Generally, the methodology for creating calculation points in the River model is as follows:

- At locations where Cross sections are present in the cross section file, h-points are always created.
- At locations where rivers are connected, a 'connection node' is created internally in the simulation engine and h-points will be automatically created at these locations as well (in case connections are defined where no Cross sections are defined)
- If the defined value of *Maximum dx* is smaller than the distance between cross sections, a number of additional h-points are automatically inserted at engine defined locations to ensure that the maximum distance between h-points is less than or equal to the defined *Maximum dx*. The additional h-points generated from the *Maximum dx* criteria in the model utilise linearly interpolated values for hydraulic parameters (Area, Radius, Width etc.) from the nearest physically defined up- and down-stream cross sections.

Maximum dx may therefore be used to increase the spatial resolution of the river model by forcing an additional number of calculation points in between




physically defined cross sections. (The lower *Maximum dx*, the higher the resolution).

Maximum dx always utilise a global value, which per default is rather large (10 km). It is possible to define branch specific local variation of *Maximum dx* in the Local values table.

Global value

The Global value of *Maximum dx* will be applied throughout the entire river network except for branches defined with a local values.

Local values

It is possible to define a *Maximum dx* for each individual river branch in the model setup. To define a local value, click the Append button  above the Local values table, select a Branch from the drop-down menu and enter a value for *Maximum dx*. For areas that are not assigned a local value, the global value of *Maximum dx* will automatically be applied.

Wave approximation

Wave approximation refers to the numerical solution and number of physical terms included in the Momentum equation applied in the Hydrodynamic simulation. For further details, please see the MIKE 1D reference manual.

Four different wave approximation (or flow description) types are available.

- **Dynamic:** The 'Dynamic' wave approximation should be used where the inertia of the water body over time and space is important. This is the case for all tidal flow situations and in river systems where the water surface slope, the bed slope and the bed resistance forces are small.
- **Dynamic, high order friction:** Similar to 'Dynamic' (see above), but in addition, the 'Dynamic, high order friction' wave approximation contains specific high order and upstream centred friction terms in the momentum equation. This modification typically allows simulations to be performed at longer time steps than the 'Dynamic' wave approximation.
- **Diffusive:** The diffusive wave approximation is a simplification of the 'Dynamic' wave approximation. It assumes that there are no inertial forces (i.e. the inertial terms are dropped from the momentum equation). It is suitable for backwater analysis, slow propagating flood waves and for cases where the bed resistance forces dominates. It is not suitable for tidal flows.
- **Kinematic:** The kinematic wave approximation assumes a balance between the friction and gravity forces on the flow. It is suitable for relatively steep rivers without backwater effects.

In general it is recommended to use 'Dynamic' or the 'Dynamic, high order friction' wave approximation. 'Diffusive' or 'Kinematic' wave approximation



are simplifications of the 'Dynamic' equations and should only be used when it can be clearly shown that they are adequate. It is important that the assumptions described above are respected. In that case, selecting 'Diffusive' or 'Kinematic' wave approximation can improve the computational efficiency of the model.

The wave approximation can be selected globally for the system and/or locally for individual Branches.

Computation parameters

The Computational parameters includes a range of variables targeting the numerical solution of the hydrodynamic simulation. All variables are defined with default values.

Shallow water equation parameters

- **Time centering coefficient for gravity term (Delta):**
A value of 0.5 will produce the most accurate calculations provided there are no numerical instabilities, but instabilities are less likely with higher Delta values. Large value of Delta (towards 1.0) has a dissipative effect which can significantly influence model dynamics. High values of Delta should therefore be avoided especially for applications with strong and systematic dynamics such as in e.g. a tidal range application.
- **Velocity distribution coefficient (Alpha):**
Used in the convective acceleration term of the momentum equation.
- **Weighting factor, momentum equation (Theta):**
Used in the quadratic part of the convective acceleration term of the momentum equation.
- **Threshold water level slope for diffusive wave approx. (Eps):**
If the water surface slope becomes greater than Eps, the computational scheme will become fully forwarded upstream. The parameter can be used to control the stability of the computation.
- **Enhanced formulation of convective suppression (EnConvSup):**
For situations with high Froude numbers combined with small grid spacing the enhanced formulation can be applied (see the MIKE 1D Reference Manual).

Two parameters must be specified:

- **Froude Max:**
'Froude Max' is the parameter 'a' in the enhanced formulation of the suppression term applied to the convective acceleration term in the momentum equation.
- **Froude Exp:**
'Froude Exp' is the parameter 'b' in the enhanced formulation.



- **Alternative utilization of the enhanced convective suppression (AIConvSup):**
If 'Dynamic' or 'Dynamic, high order friction' has been selected as wave approximation (see Wave Approximation), the 'Alternative utilisation of the enhanced convective suppression' may be used to increase the numerical stability of the algorithm.

Structure parameters

- **Threshold water level difference below which flow is linearized (Delhs):**
The Delhs value defines the threshold for water level difference across a weir for which the solution of flow across the structure changes between a normal structure equation and a linearised solution. If the water level difference is less than the threshold value, a linearised solution will be applied to obtain improved stability of the structure solution.
- **Minimum head loss coefficient (ZetaMin):**
The minimum head loss coefficient allowed in the computation of flow over structures.
- **Max. number of iterations at structures (MaxIterStruc):**
The maximum number of iterations permitted at each time step to obtain a solution at a structure.
- **Use pre-processed h-Q-h files for bridges generated from previous simulations:**
The generation of h-Q-h relations files for bridge-structures, during the initialisation of the simulation, may be time consuming. When this option is checked, the files generated during a previous simulation are re-used, when they exist, but are not updated. If they don't exist, they are generated again during the initialisation of the following simulation.

Miscellaneous

- **Threshold depth for slot creation (Delh):**
The Delh factor controls the dimensions of an artificial 'slot', which is introduced to a cross section to prevent 'drying out' of the section. The artificial slot is a small void introduced at the base of the section and allows a small volume of water to remain in the section preventing computational instabilities at low flows. The slot is inserted at height Delh above the river bottom and extends to a depth of $5 \cdot \text{Delh}$ below this level.



- **Number of iterations at each time step (Nolter):**
Each time step in a simulation includes as a minimum one sequence of solving the hydrodynamic equations. The Number of iterations at each time step variable is therefore used to define the number of additional iterations performed in each time step.
The default value is 1, which means that one additional iteration is made, which results in two calculations in each time step. One additional iteration generally provides the accurate and adequate results. Zero additional iterations is generally not recommended (less accurate) whereas the benefit of adding multiple additional iterations also generally is very small when it comes to enhanced accuracy of model results.
- **Max. exceedence factor for depths above bank level (MaxBank-Depth):**
The maximum allowed exceedence factor for simulated water depths compared to the max depth in cross sections can be controlled by adjusting the variable. The factor is calculated as a multiplum of the max allowed depth in a cross section (calculated from bottom level to the highest of the left or right embankment levels).



5 Map Configurations

The *Map view* is the focal point of each MIKE HYDRO model session, and the Map configurations option enables a user-defined customisation of the *Map view* appearance to best suit the user's preferences and requirements.

Map configurations include the options:

- Coordinate System (Map view and Feature coordinate systems)
- Background Map (Map view background)
- Background layers (Shapefile, raster, and image definitions)
- Digital Elevation Model (DEM) (Background and processing)
- Working Area (Map view working area definition)

5.1 Coordinate System

The user can select a coordinate system for the *Map view* in the Setup tree view. The *Map view* coordinate system is selected using the drop-down menu labeled "Map view coordinate system".

If the user would like for features of the model (i.e., branches, water users, catchments, etc.) to use a different coordinate system than the *Map view* coordinate system, then it is possible to select a different coordinate system using the drop-down menu labelled "Coordinate system for features stored in setup file".

If two different coordinate systems are selected, the coordinates of the model features are automatically converted into the map coordinate system when displayed in the *Map view*. However, model features are saved in the model feature coordinate system.

In some cases, the coordinate system used for model features may be based on a different datum than the datum used in the *Map view* coordinate system. In these cases, a datum shift can be applied to locate model features correctly in the *Map view*. Two approaches are available: a three-parameter Molodensky transformation, and a seven-parameter 3D Helmert transformation. Details of the two approaches are documented in the geodesy manual supplied with the installation (MIKE_Zero_Geodesy.pdf).

See also: <http://www.arcwebservices.com/arcwebonline/services/dat-trans.htm>.

A different coordinate system for model features should be considered if the study area is located in an area where distances are distorted significantly in the *Map view* coordinate system. For example, using a Sphere Mercator coordinate system in the north of Alaska would result in significant errors in calculated river reach lengths.



If the model working area is located in a region where the risk of significant errors due to distortion is large, the user will receive a validation warning. Validation messages will not be issued unless a working area has been defined.

5.2 Background Map

A background map can be selected in the Setup tree view. Three options are available: None, a Google map background, and a Countries/Coastline shapefile. If the Google map background is selected, the user is prompted to select from a drop-down list of available map types. Both the Google map and Countries/Coastline shapefile can be disabled in the *Map view* by unchecking the Visible check box.



Note: In order to use Google map background, an internet connection must be available. Google maps images are automatically rectified to the *Map view* coordinate system.

5.3 Background Layers


Background layers including shape files, images, and rasters can be added in the Setup tree view. To add a new background layer, click on the  icon and select the layer type.

Image area coordinates can be read from a world file or else specified by the user. If using a world file, the world file must be located in the same directory as the image file, with the same name. Otherwise, image coordinates can be entered by hand.

Shape file background layers require specification of a projection type. If a projection file (PRJ file) exists, the projection type is automatically read. Otherwise, a user-specified map projection must be selected.

The raster background layer is used when the user would like to use a digital elevation model (DEM) or other raster as a background layer in the Map view. Four raster formats can be selected:

- **ESRI ASCII:** a text file format, which requires that the coordinate system as well as the elevation unit are specified manually.
- **dfs2:** for a dfs2 raster file, the coordinate system and units are obtained from the file's metadata.
- **GeoTIFF:** a special format of tiff file, which also stores the coordinate system and units as metadata.
- **ESRI binary grid:** a binary format of raster, composed of multiple files with extension .adf. The raster is actually the folder containing the .adf files.



Background layers are displayed in the *Map view* in the order given in the list of layers found at the bottom of the tabular view (when Background layers is selected in the Setup tree view). The appearance of background layers follows the order in the list of images (i.e. the image in line 1 is on top, then image 2 is below image 1, then image 3 is below image 2, etc.). It is possible to rearrange the order of images by selecting an image and using the arrows to move the selected image up or down in the list.

Display settings for background layers can be modified in the Symbology tree view. In this view, layer transparency settings can be changed so that underlying background layers are visible in the *Map view*. It is also possible to remove layers from the map view by unchecking the Visible box. Other aspects of symbology can also be modified, such as colour schemes and feature labels. For more details about symbology options, please see the discussion of symbology in Section 2 (Working With MIKE HYDRO).

5.4 Digital Elevation Model (DEM)

A Digital Elevation Model (DEM) may be included in the model setup for river tracing and catchment delineation.

When a DEM is to be used, it must be selected amongst the list of raster files loaded as background layers. Therefore, any DEM must first be added in the Background layers menu before it can be used as a DEM for e.g. river tracing or catchment delineation.

5.4.1 Cross section creation

When loading a DEM new cross sections are automatically extracting elevations from the DEM. By setting the checkmark 'Use DEM for cross section creation' specifications for the extraction can be specified.

For following options are available for cross section digitisation:

- **Use bilinear interpolation:** If '*Use bilinear interpolation*' is checked bilinear interpolation is used when extracting cross section z-values from the DEM cells. If unchecked the z-value in the cross section profile will be the exact value from the DEM cells that are intersected, whereas if bilinear interpolation is used, z-values will be the interpolated z-value from the 4 closest neighbouring DEM grid cells.
- **Use half cell size for spacing between points:** The maximum spacing between points can either be user-specified or assumed as half the cell size of the DEM.
- **Maximum number of points:** Should the specified maximum spacing between points result in more than the allowed maximum points the spacing is increased to obtain the maximum number of points.



When plotting existing cross sections the corresponding DEM elevations can be displayed together with the cross section by setting the checkmark 'Show DEM together with cross section plots'. The DEM elevations can be displayed with a user-specified spacing or use the cell size of the DEM.

5.4.2 River tracing and catchment delineation

If the checkbox, 'Use DEM for river tracing and catchment delineation' is selected, the DEM can be used to delineate rivers and catchments based on the elevations in the DEM. Delineation is carried out using the 'Add trace' and 'Add delineate' tools in 'Branch' and 'Catchment' ribbons. More details about these tools are available in Section 6 (River Network) and Section 7 (Catchments). Before these tools can be used, however, the DEM must be processed.

Processing elevation data

The DEM is preprocessed to create a raster file with flow direction information that is used to delineate river branches and catchments.



Note: It is recommended that the size of the DEM should not exceed 25 million pixels. A warning message will be shown if the size of the DEM is above this limit.

The size of the DEM can be reduced by changing the spatial extent or by resampling.

Spatial extent

The spatial extent of the DEM can be limited to the defined working area (see below), thereby reducing the number of pixels.

Resampling

Resampling requires specification of a resampling factor. If a resampling factor of 2 is used, then the minimum elevation of a 2x2 set of pixels is assigned to one new pixel with the same area as the 2x2 set. A resampling factor of 3 assigns the minimum value of a 3x3 pixel set to one new pixel with the same area as the 3x3 set.

Preprocess digital elevation data

In order to use the river tracing and catchment delineation on the map, the digital elevation data must be preprocessed by clicking the button 'Preprocess digital elevation data'. After changing the spatial extent or resampling the elevation data, the DEM must be preprocessed again.

The processed data is saved in a ppdd-file. If no location is specified, then the ppdd file is saved by default in the same location as the model setup.



Many DEMs contain inaccuracies because of their resolution or artifacts resulting from the method of creation. When delineating catchments and riv-



ers from the DEM, you may find some unexpected results. This can happen in flat areas where DEMs may not have the vertical resolution to correctly delineate the rivers and catchments.



Note: In many digital elevation models, all pixels in lakes have undefined (or blank) pixel values.

If the option '*Assume internal undefined areas as local depressions*' is set, all internal undefined areas are handled as a local depression. When a traced branch is added, an artificial channel will be traced through the depression along the shortest path between the upstream and downstream ends. For example, if a lake is represented in a DEM as an undefined area, then an artificial flow path will be created through the lake along the shortest path to the outlet.

If undefined areas are not set as local depressions, a traced branch will stop upon reaching an undefined area.

5.5 Working Area

A working area may be defined by clicking '*Draw rectangle on the map*' and dragging a rectangle on the map. Once defined, the working area may be edited (and edits must be saved) but it cannot be deleted.

If no working area is defined, the maximum extent of features is used as default working area.



6 River Network

The River network forms the basis of all River applications. The river network is defined as a combination of connected river segments and computational points, and module specific features are then generally added to the model using the river network as a base (river network example illustrated in Figure 6.1)

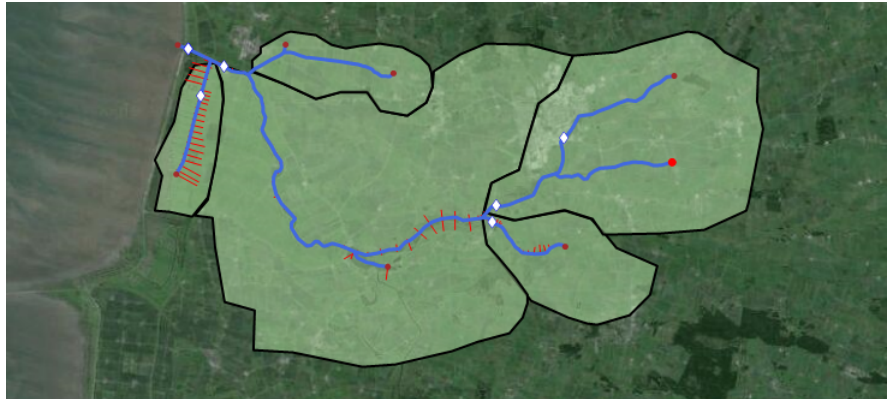


Figure 6.1 Example of river network in MIKE HYDRO.

Additional river network details:

- River segments are called **Branches** and branch definitions include options for connecting river branches as well as defining User defined chainage points.
- Calculation points in the hydrodynamic module include **h- and Q-points**:
 - **h-points**: Water level (h) calculation points. All **Cross sections** as well as **start- and end-points of river branches** automatically creates h-points in a model. Furthermore, h-points are automatically inserted to ensure that the maximum distance between h-points (grid spacing) is less than or equal to the length defined as the Maximum dx variable (see Grid spacing (p. 60))
 - **Q-points**: Discharge (Q) calculation points. Q-points are automatically inserted mid-ways between h-points in the entire model area. Additionally, **hydraulic structures** are included as Q-points in the numerical model.
- Inflow to the River network is specified through Boundary conditions (see 12.1 Standard boundaries (p. 230)).

This chapter describes the following topics in further detail:

- Creating a River Network
- Branches
- Routing
- Alignment lines
- Cross Sections
- Structures
- Control
- Storages
- Decoupled simulation

6.1 Creating a River Network

The Branches segments of a River network including associated River nodes and Branch connections can be created in three different ways:

- Method 1: Digitising a River network
- Method 2: Importing a River network
- Method 3: Derive the River network from a DEM

In either way, it is recommended to reference your network geographically, so that features are located at their correct geographical coordinates (cf. Section 5.1 Coordinate System (p. 65)).

6.1.1 Method 1: Digitising a River network

For schematic models and small river networks, the network can be digitised directly on the map. A background map (cf. Section 5.2 Background Map (p. 66)) can be selected to assist when digitising.

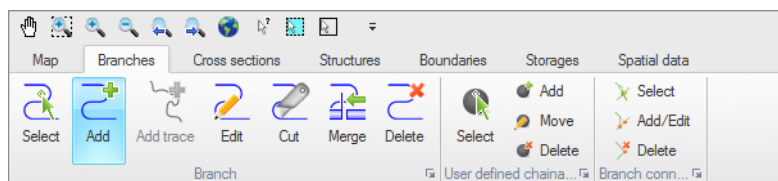


Figure 6.2 Add Branches and Branch Connections

To digitise a Branch, go to *Map view*, select the *Branches* ribbon and click on the 'Add' button. Use the left mouse button and click at each point along the river. To finish the Branch, double-click on the last point.

To edit the shape of an existing branch click on the 'Edit Branch' button and click on the branch to be edited. All points on the branch are shown with a



small square. To add a new point click on branch at the location of the new point. To move an existing point click on the square and drag to the new location. To remove an existing point double-click on the point.


Branches may be connected by using the '*Add/Edit Branch Connection*' button.

Note that the snap distance has been hard-coded as 5 pixels.

If you make a mistake while you are still digitising you can use the *Esc* button to cancel the operation.

6.1.2 Method 2: Importing a River network

The River network may be imported from an existing shape file by selecting '*Import*' in the *File* drop-down menu and clicking in '*Network from shape file*'. This opens the '*Import Branches*' dialogue.

Click on the  button and browse to the existing shape file. Branches can be auto-connected by enabling '*Auto connect branches*' and specifying a maximum distance for auto-connection.



Note: Existing MIKE 11 models can be imported by selecting '*Import*' in the *File* drop-down menu and clicking in '*Network from MIKE 11*'.

6.1.3 Method 3: Derive the River network from a DEM

A branch may be added using a DEM see Section 5.4 (Digital Elevation Model (DEM)).

When a DEM has been loaded the '*Add trace*' button is enabled on the *Branch* ribbon. Clicking on the upstream location of the river will trace down to the outlet of the river based of the DEM data. Subsequent branches will be traced down to the point where they flow into existing river branches.



Note: In local depressions (areas with local minimum values, e.g. lakes or flat areas) the tool connects the 'inflow' and the 'outflow' of the depression using an algorithm that ensures the river is traced within the depression. This line may not represent the actual location of the river.

Moving the cursor over the map will automatically display the traced river according to the cursor position.

6.2 Branches

The *Branches* properties dialogue contains the following tabs:

- General

- Link Channels (present if depending on branch type selection)
- User defined chainage points

6.2.1 General

The properties of Branches are specified under the *General* tab. The lower part of the dialogue window contains a list of all Branches in the River network. Select a Branch from the list and specify the properties.

Definitions

Branch name

A Branch can be given any name.

Topo ID

Topo ID is an abbreviation of 'Topographical Identifier', which refers to the same variable defined as part of the Cross section definitions. The cross section file offers the possibility to store several measurement campaigns (e.g. from different years or before and after the construction of a structure), and these are identified through different Topo ID's. The main purpose of this is to enable simulation of alternative scenarios once the network has been configured. The topographical data used in a simulation is defined by the Topo ID.



Note that the String defined for the Topo ID must be identical to a Topo ID present in the cross section file for the actual river branch. If not, cross sections in the branch are not recognised and the calculation cannot start.

Start chainage

The chainage of the first point in the branch. To change this chainage, it is required to insert a User defined chainage point in the first point of a branch. The chainage can then be specified under the *User defined chainage points* tab (see below).

End chainage

The chainage of the last point in the branch. To modify this chainage, a User defined chainage point must be inserted in this last point of the branch. The chainage can then be specified under the *User defined chainage points* tab (see below).

Flow direction

The flow direction may be set as positive or negative, and indicates how the chainage definition is interpreted with respect to the flow direction. If flow direction is selected as positive, the actual flow direction will be assumed to go from low chainages to high chainages in the river branch. If flow direction is selected as negative, the actual flow direction is assumed to be defined from high chainages to low chainages.



The default assumption is to define flow direction positive with increasing chainages looking downstream of the river. Hence, if a river has been defined with chainage 0 (zero) at the river mouth and chainages increasing upstream, the flow direction must be defined as negative.

Branch type

- There are three different Branch types:
- Regular. A Regular Branch is a 'normal' Branch consisting of a number of computational points defined from Cross sections. A minimum of one Cross section is required. The hydrodynamic simulation in a regular branch is based on a solution of the de Saint-Venant equations.
- Link Channel. A Link Channel is typically a short Branch used to connect a flood plain to the main river branch. Link channels do not require Cross sections to be specified but all parameters given in the Link channels dialogue (see below) must be specified.
- Routing. No cross sections are required. Only the flow is calculated and routing is a simplified hydraulic calculation. The concept of routing is basically the transformation of a hydrograph entering at the upstream end of a branch into a downstream outflow.

Connections

Information about Branches and their upstream and downstream linkages are provided.

Edit coordinates of vertices

In some cases it might be necessary to update the coordinates on the map view of the vertices of one or more branches.

When a specific branch is selected in the table and the "Edit coordinates of vertices..." button is pressed, a table-dialogue pops up. This table contains the X and Y coordinates of the points of the selected branch, which can be edited (copy and paste is also supported). No new rows can be added, nor deleted. When the update is done and OK is pressed, the map view is updated.

Load shape

It is possible to import/load the shape of an existing branch directly. Select/activate the actual branch in the tabular view and Click '*Load shape*', select the relevant shape file and choose the shape item that represents the branch.



Note: this functionality will only edit the shape of the active branch on the map. It will not create any new branch. Instead, it is possible to import and

create one or multiple branches at once from a shape file. To achieve this, please see section Import from Shapefile, page 289.

6.2.2 Link Channels

A branch may be defined as a 'Link Channel' (see Branch Type above), where a link channel typically is a short branch used to connect a flood plain to the main river branch and therefore generally represents the embankment geometry between parallel rivers (e.g. main river branch and flood plain branch). Link channels are modelled as a single structure branch of only three computational points (h-Q-h).

A link channel doesn't require cross sections to be defined: the h-points are automatically assumed to be made of very wide geometries.

The Q-point in the middle of link channel branch is defined through an 'open culvert' type of structure. That is, an overflow weir with a length to include the friction loss component. The friction loss is computed using the length defined by the upstream and downstream chainages of the branch.

Typical link channels representation is illustrated in Figure 6.3.

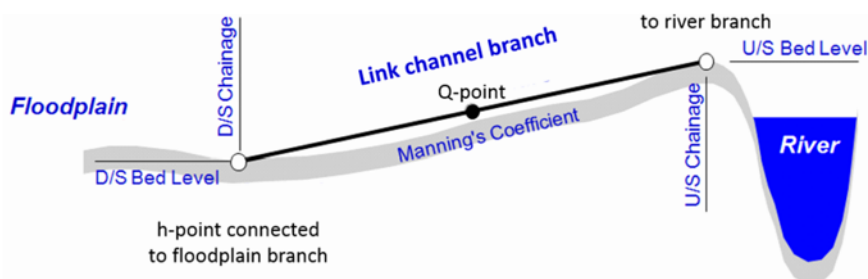


Figure 6.3 Longitudinal representation of a link channel

Note: Following the internal formulation of the link channel where no true cross sections are defined in a link channel branch, the usage is restricted to Hydro-Dynamic simulations ONLY. Thus, for Advection-Dispersion, Water-Quality and Sediment-Transport calculations the set-up should be void of link channels. Link channels should therefore be replaced by 'regular' branches with actual cross sections defined and eventually a weir to describe embankment levels if this is what should be represented with the connecting branch.

Note: since a link channel do not contain cross sections, it doesn't have storage capacity at nodal points where the link connects to another branch.

If a branch type is set to Link Channel, the *Link Channels* tab automatically appears and the following properties must be specified.



Link geometry

The link channel geometry comprises the definition of the longitudinal and transversal geometry of the embankment along the river. The geometry is defined from the following parameters:

Type

The Type describes the way the embankment geometry (transversal to the flow direction through the link channel) is defined. The following options are available:

- Depth-Width: the cross sectional geometry in the link channel is defined with a number of relationships between the depth above the upstream invert and the width of the channel.
- Level-Width: the cross sectional geometry in the link channel is defined with a number of relationships between the level and the width of the channel.
- Cross section DB (cross section database): the cross sectional geometry is read from a cross section specified in the 'Cross sections' menu.

Datum

The Datum may be used for adjusting the levels of the link channel, e.g. to conform with a specific reference datum in the model area or to conduct sensitivity tests. The datum value is added to upstream levels (either the Upstream invert for a Depth-Width type of geometry, or the levels from the table for a Level-Width type) as well as to the Downstream invert.

Note: when a value is inserted for the Upstream invert or Downstream invert, this value is automatically updated with the Datum, to show the value actually used in the calculation.

The Datum is not used for the 'Cross section DB' option. If any datum should be applied with this option, it should be specified using the equivalent Datum value in the cross section definition.

Upstream invert

Upstream bed level of the link channel. Available for the Depth-Width link channel type only.

Downstream invert

Downstream bed level of the link channel.

Depth/Level-Width table

This depth/level-width table defines the cross section geometry of the flow area in the link channel. Both the depth and the width must be increasing. This table is only active for 'Depth-Width' or 'Level-Width' link geometry types.



Branch name, TopoID and Chainage

When the link geometry type is set to 'Cross section DB', the branch name, TopoID and chainage of the cross section defining the geometry must be specified here.

Flow conditions

Allow for recalculation

When this option is ticked, the values in the Q/h relations table may recomputed from the menu Run \ Recalculate flow conditions...

This option can be unticked in order to keep the Q/h relations table unchanged while recomputing other tables from the Run \ Recalculate flow conditions... menu.

No of Q/h relations

The number of relations to be used in the Q/h relations table is specified here. This number is only used when the geometry in the 'Link geometry' options is described with 8 levels or less.

If the geometry in the 'Link geometry' options is described with more than 8 levels, it is assumed that the cross sectional geometry is accurately defined and that the Q/ relations should be computed at the exact levels defining the geometry, therefore ignoring the specified 'No of Q/h relations'.

Q/h Relations table

Free outflow Q/h relations must be calculated prior to the simulation. To calculate the Q/h relationships, specify the number of relationships required and press the 'Calculate Q/h relations' button. The result of the calculation will appear in the table.

Table values in the Q/h relations table can be changed if required but general recommendation is to leave the table as is from the automatic calculation when pressing the 'Calculate Q/h relations' button. Table parameters are:

- y: water depth in link channel structure
- Area: flow area in link channel structure
- Radius: hydraulic radius in link channel structure
- Conveyance: conveyance indicator in link channel structure
- Qc: critical flow in link channel structure
- hUS: water level in h-point upstream of link channel structure
- US Type: indicator of upstream flow type derived from the calculated flow parameters in the link channel (can not be changed)
- hDS: Water level in h-point downstream of link channel structure
- DS Type: indicator of downstream flow type derived from the calculated flow parameters in the link channel (can not be changed)



Note : If any of the parameters defining the link channel geometry or loss coefficients is changed, the Q/h relations must be re-calculated so that the change is taken into account in the simulation.

Head loss and Bed resistance

Type. The bed resistance along the length of a link channel can be described using Manning's M or Manning's n.

Value. Bed resistance value, expressed in the unit type selected above.

Positive flow and Negative flow. Head loss factors must be defined for Inflow (contraction), Outflow (expansion), Free overflow and Additional losses. All four factors are dimensionless and must be within the range 0.00 - 1.00. Each factor may be given a different value depending on the flow direction across the link channel.

6.2.3 User defined chainage points

A user defined chainage point is a specific location with a controlled chainage.

If a branch does not contain any user defined chainage points the start chainage is per default 0 and the end chainage equivalent to the length of the branch calculated based on the geographical length of the branch shape. Both the start and end chainage can be controlled by inserted user defined chainage points.

The chainage along a branch between two user defined chainage points are linearly calculated between the two defined chainage values. The chainage along a branch between a user defined chainage point and a branch end is calculated as the geographical distance from the defined chainage value.

User defined chainage points are added to the River network by using the 'Add' button in the 'User defined chainage point' section under the *Branches* ribbon in the *Map view*.

If user defined chainage points have been added in *Map view* their chainage may be edited under the *User defined chainage points* tab.



Note: If a user defined chainage point is moved the defined chainage is fixed and hence moving the point will update all chainages along the branch.



Example: A branch without user defined chainage values are created with a geographical length of 1000 m. The start chainage will be 0 and the end chainage will be 1000.



A user defined chainage point is added at 500 m (geographically) along the branch and given the chainage 750. The start chainage is now automatically updated to 250 and the end chainage to 1250.

Another user defined chainage point is added at 250 m (geographically) along the branch and given the chainage 100. The start chainage is now automatically updated to -150 and the end chainage is unchanged at 1250.

6.3 Routing

In the *Routing method properties* dialogue, routing of flow as well as water level calculations can be included.

Routing can be used to model the hydraulics of upstream tributaries and secondary river branches, where the main concern is to route water to the main river system.

Routing is a simplified hydraulic calculation. Normally, simulation of hydrograph mitigation along a branch is based on the shallow water equations, which require cross section information. However, routing doesn't require such cross sections. The concept of routing is basically transformation of a hydrograph, i.e. calculating the modified hydrograph (different peak value and adjusted shape) at a given location based on the hydrograph at the upstream point and simple parameters. Typically, routing may be used to represent a reach of a river, a reservoir or a structure.

Note that the following applies to routing branches:

- Routing branches cannot be used to model a looped part of a river network.
- Use of routing branches requires that all branches located upstream of a routing branch are also routing branches. Routing can however be combined with regular branches, if these are located downstream the routing branches.
- Runoff from catchments can be included in a routing branch.
- The routing method does not support the use of structures at routing branches.
- The routing method does not account for backwater effects.
- If no routing point is inserted in a routing branch, the outflow will equal the inflow.

To allow for the insertion of routing components into a branch, the branch type must be defined as "Routing", in the 'Branches' menu. Routing locations can be added or deleted using the Append '+' or Delete '-' buttons above the overview table located in the lower part of the window.



6.3.1 Routing Method

The routing method definitions specify the location and properties for routing elements defined at specific locations in the river system. Routing elements are recognized as locations in which a routing procedure is active, that is where change in hydrograph appearance occur.

Multiple routing elements can be defined within the same branch if required.

Location

Specify the location of a Routing element

Branch name. The branch name of where the routing method is applied can be chosen from the drop down list.

Chainage. Chainage at which the routing method is applied. The chainage of a routing element will have effect on the downstream node in the branch where the routing element is defined.

ID. String identification of the routing location.

Flow routing method

Three options for routing methods are available These methods are:

- **Muskingum:** a routing method that requires two parameters to be specified to describe the delay and the shape of the modified hydrograph. These parameters are fixed in time and space.
- **Muskingum-Cunge:** a modified version of the Muskingum routing method, for which the two parameters are automatically computed based on the diffusive wave model. The parameters are recomputed at each time step of the simulation. Hence, this method doesn't require any additional parameter to be specified by the user.
- **No flow routing:** the hydrograph is not transformed while passing the current routing element.

When the Muskingum-Cunge method is selected, the two additional parameters below are must also be specified.

Delay parameter. Time for the hydrograph to travel between the selected river node and the next downstream node. Its value may also be estimated as the observed travel time of the flood peak between the nodes.

Shape parameter. A dimensionless factor controlling the shape of the modified hydrograph. This factor depends on the shape of the modelled wedge storage and must be between 0 and 0.5. In natural rivers, it usually has a value between 0 (reservoir type storage) and 0.3, with a mean value of 0.2.



Note: the Muskingum-Cunge method computes the delay parameter and the shape parameter using the flow width and the flow area, and it therefore requires that a cross section exists at the routing location.

6.3.2 Water level Calculation

Routing branches does not per default consider any water levels as the primary purpose is to route a hydrograph through the river system and hence, only the discharge is a key state variable. It is, however, possible to determine water levels in case this should be necessary for specific applications (e.g. coupling of MIKE HYDRO to MIKE SHE where head elevations are essential to calculate the correct discharge/recharge interaction between surface and sub-surface water).

Two methods are available for Water level calculations:

- Rating curve approach (Q - h table)
- Manning formula.

Both apply for steady-state flow and are thus approximations, but they are usually reasonable estimates as long as the water level does not change rapidly (e.g. during a flood). The items in the dialogue window change depending on the option chosen.

Rating curve approach

When the Rating Curve approach is used, it looks up the water level in a user specified Q - h table. The table must cover the range of discharges encountered during a simulation.

Manning formula

The Manning formula is:

$$Q = 1/n \cdot A \cdot R^{(2/3)} \cdot \sqrt{S} \quad (6.1)$$

Where n is the Manning's number n , A is the cross-sectional area, S is the water surface slope, R is the hydraulic radius or resistance radius:

The hydraulic parameters in river modelling are required and consequently, it is required to specify cross sections in the routing branch in order to extract hydraulic parameters from the processed data. If cross sections does not exist, it is not possible to use the Manning formula option.



6.4 Alignment lines

The purpose of using alignment lines is to use geo-referenced polylines, e.g. describing river banks, to update information in the cross sections. Alignment lines will affect the simulation results only when alignment lines positions are transferred to the cross sections, and such transfer must be triggered by the user using the 'Trim or extend cross sections' tool available in the 'Tools' menu.

Transferring alignment line information will update the positions of markers in the cross sections as described below:

- Marker 1 will be placed at intersection with left extent alignment line
- Marker 4 will be placed at intersection with left bank alignment line
- Marker 5 will be placed at intersection with right bank alignment line
- Marker 3 will be placed at intersection with right extent alignment line.

Alignment lines can intersect multiple cross sections.

Alignment lines can only be created from the Map view. Select the 'Spatial data' tab and then choose 'Alignment line' from the 'Type' drop-down menu. The ribbon menu then allows you to:

Select. Select a line on the map and synchronize the selection with the tabular view.

Add. Add a line to the map. This is done in the same way as branches are inserted, but without connecting to any existing element in the map.

Edit. Edit vertices, in a similar way to what the command does for a branch.

Cut. Removes segment between closest vertices. Both lines keep the same type as the original line. The first line also keeps the name. The second line is given the same name as the first with a prefix of "_2".

Merge. Inserts segment between two lines. The name and type of the new line is the same as the first line selected (click the 'Merge' button, then select one line's end and drag the mouse to the second).

Delete. Delete the entire line.

Alignment lines menu

The lower part of the dialogue window contains a list of all alignment lines in the river network. Alignment lines can be deleted from the list using the Delete '-' button above the table.

Select an alignment line from the list and specify the properties below.

Location

Name. Name of the alignment line.

Type. The following list of types of alignment lines is available:

- Left and Right bank: these locations indicate the left and right levees, corresponding to the limits of the river bed.
- Left and right extent: location of the left and right extent of the full cross section, including the flood plains if existing.

6.5 Cross Sections

Cross sections can be edited and shown graphically in the *Map view* and in the *Tabular view*, but the data are stored in an external cross section file with the extension xns11.

When working from the *Tabular view*, the numerical data and the plots are presented under a number of windows, as shown on in Figure 6.4.

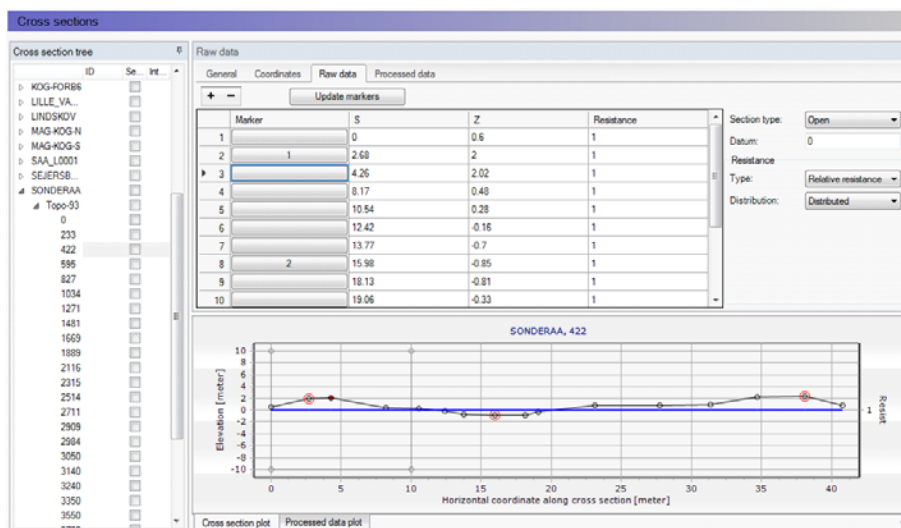


Figure 6.4 Tabular view for the Cross sections menu



Note: Although it is possible to edit the cross sections data from MIKE HYDRO, it is also possible to edit the cross section file from the cross section editor (from 'File \ Open \ File'). This will open the cross section file as it is saved to the disk and will not contain unsaved changes made in MIKE HYDRO. The cross section editor will not automatically refresh if you save cross section changes from MIKE HYDRO while the cross section editor is



opened. If editing cross sections both from MIKE HYDRO and from the cross section editor it is recommended to close the cross section editor each time changes are made from MIKE HYDRO, or use the 'Refresh' option available in MIKE HYDRO in the 'General' tab of the cross sections menu.

Inserting a cross section in Map view

To insert a cross section from the map select the 'Add' button in the *Cross sections* ribbon in the *Map view* (see Figure 6.5).

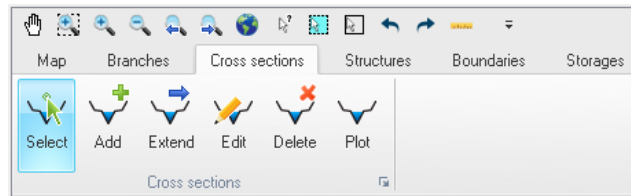


Figure 6.5 Map view ribbon for adding and editing cross sections

Cross sections are defined as polylines. To add a new cross section line click firstly on the required location of the bank on one side of the river and then click at all locations of intermediate points of the polyline. To stop digitising the cross section, double-click at the location of the bank on the other side of the river. The digitised cross section must always intersect with a branch. The branch name and chainage defining the location of the cross-section are automatically defined by this intersection. The created cross section is automatically shown in the Cross Section window.



Note that inserting a cross section on the map will set marker 1 in the point located to the left of the branch (taking the branch direction into account), marker 3 in the point located to the right of the branch and marker 2 where it crosses the branch.

If a DEM has been added to the map the cross section is defined based on the DEM values along the digitised cross section, and using the digitisation options available in the 'Digital Elevation Model' menu. If a DEM is not added to the map a default cross section is created with the horizontal extent corresponding to the digitised cross section.

Cross sections that are drawn on the map will automatically be created with coordinates for each digitised points, linking them to the position specified. The use of these coordinates can be disabled in the *Tabular* view.



Note: Cross sections with coordinates are fixed at the given location. This means that the chainage of the cross section used in the calculation may differ from the position shown on the map e.g. if the chainages have been edited after the cross sections have been drawn. Chainages may be updated using the tool 'Edit multiple cross sections'.



Extend. Select this icon to extend a cross section on its left and/or right side. This tool keeps the raw data along the original extent of the cross section unchanged, and adds additional data following the same principles as when adding a new cross section. That is, if a DEM is available the cross section is extended using the elevation data from the DEM. In the latter case, the number of points being added is controlled by the spacing between points as well as the maximum number of points specified in the 'Digital Elevation Model (DEM)' menu, where this maximum number of points corresponds to the total number of points in the extended cross section, including original points. When extending a cross section, markers 1 and/or 3 are moved to the new cross section's end, while marker 4 and/or 5 is assigned to the original end of the cross section (except if marker 4 and/or 5 was already assigned, in which case it's kept unchanged).

Edit. To edit a cross section on the map, select the 'Edit' icon on the ribbon and click once on the cross section on the map. The vertices of the polyline are then shown on the map, and you can click and drag these vertices to edit the location of the cross section. Clicking between two vertices will add a new vertex to the cross section.

Delete. Select this icon and click on a cross section on the map to delete it.

Plot. To display a cross section in the Cross Section window, select the 'Plot' icon and select the cross section to be displayed on the map.

6.5.1 Cross section in tabular view

Cross section tree

The Cross Section Tree window in the Tabular view shows the list of all cross sections in the setup. In the first column, cross sections are identified by their branch name, topo ID and chainage.

Clicking a cross section in this tree view will show the details of this cross-section on the right-hand side of the Tabular view.

Right-clicking in the first column gives access to options to edit the cross sections. Notice that the options offered in the contextual menu depend on where you actually clicked in the tree view. For instance clicking on a single chainage allows editing the corresponding cross section only, whereas clicking on the topo ID or on the branch name allows editing all or a selection of cross sections.

The 'Move...' feature allows to move cross sections to different locations, by changing either the branch name, topo ID and/or chainage. When selecting the 'Move...' feature, the dialogue in Figure 6.6, where the 'From' groupbox indicates which cross sections are being moved and where the 'To' groupbox indicates the final destination which has to be specified by the user.



Figure 6.6 Dialogue for moving a cross section

The 'From' groupbox shows the original chainage only when moving a single cross section. It shows the topo ID only when moving a single cross section or a group of sections from a given topo ID.



Note: The upper part of the dialogue is only active when selecting 'Move...' from a branch name or a topo ID. It allows selecting between moving all the cross sections of the branch / topo ID, or only the selection. These options are therefore not relevant for moving a single cross section.

The 'Copy...' feature is similar to the 'Move...' one except that the original cross sections are kept at their original location.

To insert a cross section, it is possible to use the 'Insert blank cross section' feature, which allows creating a cross section on any branch and for any topo ID, which therefore have to be specified as shown in Figure 6.7.

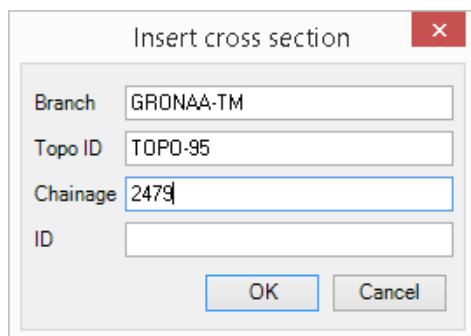


Figure 6.7 Dialogue for inserting a blank cross section

Alternatively, right-clicking on the branch name or topo ID allows inserting a cross section respectively in the corresponding branch or the corresponding topo ID. In that case the branch name and eventually the topo ID are automatically filled, as illustrated in Figure 6.8.

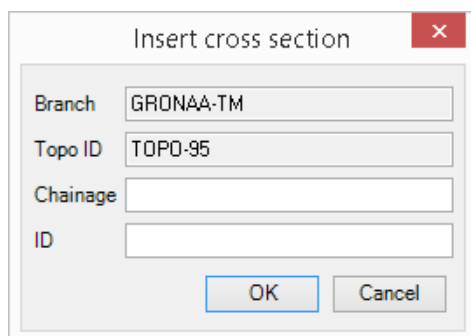


Figure 6.8 Dialogue for inserting a cross section in a selected topo ID

It is possible to insert an interpolated cross section, by right-clicking and selecting 'Insert interpolated cross sections'. This opens up the dialogue illustrated in Figure 6.9, where the branch name and Topo ID where the interpolation is to be conducted must be selected. It is possible to interpolate a single cross section at a given chainage or multiple cross sections. In the latter case a maximum distance between the interpolated cross sections must be specified, along with the range of chainages.

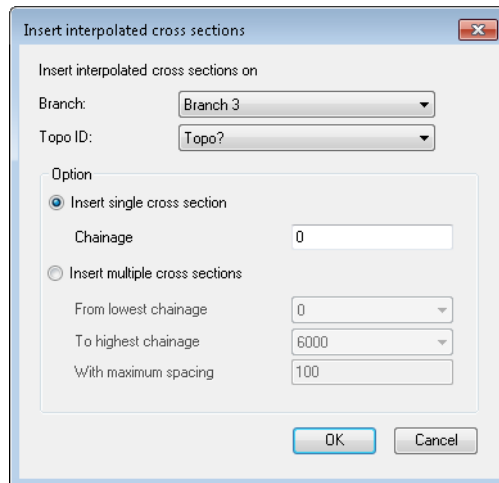


Figure 6.9 Dialogue for inserting an interpolated cross section

Similarly there are multiple options for deleting cross sections. Right-clicking on a single cross section gives access to the 'Delete this cross section' feature. Clicking on the topo ID allows deleting either all cross sections under this topo ID (using the feature 'Delete topo ID') or only the selected sections under this topo ID (using the feature 'Delete selected in this topo ID'). Finally, clicking on the branch name allows deleting either all cross sections under this branch (using the feature 'Delete river') or only the selected sections under this branch name (using the feature 'Delete selected in this river').

ID. The second column contains user defined identifiers. To add or edit an ID, click on the field for the relevant cross section.

Select. The third column contains a checkbox that allow selecting and unselecting cross sections. Selections may later be used to process multiple cross sections at the same time, for instance to recompute the processed data for all selected cross sections.

Interp. The last column contains a symbol for cross sections created by interpolation of other cross sections. This allows keeping track of the origin of data.



Note: Per default, cross sections are sorted by increasing chainage under each branch and topo ID. It is however possible to sort them by decreasing chainage by clicking in the header of the first column, or sort them by their attributes (ID, selected/unselected, interpolated) by clicking in the appropriate column's header.

Cross section properties

General

The General tab contains options and data which relevant for all or part of all the cross sections.

Recompute all. The 'Recompute all' button recomputes processed data for all the cross sections.

Recompute selected. The 'Recompute selected' button recomputes processed data for the selected cross sections (those having the 'Select' checkbox checked) only.

Cross-section filename. Cross sections are stored in a cross section file, with the xns11 file extension. Click the '...' button to either select an existing file, create a new one or refresh the content of the file.

MIKE HYDRO River has been designed in such a way that cross sections should primarily be edited within MIKE HYDRO. The cross section file can nevertheless be edited in the Cross Section Editor. The 'Refresh' function can be used to update the MIKE HYDRO data, if the cross section file has been edited in the Cross Section Editor while the MIKE HYDRO file was opened.

Draw history on plots. When this option is checked, watermarks are added as a history of previous cross sections drawn on the 'Cross section plot' and the 'Processed data plot'. This feature allows comparison of multiple cross sections on a single scale.

Coordinates

The 'Coordinates' tab provides the list of vertices defining the location of the cross section (i.e. the polyline shown on the map). Each row describes a point identified with its X and Y coordinates expressed in the coordinate system used for features in the setup. The 'S' column provides the horizontal distance of each vertex along the polyline from its left end. These vertices don't have to match the list of points provided in the 'Raw data' tab.

When cross sections have been created from the *Map* view, the table is automatically filled with all vertices defining the location of the polyline and one point at the intersection with the branch.

Apply. When coordinates are provided in the table, the 'Apply' option can be checked. When it's checked, cross sections are displayed on the map based on the defined coordinates otherwise the cross section is displayed perpendicular to the branch at the specified chainage.

Create geometry. When the coordinates are not applied, the 'Create geometry' button is active. Pressing this button fills the table with three points: one at each end of the cross section and one at the intersection with the branch.



Note: Cross sections with coordinates are fixed at the given location. This means that the chainage of the cross section used in the calculation may differ from the position shown on the map e.g. if the coordinates have been edited after the cross sections have been drawn.



Raw data

The 'raw data' tab provides the list of points defining the topography of the river bed along the cross section. These points don't have to match the list of vertices provided in the 'Coordinates' tab.

The 'S' column provides the horizontal distance of each point along the cross section, from the left end of the cross section. The 'Z' column provides the elevation of the points.

The '+' button above the table can be used to insert a new line at the bottom of the table, while the '-' button can be used to delete the active line.

Markers. Markers may be assigned to points in the 'Marker' column of the table. Markers can be assigned in two different manners: the first one is to click an element in the 'Marker' column, which opens a marker dialogue as shown in Figure 6.10 from which a requested marker number can be assigned for the selected point.

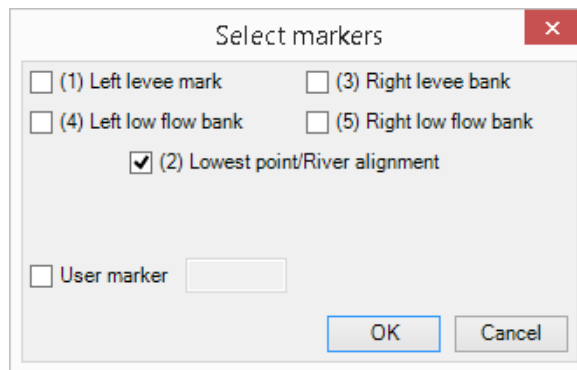


Figure 6.10 Select marker dialogue

The second manner is to select a point on the cross section plot, and press the marker number on the keyboard. Pressing the 'Ctrl' key and the marker number will remove the marker for the selected point on the cross section plot.

A number of markers may be set in this dialogue:

- **Left and right levee mark (Markers 1 & 3):** defines the extent or the active part of the cross section used for the calculations. Default placement of marker 1 and 3 is to apply marker 1 in the very first point in the raw data and marker 3 at the very last point of the raw data. Placing any of these markers at different locations will limit the extent of the active part of the cross section such that only the part of the cross section in between markers 1 and 3 is included in the simulation (that is, Processed data are only calculated for cross section data in between these markers).

- **Left and right low flow bank (Markers 4 & 5):** defines the extent of the low flow channel. The markers influence the calculation of the processed data. If defined, the section is internally divided into three major 'slices' at markers 4 and 5 positions and the resulting processed data for such a section is a sum of integration results of three sub-parts of the section instead of calculating a result from one single, large section. Additionally, markers 4 and 5 can be used to define the extent of the low flow channel which is used with the 'High/low flow zones' description of the resistance distribution in the raw cross section data.
- **Lowest point/River alignment (Marker 2):** marker 2 typically define the lowest point of the river section, or the location of the intersection with the branch. Marker 2 settings does not affect the calculations at all. Instead, it serves primarily the Map view for placement of cross sections which have no coordinates defined. It is therefore recommended to define the correct position of marker 2 in all sections.
- **User marker:** any number above 7 may be used as a user marker. User markers do not impact the simulation results. They are an option for indicating a specific point in a cross section e.g. the location of a measurement gauge.



Note: Marker locations must be defined such that marker 1 is defined to the left of marker 3 in the raw data table.

Update markers. This button updates markers 1, 2 and 3 in the current cross section, which are respectively placed at the left end point, lowest point and right end point.

Section Type. The type of cross section is set here. Four options are available:

- Open section: the typical setting for river cross sections.
- Closed irregular: closed sections with arbitrary shape.
- Closed circular: closed circular section shape where the geometry is only defined by the diameter.
- Closed rectangular: closed rectangular section shape where the geometry is only defined by the width and height.

Datum. A datum value may be entered here. The datum is normally used for adjusting the levels of the cross sections such that they conform to a specific reference datum in the model area. The datum value is added to all elevations in the 'Raw data' tab. The datum is also used for circular and rectangular sections, to set the elevation of the bottom level of the cross section.

Resistance – Type. Multiple options exist for defining the desired type of resistance method in the cross sections. The following types are available:



- **Relative resistance:** the resistance is given relative to the resistance number specified in the 'Bed resistance' menu. The resistance value specified in the cross section for this resistance type is therefore a coefficient. A coefficient higher than 1 will increase the actual roughness of the river bed, whereas a coefficient lower than 1 will decrease the actual roughness. So when the resistance type is Manning (M) in the 'Bed resistance' menu, then the Manning's M value is divided by this coefficient. When the resistance type is Manning (n), then the Manning's n value is multiplied by this coefficient.
- **Manning's n:** the resistance number is specified as Manning's n in the unit $\text{s/m}^{(1/3)}$.
- **Manning's M:** the resistance number is specified as Manning's M in the unit $\text{m}^{(1/3)}/\text{s}$ (Manning's $M = 1/\text{Manning's } n$).
- **Chezy number:** the resistance number is specified as Chezy number in the unit $\text{m}^{(1/2)}/\text{s}$.
- **Darcy-Weisbach (k):** the resistance is specified in the form of an equivalent grain diameter.



Note: Resistance numbers have to be defined in the 'Bed resistance' menu, as a global value and eventually local values. However, if absolute resistance numbers are defined in one or more cross sections as either Manning's n, Manning's M or Chezy number, then these numbers will have first priority and overrule any resistance values defined in the 'Bed resistance' menu.

Resistance – Distribution. This distribution type defines the description of the transversal resistance across the cross section. Three options are available:

- **Uniform:** a single resistance number will be applied uniformly throughout the cross section.
- **High/Low flow zones:** three resistance numbers are to be specified. The 'Left high flow' number applies between markers 1 and 4, the 'Right high flow' number applies between markers 5 and 3, and the 'Low flow' number between markers 4 and 5. If marker 4 and 5 do not exist the low flow resistance number will apply uniformly throughout the cross section.
- **Distributed:** the resistance number is to be specified for each point, in the raw data table in the 'Resistance' column. The value specified for a given point applies uniformly between this point and the previous one.

Sediment transport – Flood level definition. This controls the level of the floodplain above which no sediment transport calculation is considered. It is only used at locations where the morphological update method is Proportional to depth. Three options are available:



- **None (top of cross section):** the level is set to the highest level of the cross section, meaning that the sediment transport calculation is also performed in the floodplain when the cross section covers this floodplain.
- **Min. of markers 4 and 5:** the level is set to the lowest value between levels of markers 4 and 5.
- **User defined:** a user defined value is used.

The *Floodplain level definition* is therefore only used when the Sediment Transport module is active in the simulation, and when it includes morphological updates.

Sediment transport – Level. The level of the floodplain, when the *Floodplain level definition* is set to User defined.

Processed data

The 'Processed data' tab displays the hydraulic characteristics of the cross section which are used during the simulation. These processed data provide the values of cross section area, radius, width, bed resistance and conveyance as a function of the water level. The details of these variables are provided below:

- **Level:** levels for which processed data are calculated in the cross section. Default levels definition range from the lowest z-value and up to the highest z-value in the raw data table.
- **Cross section area:** effective cross sectional flow area calculated from the raw data. Effective area is determined from the total flow area adjusted by eventual relative resistance values different from 1 in the raw data tab (see MIKE 1D Reference manual).
- **Radius:** a resistance or hydraulic radius depending on the selected type in the 'Radius type' drop-down list.
- **Storage width:** width of the water surface for the given water level.
- **Resistance:** this factor can be used to apply a level dependent, variable resistance in the cross section. The resistance factor can contain the following two types of values depending on the Resistance Type definition in the raw data tab:



- Resistance type defined as relative resistance factor: in this case, the resistance value is interpreted as a factor by which the resistance numbers defined in the 'Bed resistance' menu will be multiplied or divided during the calculation, in order to establish a level-dependent resistance in the section. That is, the resistance factor works as a level dependent resistance scaling factor in the current section.
It is important to notice in the case of relative resistance type, that a factor higher than 1 will increase the actual roughness of the river bed, whereas a factor lower than 1 will decrease the actual roughness. So when the resistance type is Manning (M), then the Manning's M value is divided by this factor. When the resistance type is Manning (n), then the Manning's n value is multiplied by this factor.
- Resistance type defined as absolute resistance number (Manning's n, Manning's M or Chezy number): in this case, the resistance column contains the actual resistance number applied in the simulation. The resistance column can therefore have values of either Manning's M, Manning's n or Chezy numbers in this case.
- Conveyance: the conveyance values are not used in the simulation but is primarily displayed as part of the processed data for the purposes of checking that the conveyance is monotonously increasing with increasing water level, which is one of the key assumptions for the open water hydraulics.

Additionally, an additional storage area may also be defined manually, again as a function of the water level. The purpose of the additional storage area is to include an additional volume of storage in the cross section, which is not represented by the geometry in the raw data. The calculated water level in this additional storage remains strictly the same as in the cross section. This is useful for representing small storage's associated with the main branch such as a lakes, bays and small inlets. The additional storage area values describe the area of the water surface for a given water level.

Additional storage areas are always user-defined: they will never be given a value from the automatic processing of the raw data.

During the simulation, the processed data will be interpolated in order to cover the full range of water levels encountered during the simulation.



Note: Processed data are essentials in the simulation, as they describe the hydraulic aspects of the cross sections. Hence, it is important to inspect processed data and make sure that accurately describe these hydraulic parameters. It is for example important to make sure that their plots are smooth in order to correctly reproduce the progressive changes with changing water levels. If the plots show abrupt changes, it may be necessary to edit the levels at which processed data are computed. Additionally, a situation where the conveyance column is not monotonically increasing with water levels can relatively easily occur, especially in the case of some closed sections or in situa-

tions where the section geometry includes a sudden width increase and the radius type has been selected to 'Hydraulic radius'. Should this situation occur, then it is strongly recommended (not to say a strict requirement) that time is spent on adjusting the section characteristics such that a monotonically increasing conveyance curve is obtained. If not, there is a very significant risk of obtaining instabilities in the simulation for water levels in the range where the non-increasing conveyance values are present.

Typical options for optimising the cross section characteristics in the situation of an open section is to use the 'Resistance radius' type instead. Alternatively, an option using the 'Hydraulic radius' type is to manually subdivide the section into several 'slices' by adjusting the relative resistance numbers in the raw data at locations where the section's shape significantly changes (e.g. changing a relative resistance value from 1.000 to 1.001 'forces' the processed data calculator to divide the integration of the processed data into several slices and the non-monotonically increasing conveyance curve can normally be resolved from this.



Note: It is important to notice that the conveyance numbers presented in the conveyance column are actually not the 'True' conveyance values. Depending on the choice of resistance type in the 'Processed data' tab, the 'True' conveyance may depend on the resistance values specified in the 'Bed resistance' menu. However it has been decided to present conveyance values which does not include these resistance number. Consequently, the conveyance shown in the processed data does not reflect the true conveyance, but is primarily offered as a possibility for analysing the 'conveyance trend' as a function of water levels in the cross sections. And these should be monotonically increasing with water levels to secure a healthy output from the simulations.

The '+' button above the table can be used to insert a new line in the table, while the '-' button can be used to delete the active line.

Allow for recalculation. When this option is checked, the table may be automatically recomputed. Data are recomputed when changes are applied to the cross section's properties, when the 'Recompute' button in the current window is pressed, or when the 'Recompute all' or 'Recompute selected' buttons in the 'General' tab are pressed. In case the processed data have been manually adjusted, it may be necessary to uncheck this option in order to make sure to keep them unchanged afterwards.



Note: Processed data are also recomputed when the setup is saved when 'Allow for recalculation' is checked.

Recompute. This button is only active when the option 'Allow for recalculation' is checked. Pressing this button recomputes all the processed data in the table.

Radius type. The radius type may be chosen between the three following options:



- Resistance radius: a resistance radius formulation is used.
- Effective area, hydraulic radius: a hydraulic radius formulation where the area is adjusted to the effective area according to the relative resistance variation.
- Total area, hydraulic radius: a hydraulic radius formulation where the total area is equal to the physical cross sectional area.

Number of levels. The desired number of processed data levels. The automatic level selection method may not use the full number of level specified. This will occur when a smaller number of levels is sufficient to describe the variation of cross sectional parameters.

Edit levels. This button is available only if the level selection method is user-defined. The required levels are entered manually or pasted into the table.

Angle correction. An angle correction may optionally be applied to the cross section. The correction may be used for situations where the cross section profile isn't perpendicular to the center line of the river. To activate the correction, the 'Apply' checkbox must be checked, and the angle must be manually specified.

The correction applied is simply a projection of the cross sectional profile on the normal to the thalweg of the river i.e. the correction reads

$$x_{\text{cor}} = x \cos \theta \quad (6.2)$$

where θ is illustrated below

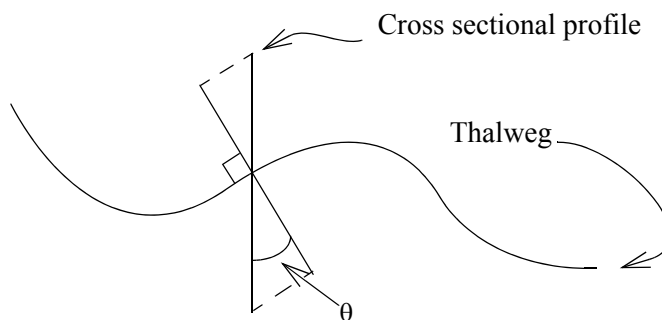


Figure 6.11 Definition sketch of the correction angle

Please note that the correction of X-coordinates is not reflected in a change of S values in the raw data table, but only in the processed data table.

Compute angle. The use of this button requires that the option 'Apply' is checked in the Coordinates tab. Pressing the button computes the angle

between the actual cross section, based on its coordinates, and the normal to the branch.

Cross section plot

The graphical view presents either a single plot for the current cross section, or eventually a number of plots from different sections if the 'Draw history on plots' option in the 'General' tab is active. The curve represents the values defined in the raw data table, with the X axis describing the S values and the Y axis describing Z values plus the datum value.

Points shown with red circles on the plot indicate the locations of markers 1, 2 and 3.

The blue curve describes the resistance value for the current cross section.



Note: To remove the history of cross sections right-click on the plot and choose 'Clear history'.

To control the settings and appearance of the plot, a number of facilities are available through a contextual menu. To open the pop-up menu point to the graphical view with the mouse cursor and press the right mouse button. A pop-up menu as presented in figure below will appear.

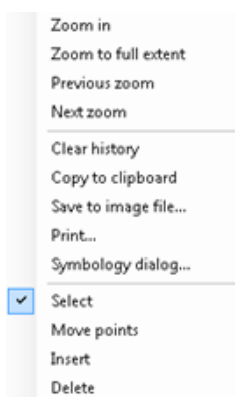


Figure 6.12 Contextual menu for the cross section plot

The pop-up menu includes the following three feature groups:

- The first group of features relates to the zooming facilities: from here the zoom in, zoom to full extent and the previous / next zoom facilities are available.



- The second group of features relates to the appearance and export of the graphical view. From here you can therefore export the image to the clipboard or to an image file on the disk, and you can also print it. Additionally the symbology dialogue allows changing the display settings of the plot.
- The third group of features relates to editing the active cross section's raw data on the plot. The following functions are available:
 - Select: when this mode is active, it is possible to select a cross section's point on the plot, which makes this point active in the raw data table. Pressing a number on the keyboard will assign the corresponding marker value to the selected point. Pressing the 'Ctrl' key + a number will remove the corresponding marker.
 - Move points: when this mode is active, it is possible to move a point graphically from the plot. The raw data table will be updated accordingly.
 - Insert: when this mode is active, new cross section's point may be added. Inserted points are interpolated between two existing points, and may be moved afterwards.
 - Delete: when this mode is active, points may be deleted from the plot.

Processed data plot

The graphical view presents either the data for the current cross section only, or eventually for a number of cross sections if the 'Draw history on plots' option in the 'General' tab is active. The curve represents the values defined in the processed data table, with the Y axis describing the level values and the X axis describing one of the other items from this table (either area, radius, width, additional storage area, resistance or conveyance). The plotted item is controlled by the drop-down list above the plot.

To control the settings and appearance of the plot, a number of facilities are available through a contextual menu. To open the pop-up menu point to the graphical view with the mouse cursor and press the right mouse button. A pop-up menu as described in Cross section plot (p. 98).

6.6 Structures

Man-made interventions and hydraulic structures in general are vital in most river application in the attempt to achieve accurate and reliable model results. Hydraulic structures impose significant internal control points to a river model and secures that it is possible to achieve the correct flow dynamics and back water effects around structures in the model.

The current version supports the use of the following types of hydraulic structures:

- Weirs

- Culverts
- Bridges
- Pump
- Gates
- Direct discharges
- Dambreak
- Energy losses
- Tabulated

6.6.1 Structure details

Structures are always located in Q-points in which the momentum equation is normally solved, and the reason for including a structure in a model is always to replace the momentum equation with something more suitable for the structure in question.

The effect of the bed friction is generally not taken into account thus it is required that the h-points (cross sections) up- and downstream are situated close to the structure.

Inserting a structure

To insert a structure in the model first choose the relevant structure type in the *Structures* ribbon in *Map view* (see Figure 6.13).

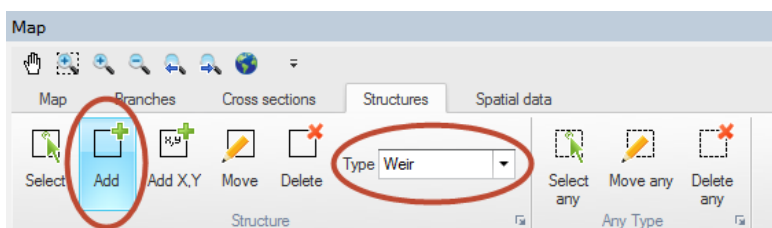


Figure 6.13 Map view ribbon for inserting and editing hydraulic structures

Select the 'Add' button and digitise the desired location of the structure by clicking on one of the rivers in the setup.






Note: The structure must be located on a river branch.

Also note that the mouse cursor changes to a cross when the cursor is properly placed.

To insert a structure at a specific geographical location select 'Add X, Y' and specify the coordinates using the map coordinate system. If specified location is not located on a branch the structure will be placed at the closest valid location.



Alternatively, structures can be inserted in the river network in *Tabular view*, where their properties can be specified. To add or remove a structure in *Tabular view*, use the 'Append' and 'Delete' buttons   at the bottom of the page. The table is synchronized with the attributes on top of the page, so the properties of the structure can be entered either in the table or at the appropriate dialogues of the tabular view.

To display the properties of a specific structure to the *Property view*, select the checkbox of that item in the table. More than one item can be selected at a time. When several table items are checked, it is possible to unselect all of them simultaneously by clicking on the 'Clear selection' button .

Composite structures definition

Composite structures are easily defined using an approach of combining a number of individual structure types. A composite structure such as a bridge with several openings and possible occasional road overtopping can be defined as a combination of multiple culverts and an overflow weir. The only requirement for creating a composite structure is, that all individual structures in the composite structure is defined at the exact same location (same River Name and Chainage). This example is illustrated in Figure 6.14.

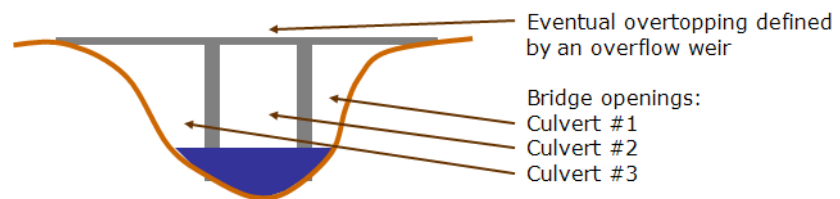


Figure 6.14 Composite structure illustration. Can be defined with a number of individual structures defined at exact same location.

One of the benefits of defining a composite structure as illustrated above is, that it is possible to obtain and analyse flow results through/over each of the individual structures forming the composite structure.

Structure types definition

Most hydraulic structures available can be defined as one of three different types; 'Regular', 'Side Structure' or 'Side structure with storage'.

Regular

Regular structures are internal structures that specify a flow in the branch typically based on up- and down- stream flow conditions.

Side structure

Side structures are special type of a regular structure. Side structures takes water out of the river network at the location where the side structure is defined.

Internally in the calculation engine, side structures are handled through an automatic generation of an 'artificial' side branch with the side structure included as a regular structure placed midway in the side branch as illustrated in Figure 6.15.

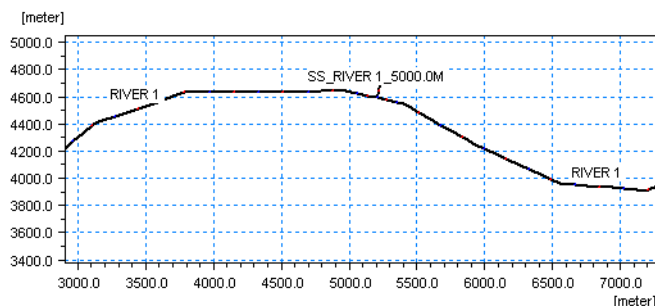


Figure 6.15 Illustration of automatically generated side structure branch

Specific characteristics of a Side Structure includes:

- Head losses are excluded from the flow calculation for side structures by setting internally the flow areas of the cross sections in the side structure branch as extremely large ($1 \times 10^{20} \text{ m}^2$) and hence, the velocity component in these sections ($V=Q/A$) is a negligible number.
- For culverts and weirs, an assumption of free out- and over-flow respectively is implemented through an internally defined water level boundary condition for the outflow point of the artificial side structure branch. The boundary condition is defined as a water level boundary with a low water level.
- Naming convention used for the artificial side branch are: "SS_"<original branch name>"_ "<original chainage>"
- In case of multiple side structures defined at the same location, these will all be included in the same side branch as a 'composite side structure'.
- A side structure branch is a simple h-Q-h branch with two cross section h-points in each end of the branch and a structure in the centre Q-point.
- Up- and downstream cross sections in the Side Structure Branch are automatically assigned with a shape identical to the main river cross section located immediately downstream of the side structure location. Upstream cross section of the Side Structure branch is identical to the section in the main river, but the downstream section has been shifted 10 m down in order to obtain a free outflow condition in the Side Structure. See Figure 6.16.

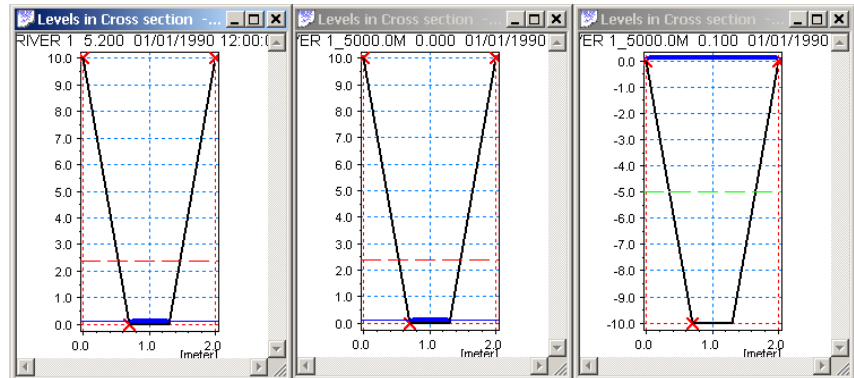


Figure 6.16 Cross sections as defined in the artificial Side Structure branch. Left part illustrates the cross section in main river downstream of the chain-age defining the Side Structure. Middle part illustrates the upstream cross section at chainage 0 of the side structure branch (=copy of main branch section) and right part illustrates the downstream cross section of the Side Structure Branch, which is identical shape as the other sections but only shifted -10 m.

Side Structure with Storage

Including side structures with storage in a branch results in automatic generation of a side branch with the structure placed midway as described for side structures without storages. In addition to this, the downstream cross section of the side structure branch has a user-defined, additional storage area (reservoir). The significant difference between a structure defined as 'Side structure' and 'Side structure with storage' is, therefore, that in the latter case, the water conveyed through the side structure is stored in the system, the reservoir storage can be continuously measured and water from the storage can potentially return to the main river through the side structure in case water levels in storage gets higher than the water level in the river. For the structure type without the storage, the water simply leaves the river model and disappears completely.

When using a side structure with reservoir an additional tab 'Storage' will appear next to the 'General' tab for specification of reservoir details.

Storage capacity type. The storage capacity type of the reservoir must be defined using the drop-down menu. There are two options:

- **Elevation-area:** Additional storage area is derived directly from the Elevation-Area relationship by using the water level in the reservoir. An Elevation-Area relationship is specified by entering corresponding values of Elevation and Area in the table. Click the Append button to add a row to the table.



- **Elevation-volume:** Additional storage area is calculated by using the Elevation-Storage volume relationship and the following formula:

$$FIA_{i+1} = \left(\frac{\Delta V_{Table}}{\Delta H_{Table}} - FIA_i \right) \cdot 2 \quad (6.3)$$

where:

- FIA: Flooded or additional storage area at level 'i' or 'i+1'
- ΔV_{Table} and ΔH_{Table} are differences in volumes and levels respectively from values defined in the Elevation-Storage volume table.

Initial surface area. The initial surface area must be defined if *Elevation-volume* storage capacity type is selected.

Apply Coordinates. The Apply coordinates offers a possibility for defining coordinates for the reservoir - which in the case of the artificially generated side structure branch is identical to the end point of the side structure branch.

Additional specific features of the Side Structure with Reservoir:

- Head losses are excluded from the flow calculation for side structures with reservoir by setting internally the flow areas of the cross sections in the side structure branch as extremely large (1e20 m²) and hence, the velocity component in these sections ($V=Q/A$) is a negligible number.
- The boundary introduced through the inclusion of the side structure is internally specified as a no flow boundary. Flow may be included through specification of point source boundaries at the boundary.
- More side structures with reservoir in the same location will result in one side branch for each.
- The side branches will be named with the naming convention: "SSPR"<original branch name>"_"<original chainage>"_"<Structure ID>.
- The cross sections applied in the artificially generated Side Structure Branch with Reservoir are defined such that the upstream cross section is a copy of the section in the main river downstream of the chainage defining the side structure and the downstream cross section is defined as a triangular cross section shape as illustrated in Figure 6.17.

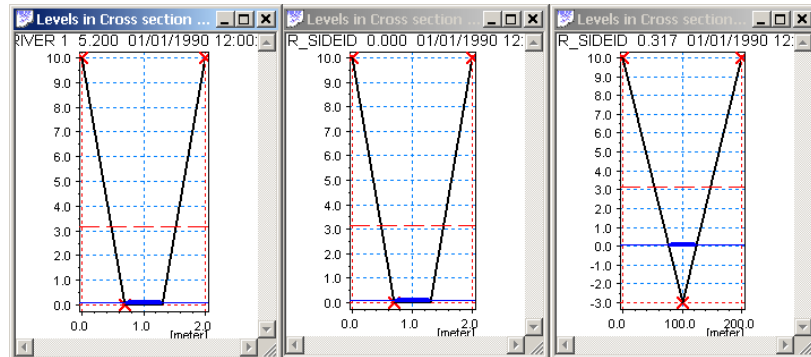


Figure 6.17 Cross sections as defined in the artificial Side Structure with Reservoir branch. Left section illustrates cross section in main river downstream of chainage defined for Side Structure. Middle section illustrates the upstream section of the side structure branch (copy of main branch section) and right section illustrates triangular section applied at downstream h-point of Side Structure Branch.

Levels of the cross section is adapted from the defined Level-Area/Volume table, whereas the widths (0 m at bottom and 100 m in top of section are just artificial numbers applied in order to draw a cross section in result presentation tools. The calculated storage volume of the section is defined from the table-values.

Structure geometry definition

Some of the available structure types require a Structure Geometry definition to complete the structure input. These include the Weir and Culvert structure types.

It is important to notice, that the Structure Geometry to be defined is the geometry - or level-width relation - of the sections of the hydraulic structure where the water can potentially flow. That is, the Flow Area definition as illustrated in Figure 6.18 marked with the blue section.

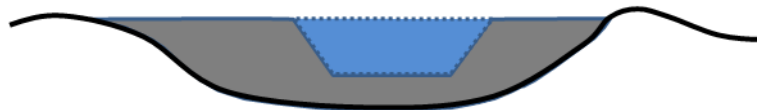


Figure 6.18 Illustration of Structure Geometry definition. The blue area which potentially conveys water over/through the structure is the geometry which must be defined.



Example: Assuming the following geometrical definition of the blue flow area in Figure 2.36: Crest level of structure is at level 3 m (crest level = bottom of blue area in figure), bottom width is 5 m, top-width is 7 m at a level of 5 m. A

structure geometry definition in this case is normally defined as presented in Table 6.1 below.

Table 6.1 Weir Geometry definition

Level [m]	Width [m]
3.00	0.00
3.05	5.00
5.00	7.00

Structure plotting

For structures where specification of dimensions and elevations is part of defining the structure, it is possible to make a plot of the structure together with the upstream and downstream cross section. This includes all weirs, culverts and gates. Plotting structures and their neighbouring cross sections together helps ensuring that structure dimensions and elevations are specified correctly.



Note: A typical source of model instability is a structure which is larger or lower than the neighbouring cross section.

For structures which can be plotted the dialogue contains a section named 'Graphics' with a 'Plot/Refresh' button. Clicking this will open a separate graphical window showing two plots. The left plot being the upstream cross section and the structure, and the right plot being the downstream cross section and the structure. Initially, the centre of the structure is horizontally aligned to marker 2 of the cross section.

The following figure shows a typical structure plot for a weir.

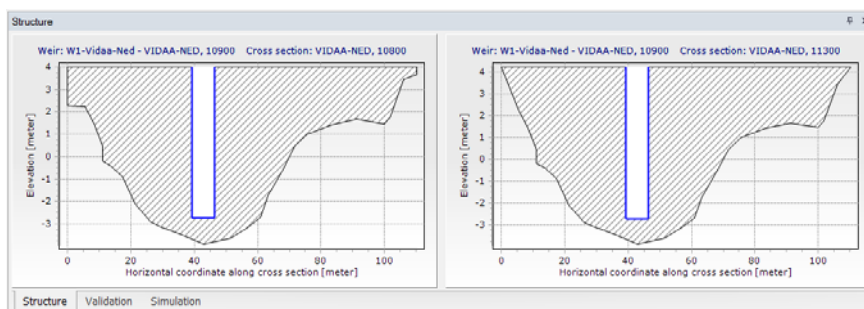


Figure 6.19 Structure Plot



6.6.2 Weirs

General

The *General* tab includes that majority of definitions for structures. The following parameters can be specified.

Location

ID.

String identification of the weir. ID will be used in structure results and hence makes it easier to identify a specific structure if setup contains many structures, or if multiple structures are defined at the same location.

Branch name.

The Branch name is automatically registered where the weir is located.

Chainage.

Chainage at which the weir is located.

Type.

The location type may be defined as Regular, Side Structure or Side Structure with Reservoir. See Structure types definition (p. 101) for details.

Graphics

Horizontal offset from marker 2.

The horizontal position of the weir within the cross section can be defined. By default the centre of the weir is positioned at marker 2. Note that the position is used only to plot/display the weir and has no impact on the simulation.

Geometry

Here, the geometry of the weir is defined. The exact contents of the dialogue window depends on the type of weir.

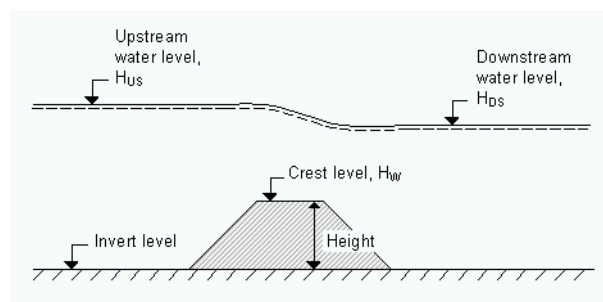


Figure 6.20 Definition sketch for Weir formula



Note: Since a weir is defined as a structure causing a contraction loss and subsequently an expansion loss, the geometry of the weir must be such that



the cross sectional area at the weir is less than the cross sectional area at both the upstream and the downstream cross section for all water levels.

Broad Crested weirs

Type.

Broad Crested weirs have two different ways for defining the weir geometry:

- **Level-Width:** The weir geometry is specified as levels (relative to the datum) and corresponding flow widths.

Datum: Offset which is added to the level column in the level/width table.

Level/Width table: This table includes user defined levels and corresponding flow widths. Click on the Append button to add rows to the table. Note that values in the 'Level' column must be increasing.

- **Cross Section DB:** The weir geometry is specified in the cross section editor. Select the branch name, TopoID and chainage of the cross section, which describes the structure's geometry.

Villemonte formula

Width.

Width of the flow.

Height.

Weir height. See Figure 6.20.

Invert Level.

Bottom datum level. See Figure 6.20.

Honma formula

Width.

Width of the flow.

Crest level.

Weir level. See Figure 6.20.

Extended Honma formula

Width.

Width of the flow.

Height.

Weir height. See Figure 6.20.

Crest level.

Weir level. See Figure 6.20.



Attributes

Weir type.

The Weir type defines the type of Weir formulation used to calculate the flow dynamics over the structure:

- **Broad Crested:**
For Broad Crested weirs a hydrostatic pressure condition is assumed and the Q/h relationship is calculated automatically.
- **Villemonte Formula:**
A standard weir expression reduced according to the Villemonte formula is applied. See the MIKE 1D Reference Manual.
- **Honma Formula:**
The Honma weir expression. See the MIKE 1D Reference Manual.
- **Extended Honma Formula:**
An extended version of the Honma weir formula is applied. See the MIKE 1D Reference Manual.

Valve.

Valves may be included to control the direction of the overflow.

- **None:**
No valve regulation applies.
- **Only Positive Flow:**
Only positive flow is allowed, i.e. whenever the water level downstream is higher than upstream the flow through the structure will be zero.
- **Only Negative Flow:**
Only negative flow is allowed, i.e. whenever the water level upstream is higher than downstream the flow through the structure will be zero.
- **No Flow:**
The flow through structures will always be zero, regardless of upstream and downstream water levels.

Flow blockage

Apply flow factor

When this option is active, the discharge computed through the weir is multiplied by a flow factor. This factor's value is specified in the Flow factor field. The factor is a dimensionless factor, and a value of 1 means that no change is applied to the computed discharge. A value lower than 1 can typically be used to describe the reduction of the flow through the structure due to obstacles, like debris, restricting the flow area in the structure.

Head loss

The factors determining the energy loss occurring for flow through hydraulic structures (only active for broad crested weirs). Head loss factors can be



defined with different values depending on the flow direction across the weir (positive or negative flow direction).

Weir formula parameters

Here the Weir formula parameters are specified. The contents of the dialogue depends on the type of weir (and is not relevant for Broad Crested weirs).

Villemonte formula

The Villemonte formula reads:

$$Q = W \cdot C \cdot (H_{us} - H_w)^k \cdot \left[1 - \left(\frac{H_{ds} - H_w}{H_{us} - H_w} \right)^{k-0.385} \right] \quad (6.4)$$

Where Q is discharge through the structure, W is width, C is weir coefficient, k is the weir exponential coefficient, H_{us} is upstream water level, H_{ds} is downstream water level and H_w is weir level.

Weir coefficient.

Weir coefficient is the multiplication coefficient 'C' in the weir formula.

Weir exponential coefficient.

Weir exponent coefficient is the exponential coefficient 'k' in the weir formula.

Honma formula

The Honma formula reads:

$$Q = \begin{cases} C_1 W (H_{us} - H_w) \sqrt{(H_{us} - H_w)} & \text{for } (H_{ds} - H_w)/H_{us} < 2/3 \\ C_2 W (H_{ds} - H_w) \sqrt{(H_{us} - H_{ds})} & \text{for } (H_{ds} - H_w)/H_{us} \geq 2/3 \end{cases} \quad (6.5)$$

Where Q is the discharge through structure, W is the width, C_1 is the first weir coefficient, $C_2 = (3/2)\sqrt{3}C_1$ is the second weir coefficient, H_{us} is the upstream water level, H_{ds} is the downstream water level and H_w is weir level

Weir coefficient.

The Weir coefficient is the multiplication coefficient 'C1' of the Honma weir formula.



Extended Honma formula

The Extended Honma formula calculates the flow in three flow regimes: 1) perfect flow, 2) imperfect flow, and 3) submerged overflow. The equations reads:

$$Q = \begin{cases} \left[C_0 W h_u^{3/2} \right. & , \text{ perfect} \\ C_0 (\text{Alpha} \cdot (h_d/h_u)^q + \text{Beta}) W h_u^{3/2} & , \text{ imperfect} \\ \left. C_0 (\text{Gamma} \cdot (h_d/h_u)^r + \text{Delta}) W h_d (h_u - h_d)^{1/2} \right] & , \text{ submerged} \end{cases} \quad (6.6)$$

where

$$C_0 = a \cdot (h_u/H)^p + b$$

W is weir width

H is weir height above cross section invert

h_d is the upstream water level above the crest

h_u is the down stream water level above the crest

Perfect overflow parameters

$(h_d/h_u)_i$ is the depth ratio limit between perfect and imperfect flow regime. Parameters a , b and p are used to calculate the weir coefficient C_0 .

Imperfect overflow parameters

Parameters Alpha, Beta and q are used in imperfect overflow calculation

Submerged overflow parameters

$(h_d/h_u)_s$ is the depth ratio limit between imperfect and submerged flow regime. Parameters; Gamma, Delta and r used in Submerged overflow calculation



Note: It is not checked if there is a continuous transition from one flow regime to the next. This has to be ensured by the user through proper selection of the parameters.

Flow conditions

For broad crested weirs an additional tab is shown for flow conditions. Free overflow Q/h relations are calculated automatically by clicking 'Calculate Q/h relations' once the no. of Q/h relations have been specified.

The results are shown in the Q/h -relations table and includes:

- **Q:** Structure discharge
- **H-Pos:** Level upstream in case of positive flow

- **H-Neg:** Level downstream in case of negative flow
- **H-Weir:** Level at structure
- **Width:** Structure width for actual structure level
- **Area:** Structure area for actual structure level

To compute the Q/h relations, the nearest upstream and downstream cross sections are used. The cross sections must be located within a distance which is smaller than the user-defined maximum grid-spacing (*'Maximum dx'*) for the branch in question. The Q/h relations cannot be calculated unless cross sections are defined.



Note that Q/h relations must be re-calculated if any changes are made to the weir and/or up/down-stream cross sections. This is done by either pressing the button *'Calculate Q/h relations'* for an individual structure or by selecting *'Recalculate flow conditions...'* in the *'Run'* drop- down menu in the MIKE Zero menu bar for recalculating all structures.

Allow for recalculation. Q/h relations can either be calculated automatically or entered/edited manually. If the Q/h relations have been modified manually it is possible to lock the relations by unchecking *'Allow for recalculation'*. This will ensure that Q/h relations for this structure are not recalculated through the menu *'Recalculate flow conditions...'*.

6.6.3 Culverts

General

Under the *General* tab the following types of information may be specified.

Location

ID. String identification of the culvert. ID will be used in structure results and hence makes it easier to identify a specific structure if setup contains many structures, or if multiple structures are defined at the same location.

Branch name. The Branch name is automatically registered where the culvert is located.

Chainage. Chainage at which the culvert is located.

Type. The location type may be defined as Regular, Side Structure or Side Structure with Reservoir. See Structure types definition (p. 101) for details.

Graphics

Horizontal offset from marker 2. The horizontal position of the culvert within the cross section can be defined. By default the centre of the culvert is positioned at marker 2. Note that the position is used only to plot/display the culvert and has no impact on the simulation.



Attributes

Valve. Valves may be included to control the direction of the flow.

- **None:** No valve regulation applies.
- **Only Positive Flow:** Only positive flow is allowed, i.e. whenever the water level downstream is higher than upstream the flow through the structure will be zero.
- **Only Negative Flow:** Only negative flow is allowed, i.e. whenever the water level upstream is higher than downstream the flow through the structure will be zero.
- **No Flow:** The flow through structures will always be zero, regardless of upstream and downstream water levels.

Head loss

Head loss variables are included in the calculation of the flow direction dependent energy loss occurring for flow through hydraulic structures. They include:

Manning's n. Manning's bed resistance number along the culvert (for calculation of the friction loss present in a structure with a certain length).

Positive/Negative flow. Headloss factors are defined for Inflow, Outflow, Free overflow and Bends. Different loss factors can be applied depending on the flow direction across the structure.

Flow blockage

Apply flow factor

When this option is active, the discharge computed through the culvert is multiplied by a flow factor. This factor's value is specified in the Flow factor field. The factor is a dimensionless factor, and a value of 1 means that no change is applied to the computed discharge. A value lower than 1 can typically be used to describe the reduction of the flow through the structure due to obstacles, like debris, restricting the flow area in the structure.

Geometry

Here, the geometry of the weir is defined.

Type. The type of cross sectional geometry of a culvert can be specified as:

- **Rectangular:** The geometry is specified by width and height.
- **Circular:** The geometry is specified by the diameter.
- **Irregular:** The geometry is specified using a level/width table or a depth/width table. Select the relevant table and click on the Append button to include rows. Note that the values in the first column (level or depth) must be increasing.

- **Cross Section DB:** The culvert geometry is specified in the cross section editor. Select the branch name, TopoID and chainage of the cross section, which describes the structure's geometry. The selected cross section must be a closed section.

Upstream invert. Invert level of the upstream end of the culvert.

Downstream invert. Invert level of the downstream end of the culvert.

Length. Length of the culvert.

No. of culverts. Number of parallel culverts.

Section type. For rectangular and irregular culverts the culvert type must be specified as either open or closed. For circular culverts it is closed and for culverts defined with a cross section the cross section definition determines if the culvert is open or closed.



Note: Since a culvert is defined as a structure causing a contraction loss, a friction loss (bend loss) and subsequently an expansion loss, the geometry of the culvert must be such that the cross sectional area at the inflow is less than the cross sectional area upstream of the culvert for all water levels. Similarly, the cross sectional area at the outflow end must be less than the cross sectional area immediately downstream of the culvert.

Flow conditions

For culverts an additional tab is shown for flow conditions. Different hydraulic relations are calculated automatically by clicking '*Calculate Q/h relations*' once the no. of Q/h relations have been specified.

Q/h relations

The Q/h relations are given as Q/y relations (where y is depth above the lowest of the Upstream and Downstream invert levels). The Q/y relations table also shows the type of flow occurring for the actual levels. Please note that the flow-type definitions in the tables are only flow indicators derived from the Q/h table calculation, and these indicators are not directly impacting the calculation results. Hence, there will be no effect of changing these flow-type definitions on the calculation results.

The possible types are:

- **No Flow:** No flow occurs at the first level ($y = 0$) and when the valve regulation flag prohibits flow in one direction.
- **Inlet C:** The flow at the inlet is critical.
- **Outlet C:** The flow at the outlet is critical. A backwater curve using a fine resolution is calculated to relate the discharge to the upstream water level in the river.



- **Orifice:** The flow at the culvert inlet has an orifice type formation.
- **Full:** The culvert is fully wet with a free discharge at the outlet.

Orifice flow coefficients

Orifice type discharge is calculated based on the orifice coefficient tables shown in the this menu. These coefficients can be edited, added or deleted, if required. After editing the Orifice flow coefficients it is required to recalculate the Q/h relations using the 'Calculate Q/h relations' button.

Hydraulic parameters

The hydraulic parameters is a table of level dependent parameters used in the calculation of Q/h relations. The Hydraulic parameters are also used internally in the hydrodynamic engine when calculating the structure flow for other flow conditions than the free outflow. Hydraulic parameters includes Flow Area, Hydraulic Radius and Conveyance calculated for the level 'Y' (depth above culvert invert).

To compute the Q/h relations, the nearest upstream and downstream cross sections are used. The cross sections must be located within a distance which is smaller than the user-defined maximum grid-spacing ('*Maximum dx*') for the branch in question. The Q/h relations cannot be calculated unless cross sections are defined.



Note that Q/h relations must be re-calculated if any changes are made to the culvert and/or up/down-stream cross sections. This is done by either pressing the button '*Calculate Q/h relations*' for an individual structure or by selecting '*Recalculate flow conditions...*' in the '*Run*' drop- down menu in the MIKE Zero menu bar for recalculating all structures.

Allow for recalculation. Q/h relations can either be calculated automatically or entered/edited manually. If the Q/h relations have been modified manually it is possible to lock the relations by unchecking '*Allow for recalculation*'. This will ensure that Q/h relations for this structure are not recalculated through the menu '*Recalculate flow conditions...*'.

6.6.4 Bridges

Working with bridges

When working with a bridge structure, the behavior of the overview table is different than the tables used for other structure types. With bridges, two different levels of structures are introduced:

- A primary structure, which is here called the main structure. The main structure stores information about the location only.

- A secondary structure, which is here called an opening. A main structure may have multiple openings, which describe the various apertures under the bridge. Each opening is bound to the main structure on which it has been created, and a minimum of one opening is required for each main structure.

Figure 6.21 illustrates an example of bridge containing multiple openings.

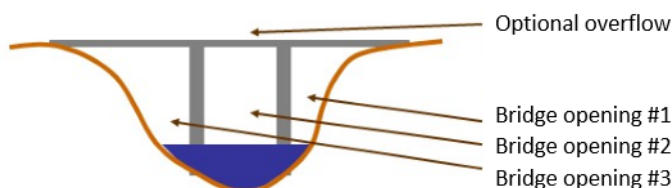


Figure 6.21 Example of bridge openings

Openings are shown at a sub level of the main structure's level, in the overview table. Figure 6.22 shows the main structures (e.g. Bridge 3) and their openings (e.g. Bridge 3 Opening 2) in the overview table. The active items are indicated by the black triangles to the left of the row numbers.

+ - x									
	ID	Branch Name	Chainage						
1	Bridge 2	GRONAA-TM	2030						
2	Bridge 3	LINDSKOV	1915						
	ID	Piers	Waterway blocked	Asymmetric opening	Skewness	Default table (skewness)	Angle	Eccentricity	
1	Bridge 3 Opening 1	<input type="checkbox"/>	0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	<input type="checkbox"/>	
2	Bridge 3 Opening 2	<input type="checkbox"/>	0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	<input type="checkbox"/>	
3	Bridge 3 Opening 3	<input type="checkbox"/>	0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	<input type="checkbox"/>	
III									
3	HwayRd	VIDAA-MEL	351						

Figure 6.22 Main structures and openings in the overview table

A main structure may be added or removed by pressing respectively the '+' or '-' button above the overview table. These buttons control only the main structures but not the openings. Removing a main structure will remove all its openings as well.

An opening can be added or removed by activating the related main structure, and by use of the buttons available in the 'Multiple openings' options.

General

Under the *General* tab the following types of information may be specified.



Location

The location options contain a maximum of three items, which availability depends on the structure type:

ID. String identification of the main structure or opening.

Branch name. The name of the branch on which the bridge is defined. The branch name is specified only for the main structures.

Chainage. Chainage at which the bridge is located. The chainage is specified only for the main structures.

Multiple openings

Three buttons are available to add or remove openings for the active main structure.

Add opening. This button adds a new opening with default settings.

Copy opening. This button adds a new opening being a copy of an existing opening, selected in the 'From' list. The new opening gets the same parameters as the copied one.

Remove opening. This button removes the active opening from the active main structure.

Flow blockage

Apply flow factor

When this option is active, the total discharge computed through the bridge (i.e. the sum of discharges through the various openings) is multiplied by a flow factor. This factor's value is specified in the Flow factor field. The factor is a dimensionless factor, and a value of 1 means that no change is applied to the computed discharge. A value lower than 1 can typically be used to describe the reduction of the flow through the structure due to obstacles, like debris, restricting the flow area in the structure.

Free surface flow

The parameters available here control the free surface flow equations applied for the active opening. The number of parameters depends on the selected method.

Method.

The method controls the equation applied to describe the free surface flow through the opening. The available methods can be divided into pure free flow methods and methods which may be combined with submergence/overflow methods. The pure free flow methods can be further subdivided into methods for piers and methods for arches. The choice of method should be based on the assumptions for the different methods and the requirements of the modelling. The available methods are:

- Energy equation: a standard step method where a backwater surface profile determination is used to calculate the discharge through the opening. The method takes the contraction and expansion loss for bridges of arbitrary shape into account. The method assumes sub-critical flow and may default to critical flow for steep water surface gradients.
- FHWA WSPRO: the Federal Highway Administration (FHWA) WSPRO method is based on the solution of the energy equation. Contraction loss is taken into account through the calculation of an effective flow length. Expansion losses are determined through the use of numerous experimentally based tables. The method takes the effect of eccentricity, skewness, wingwalls, embankment slope etc. into account through the use of these tables.
- USBPR: the US Bureau of Public Roads (USBPR) method estimates free surface flow assuming normal depth conditions. The method is based on experiments and takes the effect of eccentricity, skewness and piers into account.
- Yarnell piers: an equation derived from experiments for normal flow conditions in the sub critical flow range. The effect of piers is handled through the use of adjustment factors.
- Nagler piers: an orifice type of flow description with the effect of the piers taken into account through an adjustment factor.
- Hydraulic Research arch: the Hydraulic Research method is based on laboratory experiments of both single and multi-spanned arch bridges in rectangular channels. The method uses tables describing the relation between the blockage ratio, the downstream Froude number and the upstream water level.
- Biery and Delleur arch: an orifice type of equation is used to describe the discharge through the opening. The equation is derived under the assumption of a rectangular channel and is based on a single span arch opening. Multiple arch openings are handled by a simple multiplication factor.

Resistance type. Defines the type of resistance value specified for the opening. The type can be either Manning's M or Manning's n.

Resistance value.

The value for the resistance on the bridge structure.

This resistance value will per default apply all along the bridge's cross sections. However, if these cross sections have markers 'Left abutment' and 'Right abutment' specified, then it is assumed that the part of the cross sections in-between represents the river bed, and the resistance applied between these markers is obtained from the resistance applied to the upstream and downstream cross sections. In this case, the resistance value specified for the opening only applies from the left extent of the bridge's cross section to 'Left abutment' marker, and from 'Right abutment' marker to the right extent of the bridge's cross section.

Ratio. Selects between the channel contraction ratio ($m=1-M$) or bridge opening ratio (M) as parameter in the various tables.

Opening type. For the FHWA WSPRO method, the opening type describes the type of embankment and abutment, through four different types: I, II, III and IV.

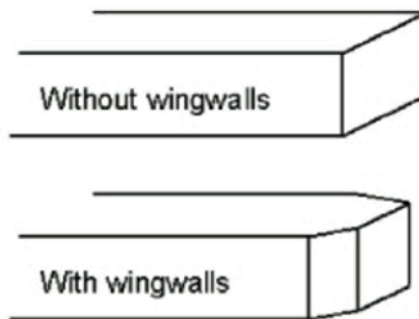


Figure 6.23 Definition sketch of opening type I (vertical embankments and vertical abutments, with or without wingwalls) (after Matthai)



Figure 6.24 Definition sketch of opening type II (sloping embankments without wing-walls) (after Matthai)

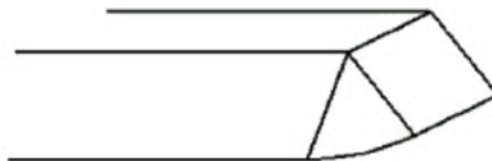


Figure 6.25 Definition sketch of opening type III (sloping embankments and sloping abutments; spillthrough) (after Matthai)

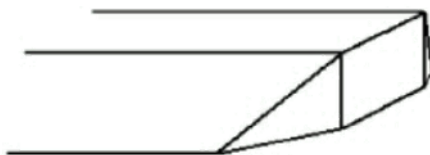


Figure 6.26 Definition sketch of opening type IV (sloping embankments and vertical abutments with wingwalls) (after Matthai)

Waterway length. Length of the waterway in the flow direction.

Level of WW length. The level at which the waterway (WW) length is measured.

Contraction loss coef. The contraction loss factor.

Expansion loss coef. The expansion loss factor.

Opening width. For an arch: opening width at the arch spring line (b , in figure below). For piers: total opening width between the piers.

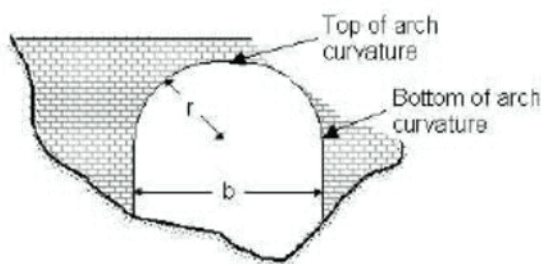


Figure 6.27 Geometrical description of an arch

Type of piers. Describes the shape of the piers:

- Squared
- Semi-circular
- Triangular
- Twin cylinder 1
- Twin cylinder 2
- Lens-shaped

Number of arches. The number of arches in the opening.



Level of bottom of arch curvature. Level of arch spring line, shown on Figure 6.27 above.

Level of top of arch curvature. Level of upper most point in the arch, shown on Figure 6.27 above.

Radius of arch curvature. Radius of the arch (r , in Figure 6.27 above).

Use default table. When this option is checked, default loss factor tables are used in the simulation. When unchecked, user-defined tables may be used, which are accessed with the 'Edit table' button.

Geometry and tables windows

User-defined loss tables are edited in pop-up windows, which content differs depending on the selected method. For the FHWA WSPRO method, the window also contains additional geometrical parameters.

Use default table. When this option is checked, the simulation uses the default loss table for the related item. When unchecked, the simulation uses a user-defined table, which appears in a separate tab. In this user-defined table, it is possible to add more rows and sometimes more columns, and edit the values.

Please refer to the reference manual for descriptions of variables used in the tables for the various methods.

Number of rows. For some user-defined loss tables, the number of rows is controlled by this value.

Number of columns. For some user-defined loss tables, the number of columns is controlled by this value.

Type (Entrance rounding, FHWA WSPRO method).

Entrance rounding type for opening type I with FHWA WSPRO method. Two types are available:

- Corner
- Wingwall.

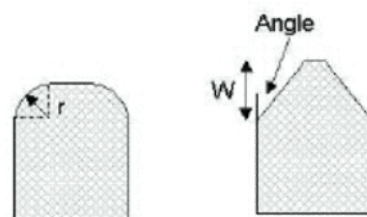


Figure 6.28 Rounding type for opening type I (Left: corner; Right: wingwall)



Radius. Radius of the corner (r , in Figure 6.28 above).

Width. Width of the wingwall (W , in Figure 6.28 above).

Angle (Entrance rounding). Angle of the corner (Angle, in Figure 6.28 above).

Embankment slope. The slope of the embankment for opening types II, III, and IV with FHWA WSPRO method. For example, use a value 2 for a 1:2 slope.

Angle (Wingwalls). Angle of the wingwall for opening type IV with FHWA WSPRO method.

Type (Base coefficient, USBPR method). The type of opening. Three types are available:

- 90° wingwall
- 45° wingwall
- Spillthrough

Options

Additional options may be added to the free surface flow calculation. Available options depend on the selected method. To include an option, simply tick the corresponding box.

Most of the options are taken into account through the use of adjustment factors defined in tables. It is possible to either use the default tables or use user-defined tables.

Piers and piles.

Used to include the effect of piers and piles on the free surface flow calculation.

Waterway blocked. The proportion of the waterway blocked by piers/piles.

Type.

For the FHWA WSPRO method, two types are available:

- Piers
- Piles.

For the USBPR method, the following types of piers are available:

- Single cylinder
- Twin cylinder 1
- Twin cylinder 2
- Series of cylinder



- Single squared
- Twin squared 1
- Twin squared 2
- Series of squared
- Semi-circular
- Series of I-formed.

Use default table. When this option is checked, the simulation uses the default loss tables for piers and piles. When unchecked, the simulation uses user-defined tables, which are accessed in extra tabs.

Please refer to the reference manual for descriptions of variables used in the tables.

Opening type. The loss tables used for piers and piles depend on the opening type, specified in the Free surface flow options. The opening type used is therefore reminded here for reference. The opening type can only be changed from the Free surface flow parameters.

Asymmetric opening

Used for individual definition of left and right abutments' properties. When this option is active, most of the opening's parameters (in the Free surface flow parameters, or in pop-up windows) have to be defined separately for the left and for the right side.

Skewness

Used when the embankments are not perpendicular to the flow direction.

Angle. Angle describing the orientation of abutments.

Use default table. When this option is checked, the simulation uses the default loss table for skewness. When unchecked, the simulation uses a user-defined table, which is accessed through the 'Edit table' button.

Please refer to the reference manual for descriptions of variables used in the table.

Eccentricity

Used when the bridge opening is eccentrically located in the river.

Use default table. When this option is checked, the simulation uses the default loss table for eccentricity. When unchecked, the simulation uses a user-defined table, which is accessed through the 'Edit table' button.

Please refer to the reference manual for descriptions of variables used in the table.



Spur dykes

Used to include the effect of spur dykes on the free surface flow calculation.

Use default table. When this option is checked, the simulation uses the default loss tables for spur dykes. When unchecked, the simulation uses user-defined tables, which are accessed in additional tabs.

Please refer to the reference manual for descriptions of variables used in the tables.

Length. The length of the spur dyke.

Type. The type of the spur dyke, which can be either straight or elliptical.

Angle. The angle, for an elliptical spur dyke.

Offset. The offset, for a straight spur dyke.

Opening type. The loss tables used for spur dykes depend on the opening type, specified in the Free surface flow options. The opening type used is therefore reminded here for reference. The opening type can only be changed from the Free surface flow parameters.

Submergence and overflow

Under the *Submergence and overflow* tab, it is possible to optionally include either submergence, or submergence and overflow. It is not possible to include overflow without submergence.

These options are only available in combination with the Energy equation, FHWA WSPRO or USBPR method for free surface flow calculation.

Submergence

Method. The method controls the equations applied to describe the submerged flow through the opening. The choice of method should be based on the assumptions for the different methods and the requirements of the modeling. The available methods are:

- FHWA WSPRO: the Federal Highway Administration (FHWA) WSPRO method for pressure flow is based on two orifice equation descriptions. One for situations when the orifice is submerged downstream and a modified equation for situations when only the upstream part of the orifice is submerged.
- Energy equation: the flow under the opening is determined through a standard backwater step method. The flow is assumed to be in the sub-critical range and thus the method may default to critical flow. Both contraction and expansion losses are taken into account.



- **Culvert:** a standard culvert description may be chosen for submergence flow, where the culvert is specified in the 'Culverts' page. The culvert is only active if submergence occurs.

Coefficient of discharge. The discharge coefficient in the FHWA submergence equation.

Use default table. When this option is checked, the simulation uses the default table for coefficient of discharge. When unchecked, the simulation uses a user-defined table, which is accessed with the 'Edit table' button.

Please refer to the reference manual for descriptions of variables used in the table.

Contraction loss. The contraction loss coefficient.

Expansion loss. The expansion loss coefficient.

Bridge soffit level. The soffit (underside) level of the opening.

Culvert. The ID of the selected culvert to be used, from the list of valid structures in the 'Culverts' page. The culvert must be located on the same branch and the same chainage as the bridge.

Overflow

Method. The method controls the equations applied to describe the flow over the bridge. The choice of method should be based on the assumptions for the different methods and the requirements of the modelling. The available methods are:

- **FHWA WSPRO:** the Federal Highway Administration (FHWA) WSPRO method is based on a weir equation taking tail water submergence into account through the use of a submergence coefficient. The method may be used for both gravel and paved surfaces.
- **Energy equation:** the flow over the bridge is determined through a standard backwater step method. The flow is assumed to be in the subcritical range and thus the method may default to critical flow. Both contraction and expansion losses are taken into account.
- **Weir:** a standard weir description may be chosen for overflow, where the weir is specified in the 'Weirs' page. The weir is only active if overflow occurs.

Bridge deck level. The deck (underside) level for the opening.

Length. The length of top of embankment in the flow direction.

Use default tables for coefficient of discharge. When this option is checked, the simulation uses the default tables. When unchecked, the simulation uses user-defined tables, which are accessed with the 'Edit table' button.

Please refer to the reference manual for descriptions of variables used in the tables.

Surface. The surface of the top of the bridge, required when using default tables, and which can be of type:

- Paved
- Gravel.

Weir. The ID of the selected weir to be used, from the list of valid structures in the 'Weirs' page. The weir must be located on the same branch and the same chainage as the bridge.

Bridge's cross sections

Under the *Bridge's cross sections* tab, it is required to specify the geometry of the bridge opening, on both its upstream and its downstream side. The corresponding bridge's cross sections should describe the bottom of the opening as well as its lateral walls, and are used for the free surface flow calculation. When submergence and overflow are included, the levels of submergence and overflow are specified in the Submergence and overflow tab, and therefore don't have to be included in the bridge's cross sections.

The upstream and downstream bridge's cross sections are illustrated on Figure 6.29, representing a plan view of the bridge, respectively as sections 2 and 3. Cross sections 1 and 4 are the closest cross sections defined in the Cross sections menu.

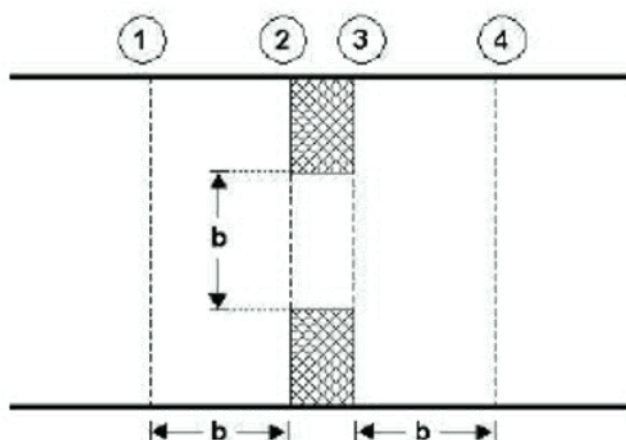


Figure 6.29 Location of up- and downstream cross sections.

The location of the cross sections outside the bridge should be so that any potential contraction or expansion loss is taken into account. In other words,



the optimal location is where the stream lines are parallel prior to a contraction and post a possible expansion. As a rule of thumb, the distance between the bridge and the cross sections should be in the order of one opening width.

Upstream section, free surface flow

The upstream cross section of the opening is defined in a table, which provides a list of points describing the topography along the lateral walls and the bottom of the opening.

The 'S' column provides the horizontal distance of each point along the cross section, from the left end of the cross section. Note that the S-values are compared to the S-values of the upstream cross section, and it is therefore important that S-values are consistent between these cross sections.

The 'Z' column provides the elevation of the points.

The 'Resist.' column provides the relative resistance in the cross section point. A factor 1 corresponds to the resistance value specified in the General tab.

The 'Marker' column may optionally contain the following markers:

- 'Left extent': defines the left extent of the active part of the cross section used for the calculation. All points of the cross section located to the left of this marker are ignored. If this marker is not defined in the cross section, it is assumed that left extent corresponds to the first point in the table.
- 'Left abutment': defines the horizontal position of the bottom of the left wall of the opening.
- 'Right abutment': defines the horizontal position of the bottom of the right wall of the opening.
- 'Right extent': defines the right extent of the active part of the cross section used for the calculation. All points of the cross section located to the right of this marker are ignored. If this marker is not defined in the cross section, it is assumed that right extent corresponds to the last point in the table.

When markers 'Left abutment' and 'Right abutment' are defined in the cross section, it is assumed that the part of the cross section between these two markers corresponds to the natural river bed, and in this case the resistance applied between these markers is obtained from the upstream and downstream cross sections. If these markers are not defined, the resistance value specified in the General tab will be uniformly applied along the bridge's cross section.

The '+' button above the table can be used to insert a new line at the bottom of the table, while the '-' button can be used to delete the active line.

Datum. A datum value may be entered here. The datum is added to all Z values in the table. The datum is normally used for adjusting the levels of the cross sections such that they conform to a specific reference datum in the model area.

Copy upstr. section. The button 'Copy upstr. section' fills in the table, by copying the S, Z and resistance values from the first cross section located upstream of the bridge. This allows to easily obtain the geometry of the upstream cross section, and manually edit this cross section afterwards to describe the geometry of the bridge.

Use default upstream sub-cross section. When the main structure contains multiple openings, the entrance loss upstream of each opening is computed using a sub-part of the upstream cross section (defined in the Cross sections page), and not using the full width of the upstream cross section. The upstream sub-cross section represents the part of the cross section where the water flows towards the active opening.

When the option 'Use default upstream sub-cross section' is checked, the upstream sub-cross section is defined automatically, based on geometrical considerations only.

When the option is unchecked, the position of the upstream sub-cross is manually defined between 'From S value' and 'To S value'.

From S value. This value represents the left horizontal extent of the upstream sub-cross section. This S value refers to the S value specified in the raw data table of the upstream cross section, defined in the Cross sections page.

To S value. This value represents the right horizontal extent of the upstream sub-cross section. This S value refers to the S value specified in the raw data table of the upstream cross section, defined in the Cross sections page.

Downstream section, free surface flow

The downstream cross section of the opening may either be defined as a copy of the upstream cross section of the bridge, in which case elevations are corrected using a longitudinal slope, or defined with its own table.

Cross section. This controls the way the downstream cross section of the bridge is defined:

- Same as upstream: the downstream cross section of the bridge is defined with data from the upstream cross section, but with a correction of elevations to take the longitudinal slope into account.
- User-defined section: the downstream cross section of the bridge is defined in its own table. Please refer to the description of the table used for the upstream cross section for more details.



Slope. The slope between the upstream and the downstream cross sections of the bridge.

Datum. A datum value may be entered here when the downstream cross section of the bridge is defined in its own table. The datum is added to all Z values in this table. The datum is normally used for adjusting the levels of the cross sections such that they conform to a specific reference datum in the model area.

Copy downstr. section. The button 'Copy downstr. section' fills in the table, by copying the S, Z and resistance values from the first cross section located downstream of the bridge. This allows to easily obtain the geometry of the downstream cross section, and manually edit this cross section afterwards to describe the geometry of the bridge.

Use default downstream sub-cross section. When the main structure contains multiple openings, the exit loss downstream of each opening is computed using a sub-part of the downstream cross section (defined in the Cross sections page), and not using the full width of the downstream cross section. The downstream sub-cross section represents the part of the cross section where the water flows from the active opening.

When the option 'Use default downstream sub-cross section' is checked, the downstream sub-cross section is defined automatically, based on geometrical considerations only.

When the option is unchecked, the position of the downstream sub-cross is manually defined between 'From S value' and 'To S value'.

From S value. This value represents the left horizontal extent of the downstream sub-cross section. This S value refers to the S value specified in the raw data table of the downstream cross section, defined in the Cross sections page.

To S value. This value represents the right horizontal extent of the downstream sub-cross section. This S value refers to the S value specified in the raw data table of the downstream cross section, defined in the Cross sections page.

Graphics

Horizontal offsets. The horizontal position of the structures within the cross section are per default defined by the S values, or centred on the sub-cross sections. This position can be adjusted by use of the horizontal offset. Note that the position is used only to plot/display the structure, but the offset and this horizontal position have no impact on the simulation.

Plot/Refresh. This button opens the Structure window to plot the active opening, and its submergence and overflow levels when submergence and overflow are included. This graphical representation aims at verifying that the bridge's cross sections as well as submergence / overflow levels are valid,



but e.g. the horizontal extent of submergence and overflow may not strictly represent the extents considered in the simulation. The plot also shows the other openings of the main structure with a light blue color.

6.6.5 Pump

General

Under the *General* tab the following types of information may be specified.

Location

ID. String identification of the pump. ID will be used in structure results and hence makes it easier to identify a specific structure if setup contains many structures, or if multiple structures are defined at the same location.


Branch name. The Branch name is automatically registered where the pump is located.

Chainage. Chainage at which the pump is located.

Type. The location type may be defined as Regular, Side Structure or Side Structure with Reservoir. See Structure types definition (*p. 101*) for details.

Pump data

Pump type. The pump type is determined by the way its pump rate or discharge is defined:

- **Fixed discharge:** Pump discharge is fixed and independent of the local water head except for the start/stop control.
- **Tabulated:** Pump discharge is controlled by a specified Q/dH relationship and the water level difference (dH). To add a Q/dH relationship click the  button and a new row will appear in the table. The discharge is determined through interpolated look up in the table. The dH used for the look-up is generally calculated as the difference between downstream and upstream water level. However, if an outlet level has been defined then the effective downstream water level used in the dH calculation is the maximum of Outlet level and the downstream water level.

Note that extrapolation is not allowed. It is therefore recommended to add limit points to the Q-dH -curve. See Figure 6.30.

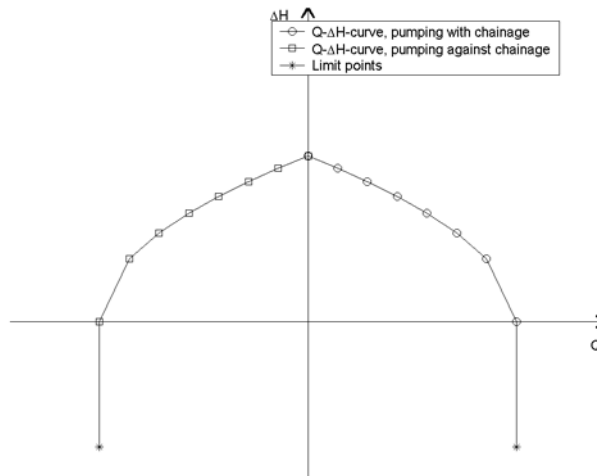


Figure 6.30 Q-dH curves for pumps



Note that a positive discharge Q is always interpreted as the pump pumping from upstream to downstream (i.e. from higher lying areas to a lower lying areas) disregarding any chainage conventions. Typically, this is from smaller chainage to higher chainage. If a negative flow direction is selected for the branch containing the structure, positive discharge is still interpreted as pumping from upstream to downstream areas (which in this situation means pumping from higher chainages to smaller chainages).

Attributes

Start level: The pump will be activated when the upstream water level exceeds the Start level.

Stop level: Water level at which the pump starts closing down.

Startup period: Period for changing pump discharge from zero to full. The pump discharge is changed linearly in time.

Close down period: Period for changing pump discharge from full to zero. The pump discharge is changed linearly in time.

Outlet level: The outlet level is used when calculating the dH value at the location of the pump. If specified with a value >0 the dH will be calculated as the difference of upstream water level and the maximum of downstream water level and the outlet level.



6.6.6 Gates

Gates may be used whenever structure flow is to be regulated by the operation of a movable gate forming part of the structure. The *Gates* menu contains the physical properties of the structures, while the control rules regulating it are defined under the *Control* menu.

General

Under the General tab the following types of information must be specified.

Location

ID. String identification of the gate. ID will be used in structure results and hence makes it easier to identify a specific structure if setup contains many structures, or if multiple structures are defined at the same location.

Branch name. The Branch name of where the gate is located can be chosen from the drop down list.

Chainage. Chainage at which the gate is located.

Type. The location type may be defined as Regular, Side structure or Side structure with storage. See Structure types definition (pg. 101) for details.

Graphics

Horizontal offset from marker 2. The horizontal position of the gate within the cross section can be defined. By default the centre of the gate is positioned at marker 2. Note that the position is used only to plot/display the gate and has no impact on the simulation.

Attribute

Gate Type. The following options are available to define the gate type:

- **Overflow:** This gate type corresponds to a variable crested weir.
- **Underflow:** This gate type corresponds to a vertical sluice gate.
- **Radial gate:** This gate type corresponds to a Tainter gate. In contrast to the other gate types a radial gate does not need any information about head loss factors. Instead a number of radial gate parameters must be entered, see the Radial gate parameters section.
- **Sluice gate:** This gate is physically the same as an underflow gate but instead of using the energy equation a set flow formulas are applied, see the Sluice gate parameters section.

Max speed. Defines the maximum allowable change in gate level per time.

Max speed represents the physical maximum speed of the gate, and may be used to avoid very rapid changes in gate level. Such rapid changes could



appear from some control rules but in case they are not realistic they would potentially create instabilities in the computation.

Initial level. The value specified will be used as initial gate level at the beginning of the simulation.

Contraction coef. This is the contraction coefficient used for underflow gates only. The default value is 0.63.

Geometry

Here, the geometry of the gate is defined. The exact content of the dialogue window depends on the type of the gate.

No. of gates. The number of identical gates is entered here. This variable is used when a series of identical gates are simulated. When multiple gates are simulated, they will always have the same gate level and obey to the same control rules.

Gate width. The width of the gate.

Sill level. The level of the sill just upstream of the gate.

Max level. The highest possible gate level.

Gate height. Parameter specified for radial gate and sluice gate. Height of the overflow crest above sill level when the gate is closed. See Figure 6.20 for radial gate.

Trunnion. Parameter specified for radial gates only. Height of the centre of gate circle above sill level, see Figure 6.20.

Radius. Parameter specified for radial gates only. Radius of gate, see Figure 6.20

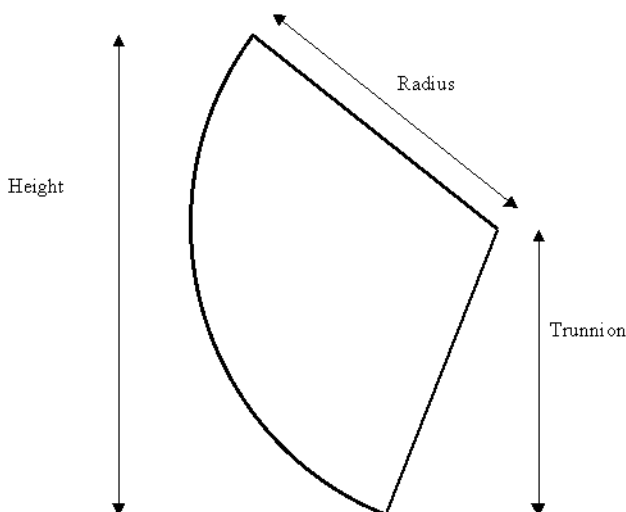


Figure 6.31 Definition of a radial gate

Head loss

Head loss variables are included in the calculation of the energy loss occurring for flows through hydraulic structures. These are specified for overflow and underflow gates and may be distinguished according to the flow direction.

Positive/Negative flow. Head loss factors are defined for Inflow, Outflow and Free overflow. Different loss factors can be applied depending on the flow direction across the structure.

Radial gate parameters

In MIKE 1D, radial gates are automatically divided into an underflow part and an overflow part. When specifying gate levels for a radial gate, the user should specify the level for the underflow part, i.e. the level of the bottom of the gate. The gate level for the overflow part is then calculated based on geometric considerations.

Flow through a radial gate is calculated as a function of the flow regime of the gate (either; 'Free', 'Submerged' or 'Transition'). Free and Submerged flow regimes are calculated individually as described in 'Hydraulic Aspects - Radial Gates' in the MIKE 1D Reference Manual and flow in the transition zone is calculated as an interpolated value between the calculated Free and Submerged flows.

Tune factor. Discharge calibration factor. The Tune factor is applied as a multiplication factor solely on the part of the discharge that flows below the radial gate (not applied to eventual overtopping gate flow).



Weir coeff. Coefficient used in the calculation of eventual flow above the top of the radial gate (considered as weir flow above the top of the radial gate).

Weir exp. Exponent used in the calculation of eventual flow above the top of the radial gate (considered as weir flow above the top of the radial gate).

Tran. depth. The Tran. depth parameter is used to define the depth (or height) of the transition zone between free flow and submerged flow regimes and consequently is used to define the level where the flow regime changes to submerged flow. Corresponds to $y_{Tran,Depth}$ as defined in 'Hydraulic Aspects - Radial Gates' in the MIKE 1D Reference Manual.

Tran. Bottom. The Tran. bottom parameter is an offset used to define the level where a free flow regime for the gate changes to a transition zone between free flow and submerged flow regime. It corresponds to $y_{Tran,Bottom}$ as defined in 'Hydraulic Aspects - Radial Gates' in the MIKE 1D Reference Manual.

Sluice gate parameters

With this gate type the flow under the gate is divided into four flow regimes, the choice of which depends on the upstream and downstream water level. The four flow regimes are:

- – Controlled submerged
- – Controlled free
- – Uncontrolled submerged
- – Uncontrolled free

Additionally flow over the top of the sluice gate is taken into account when the water level up stream and/or downstream exceeds the gate level plus the gate height.

The parameters to specify for the four flow regimes are:

Coef. a. Coefficient a of the flow equation for the four different flow regimes.

Exp. b. Exponent b of the flow equation (it does not apply to the uncontrolled free flow).

High limit and Low limit. High limit and low limit parameters used to smoothen the transitions between flow regimes (they do not apply to the uncontrolled free flow).

Please refer to the chapter 'Hydraulic Aspects - Sluice Gates' in the MIKE 1D Reference Manual for further details on the flow at sluice gates and to see the equations used in the computations.



Flow blockage

Apply flow factor

When this option is active, the discharge computed through the gate is multiplied by a flow factor. This factor's value is specified in the Flow factor field. The factor is a dimensionless factor, and a value of 1 means that no change is applied to the computed discharge. A value lower than 1 can typically be used to describe the reduction of the flow through the structure due to obstacles, like debris, restricting the flow area in the structure.

6.6.7 Direct discharges

A direct discharge structure is a structure which is not represented by its geometry, but instead directly by the discharge flowing through it. This type of structure may for example be used when a regulation is to be applied directly to the discharge, and not to e.g. a gate level.

The *Direct discharge* menu contains some attributes of the structures, while the control rules regulating it are defined under the *Control* menu.

General

Under the General tab the following types of information must be specified.

Location

ID. String identification of the gate. ID will be used in structure results and hence makes it easier to identify a specific structure if setup contains many structures, or if multiple structures are defined at the same location.

Branch name. The Branch name of where the direct discharge is located can be chosen from the drop down list.

Chainage. Chainage at which the direct discharge is located.

Type. The location type may be defined as Regular, Side structure or Side structure with storage. See section Structure types definition (p. 101) for details.

Geometry

No. of structures. Number of identical parallel discharge structures. When multiple structures are included, the control rules define the discharge through a single structure. This discharge is then multiplied by the number of structures.

Attribute

Max. discharge change. This variable defines the maximum allowable change in discharge per time. This variable may be used to avoid very rapid changes of discharge. Such rapid changes may happen with some control rules whereas they wouldn't be realistic and could create instabilities in the computation.



Initial discharge. The value specified will be used as initial discharge at the beginning of the simulation.

Min. discharge. This value is the minimum discharge through the structure.

Max. discharge. This value is the maximum discharge through the structure.

6.6.8 Dambreak

Most dambreak setups consist of a single or several channels, a reservoir, the dam structure and perhaps auxiliary dam structures such as spillways, bottom outlets etc. Further downstream the river may be crossed by bridges, culverts etc. It is important to describe the river setup accurately in order to obtain reasonable results. There is no limit to the number of dambreak structures in a MIKE HYDRO River model.

Due to the highly unsteady nature of dambreak flood propagation, it is advisable that the river topography be described as accurately as possible through the use of as many cross-sections as necessary, particularly where the cross-sections are changing rapidly.

Another consideration is that the cross-sections themselves should extend as far as the highest modelled water level, which will normally be in excess of the highest recorded flood level. If the modelled water level exceeds the highest level in the cross-section for a particular location, MIKE HYDRO River will extrapolate the PROCESSED data using an assumption of 'vertical walls' (keeping the top-width of cross sections when calculating hydrodynamic parameters above top-level of sections).



Note: When performing dambreak simulations, initial conditions should be set up with care, in order to obtain relevant results. Indeed, the discharge through the breach may highly depend on the downstream water level, therefore if the breach fails early in the simulation, the results may be influenced by the initial condition. It is advised to make sure that water levels and discharge throughout the domain, just before the failure, correctly represent the expected conditions. Additionally, initial conditions may create oscillations at the beginning of the simulation, which can unexpectedly trigger the failure of the dam in case the failure moment is controlled by the upstream water level. A hotstart initial condition may be used to achieve this.

General

The General tab is used for specifying the main properties of the dam and its break. The page consists of a number of dialogue boxes whose functionality is described below.



Location

ID. String identification of the structure. It is used to identify the structure if there are multiple structures at the same location. It is recommended always to give the structure an ID.

Branch name. Name of the river branch in which the dambreak is located.

Chainage. Chainage at which the dambreak is located.

Type: The location type may be Regular, Side structure or Side structure with storage. See section Structure types definition (p. 101) for details.

Breach formula

The calculation of the dam break may be carried out using two different formulas. These will also affect the available options for geometry and failure mode. The breach formula available are:

- **Energy equation.**
- **NWS DAMBRK** equation (from US National Weather Service).

Failure mode

The development of the breach can take place in different ways. The failure mode of the structure is affected by the chosen breach formula.

When the energy equation is used to compute the dam break, the failure mode choices are:

- **Time dependent.** The development of the dam breach is specified by the user in terms of breach level, width and slope as functions of time (these are specified in the Breach details tab).
- **Erosion based.** The breach development is calculated by use of a sediment transport formula for which the parameters are specified in the Breach details tab.

If the NWS DAMBRK equation is selected the options are:

- **Breach failure.** The dam break initiates as a breach of the crest.
- **Piping failure.** The dam break initiates as a piping failure, the shape of the pipe being trapezoidal.

Geometry

Crest level before failure. The initial crest level of the dam before failure.

Crest length before failure. The initial crest length (perpendicular to the flow) of the structure before failure.



Additionally, when the Energy equation is selected as breach formula, in combination with the Time dependent failure mode, the following must be specified:

Apply limiting section. If you select this option the development of the breach is limited (e.g. due to solid rock below the dam), otherwise it will be unlimited. The shape of the limitation should be specified in the

Cross section dialogue. When this option is selected, the following parameters should also be specified:

- **Branch name.** Name of the river branch in which the limiting section in the cross section dialogue is located.
- **Topo ID.** Topo ID in which the limiting section in the cross section dialogue is located.
- **Chainage.** Chainage of the limiting section in the cross section dialogue.
- **S-coordinate of breach's centre.** The S-coordinate of the limiting section which is aligned with the breach centreline.

When the Energy equation is selected as breach formula, in combination with the Erosion based failure mode, a limiting breach geometry must be specified using the following parameters:

- **Minimum bottom level.** Minimum bottom level of the trapezoidal shape describing the breach.
- **Maximum bottom width.** Maximum bottom width of the trapezoidal shape describing the breach.
- **Maximum breach slope.** Maximum transversal slope (horizontal: vertical) on the sides of the trapezoidal shape describing the breach.



Note: The discharge through the dam is continuously computed through two separate structures, respectively describing the crest of the dam and the breach. After the failure, the width of the crest varies in time, and is equal to the *Crest level before failure* minus the instantaneous breach width. For this reason, the breach width cannot become longer than the *Crest level before failure*.

Failure moment

The moment at which the dam failure commences can be defined in the following ways:

- **Hours after start.** The failure is specified to take place at a specified number of hours after the start of the simulation.
- **Date and time.** The failure time is specified as a specific date and time.



- **Reservoir water level.** The failure is specified to take place when the water level in the reservoir (assumed to be the grid point immediately upstream of the dam) exceeds the specified level.
- **Control rule.** This option allows to use a user-defined rule to define the failure moment. This rule is defined in the *Control rules* menu, and may depend on e.g. instantaneous results or external data.

Head Loss Factors

The factors determining the energy loss occurring for flow over/through the hydraulic structure. These input parameters are only required for the energy loss method and must be specified for both Positive and Negative flow.

Inflow, Outflow and Free overflow. Energy loss coefficient for the specific types of flow.

Flow blockage

Apply flow factor.

When this option is active, the discharge computed through the dambreak is multiplied by a flow factor. This factor's value is specified in the Flow factor field. The factor is a dimensionless factor, and a value of 1 means that no change is applied to the computed discharge. A value lower than 1 can typically be used to describe the reduction of the flow through the structure due to obstacles, like debris, restricting the flow area in the structure.

Breach details

This tab contains different parameters depending on the choice made for the breach formula and failure mode.

For the Erosion based failure mode, which can be selected when the Energy equation is used as breach formula, the following dialogues and parameters are available in the Breach details tab.

For this case, the vertical development of the breach is calculated using the Engelund-Hansen sediment transport formula. Breach width is determined from the product of breach depth and the side erosion index specified by the user.

Dam geometry

Up stream slope. Longitudinal slope (horizontal:vertical) of the upstream face of the dam structure.

Down stream slope. Longitudinal slope (horizontal:vertical) of the downstream face of the dam structure.

Top width. The top width of the dam crest, in the longitudinal direction.

Material properties

Grain diameter. Representative grain diameter of the dam core material.



Specific gravity. Relative density of the dam core material. Usually ranging between 2.5 - 2.7.

Porosity. Porosity of the dam core material. 0.3 - 0.5.

Critical shear stress. Critical shear stress of dam core material used for sediment transport estimation (Shields criteria). Usually ranging between 0.03 - 0.06.

Side erosion index. Multiplication factor used to calculate breach width erosion rates from breach depth predictions.

Initial failure

Type of initial failure. The failure of the dam can initially take place in two ways:

- **Breach failure:** a breach starting at the crest of the dam
- **Piping failure:** a piping failure through the dam.

For the Breach failure type, the additional parameters below must be specified:

- **Initial level after failure.** The bottom level of the trapezoidal breach at the first time step after the failure moment.
- **Initial width after failure.** The bottom width of the trapezoidal breach at the first time step after the failure moment.

For the Piping failure type, the additional parameters below must be specified:

- **Starting level.** The pipe's centre level at which piping failure begins to occur.
- **Initial diameter.** The diameter of the piping breach at the first time step after the failure moment.
- **Roughness.** Pipe roughness used to calculate the Darcy friction factor.
- **Collapse ratio (D/y).** When the ratio between the diameter of the pipe (D) and the distance from the top of the dam to the top of the pipe (y) is larger than the collapse ratio, the pipe collapses. After it has collapsed, the breach is modelled using a trapezoidal breach.
- **Volume loss ratio.** When the dam collapses some of the material may be carried out without depositing on the bed of the breach. The volume loss ratio is the fraction of the material to be washed out immediately after collapse. It ranges in the interval 0 – 1.

- **Calibration coefficient.** Calibration multiplication factor used to adjust the calculated change in pipe radius.

In all the other cases, i.e. when the Energy equation is used in the breach formula and the failure mode is Time dependent, as well as when the NWS DAMBRK equation is selected as breach formula, only the parameters of the temporal description of the dam geometry must be specified.

Temporal description of geometry

When using the NWS DAMBRK equation, or the Energy equation with the Time dependent failure mode, the breach geometry is defined with the following parameters:

- **Breach bottom level.** The bottom level of the trapezoidal shape of the breach.
- **Breach bottom width.** The bottom width of the trapezoidal shape of the breach.
- **Transversal slope.** The transversal slope (horizontal:vertical) on both sides of the breach.
- **Pipe top level.** Upper level of the pipe. This option is available only if NWS piping failure is selected.

All these geometry features can be defined either as constant values or as time varying ones. For the latter it is necessary to select a dfs0 file, and select the appropriate item if the file contains more than one item.

Dambreak reservoir description and auxiliary structures

To obtain an accurate description of the Dam reservoir storage characteristics, the reservoir can be modelled either as a single h-point in the model or as a branch with a number of cross sections correctly defining the topography and hence, volume characteristics along the reservoir.

In case the reservoir is represented by only one h-point, this will also corresponds to the upstream boundary of the model where inflow hydrographs are specified.

The description of the reservoir storage in this case is defined in the processed data of the cross section in this point. The columns relevant to consider for defining a level storage relation of the reservoir are water level and the additional flooded area columns.

Thereby, the surface storage area of the dam is described as a function of the water level. The lowest water level should be somewhere below the final breach elevation of the dam, and should be associated with some finite flooded area. (This first value, hence, describes a type of 'slot' in the reservoir).



The cross-sectional area is set to a large finite value. It is only used when calculating the inflow headloss into the breach.

It may be practical to locate the dambreak structure on a separate branch containing only three calculation points, as shown in Figure 6.32.

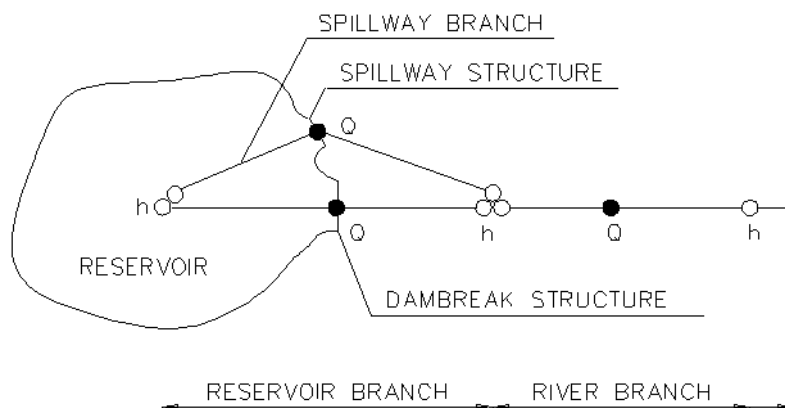


Figure 6.32 Example of Dambreak simulation setup schematisation

Spillways and other structures

If a spillway is added to the dam itself, it could be described as a separate branch, see Figure 6.32.

At the node where the two branches meet, the surface flooded area is taken as the sum of the individual flooded areas specified at each h-point.

Hence, if the reservoir storage has already been described in the reservoir h-point, the spillway h-point should contain no additional surface areas. In this case both the width and the additional flooded areas should be set to zero. The cross-sectional area, hydraulic radii, etc. can be given as for the reservoir branch.

It is not a requirement that a separate branch for the spillway structure is defined. The dambreak and the spill way structure can be located in the same grid point, i.e. as a composite structure. The advantage of having two separate branches is that the discharge through the spillway and the dambreak structure is given as two separate time series in the result file.

6.6.9 Energy losses

General

The Energy losses property page is used to define energy losses associated with local flow obstructions such as sudden flow contractions or expansions



and gradual or abrupt changes in the river alignment. Moreover, a user defined energy loss coefficient can be defined.

Location

ID. String identification of the energy loss point. ID will be used in structure results and hence makes it easier to identify a specific structure if setup contains many structures, or if multiple structures are defined at the same location.

Branch name. The Branch name is automatically registered where the energy loss point is located.

Chainage. Chainage at which the energy loss point is located.

Energy losses

At each specified energy loss point a discharge grid point is inserted at runtime. At each time step of the computation the discharge at energy loss points is computed by use of the energy equation:

$$\Delta H = \zeta \frac{Q^2}{2gA^2} \quad (6.7)$$

in which ΔH is the energy loss, g is the acceleration of gravity, Q is the discharge, A is the cross-sectional wetted area and ζ denotes the energy loss coefficient.

Five different types of energy losses may be included in the simulation:

- User defined loss
- Contraction loss
- Expansion loss
- Abrupt change in alignment
- Gradual change in alignment

For user defined loss, contraction loss and expansion loss an energy loss coefficient must be defined for both positive and negative flow.

For alignment change, the angular change in river alignment at the energy loss point must be specified. Additionally, for gradual change the roughness coefficient must be defined. The roughness coefficient is of the order of 0.2 for rough pipes and of the order of 0.1 for smooth pipes.



6.6.10 Tabulated

General

The Tabulated Structure property page is used for defining a structure regulated by a user defined relationship between the discharge through the structure and the up- and downstream water level (Q/dh).

Location

ID. String identification of the structure. ID will be used in structure results and hence makes it easier to identify a specific structure if setup contains many structures, or if multiple structures are defined at the same location.

Branch name. The Branch name is automatically registered where the structure is located.

Chainage. Chainage at which the structure is located.

Type. The location type may be defined as Regular, Side Structure or Side Structure with Reservoir. See Structure types definition (p. 101) for details.

Calculation mode

Q as a function of h (upstream) and h (downstream): The discharge is given as a function of the up- and downstream water level. The upstream water level 'h(upstream)' must be tabulated in the first column and the downstream water level 'h(downstream)' must be tabulated in the first row in the table. Then the corresponding discharges must be tabulated.

The upstream water level must increase in the downward direction and the downstream water level must increase in the right direction. The discharge cannot increase in the right direction and it cannot decrease in the downward direction.

h (upstream) as a function of h (downstream) and Q: The upstream water level is given as a function of the discharge and downstream water level. The downstream water level 'h(downstream)' must be tabulated in first column and the discharge must be tabulated in the first row in the table. Then the corresponding upstream water levels must be tabulated.

The discharge must increase in the right direction and the downstream water level must increase in the downward direction. The upstream water level must increase in the right and the downward direction.

h (downstream) as a function of h (upstream) and Q: The downstream water level is given as a function of the discharge and upstream water level. The upstream water level 'h(upstream)' must be tabulated in the first column and the discharge must be tabulated in the first row in the table. Then the corresponding downstream water levels must be tabulated.

The discharge must increase in right direction and the upstream water level must increase in the downward direction. The downstream water level must decrease in the right direction and increase in the downward direction.

Tabulated data

No. of rows: Set number of rows in the table. The number of rows must be 4 or higher. A large number of rows will increase the accuracy and the stability of the results.

No. of columns: Set number of columns in the table. The number of columns must be 4 or higher. A large number of columns will increase the accuracy and the stability of the results.

Discharge factor: The discharge factor is multiplied to the discharge in the table.

Water level datum: The water level datum is added to the up- and downstream water level in the table.

6.7 Control

The control options may be used to define the regulating rules that apply to regulated structures. These regulated structures are:

- Gates
- Direct discharges
- Dambreaks
- Pumps

Regulations are defined with sensors, control rules and eventually PID parameters.

6.7.1 Sensors

A sensor is a time-varying variable which is used as a parameter for a control rule. A sensor may be a result of the ongoing simulation, or an external item read in a time series (e.g. observed data).

Sensors are added or deleted using the Append '+' or Delete '-' buttons above the overview table. A sensor cannot be added on the map.

Each sensor is identified using the data below.

ID. String identification of the sensor. It is used to identify the sensor if there are multiple sensors at the same location.



Type. Type of variable. The type 'External' may be used to read the variable in an external time series. When the sensor is a result of the simulation, its type can be:

- Water level
- Discharge
- Velocity
- Water depth
- Area
- Volume
- Structure level
- Concentration (only relevant for advection-dispersion and water quality simulations)

The sensor's location is defined using the data below:

Type. Six types are available:

- **Branch:** for this location type, the location is defined with a branch name and chainage
- **Structure:** for this location type, the location is defined at the location of a selected structure. This location type only applies to Discharge type of sensor.
- **Upstream structure:** for this location type, the sensor is automatically located at the first calculation point upstream of a structure
- **Downstream structure:** for this location type, the sensor is automatically located at the first calculation point downstream of a structure
- **Gridpoint:** this type is only available for the Volume type of sensor, and indicates that the volume is to be considered in a single calculation point.
- **Storage:** for this location type, the location is defined by a storage. This location type only applies to Water level type of sensor.

Branch name. Name of the river branch in which the sensor is located, when the location type is Branch.

Chainage. Chainage at which the sensor is located, when the location type is Branch. When the sensor type is Volume, both an upstream and a downstream chainage have to be specified; these two chainages define the reach on which the total volume is considered.

Structure type. Identifies the type of structure, when the location type is either 'Structure', 'Upstream structure' or 'Downstream structure'.

Structure. ID of the structure, when the location type is either 'Structure', 'Upstream structure' or 'Downstream structure'.

Component. Component to be considered, when the sensor type is 'Concentration' (only relevant for advection-dispersion and water quality simulations).

Time series file. The time series containing the external data, when the sensor type is External. The button to the right may be used to either browse, create, edit or plot the time series.

Time series item. This field shows the name of the item selected in the time series.

Storage ID. Name of the storage in which the water level sensor is located, when the location type is Storage.

6.7.2 Variables

The *Variables* page allows defining any user-defined variable, which can be used as a parameter for a control rule and which can be saved as an additional result item. A variable will usually be a function of sensors and may also be a function of other variables.

The primary purpose is to define complex control rules using variables: instead of defining a complex condition for a control rule, it is possible to define a complex variable and then call it in a simpler condition for the control rule. The variable's value can be saved in the result file during the simulations, and it is therefore easier to check that the correct behavior of the regulated structure is modelled, by comparing the applied control strategy with the variable.

Variables are added or deleted using the Append '+' or Delete '-' buttons above the overview table. A variable cannot be added on the map.

Each variable is identified using the data below.

ID. String identification of the variable. It is used to identify the variable and eventually call it when defining another variable.

Description. Any text description relevant for the user may be supplied here.

Expression. The expression defining the mathematical expression of the variable must be typed in this field. The expression can be typed manually or using the Edit button. This button opens a Edit variable window with a list of available sensors, variables, functions and operators, which can be added to the expression by double-clicking them. The Edit variable window also contains a Validate button which evaluates whether the syntax of the expression is correct or not. A full list of functions available can be found in A.1 Control Functions (p. 323).



Example: An example of a syntax for a variable deriving the energy through a structure may be as provided below:

$9.81 * [dh] * [Q] * [efficiency]$

Where [Q] is a sensor returning the discharge value through the structure and [dh] and [efficiency] are other user-defined variables.



Note: The condition must be written with the syntax from C# programming language. Therefore, the few rules below should apply:

- A path to file must be written with double \. For example, if a variable looks up the time series C:\Data\TS.dfs0, the file path in the function must be written C:\\Data\\TS.dfs0.
- The decimal separator must always be a point.



Note: Circular references (i.e. interdependency of variables) are not supported. That is, a first variable can depend on a second one, but in this case it is not allowed that this second variable also depends on the first one.

6.7.3 PID parameters

PID parameters are necessary when a control rule uses a PID action.

Sets of PID parameters are added or deleted using the Append '+' or Delete '-' buttons above the overview table.

For each set of PID parameters, the data below must be provided.

Name. Name of the set of PID parameters. This name will be referred to in the action definition for the control rule.

Integration time, Ti. Corresponds to Ti in equation (6.8).

Integration time, Td. Corresponds to Td in equation (6.8).

Proportionality factor, K. Corresponds to K in equation (6.8).

Weighting factor for time step 1, a1. Corresponds to a1 in equation (6.8).

Weighting factor for time step 2, a2. Corresponds to a2 in equation (6.8).

Weighting factor for time step 3, a3. Corresponds to a3 in equation (6.8).

6.7.4 Control rules

A control rule defines the way a given structure is regulated. For each structure to be regulated, different actions may be performed based on instantaneous values from sensors.



Control rules are added or deleted using the Append '+' or Delete '-' buttons above the overview table.

General

The General tab contains the general characteristics of the control rule:

ID. String identification of the control rule.

Type. Type of structure to be regulated. Note that the Energy loss and Tabulated types of structures cannot be regulated.

Structure ID. ID of the structure to be regulated.

Controllable parameter. This is the item which is to be regulated in the selected structure. The available parameters depend on the selected type of structure:

- **Gate position:** instantaneous level of the gate. Only available for Gate type of structure
- **Minimum level:** minimum level of the gate. Only available for Gate type of structure
- **Maximum level:** maximum level of the gate. Only available for Gate type of structure
- **Flow:** instantaneous discharge through the structure. Only available for Direct discharge and Pump types of structure
- **Minimum flow:** minimum discharge through the structure. Only available for Direct discharge type of structure
- **Maximum flow:** maximum discharge through the structure. Only available for Direct discharge type of structure
- **Start breach:** failure moment of a breach. Only available for Dambreak type of structure
- **Start level:** the pump is activated when the upstream water level exceeds this level. Only available for Pump type of structure
- **Stop level:** water level at which the pump starts closing down. Only available for Pump type of structure.
- **Flow factor:** instantaneous flow factor applied to the discharge through the structure. Only available for Weir, Culvert, Bridge, Gate and Dambreak types of structure. A control rule controlling a flow factor will only be applied during a simulation if the *Apply flow factor* option for the selected structure is ticked, and will be ignored otherwise.



- **Crest level:** instantaneous crest level of the weir. Only available for Weir type of structure, if using the Honma formula. This control will overwrite the value specified in the weir definition page, except for the initial condition (at the first time step).
- **Weir coefficient:** instantaneous weir coefficient of the weir. Only available for Weir type of structure, if using the Honma formula. This control will overwrite the value specified in the weir definition page, except for the initial condition (at the first time step).

Description. Any text description relevant for the user may be supplied here.

Rules

Structures are regulated with an arbitrary number of rules. For each rule, two things must be specified:

1. **A condition:** one or more criteria that must be fulfilled for the rule to be executed, and
2. **An action:** the action performed by the structure when the condition is evaluated to “true”.

Condition

All the rules for a given structure can potentially be evaluated at each time step. They are all evaluated following the specified priority, and the first one for which the condition is verified (or evaluated as “true”) will be applied at the considered time step. Priorities assigned to the rules are therefore of significant importance.

For each regulated structure, all rules are added or deleted using the Append ‘+’ or Delete ‘-’ buttons above the table in the Rules tab. The Move Up and Move Down buttons also located above this table are used to change the priority of the various rules. The first rule in the table is assigned priority 1, the second rule has priority 2, and so on.

The condition which must be fulfilled for the selected rule can be manually typed in this field.

The condition can also be defined using the *Edit* button. This button opens a ‘*Edit condition*’ window with a list of available sensors, variables, functions, operators and connectors, which can be added to the condition by double-clicking them. The *Edit condition* window also contains a ‘New’ button, allowing to create a new sensor from within this menu directly. It also contains a ‘Validate’ button which evaluates whether the syntax of the condition is correct or not. A full list of functions available for the Control module can be found in Appendix A.1 Control Functions (p. 323)).

Example: An example of a syntax for the condition; *[(WL_{Bridge 2} larger than 12m AND Q_{Weir 5} smaller than 3 m³/s) OR Time(Year) is later than 2001]* is provided below:

```
( [WL at bridge 2] > 12 && [Discharge at weir 5] < 3 ) ||
Year (SimTime()) > 2001
```

Where [WL at bridge 2] and [Discharge at weir 5] are user-defined sensors, && is the AND connector and || is the OR connector.

Example: An example of syntax for the condition *Time since last gate level change at gate 1 is smaller than 2 hours* is provided below:

```
TotalHours ( TimeSinceChange ( [GL at gate 1] ) ) < 2
```

Where [GL at gate 1] is a user-defined sensor monitoring the gate's level.



Note: The condition must be written with the syntax from C# programming language. Therefore the few rules below should apply:

- An equality check is written ==. For example, the condition for verifying if the simulation time is in August is: Month(SimulationTime()) == 8
- The AND condition is written &&, whereas OR is written ||. This is shown in the 'Connectors' drop-down list, but the text AND and OR should not be used in the condition.
- A path to file must be written with double \. For example, if a condition looks up the time series C:\Data\TS.dfs0, the file path in the function must be written C:\\Data\\TS.dfs0.
- The decimal separator must always be a point.

Action

The action to be performed, when the related condition is fulfilled, is defined in this section, using the specifications as described below.

Control type. This controls the type of action which is performed, and also controls the eventual additional parameters to be specified.

The following options are available:

- **Unchanged:** the controlled parameter (e.g. the gate position or the flow) remains unchanged, that is equal to its previous value.
- **Direct setting:** the action directly sets the value of the controlled parameter (e.g. Gate level, structure discharge). Available for all structure types except Dambreak.
- **PID:** the value of the controlled parameter is assessed indirectly, by minimizing the deviation from the expected value. Available for all structure types except Dambreak.



- **Natural flow:** the structure is ignored and the flow is computed as in a natural channel. Available for all structure types except Dambreak.
- **Start breach:** the action triggers the breach for the Dambreak structure. Only available for Dambreak structure type.

Direct setting control

When choosing a direct setting control the additional parameters below must be specified.

Value type. Defines how the value of the controlled parameter is expressed. The following options are available:

- **Absolute value:** the specified value in the 'Value' field is an absolute value (e.g. level for a gate position or discharge for a flow).
- **Delta value:** the specified value is a relative change from the previous state.
- **Close:** the structure is closed (e.g. automatic setting of no discharge for a pump, maximum gate level for an overflow gate, etc.) and no value has to be specified.
- **Fully open:** the structure is open and no value has to be specified.
- **Time series:** the time-varying absolute value is specified in a time series.
- **Tabulated:** the absolute value is specified in a table, containing the relationship between an input variable and the desired value (output).
- **Time tabulated:** the absolute value is specified in a table, containing the relationship between an input variable and the desired value (output), where this relationship is a function of time. If the input value obtained during the simulation is smaller than the smallest input value specified in the table, then the column related to the smallest input value is used. If the input value obtained during the simulation is greater than the greatest input value specified in the table, then the column related to the greatest input value is used.

Value. The absolute value or delta value (relative change) to be applied. The value can either be a simple numeric value or derived from an expression. The 'Edit' button may be used to specify an expression. The same syntax as described above for the condition must be used.

Time series. The time series containing the values to be applied.

Table input. Indicates the variable used as input in the table for the 'Tabulated' or 'Time tabulated' type. The 'Edit' button may be used to select e.g. a sensor. The same syntax as described above for the condition must be used.

Scale. The value will be multiplied by this factor, which can be either a simple numeric value or scaling defined from an expression. The same syntax as described above for the condition must be used.

No. of input columns. The number of columns in the table for the 'Time tabulated' type.

Values specified as yearly variation. This may be used to apply the same yearly pattern to multiple years. When active, values will be interpolated between the first and the last values specified within the year, and between the last and the first values. When inactive, if the time span of the table does not cover the whole simulation period, the first values of the table will be used for earlier dates and times, and the last values of the table will be used for later dates and times.

PID Control

When choosing a PID control the deviation is minimized using the following equation:

$$u^n = a1K \left\{ 1 + \frac{T_s}{T_i} + \frac{T_d}{T_s} \right\} \{ y_{ref}^n - y^n \} - a2K \left\{ 1 + 2\frac{T_d}{T_s} \right\} \{ y_{ref}^{n-1} - y^{n-1} \} - a3K \frac{T_d}{T_s} \{ y_{ref}^{n-2} - y^{n-2} \} - u^{n-1} \quad (6.8)$$

where:

- u^n is the gate level (or discharge in case of a Direct discharge structure) at the n th time step,
- K a factor of proportionality,
- T_i the integration time,
- T_d the derivation time,
- T_s the sampling period, i.e. the simulation time step,
- y_{ref}^n is the required value of the target point at the n th time step,
- y^n the actual value of the target point at the n th time step,
- $a1$, $a2$ and $a3$ are weighing factors. In this way $\{y_{ref} - y\}$ represents a deviation from the desired situation. This deviation is minimized by the PID algorithm in (6.8).
- The variables K , T_d , T_i , $a1$, $a2$ and $a3$ are specified in the *PID parameters* menu.

The additional parameters below must be specified when the Control type is 'PID'.

PID parameters. The set of PID parameters used in equation (6.8) is chosen here. These sets are defined in the *PID parameter* menu.



Setpoint type. This defines how the setpoint value is defined. The following options are available:

- **Absolute value:** the specified value in the 'Setpoint' field is an absolute value
- **Time series:** the time-varying absolute value of the setpoint is specified in a time series
- **Tabulated:** the absolute value is specified in a table, containing the relationship between an input variable defined by the *Table input* and the output which is identical to the desired setpoint

Control input. May also be known as the Process Variable in general PID definitions. The variable indicates the target variable, for which the deviation to the desired value is minimized using the PID equation (6.8). The 'Edit' button may be used to select e.g. a sensor or specify an expression.

Setpoint. The *Setpoint* is the desired (target) value for a variable in the system. Examples are; Reservoir water level, minimum environmental stream-flow, irrigation abstraction requirements etc. *Setpoint* defines the target value for the selected *Setpoint type*. The setpoint can either be a numeric value or an expression. The 'Edit' button may be used to specify an expression.

Setpoint file. The time series containing the values for the setpoint.

Table input. Indicates the variable used as input in the table for the 'Tabulated' type. The 'Edit' button may be used to select e.g. a sensor or specify an expression.

Setpoint scale. The setpoint will be multiplied by this factor. The scale can either be a numeric value or an expression. The 'Edit' button may be used to specify an expression.

6.8 Storages

Storages are points describing any type of storage capacity, which can for example be a reservoir, a lake or a pond.

A storage can be defined anywhere on the map, and is connected to a branch using a storage connection. In the current version, a storage can only be connected to a branch end.



Note: A storage can be connected to multiple branches. A branch can however support only one connection at each end, and it is therefore not possible to connect two storages to the same branch end.

6.8.1 Inserting a storage

A storage can only be added from the map. To insert and/or edit a storage on the map, use the icons available in the *Storages* ribbon in *Map view* (see Figure 6.33).

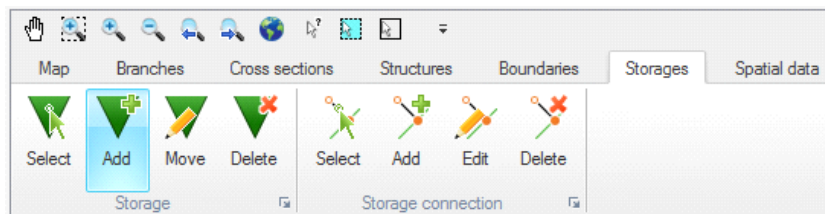


Figure 6.33 Map view ribbon for storages

Select. This button can be used to select one or more storages on the map. To select multiple storages, press the '*Ctrl*' key on the keyboard while selecting the items on the map.

Add. Select the '*Add*' button and click the desired location on the map to create a new storage.

Move. To change the location of a storage, first click the '*Move*' button, then click once on the storage to be moved, and then drag it to the new location. Click once again to stop editing.

Delete. To delete a storage, click this button and click on the storage on the map.

6.8.2 Connecting a storage

A storage can be connected to a branch either on the map or from the *Tabular* tab.

To create a connection from the map, use the buttons available in the *Storages* ribbon, in the '*Storage connection*' group.

Select. This button can be used to select one or more storage connections. To select multiple storage connections, press the '*Ctrl*' key on the keyboard while selecting the items on the map.

Add. Select the '*Add*' button to connect a storage to the end of a branch. After pressing the '*Add*' button, click on the relevant branch end and drag to the storage. It is not possible to start drawing the connection from the storage.



Edit. This button may be used to connect a branch to another storage than the one it is currently connected to. To edit the connection, first click the ‘*Edit*’ button and click once on the connection, then drag the white square above the storage to the new storage to be connected. Click once again to stop editing.

Delete. To delete a storage connection, click this button and click on the storage connection on the map.



Note: Only one connection is allowed for each branch end. If a branch end is connected to another branch or storage, and if a new storage is then connected to this branch end on the map, the branch will be disconnected from the other branch or storage.

To create a connection from the *Tabular* tab, go to the branch definition in the *Branches* menu. In the *Connections* group, set the Connection type to ‘*Storage*’ for either the upstream or downstream end of the branch, depending on where the storage is to be connected. Finally, select the appropriate storage to be connected in the ‘*Storage ID*’ drop-down list.

Connections				
	Connection type	Branch Name	Chainage	Storage ID
Upstream	Storage ▼			SEJERSBK_STC ▼
Downstream	Branch ▼	SEJERSBK-NED	320	▼

Figure 6.34 Tabular view for branch connection to storages

6.8.3 Storage properties

ID. An identifier must be specified for the storage here.

Storage capacity type. The storage capacity type of the reservoir must be defined using the drop-down menu. There are two options:

- **Elevation-area:** Additional storage area is derived directly from the Elevation-Area relationship by using the water level in the reservoir. An Elevation-Area relationship is specified by entering corresponding values of Elevation and Area in the table. Click the Append button to add a row to the table.
- **Elevation-volume:** Additional storage area is calculated by using the Elevation-Storage volume relationship and the following formula:

$$FIA_{i+1} = \left(\frac{\Delta V_{Table}}{\Delta H_{Table}} - FIA_i \right) \cdot 2 \quad (6.9)$$

where:

- FIA: Flooded or additional storage area at level 'i' or 'i+1'
- ΔV_{Table} and ΔH_{Table} are differences in volumes and levels respectively from values defined in the Elevation-Storage volume table.

Initial surface area. The initial surface area must be defined if *Elevation-volume* storage capacity type is selected.

6.9 Decoupled simulation

Advection-Dispersion and Hydrodynamic simulations are two different simulations, but are often performed simultaneously: in that case, the Advection-Dispersion simulation uses the raw results from the Hydrodynamic simulation. The present decoupled option is in that case not used.

It is however possible to run them in a decoupled mode: in that case, only the Advection-Dispersion simulation is performed and it reads the Hydrodynamic data from a result file. In such a decoupled simulation, only the Advection-Dispersion calculations are performed and the simulation time is highly reduced. To apply this mode, enable the 'Decouple Advection-Dispersion and Hydrodynamic simulations' option, and then select the hydrodynamic result file using the Browse '...' button.



Note: The decoupled option is only available when the 'Data' checkmark is ticked and the 'Sim.' checkmark is not ticked for the Hydrodynamic module, in the Modules menu.



Note: When running a decoupled simulation, it is recommended to pay attention to the selected input hydrodynamic result file. Using a regular result file, with instantaneous Water level and Discharge items, there is a risk to introduce mass errors in the Advection-Dispersion simulation if the hydrodynamic result file doesn't store all simulation time steps. To avoid this, the result file should either store all time steps from the Hydrodynamic simulation, or should be a special result files tailored for the decoupled simulation and containing the average discharge for each saved time step. This special result file can be saved during the Hydrodynamic simulation, from the 'Decoupling' menu.

7 Catchments

Catchments are generally included in a model to provide catchment runoff as an inflow to the River network. A model may contain any number of catchments.

Catchments may be represented schematically, or by their delineated boundaries as illustrated in figure below. The primary difference between schematic and delineated catchments in a model setup is, that the catchment surface area is directly derived from the delineation of catchments whereas the surface area must be specified manually for the schematic catchments.



Figure 7.1 Illustration of catchments in a MIKE HYDRO model

The runoff from a catchment is calculated using one of several rainfall runoff models available. The way the runoff from a catchment is added to a river network is defined by the so-called *River links*.

Creating a catchment

There are several ways of creating catchments, each of them being related to a specific *Add* button under the *Catchments* ribbon in *Map view*.

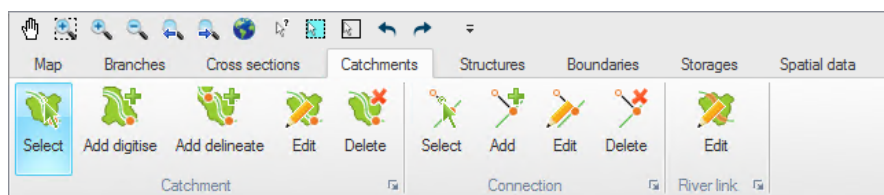


Figure 7.2 Catchment ribbon in Map view

MIKE HYDRO offer multiple options for catchments definitions

- **Delineated catchments** may be created by using a DEM (cf. Section 5.4 (Digital Elevation Model (DEM))). This type of catchment is added by pressing the '*Add delineate*' button and clicking on the branch at the downstream end of a catchment. The catchment is automatically delineated based on the elevations from the DEM, and a catchment node is automatically inserted.
- **Digitized catchments** may be drawn on the map using the '*Add digitize*' button. For this type of catchment, the connection to the branch has to be manually defined with a connection. The catchment connections can be created and edited using the buttons from the *Connection* group in the ribbon.



Note that a catchment created from the map, regardless of its type (delineated or digitized), can have its shape corrected from a shape file using the '*Load shape*' button in the *Tabular* tab. This however does not change the catchment connection.



Note that delineated catchments starting in a local depression (area with local minimum values) can have an unexpected result as an artificial flow pattern in the lake is used for defining the catchment boundary.

If '*Ctrl*'-button is pressed when inserting the catchment outlet the catchment will include both the entire lake and the catchment to the lake.

Editing a catchment

The '*Edit*' button from the *Catchment* group in the ribbon may be used to edit the geometry of a catchment on the map. After selecting this button and clicking on the desired catchment, all the vertices defining the polygon of the catchment become visible.

To move a vertex, click and drag the selected vertex.

To add a new vertex, click on the desired location along the edge of the catchment.

To delete a vertex, double click the selected vertex.



Note that to delete multiple vertices at the same time, zoom out in such a way that these vertices appear almost superimposed. After double clicking, all the vertices in the vicinity of the selected location will be deleted.

Deleting a catchment

To delete a catchment from the map, select the '*Delete*' button in the *Catchment* group in the ribbon and click on the desired catchment.

To delete one or more catchments from the *Tabular* tab, highlight the catchment(s) to be deleted in the overview table and press the '*Delete*' button above the table.



Connecting a catchment

Connections of catchments to branches are controlled by *River links* and *Connections*.

A river link is a reach, that is to say a part of a branch, or a point where the runoff from the catchment is distributed to the river. The spatial extent of the river link therefore controls how the runoff from the rainfall-runoff simulation is used in the hydrodynamic simulation.

A connection is a straight line on the map linking a catchment to a river link. The purpose of this line is to show the connection between the catchment and the river branch on the map. The location of this line has no effect on the simulation: its position along the catchment may therefore be adjusted for presentation purpose but does not affect the simulation.

The following buttons from the *Connection* group in the ribbon may be used to work with connections and river links on the map.

Select. This button can be used to select one or more catchment connections. To select multiple connections, press the '*Ctrl*' key on the keyboard while selecting the items on the map.

Add. Select the '*Add*' button to connect a catchment to a branch. After pressing the '*Add*' button, click on the border of a catchment and drag to the branch. This creates both a connection and a river link.



Note: The created river link has a '*Distributed*' type per default, with its upstream and downstream chainage being equal: it will hence act as a '*Point*' type per default. Once created from the map, this new river link can be further edited from the *Tabular* tab (cf. "River links" on page 196) or using the *Edit* button from the *River link* group in the ribbon.

Edit. This button may be used to move the connection to another location on the catchment. To edit the connection, first click the '*Edit*' button and click once on the connection, then drag the white symbol to its new location. Click once again to stop editing.

Delete. This button may be used to delete both the *Connection* and the *River link*.

The following button from the *River link* group in the ribbon only applies to the river links.

Edit. This button may be used to move the river link along the branch. To edit the connection, first click the '*Edit*' button and click once on the river link location, then drag the white square symbol to its new location. Click once again to stop editing.



Note: When editing a river link with a '*Point*' type, this single point is moved along the branch. When editing a river link with a '*Distributed*' type, only the



upstream or downstream end can be moved at a time, which therefore affects both the location and the length of the river link.

7.1 Catchment definitions

Depending on the selected options, the *Catchment definitions* dialogue may contain the following tabs:

- General
- Rainfall-Runoff

7.1.1 General

Under the *General tab* the following catchment properties are specified:

- Area
- Rainfall-runoff model
- Calibration plot
- Load shape

Area

An accurate specification of the catchment area is required in order to calculate the runoff.

Catchment areas can either be assigned manually, or calculated automatically based on the catchment shapes on the map by enabling '*Use catchment shape for area calculation*'. By default the calculated catchment areas are used. To manually assign catchment area, the option '*Use catchment shape for area calculation*' must be disabled.

Rainfall-runoff model

The following types of rainfall-runoff models are offered:

- NAM
- UHM
- Time-Area
- Kinematic wave

Calibration plot

It is possible to save the calibration plot by enabling '*Calibration plot*'. The calibration plot shows observed and simulated runoff for the catchment, cf. Section 7.1.2.



Load shape

It is possible to import/load the shape of a catchment directly. Click '*Load shape*', select the relevant shape file and choose the shape item that represents the catchment.



Note: this functionality will only edit the shape of the active catchment on the map. It will not create any new catchment. Instead, it is possible to import and create one or multiple catchments at once from a shape file. To achieve this, please see section Import from Shapefile.

7.1.2 Rainfall-Runoff

Rainfall-runoff modelling is a first stage of the simulation that creates the runoff time series for the individual catchments. The simulation of the flow in the river branches is performed at a second stage.

- There are several rainfall-runoff models available:
- **NAM.** A lumped, conceptual rainfall-runoff model, simulating the over-land flow, interflow, and baseflow components as a function of the moisture contents in four storages.
- **UHM.** The Unit Hydrograph Model includes different loss models (constant, proportional) and the SCS method for estimating storm runoff.
- **Time-area.** A conceptual model where the runoff amount is controlled by an initial and a continuous hydrological losses, and where the shape of the hydrograph is controlled by a concentration time and a time-area curve.
- **Kinematic wave.** A non-linear model based on the kinematic wave approach. The surface runoff is computed as flow in an open channel describing the catchment, whereas infiltration is calculated using a modified Horton equation.

7.1.3 NAM

The NAM model is a deterministic, lumped and conceptual rainfall-runoff model accounting for the water content in up to four different storages. NAM can be prepared in a number of different modes depending on the requirement. As default, NAM is prepared with nine parameters representing the *surface zone*, *root zone* and the *groundwater* storages.

In addition, NAM contains provision for:

- Extended description of the groundwater component.
- Two different degree day approaches for *snow melt*.



The NAM Rainfall-runoff component is accessed once the NAM model is selected under the *General* tab.

More details on the NAM model can be found in the 'MIKE 1D Reference' manual.

In the NAM model, the following sections can be found:

- Surface-rootzone
- Groundwater
- Snow melt
- Elevation zones
- Irrigation
- Initial conditions
- Autocalibration
- Seasonal variation
- Time series

The parameters of each section are specified for each representative catchment.

The NAM Rainfall-runoff simulation covers the period as specified in the *Simulation period* dialogue ("Simulation Period" on page 56).

Surface-rootzone

Storages

- **Maximum water content in surface storage (Umax).** It represents the cumulative total water content of the interception storage (on vegetation), surface depression storage and storage in the uppermost layers (a few cm) of the soil. Typical values are between 10-20 mm.
- **Maximum water content in root zone storage (Lmax).** It represents the maximum soil moisture content in the root zone, which is available for transpiration by vegetation. Typical values are between 50-300 mm.

Runoff parameters

- **Overland flow runoff coefficient (CQOF).** It determines the division of excess rainfall between overland flow and infiltration. Values range between 0.0 and 1.0
- **Time constant for routing interflow (CKIF).** It determines the amount of interflow, which decreases with larger time constants. Values in the range of 500-1000 hours are common.



- **Time constants for routing overland flow (CK1, CK2).** They determine the shape of hydrograph peaks. The routing takes place through two linear reservoirs (serially connected) with different time constants, expressed in [hours]. High, sharp peaks are simulated with small time constants, whereas low peaks, at a later time, are simulated with large values of these parameters. Values in the range of 3-48 hours are common.
- **Root zone threshold value for overland flow (TOF).** It determines the relative value of the moisture content in the root zone (L/L_{max}) above which overland flow is generated. The main impact of TOF is seen at the beginning of a wet season, where an increase of the parameter value will delay the start of runoff as overland flow. Threshold value range between 0 and 70% of L_{max} , and the maximum value allowed is 0.99.
- **Root zone threshold value for interflow (TIF).** It determines the relative value of the moisture content in the root zone (L/L_{max}) above which interflow is generated.

Groundwater

For most NAM applications only the *Time constant for routing baseflow* (CKBF) and possibly the *Root zone threshold value for groundwater recharge* (TG) need to be specified and calibrated. However, to cover also a range of special cases, such as groundwater storages influenced by river level variations, a number of additional parameters can be modified.

The groundwater parameters are:

- **Root zone threshold value for GW recharge (TG).** It determines the relative value of the moisture content in the root zone (L/L_{max}) above which groundwater (GW) recharge is generated. The main impact of increasing TG is less recharge to the groundwater storage. Threshold values range between 0 and 70% of L_{max} and the maximum value allowed is 0.99.
- **Time constant for routing baseflow (CKBF).** It can be determined from the hydrograph recession in dry periods. In rare cases, the shape of the measured recession changes to a slower recession after some time. To simulate this, a second groundwater reservoir may be included, see the extended components below.
- **Ratio of GW-area to catchment area (Carea).** This parameter describes the ratio of the groundwater catchment area to the topographical surface water catchment area, which is specified under the General tab. Local geological condition may cause part of the infiltrating water to drain to another catchment. This loss of water is described by a Carea less than one. Usual value: 1.0.



- **Specific yield of groundwater reservoir (Sy).** This parameter should be kept at the default value except for the special cases in which the groundwater level is used for NAM calibration. This may be required in riparian areas, for example, where the outflow of groundwater strongly influences the seasonal variation of the levels in the surrounding rivers. Simulation of groundwater level variation requires values of the specific yield Sy and of the groundwater outflow level GWLBFO, which may vary in time. The value of Sy depends on the soil type and may often be assessed from hydro-geological data, e.g. test pumping. Typically values of 0.01-0.10 for clay and 0.10-0.30 for sand are used.
- **Maximum GW-depth causing baseflow (GWLBFO).** It represents the distance in meters between the average catchment surface level and the minimum water level in the river. This parameter should be kept at the default value except for the special cases in which the groundwater level is used for NAM calibration, cf. Sy above.
- **Seasonal variation of GW-depth causing baseflow.** In low-lying catchments the annual variation of the maximum groundwater depth may be of importance. This variation relative to the difference between the maximum and minimum groundwater depth can be entered if the checkbox is ticked. The monthly values are given relative to the difference between GWLBFO and GWLBF_min [0-1]. This is done in the *Seasonal variation* page, in the column *Variation of groundwater maximum water depth*.
- **Minimum GW-depth for seasonal variation (GWLBF_min).** If the Seasonal variation of GW-depth causing baseflow is selected, the minimum GW-depth level [m] has also to be entered for the calculations.
- **Capillary flux, depth for unit flux (GWLBFI).** It is defined as the depth of the groundwater table generating an upward capillary flux of 1 mm/day when the upper soil layers are dry corresponding to wilting point. The effect of capillary flux is negligible for most NAM applications. Keep the default value of 0.0 to disregard capillary flux.
- **Use abstraction.** If this option is selected, it is possible to specify the groundwater abstraction depth from the catchment.

The input type of this variable can be:

- **Time series.** If this option is selected, a time series of abstraction depth has to be entered in the *Time series* page.
- **Seasonal variation.** This option allows to specify monthly values of abstraction depth in the *Seasonal variation* page.



- **Use lower baseflow, recharge to groundwater (Cqlow).** The groundwater recession is sometimes best described using two linear reservoirs, with the lower usually having a larger time constant. In such cases, the recharge to the lower groundwater is specified here as a percentage of the total recharge.
- **Time constant for routing lower baseflow (Cklow).** It is specified for $Cqlow > 0$ as a baseflow time constant, which is usually larger than the CKBF.

Snow melt

The snow module simulates the accumulation and melting of snow in a NAM catchment. It is included in the model when the '*Include snow melt*' checkbox is ticked.

Two degree day approaches can be applied: a simple lumped calculation or a more advanced distributed approach. The simple degree-day approach uses only two overall parameters: a *Constant Degree day coefficient* and a *Base temperature (snow/rain)*. The distributed approach allows the user to specify a number of elevation zones within a catchment with separate snow melt parameters, temperature and precipitation input for each zone.

The snow melt module uses a temperature input time series, usually mean daily temperature. This has to be specified in the *Time series* tab.

The snow melt parameters are:

- **Constant Degree day coefficient (Csnow).** The content of the snow storage melts at a rate defined by the degree-day coefficient multiplied by the temperature difference above the *Base temperature*. Typical values for Csnow are 2-4 mm/day/C.
- **Base temperature (snow/rain) (To).** The precipitation is retained in the snow storage only if the temperature is below the base temperature, whereas it is by-passed to the surface storage (U) in situations with higher temperatures. The base temperature is usually at or near zero degree C.

It is also possible to specify other input parameters for the snow melt module, if the relative checkboxes are ticked. These are:

- **Variation of degree day coefficient.** This option is selected when the melting coefficient is varying in time, instead of being constant. The input for this model can be specified in two ways:
 - **Specify as time series file.** In this case the melting coefficient is specified in a time series that has to be loaded in the *Time series* section.



- **Specify as seasonal variation.** If this option is selected, monthly values of the degree day coefficient for snow melt must be specified in the *Seasonal variation* section.
- **Radiation coefficient (Radiation file on time series page).** This option may be introduced when time series data for incoming radiation is available. The time series input file is specified separately on the *Time series* section. The total snow melt is calculated as a contribution from the traditional snow melt approach based on C_{snow} (representing the convective term) plus a term based on the radiation.
- **Rainfall degree day coefficient.** This option may be introduced when the melting effect from the advective heat transferred to the snow pack by rainfall is significant. This effect is represented in the snow module as a linear function of the precipitation multiplied by the rainfall degree coefficient and the temperature deviation above the base temperature.

Elevation zones

The *Elevation zones* tab becomes available when snow melt is included in the model. When the '*Delineate catchment into elevation zones for snow modelling*' option is selected, the snow melt distributed approach is applied. Following this approach, the catchment is divided into a certain number of elevation zones, each with specific snow melt parameters, temperature and precipitation inputs.

The general parameters that have to be specified for the snow melt model are:

- **Number of elevation zones**
- **Reference level for temperature station.** This parameter defines the altitude [m] of the reference temperature station. This station is used as a reference for calculating the temperature and precipitation within each elevation zone (the file with temperature data is specified on the Time series page).
- **Dry temperature lapse rate.** It specifies the vertical gradient [$^{\circ}\text{C}/100\text{ m}$] for adjustment of temperature under dry conditions. The temperature in the actual elevation zone is calculated based on a linear transformation of the temperature from the reference station to the actual zone, the correction being defined as the dry temperature lapse rate multiplied by the difference in elevations between the reference station and the actual zone.



- **Wet temperature lapse rate.** This parameter specifies the vertical gradient [C/100 m] for adjustment of temperature under wet conditions defined as days with precipitation higher than 10 [mm]. The temperature in the actual elevation zone is calculated based on a linear transformation of the temperature from the reference station to the actual zone, the correction being defined as the wet temperature lapse rate multiplied by the difference in elevations between the reference station and the actual zone.
- **Reference level for precipitation station.** This parameter defines the altitude, expressed in [m], at the reference precipitation station (the file with precipitation data is specified on the *Time series* page).
- **Correction of precipitation rate.** This parameter specifies the vertical gradient for adjustment of precipitation and is expressed in [percent/100 m]. The precipitation in the actual elevation zone is calculated based on a linear transformation of the precipitation from the reference station to the actual zone, the correction being defined as correction of precipitation rate [percent/100 m] multiplied by the difference in elevation between the reference station and the actual zone.

The specific parameters for each elevation zone are entered in the table at the bottom of the **page**. These are:

- **Zone.** Zone number ID automatically assigned by the programme.
- **Elevation.** The elevation of each zone is specified in the table as the average elevation of the zone. The elevation must increase from zone (i) to zone (i+1).
- **Area.** The area of each zone is specified in the table. The total area of the elevation zones must equal the area of the catchment.
- **Min storage for full coverage.** This parameter defines the required amount of snow to ensure that the zone area is fully covered with snow. When the water equivalent of the snow pack falls under this value, the area coverage (and the snow melt) will be reduced linearly with the snow storage in the zone.
- **Max storage in zone.** This value defines the upper limit for snow storage in a zone. Snow above this value will be automatically redistributed to the neighbouring lower zone.
- **Max water retained in snow.** It defines the maximum water content in the snow pack of the zone. Generated snow melt is retained in the snow storage as liquid water until the total amount of liquid water exceeds this water retention capacity. When the air temperature is below the base temperature T_0 , the liquid water of the snow re-freezes with rate C_{snow} .



- **Dry temperature correction.** The actual temperature correction for dry conditions to estimate actual temperature for the specific zone.
- **Wet temperature correction.** The actual temperature correction for wet conditions to estimate actual temperature for the specific zone.
- **Precipitation correction.** The relative correction for precipitation, expressed in percent, to estimate the precipitation for the specific zone.

The 'Calculate' button above the table fills the three columns in the table for *Dry temperature correction*, *Wet temperature correction* and *Precipitation correction*, based on the specified rates above.

Irrigation

An irrigation module is included in the rainfall-runoff model when the '*Include irrigation*' box is ticked. The irrigation module is used for adjusting the water balance in the NAM model. The purpose of the NAM irrigation module is that of simulating the runoff and groundwater recharge/baseflow from the irrigated areas, so that NAM can be calibrated for the non-irrigated part of the catchment.

Minor irrigation schemes within a catchment will normally have negligible influence on the catchment hydrology, unless transfer of water over catchment boundaries is involved. Large schemes, however, may significantly affect the runoff and the groundwater recharge through local increases in evapotranspiration and infiltration as well as through operational and field losses.

The irrigation module of NAM may be applied to describe the effect of irrigation on the following aspects:

- **The overall water balance of the catchment.** This will be affected mainly by the increased evapotranspiration and by possible external water sources for irrigation.
- **Local infiltration and groundwater recharge in irrigated areas.**
- **The distribution of catchment runoff amongst different runoff components (overland flow, interflow, baseflow).** This may be influenced by the increased infiltration in irrigated areas as well as by local abstraction of irrigation water from groundwater or streams.

The irrigation parameter is:

- **Infiltration rate at field capacity (K0-inf).** This parameter defines the rate of infiltration at field capacity [mm/h].

The *Irrigation sources in percent* are:



- **Local groundwater (PC_LGW).** Percentage of water for irrigation supplied by groundwater sources.
- **Local river (PC_LR).** Percentage of water used for irrigation supplied by a local river.
- **External river (PC_EXR).** Percentage of water for irrigation supplied by a river external from the catchment defined in NAM.

When part of the irrigation source comes from an external river (PC_EXR > 0), the *Branch name* and *Chainage* fields become active, and are used to specify the location on the river network where the water is taken from.

Another irrigation option is:

- **Include crop coefficients and operational losses.** If this checkbox is ticked, additional irrigation module parameters can be specified as monthly values in the Seasonal variation page.

These are:

- **Irrigation crop coefficient.** This parameter, also defined as Kc, allows to quantify the amount of water required (and transpired) by the crop. Different crops have different crop coefficient. Kc is multiplied by the reference ET of a standard crop (grass) to calculate the water demand of the crop of interest.
- **Irrigation operational and conveyance losses in percent of abstract water.** These represent the system losses of irrigation water for the different components/processes:
Groundwater
Overland flow
Evaporation

Initial conditions

The initial relative water contents of surface and root zone storage must be specified as well as the initial values of overland flow and interflow. Initial values for baseflow must always be specified. When lower baseflow is included, a value for the initial lower baseflow must also be specified.

Initial values of the snow storage are specified when the snow melt routine is used. When the catchments are delineated into elevation zones, the snow storage and the water content in each elevation zone are specified.

The parameters of the different subsections are:

Surface and rootzone

- **Relative water content in surface storage [0, 1] (U/UMax).** This value ranges between zero and one, where one indicates wet initial conditions and zero dry initial conditions.



- **Relative water content in root zone storage [0, 1] (L/LMax).** This value ranges between zero and one.
- **Overland flow (QOF).** The overland flow at the beginning of the simulation, which is normally estimated from the hydrograph [$\text{m}^3 \text{s}^{-1}$].
- **Interflow (QIF).** The interflow at the beginning of the simulation, which is normally based estimated from the hydrograph [$\text{m}^3 \text{s}^{-1}$].

Groundwater

- **Baseflow (BF).** The baseflow at the beginning of the simulation, which is normally estimated from the hydrograph [$\text{m}^3 \text{s}^{-1}$].
- **Lower baseflow (BF - low).** The lower baseflow at the beginning of the simulation, which is normally estimated from the hydrograph [$\text{m}^3 \text{s}^{-1}$].

Snow storage

If the *Elevation zones* option is not selected:

- **Global value.** This parameter represents the actual initial snow storage in [mm] over the entire catchment.

If the *Elevation zones* option is selected:

- **Zone, Snow storage, and Water in snow.** If the snow model is defined using elevation zones, it is required to enter for each zone the initial *Snow storage* in [mm] and the *Water in snow*, also in [mm], defining the water content in the snow pack.

Autocalibration

It is possible to use an automatic optimisation procedure to calibrate the 12 most important parameters in the NAM model. The calibration routine used in NAM is based on a multi-objective optimisation strategy, the SCE algorithm. The procedure implemented in NAM allows to simultaneously optimise four different calibration objectives or a combination of them. A description of the SCE algorithm is given in the '*Autocal User Guide*'.

To include the Autocalibration routine in NAM, tick the '*Include autocalibration*' checkbox.

Calibration parameters

The parameters which can be included in the autocalibration routine are listed in the *Calibration parameters* table:

- Maximum water content in surface storage (Umax).
- Maximum water content in root zone storage (Lmax)
- Overland flow runoff coefficient (CQOF)



- Time constant for interflow (CKIF)
- Time constant 1 for routing overland flow (CK1)
- Time constant 2 for routing overland flow (CK2)
- Root zone threshold value for overland flow (TOF)
- Root zone threshold value for interflow (TIF)
- Root zone threshold value for groundwater recharge (TG)
- Time constant for routing baseflow (CKBF)
- Lower baseflow, recharge to lower reservoir (Cqlow)
- Time constant for routing lower baseflow (Cklow).

If the '*Fit*' checkbox is ticked, the parameter is included in the autocalibration. The table also contains the following columns:

- **Initial value.** It is the value for the parameter specified in the Surface-rootzone or Groundwater page and used in the first model simulation.
- **Lower Bound** and **Upper Bound** are the minimum and maximum values that the parameter can assume and therefore define its range of variation.

Objective functions

In automatic calibration, the calibration objectives have to be formulated as numerical goodness-of-fit measures that are optimised.

Four calibration objectives are defined as numerical performance measures in the Autocalibration routine of NAM. These are selected and used by ticking the relative checkboxes.

The objective functions are:

- **Overall water balance.** This defines the agreement between the average simulated and observed catchment runoff overall volume error.
- **Overall root mean squared error.** This measure defines the overall agreement of the shape of the simulated hydrograph with the observed one.
- **Peak flow RMSE.** This optimisation measure defines the agreement of simulated and observed peak flows events. If this measure is selected, the minimum river discharge value above which the flow is defined as peak flow has to be specified in the *Peak flow* field.
- **Low flow RMSE.** This optimisation measure defines the agreement of simulated and observed low flows events. If this measure is selected, the maximum river discharge value below which the flow is defined as low flow has to be specified in the *Low flow* field.

Other input parameters are:

- **Maximum number of evaluations.** Here one of the stopping criteria of the auto calibration routine is specified. The automatic calibration will stop either when the optimisation algorithm ceases to give an improvement in the calibration objective or when the maximum number of model evaluations is reached.
- **Initial number of day excluded.** This is a warm up period that will be disregarded when calculating the objecting function.

Running the autocalibration

After having entered the autocalibration parameters, the autocalibration procedure starts when launching a normal simulation. When the optimisation routine is completed, the list of the parameters generated during the optimisation and the relative objective functions are made available in the *CatchmentName-Autocal.txt* file. If the Calibration plot option has been selected in the *General* tab, a *CatchmentName.plc* file is also produced. This file shows the differences between the observed and the best simulated runoff data, and can be opened in the Plot Composer.



Note: The various NAM parameters shown in the *Rainfall-runoff* tab will not be automatically updated with the optimal parameter set. To visualize the optimized parameters, either close (without saving) the MIKE HYDRO file and re-open it, or check the created text file *CatchmentName-Autocal.txt*.

Seasonal variation

In this section it is possible to enter the values of some selected time varying parameters. Monthly values have to be assigned to the parameters. Whether a parameter has a constant or a seasonal varying value is determined in the different sections of the NAM model.

It is possible to specify a seasonal variation for the parameters:

- **Variation of groundwater maximum water depth.** Seasonal variation of GW-depth causing baseflow. From *Groundwater* tab.
- **Abstraction.** Groundwater abstraction depth. From *Groundwater* tab.
- **Degree day coefficient for snow melt.** Melting coefficient for snow melt. From *Snow melt* tab.
- **Irrigation crop coefficient.** Parameter to quantify the amount of water required (and transpired) by the crop. From *Irrigation* tab.
- **Irrigation operational and conveyance losses in percent of abstracted water.** System losses of irrigation water for the different components/processes: **Groundwater**, **Overland flow** and **Evaporation**. From *Irrigation* tab.



Time series

In this section the input time series of the rainfall-runoff model are entered. Depending on which module of the model is used, these are:

- **Rainfall.** Always to be entered. A time series, representing the average catchment rainfall. The time interval between values may vary through the input series. The rainfall specified at a given time should be the rainfall volume accumulated since the previous value.
- **Evaporation.** Always to be entered. The potential evaporation is typically given as monthly values. Like rainfall, the time for each potential evaporation value should be the accumulated volume at the end of the period it represents. The monthly potential evaporation in June should be dated 30 June or 1 July.
- **Observed discharge.** This time series has to be entered if the *Autocalibration* module is used or the '*Calibration plot*' option has been selected in the *General* tab. The inclusion of the observed discharge will automatically enable additional outputs which include a calibration plot with comparison of observed and simulated discharge and calculation of statistical values.
- **Temperature.** A time series of temperature, usually mean daily values, is required only if snow melt calculations are included in the simulations.
- **Irrigation.** To be entered when the irrigation module of the rainfall-runoff module is included.
- **Abstraction depth.** To be entered when the 'Use abstraction' option in the Groundwater tab is selected and 'Time series file' is specified.
- **Radiation.** To be entered when the 'Radiation coefficient' option in the *Snow melt* tab is selected.
- **Degree day coefficient.** To be entered when the 'Variation of degree day coefficient' option in the *Snow melt* tab is selected.

Weighted time series may be used for some time series by enabling '*Use weighted time series*'. This adds a new tab 'TS weighted rainfall/evaporation/temperature' to the window (see below).

Weighted time series

To use weighted time series, '*Use weighted time series*' must be enabled under the Time series tab - see above. Under the TS weighted tab the following may be defined:



Allow gap in time series

If 'Allow gap in time series' is enabled, simulation will proceed despite missing data (gap) in the time series. If 'Allow gap in time series' is disabled, simulation will terminate with an error message if missing data (gaps) are encountered in the time series.

Distribution in time

If data is available from stations reporting at different frequencies, e.g. both daily and hourly stations, the *Distribution in time* of the average catchment rainfall may be determined using a weighted average of the high-frequency stations. You may, for example, use all daily and hourly stations to determine the daily mean rainfall over the catchment and subsequently use the hourly stations to distribute (desegregate) this daily rainfall in time.

Weighted average combinations

To specify a weighted average combination, add time series by clicking the Append '+' button above the table to add a new row, then click the browse '...' button to select a time series or create a new file and enter the weight to be assigned to that time series. Repeat the steps for all relevant time series/stations.

The weighted average values $T_{weighted}$ are computed using the following formula:

$$T_{weighted} = \frac{T_0 \cdot w_0 + T_1 \cdot w_1 + \dots + T_n \cdot w_n}{w_0 + w_1 + w_1 + \dots + w_n} \quad (7.1)$$

where T_n is the value from time series n and w_n the corresponding weight.

If data are missing from one or more time series/stations, and 'Allow gap in time series' is enabled, the Mean Area Weighting algorithm will ignore the time series with missing data.

In the present version it is possible to specify only one weighted average combination.

7.1.4 UHM

The UHM (*Unit Hydrograph*) model constitutes an alternative to the NAM model for flood simulation in areas where no streamflow records are available or where unit hydrograph techniques are already well established.

The model includes a number of simple unit hydrograph models to estimate the runoff from single storm events by the use of the well-known unit hydrograph technique. The models divide the storm rainfall in excess rainfall (or runoff) and water loss (or infiltration).



More details on the UHM model can be found in the 'MIKE 1D Reference' manual.

UHM parameters

The following parameters are used for all types of UHM models:

- Adjustment and baseflow
- Hydrograph
- Enlargement and loss model
- Lag time

Adjustment and baseflow

- **Area adjustment factor.** This adjustment factor (different from 1.0) may be applied if the catchment rainfall intensity is assumed to differ from the input rainfall data series by a proportional factor.
- **Baseflow.** If different than zero, the value specified represents a constant baseflow that is added to the runoff.

Hydrograph

The distribution of the runoff in time can be described using different methods.

Three different options can be selected for the hydrograph type:

- **SCS triangular.** This is a standard hydrograph in which the time to peak is assumed to be half the duration of the excess rainfall plus the lag time t_l .
- **SCS dimensionless.** This type of hydrograph has been derived from a large number of natural unit hydrographs from catchments of varying size and location. The flow values are expressed in Q/Q_p , where Q_p is the peak discharge, and the time in T/T_p , where T_p is the time from the start of the hydrograph rise to the peak.
- **User defined.** This hydrograph should be specified in its dimensionless form, i.e. Q/Q_p as a function of T/T_p , as for the SCS dimensionless hydrograph above. But in this case this information has to be provided in the **Hydrograph table (Dimensionless from Q/Q_p as a function of T/T_p). The hydrograph table must be provided in a time series file, where the axis type is either 'Equidistant Relative Axis' or 'Non-Equidistant Relative Axis' and containing one item with the type 'Dimensionless factor'.**

Enlargement and loss model

Four different options are available to represent the loss model, each of them requiring different inputs.



These types of loss models are:

- **Constant loss.** In this case the infiltration is described as an *Initial Loss* at the beginning of the storm followed by a *Constant Loss* term caused by infiltration.
- **Proportional loss.** If this model is selected, a *Runoff coefficient* should be specified as the ratio of runoff to the rainfall.
- **SCS method.** The SCS Loss Model uses a *Curve number* that characterises the catchment in terms of soil type and land use characteristics. The model also requires an *Initial AMC* (antecedent moisture conditions) parameter.
- **SCS generalised.** The SCS generalised loss model does not make use of the concept of an antecedent moisture content (AMC) but applies a storage *Initial abstraction depth*. Like in the SCS method, the *Curve number* should be specified for the SCS generalised.

Lag time

The Lag time type for the UHM model can be specified in two different ways:

- **User specified.** In this case the lag time is specified directly in hours.
- **Curve number method.** In this case the lag time is calculated by the standard SCS formula and will appear in the *Derived Lag Time* box. When this method is selected, three parameters need to be specified: *Hydraulic length*, the *Slope* of the catchment and the *Curve number*.

Time series

In this section the input time series of the rainfall-runoff model are entered. The UHM only requires two input time series:

- **Rainfall.** This time series represents the average catchment rainfall. The time interval between values, may vary through the input series. The rainfall specified at a given time should be the rainfall volume accumulated since the previous value.
- **Evaporation.** The potential evaporation is typically given as monthly values. Like rainfall, the time for each potential evaporation value should be the accumulated volume at the end of the period it represents. The monthly potential evaporation in June should be dated 30 June or 1 July.

Weighted time series may be used by enabling '*Use weighted time series*'. This adds a new tab '*TS weighted rainfall/evaporation*' where time series, their corresponding weights and distribution in time may be defined (see '*Weighted time series*' previous to this section).



7.1.5 Time-area

In the Time-Area (T-A) surface rainfall-runoff method, the runoff is controlled by the initial loss, size of the contributing area and by a continuous hydrological loss. Moreover a snow storage may be added to the computation.

The shape of the runoff hydrograph is controlled by the concentration time and by the time-area (T-A) curve. These two parameters represent a conceptual description of the catchment reaction speed and the catchment shape.

Time-Area

Some of the model specific data for the T-A model are entered in the Time-Area tab.

- **Impervious area.** This parameter represents the fraction [%] of the catchment area considered to contribute to the runoff.
- **Time of concentration.** The time of concentration parameter defines the time required for the flow of water from the most distant part of the catchment to the point of outflow. The default value is 420 seconds.
- **Reduction factor.** This is a hydrological runoff reduction factor accounting for water losses caused by e.g. evapotranspiration, imperfect imperviousness, etc. on the contributing area. The default value is 0.90.
- **Initial losses.** The initial losses define the precipitation depth required to start the surface runoff. This is a one-off loss, comprising the wetting and filling of catchment depressions. The default value is 0.6 mm.
- **Time-area curve.** This curve accounts for the shape of the catchment layout, and determines the choice of the available T-A curve to be used in the computation.

Three predefined types of the T-A curves are available:

1. Rectangular catchment
2. Divergent catchment
3. Convergent catchment

The values of the time and area associated with each curve type can be found in the following table



Table 7.1 Predefined types of T/A curves

Curve type 1		Curve type 2		Curve type 3	
Time [-]	Area [-]	Time [-]	Area [-]	Time [-]	Area [-]
0	0	0	0	0	0
1	1	0.1	0.12	0.1	0.01
		0.2	0.28	0.2	0.04
		0.3	0.47	0.3	0.08
		0.4	0.61	0.4	0.15
		0.5	0.72	0.5	0.24
		0.6	0.8	0.6	0.35
		0.7	0.87	0.7	0.47
		0.8	0.93	0.8	0.62
		0.9	0.98	0.9	0.8
		1	1	1	1

Additional urban parameters

The Additional urban parameter tab allows adding to the catchment runoff extra discharge inputs and outputs, such as a constant baseflow, inflow based on the population living in the area, evaporation losses and snowmelt additional input.

Additional inflows

Constant flow. Constant (base-) flow which is being added to the runoff of the catchment throughout the entire simulation. If more than one constant inflow source is present in the catchment, their contributions should be summed up and entered here.

Load based on inhabitants. Number of person equivalents (PE). The additional inflow is generated only if this field is larger than 0. In that case the inhabitant load time series field becomes active in the Time series dialogue, where you should enter the inflow time series per inhabitant. This will be multiplied by PE to generate the inflow to the system.

Additional rainfall-runoff parameters

- **Include evaporation.** If this checkbox is selected, evaporation is included in the model. The evaporation time series file is specified in the Time series page.
- **Include snow melt.** If this checkbox is selected, snow melt is included in the model. The temperature time series file is specified in the Time series page.



- **Degree-day coefficient.** This value defines the rate of snow melting when temperature exceeds zero degrees Celsius.

Time series

In this section the input time series of the Time-Area rainfall-runoff model are entered. Depending on which processes are included in the model, these are:

- **Rainfall.** This time series represents the average catchment rainfall. The time interval between values may vary through the input series. The rainfall specified at a given time should be the rainfall volume accumulated since the previous value.
- **Evaporation.** The potential evaporation is typically given as monthly values. Like rainfall, the time for each potential evaporation value should be the accumulated volume at the end of the period it represents. The monthly potential evaporation in June should be dated 30 June or 1 July.
- **Temperature.** A time series of temperature, usually mean daily values, is required only if snow melt calculations are included in the simulations.

Weighted time series may be used by enabling 'Use weighted time series'. This adds a new tab 'TS weighted rainfall/evaporation' where time series, their corresponding weights and distribution in time may be defined (see 'Weighted time series' previous in this section).

Runoff computation

Snow storage computation

Snow can accumulate when the temperature is inferior or equal to zero degrees Celsius. During warmer periods, when temperature exceeds zero degrees Celsius, the snow storage melts at a rate given by the degree-day coefficient. The snow storage computation will be performed only if a temperature time series is provided for the catchment.

During storing periods, the snow will accumulate without limit. If the temperature is below zero degrees Celsius, it is assumed that the entire rainfall will be stored as snow.

During melting periods, the snow will melt and therefore contributes to the surface runoff at rate given by the degree-day coefficient.

The snow module is the first process occurring after rainfall, therefore all other losses will be computed afterwards.

Effective precipitation computation

The simulated hydrological processes account for specific losses. The runoff starts after the rain depth has exceeded the specified initial loss for the catch-



ment. The remaining precipitation is called effective precipitation, defined generally as:

$$I_{eff}(t) = \begin{cases} I(t) - I_w(t) & \text{for } I(t) \geq I_w(t) \\ 0 & \text{for } I(t) < I_w(t) \end{cases} \quad (7.2)$$

The actual precipitation, $I(t)$, is assumed to be uniformly distributed over the individual catchments. Otherwise, it may vary as a random time function.

The wetting, $I_w(t)$, is a discontinuous loss, also called initial loss. When the precipitation starts, a part of the precipitation is used for wetting of the surface if the surface is initially dry. When the surface is wet, the wetting loss, I_w , is set to zero. This is summarised in the following expression:

$$I_w(t) = \begin{cases} I(t) & \text{for } y(t) < y_w \\ 0 & \text{for } (I(t) = 0) \text{ OR } (y(t) \geq y_w) \end{cases} \quad (7.3)$$

Surface runoff routing computation

The continuous runoff process is discretised in time by the computational time step Δt . The assumption of the constant runoff velocity implies the spatial discretisation of the catchment surface to a number of cells in a form of concentric circles with a centre point at the point of outflow. The number of cells being equals to:

$$n = \frac{t_c}{\Delta t} \quad (7.4)$$

Where:

t_c : Concentration time [s]
 Δt : simulation time step [s]

The 1D engine calculates the area of each cell on the basis of the specified time-area curve or coefficient. The total area of all cells is equal to the specified impervious area.

A time-area curve characterises the shape of the catchment, relating the flow time i.e. concentric distance from the outflow point and the corresponding catchment sub-area. Irregularly shaped catchments can be more precisely described by the user-specified T-A curves or coefficient.

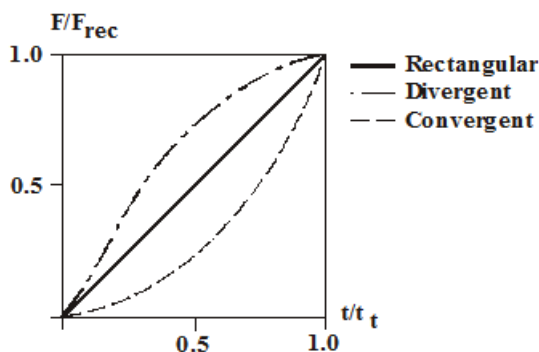
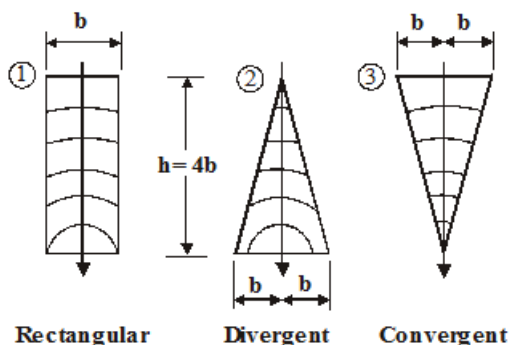


Figure 7.3 The three predefined time/area curves available

The runoff stops when the accumulated rain depth on the whole catchment surface regresses below the specified initial loss for the catchment.

At every time step after the start of the runoff, the accumulated volume from a certain cell is moved to the downstream direction. Thus, the actual volume in the cell is calculated as a continuity balance between the inflow from the upstream cell, the current rainfall (multiplied with the cell area) and the outflow to the downstream cell. The outflow from the most downstream cell is actually the resulting surface runoff hydrograph.

To account for the specified hydrological reduction, the runoff from the impervious surface is reduced by the catchments hydrological reduction factor.

Multiple-event simulations

If the Time-Area runoff model is used for a continuous simulation of multiple rainfall, a special solution can be applied for the simulation of dry periods between the consecutive events. The solution accounts for the loss of water caused by drying out of the initial loss (representing wetting and surface storage), i.e. allowing the occurrence of the initial loss at the beginning of each simulated event, in accordance with reality.



In this context, start of a dry period is defined if two conditions are fulfilled simultaneously:

- No rainfall: all connected rain gauges show no rain.
- No runoff: the runoff has fallen to zero from all catchments included in the simulation.

At the start of a dry period, the initial loss storage would be fully or partially filled up, the latter being the only case for small events of the total depth smaller than the initial loss storage depth. Recovery of the initial loss capacity, i.e. the process of surface drying is simulated as a constant “decay” rate, which replaces the actual evaporation.

The evaporation rate can be given as initial loss rate or as an evaporation time series. If the actual evaporation is not explicitly specified in the simulation (i.e. evaporation process activated and evaporation time series specified), only a decay rate will be applied during dry periods.

The recovery process is only activated during dry periods, i.e. the evaporative action during rain events is neglected.

7.1.6 Kinematic wave

The concept of this surface rainfall-runoff model is founded on the kinematic wave computation. The surface runoff is computed as flow in an open channel, taking into account the gravitational and friction forces only. The amount that runs off is controlled by the various hydrological losses and the size of the actually contributing area.

The shape of the runoff hydrograph is controlled by the catchment parameters length, slope and roughness of the catchment surface. These parameters form a base for the kinematic wave computation using the Manning equation.

Infiltration to groundwater is calculated using a modified Horton equation.

The catchment is divided into five sub-catchments that have different permeability properties of the surface.



The five surface types are:

Table 7.2 Permeability properties of sub-catchment surfaces

Permeability	Surface type
Impervious	Steep Flat
Pervious	Small Medium Large

Only relevant processes are modelled on each surface type. The model applies different hydrological parameters for each of the surface types. The total runoff is computed as the sum of these sub-catchments.

Kinematic wave

This tab contains the parameters defining the geometry and the hydrological characteristics of the catchment in which the Kinematic wave runoff model is implemented.

The parameters defined globally in the catchments are:

Length. The length parameter is defined by the catchment shape, as the flow channel. The model assumes a prismatic flow channel with rectangular cross section. The channel bottom width is computed from catchment area and length. The default value is 10 m.

Slope. It is the average slope of the catchment surface, used for the runoff computation according to Manning equation. The default value is 1.

The following parameters are defined for all the five different types of sub-catchments:

- Impervious area
- Steep
- Flat
- Pervious area
- Low impermeability
- Medium impermeability
- High impermeability



Contributing areas. Percentage that represents the fraction of the catchment surface belonging to each surface types. The default value depends on the surface type (see Table 7.3).

Manning's M. The Manning's number [$m^{1/3} s^{-1}$] describes the roughness of the catchment surface, used in hydraulic routing of the runoff (Manning's formula). The default values depend on the catchment surface category (see Table 7.3).

Wetting loss. This loss [m] accounts for wetting of the catchment surface. The default value for all surface types is $5.00E-5$ m.

Storage loss. This loss [m] defines the precipitation depth required for filling the depressions on the catchment surface prior to occurrence of runoff. It is not defined for Steep impervious areas. The default value depends on the surface type (see Table 7.3)

The parameters of Horton's equations must be defined for Pervious area surface types only:

Horton's infiltration capacity – Maximum. This parameter, also called start infiltration rate, defines the maximum rate of infiltration (Horton) [m/s] for the specific surface type. The default value depends on the surface type (see Table 7.3).

Horton's infiltration capacity – Minimum. This parameter, also known as end infiltration rate defines the minimum rate of infiltration (Horton) [m/s] for the specific surface type. The default value depends on the surface type (see Table 7.3).

Horton's exponent – Wet condition. This time factor “characteristic soil parameter” [s^{-1}] determines the dynamics of the infiltration capacity rate reduction over time during wet period. The actual infiltration capacity is made dependent of time since the rainfall start only. The default value depends on the surface type (see Table 7.3).

Horton's exponent – Dry condition. This time factor [s^{-1}] is used in the “inverse Horton's equation” and it defines the rate of the soil infiltration capac-



ity recovery after a rainfall, i.e. in a drying period. The default value depends on the surface type (see Table 7.3)..

Table 7.3 Default hydrological parameters for Kinematic wave surface runoff model

Default values for hydrological parameters					
Parameter	Impervious		Pervious		
	Steep	Flat	Small	Medium	Large
Area fraction (-)	0	0.4	0	0	0.6
Wetting (m)	5.00E-5	5.00E-5	5.00E-5	5.00E-5	5.00E-5
Storage (m)	-	6.00E-4	1.00E-3	1.00E-3	2.00E-3
Start inf. rate (m/s)	-	-	8.00E-7	8.00E-7	2.00E-5
End inf. rate (m/s)	-	-	8.00E-7	8.00E-7	3.00E-6
Wet exponent (s ⁻¹)	-	-	0	0	1.50E-3
Dry exponent (s ⁻¹)	-	-	0	0	3.00E-5
Manning (m ^{1/3} s ⁻¹)	80	70	30	30	12

Additional urban parameters

The Additional urban parameter tab allows adding to the catchment runoff extra discharge inputs and outputs, such as a constant baseflow, inflow based on the population living in the area, evaporation losses and snowmelt additional input.

Additional inflows

Constant flow. Constant (base-) flow which is being added to the runoff of the catchment throughout the entire simulation. If more than one constant inflow source is present in the catchment, their contributes should be summed up and entered here.

Load base on inhabitants. Number of person equivalents (PE). The additional inflow is generated only if this field is larger than 0. In that case the inhabitant load time series field becomes active in the Time series dialogue, where you should enter the inflow time series per inhabitant. This will be multiplied by PE to generate the inflow to the system.

Additional rainfall-runoff parameters

- **Include evaporation.** If this checkbox is selected, evaporation is included in the model. The evaporation time series file is specified in the Time series page.



- **Include snow melt.** If this checkbox is selected, snow melt is included in the model. The temperature time series file is specified in the Time series page.
- **Degree-day coefficient.** This value defines the rate of snow melting when temperature exceeds zero degrees Celsius.

Time series

In this section the input time series of the Kinematic wave rainfall-runoff model are entered. Depending on which processes are included in the model, these are:

- **Rainfall.** This time series represents the average catchment rainfall. The time interval between values may vary through the input series. The rainfall specified at a given time should be the rainfall volume accumulated since the previous value.
- **Evaporation.** The potential evaporation is typically given as monthly values. Like rainfall, the time for each potential evaporation value should be the accumulated volume at the end of the period it represents. The monthly potential evaporation in June should be dated 30 June or 1 July.
- **Temperature.** A time series of temperature, usually mean daily values, is required only if snow melt calculations are included in the simulations.

Weighted time series may be used by enabling 'Use weighted time series'. This adds a new tab 'TS weighted rainfall/evaporation' where time series, their corresponding weights and distribution in time may be defined (see 'Weighted time series' previous in this section).

Runoff computations

The model computations are based on the volume continuity and the kinematic wave equations.

The first step is the calculation of the snow storage if relevant, then the effective precipitation intensity. The effective precipitation intensity is the precipitation which contributes to the surface runoff, i.e. when the losses have been taken into account (evaporation, infiltration...).

Next, the hydraulic routing based on the kinematic wave formula (Manning) and volume continuity is applied. The sketch with schematics of the model computation is shown in Figure 7.4.

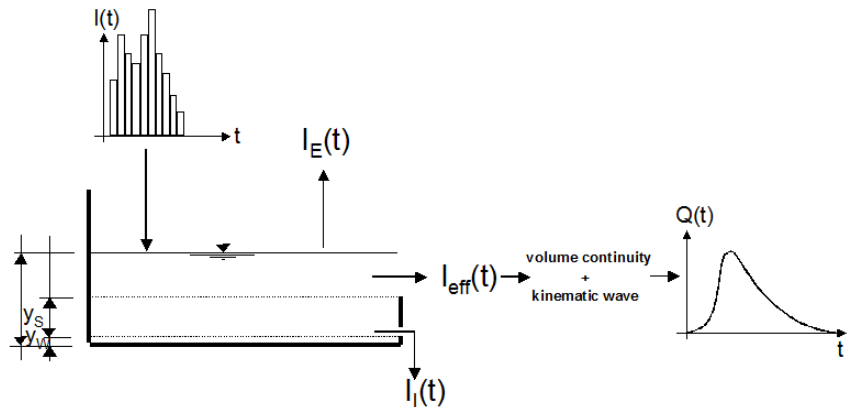


Figure 7.4 The simulated processes in Kinematic wave surface runoff model

Snow storage computation

Snow can accumulate when the temperature is inferior or equal to zero degrees Celsius. During warmer periods, when temperature exceeds zero degrees Celsius, the snow storage melts at a rate given by the snow melt coefficient. The snow storage computation will be performed only if a temperature time series is provided for the catchment.

During storing periods, or the snow will accumulate without limit. If the temperature is below zero degrees Celsius, it is assumed that the entire rainfall will be stored as snow.

During melting periods, or $T > 0^{\circ}\text{C}$ the snow will melt and therefore contributes to the surface runoff at rate given by the snow melt coefficient.

The snow module is the first process occurring after rainfall, therefore all others losses will be computed afterwards.

Effective precipitation computation

The simulated hydrologic processes account for various losses calculated - evaporation, wetting, infiltration and surface storage - according to the conventions and equations presented below. The remaining precipitation is called effective precipitation, defined generally as:

$$I_{\text{eff}}(t) = I(t) - I_E(t) - I_W(t) - I_I(t) - I_S(t) \quad , \quad I_{\text{eff}} \geq 0 \quad (7.5)$$



Where:

$I(t)$	Actual precipitation at time t
$I_E(t)$	Evaporation loss at time t . It should be noted that the evaporation loss for the catchment is accounted only if an evaporation time series is provided
$I_W(t)$	Wetting loss at time t
$I_I(t)$	Infiltration loss at time t
$I_S(t)$	Surface Storage loss at time t

The individual terms in the loss equation are fundamentally different, as some terms are continuous where others are discontinuous. If the calculated loss is negative, it is set to zero. The losses have a dimension of velocity [LT⁻¹].

The actual precipitation, $I(t)$, is assumed to be uniformly distributed over the individual catchments. Otherwise, it may vary as a random time function.

The evaporation, $I_E(t)$, is a continuous loss that is normally of less significance for single event simulations. However, on a long-term basis, evaporation accounts for a significant part of hydrological losses. If included in the computation, the evaporation is the first part subtracted from the actual precipitation, according to the following:

$$I_E(t) = \begin{cases} I_E(t) & \text{for } (I(t) \geq I_{PE}(t)) \text{ OR } (y(t) > 0) \\ I(t) & \text{for } (I(t) < I_{PE}(t)) \text{ AND } (y(t) = 0) \end{cases} \quad (7.6)$$

Where:

$I(t)$	Actual precipitation at time t
$I_E(t)$	Evaporation loss at time t
$I_{PE}(t)$	Potential evaporation at time t
$y(t)$	Accumulated depth at time t

If the actual evaporation is not explicitly specified in the simulation (i.e. Evaporation process activated – see INI file parameters – and evaporation time series specified), only a decay rate will be applied during dry periods.

The wetting, $I_W(t)$, is a discontinuous loss. When the precipitation starts, a part of the precipitation is used for wetting of the surface if the surface is initially dry. The model assumes that the precipitation remaining after subtrac-



tion of the evaporation loss is used for wetting of the catchment surface. When the surface is wet, the wetting loss, I_W , is set to zero. This is summarised in the following expression:

$$I_W(t) = \begin{cases} I(t) - I_E(t) & \text{for } y(t) < y_w \\ 0 & \text{for } (I(t) \leq I_E(t)) \text{ OR } (y(t) \geq y_w) \end{cases} \quad (7.7)$$

Where:

$I(t)$ Actual precipitation at time t

$I_E(t)$ Evaporation loss at time t

$I_W(t)$ Wetting loss at time t

y_w Wetting depth

$y(t)$ Accumulated depth at time t

The infiltration, $I_I(t)$, is the water loss to the lower storage caused by the porosity of the catchment surface. It is assumed that the infiltration starts when the wetting of the surface has been completed. The infiltration loss is calculated according to the following relation:

$$I_I(t) = \begin{cases} I_H(t) & \text{for } (I(t) - I_E(t) - I_W(t)) \geq I_H(t) \text{ \& } (y(t) \geq y_w) \\ I(t) - I_E(t) - I_W(t) & \text{for } (I(t) - I_E(t) - I_W(t)) < I_H(t) \\ 0 & \text{for } y(t) < y_w \end{cases} \quad (7.8)$$

Where:

$I_I(t)$ Infiltration loss at time t

$I_H(t)$ Horton's infiltration at time t

y_w Wetting depth

$y(t)$ Accumulated depth at time t

The infiltration is a complex phenomenon, dependent on the soil porosity, moisture content, groundwater level, surface conditions, storage capacity, etc. The model calculates the infiltration loss capacity using the well-known Horton's equation, per default in its original form:

$$I_H(t) = I_{Imin} + (I_{Imax} - I_{Imin}) \cdot e^{-k_a \cdot t} \quad (7.9)$$



Where:

$I_H(t)$	Infiltration loss calculated according to Horton
$I_{I\max}$	Maximum infiltration capacity (after a long dry period)
$I_{I\min}$	Minimum infiltration capacity (at full saturation)
t	Time since the start of the storm
k_a	Time factor (characteristic soil parameter) for wetting conditions

The surface storage, $IS(t)$, is the loss due to filling the depressions and holes in the terrain. The model begins with the surface storage calculation after the wetting process is completed. The surface storage is filled only if the current infiltration rate is smaller than the actual precipitation intensity reduced by evaporation. The actual surface storage loss is calculated according to the following:

$$I_S(t) = \begin{cases} I(t) - I_E(t) - I_W(t) - I_I(t) & \text{for } y(t) \leq (y_W + y_S) \\ 0 & \text{for } y(t) > (y_W + y_S) \end{cases} \quad (7.10)$$

Where:

$I(t)$	Precipitation intensity at time t
$I_S(t)$	Surface storage loss at time t
$I_I(t)$	Infiltration loss at time t
$I_W(t)$	Wetting loss at time t
$I_E(t)$	Evaporation loss at time t
y_W	Wetting depth
y_S	Surface storage depth
$y(t)$	Accumulated depth at time t

Surface Runoff routing computation

The runoff starts when the effective precipitation intensity is larger than zero. The hydraulic process is described with the kinematic wave equations for the entire surface at once. This description assumes uniform flow conditions on the catchment surface, i.e. equal water depth over the entire surface of certain category.



This type of runoff model is also called a non-linear reservoir model.

The surface runoff at time t is calculated as:

$$Q(t) = M \cdot B \cdot I^{1/2} \cdot y_R(t)^{5/3} \quad (7.11)$$

Where:

M	Manning's number
B	Flow channel width, computed as: $B \text{ [m]} = A \text{ [m}^2\text{]} / L \text{ [m]}$
I	Surface slope
$y_R(t)$	Runoff depth at time t

The depth $y_R(t)$ is determined from the continuity equation:

$$I_{\text{eff}}(t) \cdot A - Q(t) = \frac{dy_R}{dt} \cdot A \quad (7.12)$$

Where:

I_{eff}	Effective precipitation
A	Contributing catchment surface area
dt	Time step
dy_R	Change in runoff depth

Hydrological losses depending on surface type

The Kinematic Waves Surface Runoff Model distinguishes between up to 5 different catchment surface types. This is practically handled by the model so that the individual catchment is split into up to five sub-catchments, each with the area according to the specified percentages for specific surface categories.



For each surface type, only relevant processes are simulated. An overview of the processes associated with different surface types is shown in Table 7.8..

Table 7.4 Processes included for various types of catchment surface

Surface Type	Hydrological Loss Simulated
Impervious Steep	Evaporation Wetting
Impervious Flat	Evaporation Wetting Storage
Pervious small	Evaporation Wetting Infiltration Storage
Pervious medium	Evaporation Wetting Infiltration Storage
Pervious large	Evaporation Wetting Infiltration Storage

The model treats every area with different surface category as a sub-catchment, and the runoff computations are performed individually. The total runoff from the entire catchment is obtained then as a sum of runoffs from up to five different sub-catchments.

Definition of the sub-catchment geometry

The length and width for each sub-catchment (sub-area) are calculated so that the length/width ratio for each sub-area is kept equal to the length/width ratio of the corresponding catchment. Based on the information for the whole catchment and the principle of constant length/width ratio, equivalent values of the runoff width and length are computed for all sub-areas, as illustrated in the example below (Figure 7.5).

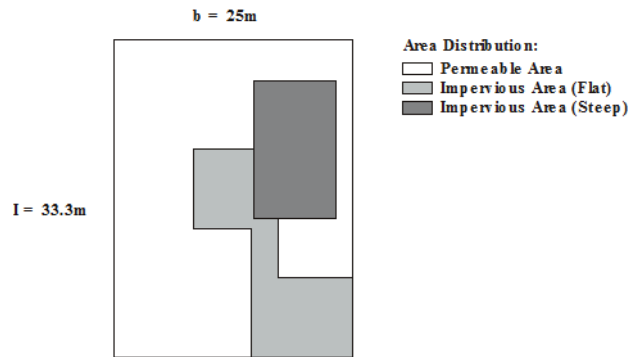


Figure 7.5 Sub-catchments with total area = 833 m²

The ratio between the catchment length and width in the given example corresponds to 1.33. 15% of the total area is impervious roof surface corresponding to 125 m². Hence the runoff length is 12.9 m and the runoff width is 9.7 m for this surface type, as $12.9 * 9.7 = 125$ and $9.7 * 1.33 = 12.9$.

Multiple-Event Simulations

If the Kinematic wave surface runoff model is used for a continuous simulation of multiple rainfall events, a special solution has been applied for the simulation of dry periods between the consecutive events. The solution accounts for the following phenomena:

Recovery of the soil infiltration capacity

According to Horton equations, the soil infiltration capacity is getting reduced as the soil gets more saturated by rain. In dry periods, an inverse process occurs, with gradual recovery of the infiltration capacity. Computation of both processes is detailed in Section 7.4.3.

As a consequence of wet and dry period alternation in a multiple event simulation, the model alternates between the two computation modes.

Switching to the “dry” mode is triggered by the exhaustion of all water available for infiltration. Consecutively, switch to the “wet” mode at the start of a new rain event.

Recovery of the initial loss capacity during dry intervals

The occurrence of the initial loss at the beginning of each simulated event is modelled in accordance with reality. During dry periods, the initial loss capacity will be recovered if no evaporation time series has been provided.



7.2 Combined Catchments

A combined catchment is a catchment made up with a number of sub-catchments, which are defined in the *Catchment definitions* menu.

The runoff from the combined catchment is found by simple addition of the simulated runoff from each of the subcatchments. This type of catchment can especially be used during the calibration process to quickly get the total runoff at a gauge location, based on all upstream sub-catchments.

Combined catchments are not shown on the map, and can only be edited from the *Tabular* tab. Use the *Append '+'* or *Delete '-'* buttons above the overview table to add or remove a combined catchment.

The various properties specified for a combined catchment are described below.

- **Combined catchment name.** The combined catchment can be given any name.
- **Total area.** The total area of the catchment is automatically computed, and is equal to the sum of the areas from all its subcatchments.
- **Subcatchments.** All subcatchments have to be added in this table using the *Append '+'* button. They can be deleted using the *Delete '-'* button. Each subcatchment has to be selected from the drop-down list amongst the catchments defined in the *Catchment definitions* menu. The area considered for each subcatchment is always its total area.
- **Calibration plot.** It is possible to save the calibration plot by enabling 'Calibration plot'. The calibration plot shows observed and simulated runoff for the catchment.
- **Observed discharge.** When the 'Calibration plot' is enabled, a time series of observed runoff has to be specified here with the Browse '...' button.

7.3 River links

A river link is a reach, that is to say a part of a branch, or a point where the runoff from the catchment is distributed to the river. The spatial extent of the river link therefore controls how the runoff from the rainfall-runoff simulation is used in the hydrodynamic simulation.

River links may be added or deleted using the *Append '+'* or *Delete '-'* buttons above the overview table.



Note: The runoff related to a river link does not necessarily have to be the full runoff from a given catchment. A river link can therefore deal with only a part



of a catchment's runoff, for example when the total runoff has to be distributed through multiple river links.

A river link is defined with the two following groups of parameters.

Catchment definition

The parameters below control the runoff value which is considered as part of the link.

- **Catchment name.** The catchment from which the runoff is computed is selected from this list.
- **Linked area type.** This parameter controls whether the part of the catchment used for the river link is expressed as an absolute area or as a fraction of the total area.
- **Linked area.** When the *Linked area type* is set to *Linked area*, this parameter indicates the area of the catchment which is considered for the river link. The resulting runoff for this river link will be the total catchment's runoff multiplied by a ratio of the specified area with the total catchment's area.
- **Linked fraction.** When the *Linked area type* is set to *Linked fraction*, this parameter indicates the fraction of the catchment which is considered for the river link. The resulting runoff for this river link will be the total catchment's runoff multiplied by the fraction. A fraction of 1 hence indicates that the total catchment's runoff is used.

River link

The parameters below control where the runoff is applied to the river network for the hydrodynamic simulation.

- **Distribute to.** When distributing to a branch, the catchment runoff is distributed along a river branch. Alternatively, the runoff may be distributed to a storage.
- **Branch name.** The branch where the runoff is applied is selected from this list.
- **Link type.** Three link types can be selected:
 - Point: the specified runoff is added as a point source in the branch. This source is defined with a single chainage.
 - Distributed: the specified runoff is distributed to a reach along the branch. The location of the reach is defined with an upstream and a downstream chainage.
 - Entire branch: the specified runoff is distributed to the whole branch, hence no chainages have to be specified.

- **Upstream chainage.** For a *Point* link type, this chainage defines the location of the point source where the runoff is applied. For a *Distributed* link type, this chainage defines the upstream extent of the reach where the runoff is applied.
- **Downstream chainage.** For a *Distributed* link type, this chainage defines the downstream extent of the reach where the runoff is applied.
- **Storage ID.** Name of the storage in which the catchment runoff is inserted, when the river link is applied to a storage.

7.4 Decoupled simulation

Rainfall-runoff and hydrodynamic simulations are two different simulations, but are often performed simultaneously: in that case, the hydrodynamic simulation uses the raw results from the rainfall-runoff simulation. The present decoupled option is in that case not used.

It is however possible to run them in a decoupled mode: in that case, only the hydrodynamic simulation is performed and it reads the rainfall-runoff data from a result file. To apply this mode, enable the 'Decouple Rainfall-runoff and Hydrodynamic simulations' option, and then select the rainfall-runoff result file using the Browse '...' button.

When running a decoupled simulation, the catchments and the river links must be defined as usual.



Note: The decoupled option is only available when the 'Data' checkmark is ticked and the 'Sim.' checkmark is not ticked for the *Rainfall runoff module*, in the *Modules* menu.



Note: When running a decoupled simulation, it is recommended to pay attention to the storing frequency used in the rainfall-runoff result file. This storing frequency should indeed be high enough in order to accurately reflect the runoff fluctuations. If that is not the case, the results of the decoupled hydrodynamic simulation will differ from those that would be obtained with a 'coupled' simulation.

7.5 Hotstart files

The initial conditions for a NAM simulation may be loaded from an existing rainfall-runoff result file by enabling 'Use hotstart files' and clicking on the 'Append' button. This will add a new row to the list of *Hotstart files*. Click on the 'Browse' button and select the required hotstart file.



Important: Initial conditions are read in the additional result files having the extension `_AddNAM.res1d`. Therefore, only this type of file can be selected here.



Note: It is possible to define multiple hotstart files covering different or overlapping areas. If some catchments are covered by more than one hotstart file the first hotstart file will be used. The 'Move up' and 'Move down' arrows may be used to modify the order of the hotstart files. If a catchment is not covered by any of the specified hotstart files, the initial conditions are those specified in the *Catchment definitions* menu.

Use simulation start time

If 'Use simulation start time' is enabled, the initial condition will automatically be extracted from the *Hotstart file* using the simulation start date and time as specified in the *Simulation period* page.

Date and Time

The date and time at which the initial conditions are loaded from the hotstart file. If 'Use simulation start time' is enabled the hotstart date and time will be taken as the simulation start.





8 Hydrodynamic Parameters

The fully dynamic Saint Venant flow equations includes a number of variables of which some are automatically calculated during the simulation and some are user defined. The *Hydrodynamic Parameters* dialogue allows the user to define specific values for parameters related specifically to the Hydrodynamic model. Parameter values can be entered as a single Global value for the entire river network or as a number of spatially varying local values.

Hydrodynamic parameters presently includes:

- Bed resistance.

8.1 Bed Resistance

Bed resistance values

Approach

Two approaches may be applied for the bed resistance. Either a uniform or a so-called multiple zones approach.

With the Uniform section approach, the bed resistance is defined by a resistance formula and associated global/local values, if the '*Resistance type*' has been set to '*Relative resistance*' for the cross section. The resulting bed resistance number applied in the simulation is the defined bed resistance multiplied by the water level dependent 'Resistance factor' which is specified for the cross sections in the cross section editor (saved in .xns11 files).

With the Multiple zones approach, the same rules apply, but it also offers a possibility for the user to divide the cross sections in a number of zones with different bed resistance values. These zones would typically represent the various areas with a local bed resistance value (e.g. a zone without vegetation at the bottom of the cross section, a vegetation zone on banks, etc.) as illustrated in Figure 8.1.

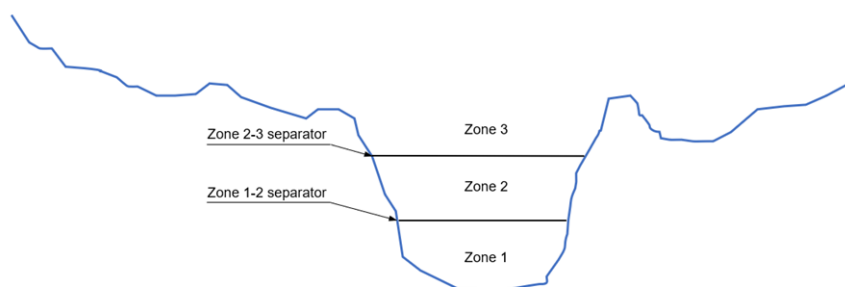


Figure 8.1 Multiple zones description of a cross-section



The resulting bed resistance numbers applied in the simulation are the zone values multiplied by the 'Resistance factors' specified for the cross sections in the cross section editor.

Zones separators, defining the threshold level or depth between two consecutive zones, must be specified in the 'Zones separators' tab.

During the simulation, the bed resistance applied to the cross section will vary in time as a function of the water level: at each time step, the bed resistance from the zone in which the water surface is, will apply uniformly to the whole cross section.



Note: Bed resistance values are actually applied to processed data of the cross section. Each row from the processed data table gets the bed resistance value from the zone corresponding to the row's level. Interpolation will apply afterwards during the simulation, between the processed data levels.

Resistance formula

Four resistance type options are available from the drop-down menu:

- Manning's M (unit: $\text{m}^{1/3}/\text{s}$, typical range: 10-100)
- Manning's n (reciprocal of Manning's M, typical range: 0.010-0.100)
- Chezy (C)
- Darcy-Weisbach (k)

For more information on the resistance formulas, please refer to MIKE 1D Reference Manual.

Resistance numbers specified in this dialogue window are a combination of a 'Global value' and possibly one or more 'Local values'.

Global value

For the Uniform approach, a global resistance value must be specified, which is applied throughout all calculation points in the network, unless local values have been defined.

For the Multiple zones approach, a global number of zones must be specified, which is applied to all calculation points of the network unless local bed resistance values with a different number of zones is specified. The bed resistance values for each of the global zones must also be specified.

Local values

Local values are applied at specific locations defined by Branch name and Chainage.

To add a local value it can either be done through the tabular view or by digitising a resistance location through the *Map view*:

- In *Map view*, open the *Spatial data* ribbon



- select '*Bed resistance*' from the '*Type*' drop-down menu.
- click the '*Add*' button and click on the desired location of the local bed resistance value.
- Location will automatically be added to the list in the Local values table and a resistance value may be entered.

If two or more local values are defined, the intermediate values will be calculated by linear interpolation.

For the Multiple zones approach, a local number of zones must be specified. This local number of zones can only apply if corresponding local zone separators also exist on the same reaches of rivers. Therefore, if a local bed resistance location is added, but if no local zone separators exist, the local bed resistance is defined with the global number of zones.



Example: A global resistance value of 'Manning's M' 30 is specified. Exceptions from the global value has been specified with Local values in other branches where resistance numbers will vary according to the defined table values between the specified chainages.

Bed resistance values

Bed resistance factors

Approach
☒ Uniform section
☐ Multiple zones

Resistance formula
Manning (M)

Global value
Resistance number: 30

Local values
+ - x

		Branch name	Chainage	Resistance number
1	<input checked="" type="checkbox"/>	CALI	0	25
2	<input type="checkbox"/>	CALI	108080	35
3	<input type="checkbox"/>	OLD_CALI	0	45
4	<input type="checkbox"/>	OLD_CALI	56350	67.5
5	<input type="checkbox"/>	OLD_CALI	95075	67.5
6	<input type="checkbox"/>	NEW_CALI	0	35
7	<input type="checkbox"/>	NEW_CALI	62934	55
8	<input type="checkbox"/>	NEW_CALI	75934	55
9	<input type="checkbox"/>	NEW_CALI	113901	60

Map Tabular

Figure 8.2 Bed resistance example

Zone separators

Global values



The global zones separators must be specified, when the Multiple zones approach is used. Global zones separators are applied throughout all calculation points in the network, unless local zones separators have been defined.

Zones separators' values may either be expressed as depths or levels. When expressed as depth, the specified depths represent the height from the bottom of the cross section up to the zone separators. Depths for the consecutive zone values must therefore increase.

Local values

Local zones separators must be specified at locations where a local number of zones is expected.

Zones separators' values may either be expressed as depths or levels. When expressed as depth, the specified depths represent the height from the bottom of the cross section up to the zone separators. Depths for the consecutive zone values must therefore increase.



Note: Local bed resistance values must be specified on all river reaches where local zones separators are specified.

Bed resistance factors

By activating the 'Include resistance factors' option, it is possible to apply factors/coefficients to the bed resistance values, which can either be constant in time or time varying. This facility e.g. allows to apply seasonal variations of bed resistance.

Applying a resistance factor higher than 1 will decrease the actual roughness of the river bed, whereas a value lower than 1 will increase the actual roughness. So when the resistance type is Manning (M), then the Manning's M value is multiplied by the factor. When the resistance type is Manning (n), then the Manning's n value is divided by the factor. This applies regardless if the cross sections use a relative or Manning resistance value: if a cross section uses a Manning resistance value, then the resistance factor is applied to its processed data.

Resistance factors specified in this dialogue are a combination of a 'Global value' and possibly one or more 'Local values'.

Global value

A global resistance factor value or time series, which is applied throughout all calculation points in the network, unless local values have been defined.

Local values

Local values are applied at specific locations defined by Branch name, Upstream chainage and Downstream chainage. For each local value, it is also possible to apply either a constant value or a time series.



9 Advection-Dispersion Parameters

The advection-dispersion (AD) module is based on the one-dimensional equation of conservation of mass of a dissolved or suspended material, i.e. the advection-dispersion equation. The module requires output from the hydrodynamic module, in time and space, in terms of discharge and water level, cross-sectional area and hydraulic radius.

The Advection-Dispersion Equation is solved numerically using an implicit finite difference scheme which, in principle, is unconditionally stable and has negligible numerical dispersion. A correction term has been introduced in order to reduce the third order truncation error. This correction term makes it possible to simulate advection-dispersion of concentration profiles with steep fronts.

9.1 The Advection-Dispersion Equation

The one-dimensional (vertically and laterally integrated) equation for the conservation of mass of a substance in a solution, i.e. the one-dimensional advection-dispersion equation reads:

$$\frac{\partial AC}{\partial t} + \frac{\partial QC}{\partial x} - \frac{\partial}{\partial x} \left(AD \frac{\partial C}{\partial x} \right) = -AKC + qC_2 \quad (9.1)$$

where

- C : concentration
- D : dispersion coefficient
- A : cross-sectional area
- K : linear decay coefficient
- C₂ : source/sink concentration
- q : lateral inflow
- x : space coordinate
- t : time coordinate

The equation reflects two transport mechanisms:

- Advective (or convective) transport with the mean flow;
- Dispersive transport due to concentrations gradients.

The main assumptions underlying the advection-dispersion equation are:

- the considered substance is completely mixed over the cross-section, implying that a source/sink term is considered to mix instantaneously over the cross-section;



- the substance is conservative or subject to a first order reaction (linear decay);
- Fick's diffusion law applies, i.e. the dispersive transport is proportional to the concentration gradient.

9.2 Components

Component names and numbers must be specified in this dialogue.

The components are user defined. Each component is modelled using a defined concentration unit and component type. Components are only required to be defined in this page when stand- alone AD-simulations are to be conducted (without any Water Quality / MIKE ECO Lab simulations included).



Note: For Water Quality simulations using the MIKE ECO Lab Framework it is not required to define any Components in this property page. Component names in an MIKE ECO Lab (Water Quality) simulation are derived from the 'State Variables' definition in the MIKE ECO Lab template file.

Component definitions includes:

Component

Here one or more component names are defined by a user-defined string. To add or remove advection-dispersion components use the '+' (append row) and '-' (delete row) buttons.

Type

The type of component must be specified. Four types are available from the drop-down menu:

- Concentration
- Temperature
- Bacteria concentration
- Salinity

Unit for results

The requested unit for each of the components is selected through the drop-down menu. The available options depend on the selected component type.

Units for component type *Concentration*:

- $\mu\text{g}/\text{meter}^3$: microgram per cubic meter
- mg/meter^3 : milligram per cubic meter



- gram/meter³: gram per cubic meter
- kg/meter³: kilogram per cubic meter
- mu-g/liter: microgram per litre
- mg/liter: milligram per litre
- gram/liter: gram per litre
- pound/feet³: pound per cubic feet
- ton/meter³: ton per cubic meter

Units for component type *Temperature*:

- degree Celsius: degrees in Celsius
- degree Fahrenheit: degrees in Fahrenheit
- degree Kelvin: degrees in Kelvin

Units for component type *Bacteria*:

- million/100 ml: million of bacterial counts per 100 ml
- per 100 ml: bacteria cells per 100 ml
- per litre: bacteria cells per liter

Units for component type *Salinity*:

- PSU: Practical Salinity Units
- per thousand: parts per thousand

9.3 Dispersion

The dispersion coefficient, D , is used for both the Advection-Dispersion and the Sediment Transport modules. It is described as a function of the mean flow velocity, V , as shown below.

$$D = aV^b \quad (9.2)$$

Where 'a' is the dispersion factor and 'b' the dispersion exponent. Typical value ranges for D : 1-5 m²/s (for small streams), 5-20 m²/s (for rivers).

Both the 'dispersion factor' and the 'dispersion exponent' can be specified. If the dispersion exponent is zero then the dispersion coefficient D becomes constant (equal to the dispersion factor). By default the dispersion is zero (i.e. there is only advective transport and no dispersion).

The 'Minimum dispersion coefficient' and the 'Maximum dispersion coefficient' parameters are used to control the range of the calculated dispersion coefficients.



Global values

Global dispersion values are defined in this group. Global values are defined in all branches and calculation points unless Local values are defined additionally.

Dispersion factor

The dispersion factor equivalent to parameter a in equation (9.2) is defined here.

Exponent

The dispersion exponent equivalent to parameter b in equation (9.2) is defined here.

Minimum disp. coefficient

Using equation (9.2), the dispersion coefficient value is depending on the velocity that vary during the simulation. This parameter defines the lowest allowed value of the dispersion coefficient during simulation.

Maximum disp. coefficient

Using equation (9.2), the dispersion coefficient value is depending on the velocity that vary during the simulation. This parameter defines the highest allowed value of the dispersion coefficient during simulation.

Local Values

MIKE HYDRO uses the values specified under global values except for those locations where local values have been specified. To add or remove these locations to the table use the Append and Delete buttons ('+' or '-'): When a local value item is included, a line is added to the local values table, where the following specification must be defined:

Branch Name

Name of the river branch with local dispersion values.

Chainage

Chainage in river with local dispersion values.

Dispersion factor

Local value of the dispersion factor.

Exponent

Local value of the dispersion exponent.

Minimum coef.

Minimum value of the dispersion coefficient at the location.

Maximum coef.

Maximum value of the dispersion coefficient at the location.



The advection-dispersion properties at a given location can be displayed in the *Property View* if the checkbox of the specific row in the table is checked. When several table items are checked, it is possible to unselect all of them at the same time by clicking on the “Clear selection” button ('x').

9.4 Decay

This page contains information for non-conservative components. These components are assumed to decay according to a first-order expression:

$$\frac{dc}{dt} = KC \quad (9.3)$$

Where K is a decay constant and C is the concentration. Both global and local values of the decay constant K can be specified.



Important: Decay constants should not be specified in this dialogue if the components selected are used for a water quality simulation (WQ) as process-equations defined in the Water Quality (or MIKE ECO Lab) template normally includes all processes affecting the State variables. Therefore, adding an additional decay in this AD Parameter Editor will normally interfere negatively with the defined equation framework of the Water Quality model.

Global values

Component

Here the component in question is selected. You can select the component from the drop-down menu.

Decay

Here the value of the decay constant is entered (default unit is [per hour]). Alternative units are

Local values

MIKE HYDRO uses the values specified under global values except for those locations where local values have been specified. To add or remove these locations to the table use the Append and Delete buttons ('+' or '-').

Branch name

The name of the river branch with the local decay constant value.

Component

Here the component with a local decay value is selected. You can select the component from the drop-down menu.

Chainage

The chainage in the river with the local value.



Decay

Here the local value of the decay constant is entered (default unit is [per hour]).



10 Water Quality

MIKE HYDRO includes options for Water Quality (WQ) simulations. WQ-simulations in MIKE HYDRO utilises the MIKE ECO Lab framework and hence, offer a versatile and extremely flexible options for WQ-modelling. Water Quality simulations can be performed using either standard, predefined MIKE ECO Lab templates or user-defined templates with tailored Water Quality models.

10.1 Notes on MIKE ECO Lab

MIKE ECO Lab is a numerical lab for Ecological Modelling. It is an open and generic tool for customising Aquatic Ecosystem models to describe water quality, eutrophication, heavy metals and ecology. The module is mostly used for modelling water quality as part of an Environmental Impact Assessment (EIA) of different human activities, but the tool is also applied in aquaculture for e.g optimizing the production of fish, seagrasses and mussels. Another use is in online forecasts of water quality.

The need for tailor-made ecosystem descriptions is big because ecosystems vary. The strength of this tool is the easy modification and implementation of mathematical descriptions of ecosystems into the hydrodynamic MIKE engines.

MIKE ECO Lab utilises so-called Template files with the filename '.ecoLab' (in the following named MIKE ECO Lab File). The MIKE ECO Lab template files are customized collections of equations and parameters required for a specific type of Water Quality simulations. The user may use one of the predefined MIKE ECO Lab Templates, which is installed together with the MIKE Zero installation - or may choose to develop own model concepts. The module can describe dissolved substances, particulate matter of dead or living material, living biological organisms and other components (all referred to as state variables in this context).

The module was developed to describe chemical, biological, ecological processes and interactions between state variables, but it can also describe the physical sedimentation process of components. State variables included in MIKE ECO Lab can either be transported by advection-dispersion processes based on hydrodynamics, or have a more fixed nature (e.g. rooted vegetation).

More details on MIKE ECO Lab can be found in the specific MIKE ECO Lab documentation, which can be accessed from the MIKE Zero Documentation Index.



10.2 Water quality definitions

Water Quality must be enabled by the user if it should be included in the simulation and in the MIKE HYDRO tree view. To activate Water Quality it is required to enable the 'Water quality' checkboxes under *Simulation specifications\Modules*.

The Water quality definitions page includes the following tab-pages:

- MIKE ECO Lab template
- Initial state variables
- Constants
- Forcings
- Output

MIKE ECO Lab template

Solution method

Water quality calculation is based on predefined MIKE ECO Lab templates which contains a model defined by a number of coupled differential equations, which is solved through numerical integration and interactions between each equation.

Solution method is a selection of the Integration Method for solving the coupled ordinary differential equations defined in the MIKE ECO Lab file. Three different built-in integration routines (solution methods) are available (please consult the MIKE ECO Lab Reference Manual for details on the methods for solving the coupled linear differential equations in the MIKE ECO Lab framework):

- **Euler:** Euler or Linear Solution
A very simple numerical solution method for solving ordinary differential equations.
- **RK4:** Fourth order Runge-Kutta.
A classical numerical solution method for solving ordinary differential equations. It has normally higher accuracy than the Euler method but requires longer simulation times. The fourth order Runge Kutta method requires four evaluations of equations per time step.
- **RKQC:** Fifth order Runge-Kutta with Quality Control
A numerical solution method for solving ordinary differential equations. The accuracy is evaluated and the time step is adjusted if results are not accurate enough. The method requires 6 evaluations at each time step to take a so-called Cash-Karp Runge Kutta step and the error is estimated as the difference between a Runge Kutta fourth order solution and the Runge Kutta fifth order solution.



The accuracy (and the computing time) varies for the three integration routines.

The most accurate result will be calculated when using RKQC. However, in some cases the same results can be obtained - using less computational time - with the less advanced options; RK4 or EULER.

In general, it is recommended to use the RKQC routine. RK4 and EULER methods are generally only applied during the set-up and initial calibration phase of a project. If the RK4 or the EULER routines are used, it is strongly recommended to run an additional simulation with the RKQC routine and compare the two results (RKQC versus RK4/ EULER) before making any conclusions based on the model.

In the case of a very dynamic model system with steep concentration gradients in one or more of the components, integration may not be possible when using the RKQC routine, and an error message will appear. Reducing the time step will help in most cases, but sometimes the gradients are so steep, that they cannot be solved accurately. The Quality Control of RKQC ensures that all components are calculated within an accuracy of 1 µg/l. Using the second best routine (RK4), where no Quality Control is included, the steep gradients can be solved in a relatively accurate way and RK4 is therefore recommended when integration is impossible with the RKQC routine.

Update frequency

The update frequency is a parameter that allows to define how often the water quality processes will be calculated during the simulation. The update frequency is defined as a multiplier of the simulation time step used for the HD- and AD-simulation and therefore determines the frequency for simulating water quality processes on top of the standard Advection and Dispersion processes which are calculated at every time step. The Update Frequency must be an integer above zero. Definition of the frequency should be based on careful considerations. The user should recognise that the dynamics of the advection dispersion is comparable to the dynamics of the process descriptions. Thus it is strongly recommended to use an update frequency of 1.

The selection of the Time Step of the MIKE ECO Lab model, and hereby the Update Frequency, has to be based on considerations of the time scales of the processes involved. Please notice that this selection can be rather decisive for the precision of the numerical solution as well as for the CPU time of the simulation. A large Update Frequency will decrease the precision as well as the CPU time. It is therefore advisable to perform a sensitivity analysis on the Update Frequency before making the final selection.


Disable calculation of processes. AD results only.

MIKE ECO Lab templates are often used to define a range of state variables (or AD components) and it therefore often beneficial to make a template that suits the need of the project. The integration of MIKE ECO Lab in MIKE HYDRO also offers the possibility for executing a simulation which does not include any water quality process solutions from the MIKE ECO Lab framework. By enabling the '*Disable calculation of processes. AD results only*' tick-mark, the simulation will include only the transport and spreading of model components (state variables) and not include any biological or chemical processes at all.

The simulation results is in that case identical to a situation where the components are defined and simulated through the Advection Dispersion module.

Enabling this checkmark may be feasible in an early stage of model creation where stability of the Hydrodynamic and Advection-Dispersion simulations are performed. Water quality processes can then be activated as a second phase of model definition.

Model selection

The water quality model to be applied is defined in a MIKE ECO Lab template file and the model selection is about selecting the MIKE ECO Lab template that must be used for the project. It is possible to select between a range of MIKE ECO Lab templates defined by DHI from the drop-down selection (press 'arrow down' in the *Select model* field. Alternatively, use the file selection button  to browse for the requested MIKE ECO Lab file.

Summary

Summary section list the overall content of the selected MIKE ECO Lab template. *Description* is the water quality model description defined in the MIKE ECO Lab template.

Initial state variables

Initial state variables enable the specification of initial values for each state variable defined in the MIKE ECO Lab template. Initial values are applied at start of simulation. Default value is 0 for all the variables unless other values are defined.

Initial state variables can be defined as Global values and Local values. Global values a per default applied for the state variables in all points in the model, and Local values can then be defined where local exceptions to the global values has to be defined.

Global

Global values of initial conditions for state variables are defined here. The table presents all state variables derived from the MIKE ECO Lab template and initial values can be defined for all state variables.



Some of the fields in the table are not editable; Name, Description, Transport, Type and Unit of the variable.

Changes can be made to the following columns:

Fixed value. Initial global value of the actual state variable.

Factor. Scaling factor for the parameter's Fixed value. The default value is 1.

Local

Local exceptions of the globally defined initial values for state variables are specified in this page.

Note that if you want to specify local variations at the same locations for different state variables, you must define this in individual lines for each State Variable.

Use the Add button '+' to create the required number of lines in the table (one line for each location where an individual state variable needs to have a local value defined). The delete button '-' deletes highlighted rows from the table.

Name. Click inside the *Name* field to open a drop down menu, from which the required state variable can then be selected.

Branch name. The branch along which the variable assumes a local value can be selected from a drop down menu listing all branches in the model setup.

Distributed. Checkbox defining if the value of the variable is specified in a single point or distributed in a certain reach of a river.

From chainage. River chainage from which the specific variable assumes a local value. if *distributed* is enabled. River chainage of local value if point value is the option.

To chainage. River chainage to which the specific variable assumes a local value in case *distributed* has been enabled

Fixed value. Initial local value of the state variable.

Factor. Scaling factor for the parameter's Fixed value. The default value is 1.

Constants

Constants can be any parameters (physical constant, coefficient, rate, etc.) defined in the expressions of in the MIKE ECO Lab model. Constants are potentially spatial varying but constant in time.

The Constants are essentially divided into two groups:

- Built-in Constants, and



- User defined Constants

The built-in Constants are automatically provided by the MIKE HYDRO engine during execution, whereas the user-specified Constants must be specified in the present dialogue.

Depending on the Spatial Variation of the Constant, it can be specified as a Global value or as Local values.

Global

Global values for Constants are defined here. The table presents all Constants derived from the MIKE ECO Lab template. Global values can be defined for all Constants defined as 'User defined' in the MIKE ECO Lab template.

Some of the fields in the table are not editable; Name, Description, Transport, Type and Unit of the variable.

Changes can be made to the following columns:

Fixed value. Initial global value of the actual Constant.

Factor. Scaling factor for the parameter's Fixed value. The default value is 1.

Interpolate. Interpolate checkbox relate to the overall definition on how to interpret the potential definition of local values for the specific variable. If Interpolate is enabled, multiple local values for the same variable at different locations in the same branch will be interpolated between the specified locations. If not enabled, values will be interpreted as local, point exceptions to the global value. That is, If this checkbox is enabled, interpolation will be activated, otherwise local values are applied only at the specific location.

Local

Local exceptions of the globally defined values for Constants are specified in this page.

Note that if you want to specify local variations at the same locations for different Constants, you must define this in individual lines for each Constant.

Use the Add button '+' to create the required number of lines in the table (one line for each location where individual Constants needs to have a local value defined). The delete button '-' deletes highlighted rows from the table.

Name. Click inside the *Name* field to open a drop down menu, from which the required Constant can then be selected.

Branch name. The branch along which the variable assumes a local value can be selected from a drop down menu listing all branches in the model setup.



Distributed. Checkbox defining if the value of the variable is specified in a single point or distributed in a certain reach of a river.

From chainage. River chainage from which the specific variable assumes a local value. if *distributed* is enabled. River chainage of local value if point value is the option.

To chainage. River chainage to which the specific variable assumes a local value in case *distributed* has been enabled

Fixed value. Initial local value of the Constant.

Factor. Scaling factor for the parameter's Fixed value. The default value is 1.

Forcings

Forcings can be defined as any input parameter (physical property, rate, etc.) in the MIKE ECO Lab model. Examples of Forcings are: Temperature, salinity, solar radiation and water depth.

Forcings are essentially divided into two groups:

- Built-in Forcings, and
- User defined Forcings

Built-in Forcings are automatically provided by the model system during model execution, whereas the user defined Forcings must be specified in the present dialogue. Depending on the Spatial Variation of the Forcing, as defined in the MIKE ECO Lab model, it can be specified as a “Constant” or a “Varying in time”.

Global

The global values of the model forcings must be entered in the table in this page. The table contains some fields that cannot be changed; these are Name, Description and Unit of the forcing. You can edit the following fields:

TS type. You can specify in this field if the value of the forcing is Constant or Time Varying.

Fixed value. Enter the constant value of a forcing.

File. If the forcing assumes a Time Varying value, you can browse from this field to the appropriate .dfs0 file or, alternatively, create and automatically upload a new .dfs0 file for the forcing.

TS item info. This field is for Time Varying forcings only. It displays which item is selected when the uploaded .dfs0 file has more than one item.



Output

Auxiliary variables

Auxiliary variables (or help processes) are intermediate calculation variables that may be defined in the MIKE ECO Lab template. If defined, they can be stored as output in an additional output result file.

Name, Description and Unit fields are derived from the template file and cannot be edited.

Selected. If this checkbox is selected, the specific auxiliary variable is included in the output results.

Process

Processes are rates describing the change of a state variable over time. Processes are defined in the MIKE ECO Lab template and may be marked an output variable. Process variables marked as output can be saved to the additional result file from the simulation.

Name, Description and Unit fields are derived from the template file and cannot be edited.

Selected. If this checkbox is selected, the specific process is included in the output results.

Derived output

Derived outputs are additional output calculations defined in the MIKE ECO Lab template. An example could be the sum of various state variables (e.g. Total N = Organic N and Inorganic N) which is useful to save in the additional output file to reduce the need for manual post-processing of the main model results.

Name, Description and Unit fields are derived from the template file and cannot be edited.

Selected. If this checkbox is selected, the specific derived outputs are included in the output results.

10.2.1 The 'Edit Filter' option

The number of state variables, constants, forcings and output variables of a MIKE ECO Lab template can become quite long, which might make it difficult to find the variables of interest in the different dialogues.

The '*Edit Filter*' option allows to filter the variables of interest according to user-specified criteria. The criteria can be edited in the window that pops up when pressing the Edit Filter button. Conditional definitions or addign or removing groups and conditions is controlled by clicking on the condition keyword and selecting from the pop-up dialogue as illustrated in Figure 10.1.

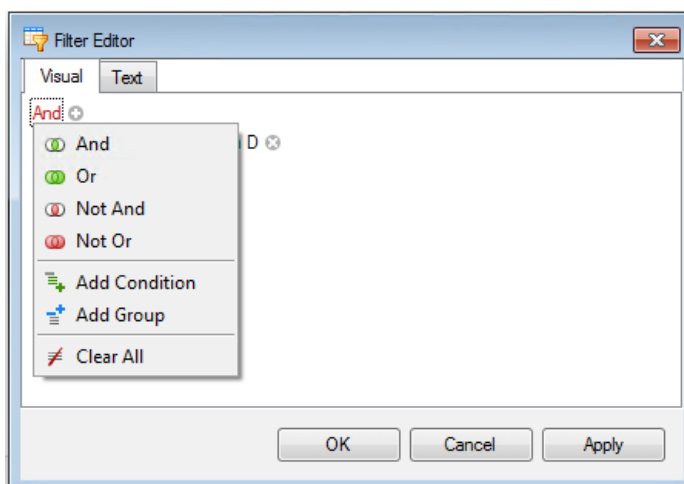


Figure 10.1 Options available for conditional definitions

In the Visual page you can create a filtering expression or a combination of them by using comparison and logical operators and applying them to the strings of the name, description and unit of the variables, their values as well as the transport option. A dialogue with options available is presented when clicking on the options field (per default set to 'Description') as illustrated in Figure 10.2.

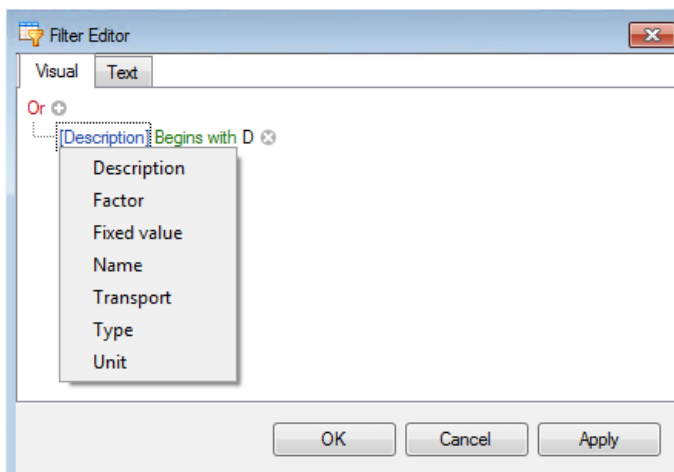


Figure 10.2 Filter editor; Options available for filtering definition

In the Text page you can visualize the expression of the filter that you have just defined.

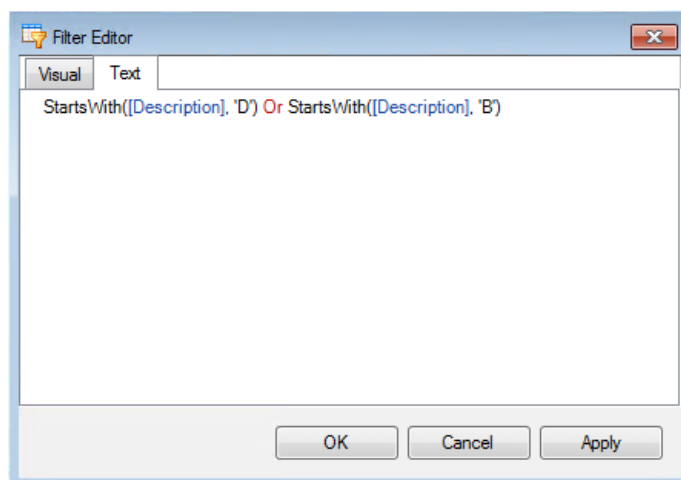


Figure 10.3 Filter editor; Defined filter expression

The defined filtering expressions will also appear below the table in the dialogue page. These filters can be selected from the drop down menu and activated and deactivated by selecting and unselecting the checkbox.



11 Sediment Transport Parameters

The sediment transport (ST) module enable the computation of cohesive and non-cohesive sediment transport, morphological changes and alluvial resistance changes of a river system. The ST module requires input from the hydrodynamic module, varying in time and space. When morphological changes are included in the simulation, the ST module also affects in return the hydrodynamic simulation.

Transport of suspended sediments uses the Advection-Dispersion equation from the AD module. For this reason, dispersion values defined in the Advection-Dispersion parameters menu are also used by the ST module.

11.1 General parameters

Sediment fractions

Number of fractions

At least one fraction must be specified. The number of fractions can be controlled using this parameter, or by adding or removing lines in the table from the Sediment fractions definitions menu. When this number of fractions is reduced, the last rows of the table are removed accordingly.

Morphological update

When morphological update is not included in the simulation, the Sediment transport module only computes a transport capacity, without any effect on the bed level.

When morphological update is included, the bed level is continuously updated depending on local erosion and deposition, and therefore affecting the calculated hydrodynamic conditions. The method used to update the bed level is specified in the Bed parameters menu.



Note: the morphological update affects the flow area, flow width and hydraulic radius values used in the hydrodynamic simulation, but it does not directly update the geometry of the physical cross sections. Therefore, the effect from the morphological update can be seen in the corresponding hydrodynamic result items (flow area, width, radius), but not on the shape of cross sections shown in the result files.

Transport properties

Maximum concentration allowed due to erosion

This parameter limits the erosion that can take place during a morphological time-step in such a way that the erosion cannot occur if it causes the sediment concentration to go above the specified maximum value. The limit applies to both bed load and suspended load.



For bed load it is applied in the form of translating the maximum concentration into a change in the sediment mass in the bed during the morphological time-step.

For suspended load the limit is used directly on the erosion function appearing on the right-hand side of the advection-dispersion equation.

The maximum concentration allowed due to erosion parameter is critical for morphological stability, but should be used with caution, as very low limits will suppress physics.

Skin friction model

The following options are available for the skin friction model:

- Total friction
- Law of the wall (full depth)
- Law of the wall (skin friction depth)

Egiazaroff hiding function

When multiple sediment fractions are simulated, the Egiazaroff hiding function can be applied.

Material properties

Kinematic viscosity

The kinematic viscosity is used for calculation of the settling velocity for the sediment.

d90/d50

The d90/d50 value is required when modelling a single fraction.

11.2 Sediment fractions definitions

The sediment properties and transport theory have to be specified for each fraction.

11.2.1 Properties

Properties

Fraction name

Each fraction is identified by a user-defined name.

Sediment type

The choice between cohesive or non-cohesive fraction type is selected from this list.



Porosity

The porosity of the sediment fraction. Porosity of the bottom sediment is usually in the range of 0.3 to 0.7.

Relative density

The relative density of the sediment fraction, compared to water density ($\rho_{solid}/\rho_{water}$). Thus, for sand, a typical value is 2.65.

Critical Shields parameter

The initiation of particle movement depends on the Critical Shields Parameter, θ_{CR} .

Fall velocity definition

The fall velocity can either be user-defined or can automatically be computed using the Rubey formula.

Fall velocity value

When using a user-defined fall velocity, the corresponding value is entered here. When applying the Rubey formula, this displays the automatically calculated value.

Grain size

Global value

A global grain size value, which is applied throughout all calculation points in the river network, unless local values have been defined.

Local values

Local grain size values can only be specified for single-fraction models. Local grain size values are applied at specific locations defined by Branch name and Chainage.

11.2.2 Transport theory

Include

For non-cohesive fractions, it is possible to select to include in the simulation either bed load or suspended load, or both. Corresponding transport theory must be defined accordingly.

Bed load

Bed load transport model

The following transport models are available for non-cohesive fractions:

- Engelund-Hansen
- van Rijn
- Engelund-Fredsoe



- Meyer-Peter & Muller
- Adjustable Smart-Jaeggi
- Yang - sand
- Yang - gravel
- Wilcock & Crowe
- Parker
- Ackers & White

Bed load factor

A bed load factor is applied to the selected transport formula. It is a simple multiplication factor applied to the calculated transport rate. The default neutral value is 1, and values can vary between 0.5 and 2.0. Beyond this range the validity of the chosen formula could be questioned.

Suspended load

Suspended load transport model

The following transport models are available for non-cohesive fractions:

- Engelund-Hansen
- van Rijn
- Engelund-Fredsøe
- Meyer-Peter & Muller
- Adjustable Smart-Jaeggi
- Yang - sand
- Yang - gravel
- Garcia & Parker
- Ackers & White
- Lane-Kalinske

Suspended load factor

A suspended load factor is applied to the selected transport formula. It is a simple multiplication factor applied to the calculated transport rate. The default neutral value is 1, and values can vary between 0.5 and 2.0. Beyond this range the validity of the chosen formula could be questioned.

Additional parameters for the Adjustable Smart-Jaeggi formula

When the Adjustable Smart-Jaeggi formula is selected for the non-cohesive fraction, the following parameters are also required, for both bed load and suspended load.



d90/d30

Describes the uniformity of sediments for the fraction.

Angle of repose

The angle of repose is specified here.

Slope dependency

The slope dependency is either defined as a function of the bed slope or the water surface slope.

Shields stress

The Shields stress is either defined as total or skin friction.

Coefficients and exponents

Parameters a1 to a8 are coefficients and exponents used in the Smart-Jaeggi formula. Refer to the MIKE 1D reference manual for more detailed information.

Suspended load, global values

The following global parameters are required for cohesive fractions. Global values are applied throughout all calculation points in the river network, unless local values have been defined.

Exponent

The exponent used in the erosion equation, from MIKE 1D reference manual.

Erosion shear stress limit

The erosion shear stress limit is specified here.

Erosion coefficient

The erosion coefficient used in the erosion equation, from MIKE 1D reference manual.

Deposition shear stress limit

The deposition shear stress limit is specified here.

Suspended load, local values

Local values may be specified in the table. Local values are applied at specific locations defined by Branch name and Chainage.

11.3 Bed parameters

Layers layout

The Layers layout define the number of layers used in the simulation:



- **Active and passive layers:** this option requires at least two sediment fractions
- **Active layer only:** this option is always available
- **Fully alluvial model (no layer):** this option can be selected when only one sediment fraction is simulated and that fraction must be non-cohesive

The bed material may be represented by two layers, an active layer in which sediment transport occurs overlying a passive layer. Each layer is divided into the number of fractions specified in the Sediment fractions definitions menu.

Target thickness of active layer

The following parameters are required for the target thickness of the active layer. Global values are mandatory and applied throughout all calculation points in the river network unless local values have been defined. Local values may be applied at specific locations defined by Branch name and Chainage. The target thickness is calculated as a factor on the current water depth, though always within the range defined by the minimum and maximum value. To give it a fixed value, set Minimum and Maximum values to the desired fixed value.

Factor on depth

The factor to multiply on the current water depth to get the target thickness of the active layer during simulation.

Minimum value

The minimum allowed thickness of the active layer during simulation.

Maximum value

The maximum allowed thickness of the active layer during simulation.

Morphological update method

The morphological update method is applied only when the morphological update is included, in the General parameters menu.

A global method must be specified. Three options are available:

- **Horizontal distribution:** sediments rest on the cross-section's bottom with a flat sediment surface. Primarily for use in cross sections with a fixed underlying bed, e.g. bottom of cross section made of concrete or sediment transported in pipes.
- **Uniform distribution:** erosion and deposition are evenly distributed along the cross section.



- **Proportional to depth:** erosion and deposition are proportional to water depth. This method uses the Floodplain level definition, specified in the cross sections. Primarily for use in cross sections with floodplain to exclude the floodplain from sediment transport calculation.

The global method is applied throughout all calculation points in the river network unless local methods have been defined. Local values may be specified in the table. Local values are applied at specific locations defined by Branch name and Chainage.

11.4 Passive branches

It is possible to exclude sediment transport calculations in selected branches when this option is active. Branches in which sediment transport should not be calculated must be listed in this table.

No sediment will enter or leave a passive branch, and therefore no morphological changes will take place in such a branch.

11.5 Non-scouring bed level

Non-scouring bed levels, where a non-erodible surface is present can be used in the simulation if this option is included. This option is only used in combination with morphological update.

During a morphological simulation, if bed erosion occurs and the bottom of the bed reaches the defined non-scouring level, no further bed erosion will take place.

Global values are applied throughout all calculation points in the river network unless local values have been defined. Local values may be specified in the table. Local values are applied at specific locations defined by Branch name and Chainage.

Apply unlimited scouring

When this option is selected, the scouring is unlimited. The combinations of global and local values can therefore be used to limit scouring only at given locations.

Non-scouring bed level

When scouring is not unlimited, it is possible to specify the global or local non-scouring bed level.



12 Boundary Conditions

Boundary conditions are a pre-requisite for solving the generally applied flow and transport equations in river modelling applications.

Boundary conditions are as a minimum required at all 'open end' locations in the river network. That is, locations where the river branch does not have any upstream or downstream connected river branches as illustrated in Figure 12.1.



Figure 12.1 River network with 'Open end' boundary locations (black circles)

In addition to the open end boundary locations, it is possible to specify a large number of other type of boundary conditions,' including:

- Internal boundary conditions (e.g. point or distributed sources)
- Global boundary definitions (e.g. wind field, Resistance factors etc.)
- Closed boundaries

Boundary conditions are defined as a combination of boundary location information and information about the parameter values to be assigned at the specific boundary. Boundary values are generally defined in a timeseries file which is then associated with the specific boundary.

The *Boundary conditions* dialogue includes the following pages for defining river specific boundary data:

- Standard boundaries
- Wind friction
- Groundwater leakage
- MIKE SHE Couplings



12.1 Standard boundaries

Standard boundary conditions for the River model are defined in the *Standard boundaries* dialogue. This page is used to specify common boundary conditions such as water levels and inflow hydrographs as well as conditions related to lateral flows along river reaches, solute concentrations of the inflow hydrographs, various meteorological data and certain boundary conditions used in connection with structures.

Boundary conditions defined in the *Standard boundaries* dialogue are visible in *Map view* with specific symbols depending on the type of boundary (Water level, discharge, Q-h relation, etc.). Using the *Boundaries* ribbon tools, boundaries can be selected, added, edited or deleted from the map. Editing a point-type boundary condition (for instance a discharge boundary with *Point source* type) will move the boundary along the river branch, whereas editing a distributed boundary (for instance a *Distributed source*) will move either the upstream or downstream end of the boundary and will therefore edit its length.

A boundary can be added to or removed from the river model by using the Append '+' and Delete '-' buttons above the overview table. Alternatively, clicking the "Create open end boundaries" button will add all the open boundaries of a river network to the boundary table simultaneously, unless they are already defined in the table.

The properties of a river model boundary can be displayed in *Property view*.

The following parameters must be entered for the river model boundaries.

12.1.1 Hydrodynamic

Boundary

ID

String identification of the boundary.

Type

Boundary Type specifies the kind of data required for the boundary. This choice also affects the Location type entry that follows.

Available options are:

- **Water level:** The Water Level boundary type is specified when a time-varying or constant water level condition is required (for the HD model).
- **Discharge:** A Discharge boundary is specified when a time-varying or constant flow hydrograph condition is required (for the HD model). Positive discharge values are considered as inflows entering the model, whereas negative values are considered as abstractions of water from the model



- **Q/h relation:** A Q/h relation boundary is specified when the relationship between the discharge and the water level is known for the HD model).
- **Free outflow:** When the Free outflow type is applied, the smallest of the critical and the natural depth is applied at the boundary
- **Closed.** The Closed boundary Type is used at free end points of the model domain where a zero flux condition across the boundary is applicable. It corresponds to zero transport across the boundary. No additional information is required except the location.
- **Rainfall.** Rainfall boundary type is specified in river reaches where the inflow of rainfall is to be represented. Rainfall can be specified globally or as a distributed source per branch.
- **Evaporation.** The Evaporation boundary type is specified in river reaches where loss of water by evaporation is to be modelled. Evaporation can be specified globally or as a distributed source per branch.
- **Runoff.** This boundary type is only available when using the Advection-Dispersion module in combination with a Hydrodynamic and Rainfall-run-off simulation. This type allows assigning a solute concentration to the runoff from a catchment, this runoff's quantity and location being defined in the River links menu

Location type

This input depends on the option selected as boundary type and is active for only some of them.

The available options are:

- **Open:** An Open boundary applies to the free upstream or downstream end of the model domain. A boundary with type Water level, Q/h relation, Free outflow or Closed can only be an open boundary.
- **Point source:** The Point source boundary condition is used for discharge boundaries when the specified discharge is to be inserted in a single point in the model domain.
- **Distributed source:** The Distributed source boundary condition is used for discharge, rainfall or evaporation boundaries when the specified inflows or outflows are to be defined all along a given reach in the model domain.



- **Global:** The Global boundary condition is applied when a rainfall or evaporation boundary condition applies over the entire model domain. In such cases it is not necessary to specify any location. It is possible to specify both a globally applicable boundary condition and a distributed boundary condition of the same boundary type: the global boundary will then be applied over the entire model area except at those locations where distributed boundaries have been specified.
- **Storage:** The Storage boundary condition is used when a discharge boundary condition applies to a storage.

Location

The location of the active boundary condition is specified here. For open and point source boundaries, the location is defined with a single point, whereas distributed source boundaries are defined for a reach, identified with an upstream and a downstream chainage.

Branch name

Name of the river branch where the boundary is located.

Chainage

Chainage where the boundary is located on the branch, for an open or a point source boundary.

Dwstr. chainage

The downstream chainage of the branch, for a distributed source boundary.

Storage ID

Name of the storage in which the discharge is inserted, for a storage location type.

HD boundary

Include boundary

The boundary condition is applied in the simulation only if the option is checked.

Input type

Here the boundary value type can be set to either Constant or Time series.

HD value

When the input type is set to Constant, the constant value for the boundary condition is specified here.

File

When the input type is set to Time series, the time series used for the boundary condition is specified here. The button to the right may be used to either browse, create, edit or plot the time series.



Item

This field shows the selected item in the time series.

Scale

The boundary condition value (constant or time series) will be multiplied by the specified Scale factor. This factor may be used to edit the boundary conditions without changing the input values, e.g. when a hydrograph representing catchment runoff needs to be applied to a number of smaller sub-catchments or when performing a sensitivity analysis.

Q/h relation table

When the boundary type is a Q/h relation, this relation is specified in this table. Rows are added and deleted using the Append '+' and Delete '-' buttons above the table. Additionally the Move Up and Move Down buttons may be used to reorder the rows.

Calculate Q/h

The Q/h relation may be automatically generated by pressing this button. This button will open a new dialog, illustrated below on Figure 12.2.

Auto calculation of Q/h table

☒ Critical flow
☐ Manning formula

Slope: 0.001
Manning's M: 40 n: 0.025

OK Cancel

Figure 12.2 Auto calculation of Q/h-relations

The Q/h relation is computed using the characteristics of the cross section located at the boundary location, and hence a cross section must exist at this location before this tool can be used. H values in the Q/h relation are extracted from the processed data of the actual cross section whereas related Q values are computed either using the critical flow or the Manning formula. If the latter is chosen the bed slope and Manning's "n" or "M" must be specified.

In case the critical flow formula is used, Q is calculated from:

$$Q(h) = A(h) \sqrt{g \frac{A(h)}{W(h)}} \quad (12.1)$$



In case of uniform flow by Manning's formula, Q is calculated from:

$$Q(h) = \text{Conv}(h) \cdot \sqrt{I} \quad (12.2)$$

where:

$Q(h)$ is the level dependent discharge

$A(h)$ is the level dependent area (from Cross section processed data)

$W(h)$ is the level dependent width (from Cross section processed data)

I is the bed slope

$\text{Conv}(h)$ is the level dependent conveyance calculated as a function of the resistance type defined in cross section, in the Raw data tab, as described below.

If the resistance type is set to Relative Resistance:

$$\text{Conv}(h) = M_{QhTool} \cdot A(h) \cdot R^{\frac{2}{3}} \quad (12.3)$$

If the resistance type is set to Manning's M:

$$\text{Conv}(h) = M(h) \cdot A(h) \cdot R^{\frac{2}{3}} \quad (12.4)$$

If the resistance type is set to Manning's n:

$$\text{Conv}(h) = \frac{1}{n(h)} \cdot A(h) \cdot R^{\frac{2}{3}} \quad (12.5)$$

If the resistance type is set to Chezy or Darcy-Weisbach:

$$\text{Conv}(h) = C(h) \cdot A(h) \cdot \sqrt{R(h)} \quad (12.6)$$

where:

M_{QhTool} is the Manning number defined in the 'Auto calculation of Q/h table' dialog

$M(h)$, $n(h)$ and $C(h)$ are the respective resistance numbers extracted from the Resistance column in the cross sections Processed data.



Runoff - AD/WQ boundary

When the boundary type is set to 'Runoff', it is required to assign a solute concentration to the runoff from a selected catchment. The options below are used to select the runoff to be used in this calculation:

Include boundary

The boundary condition is applied in the simulation only if the option is checked.

Catchment name

The catchment to be used is selected from this list. Any catchment from the 'Catchment definitions' menu may be selected.

Flow type

This option is only available when the rainfall-runoff model for the selected catchment is NAM. For these catchments, the solute concentration will apply to the selected component from the rainfall-runoff model:

- Total runoff, which is the sum of overflow flow, inter flow and base flow
- Overflow flow
- Inter flow
- Base flow
- Rainfall

For other types of rainfall-runoff models than NAM, the solute concentration will always apply to the total runoff.

12.1.2 AD / WQ boundary

Include boundary

The boundary condition is applied in the simulation only if the option is checked.

Use mixing

This option should be active for boundaries where outflows from the model area take place, and where the boundary can become an inflow during the simulation (e.g. due to tidal conditions).

When the boundary becomes an inflow, the boundary condition is adjusted according to:

$$C = C_{bf} + (C_{out} - C_{bf})e^{-t_{mix} \cdot k_{mix}} \quad (12.7)$$

Where:



C_{bf} is the concentration specified at the boundary location

C_{out} is the computed concentration at the boundary immediately before the boundary became an inflow

K_{mix} is the mixing coefficient specified for the boundary

T_{mix} is the time since the boundary became an inflow.

When outflows occur at the boundary, the boundary condition is defined as:

$$\frac{\partial^2 C}{\partial x^2} = 0 \quad (12.8)$$

Mixing coefficient

The mixing coefficient required when the option 'Use mixing' is active is specified here.

AD/WQ boundary overview table

The lower table is used to define the boundary values for each component to be described in the active boundary. Components are added or deleted from this table using the Append '+' or Delete '-' buttons above the table. For each component, the following data must be specified. Additionally, the 'Copy to selected boundaries' button may be used to add the content of the table from the active boundary to the selected boundaries.

Component

The component is selected from this list. Any component from the 'Components' menu may be selected.

Unit

Unit in which the boundary value is expressed. This field cannot be changed and is inherited from the 'Components' menu.

Boundary type

The boundary may be defined using one of the two types below:

- **Concentration:** Boundary input is specified as a concentration (mass/volume) and the simulation engine will multiply the HD inflow at the boundary location to define the total mass entering the river at the boundary location
- **Load:** Boundary input is specified as a specific load (mass) which is directly applied in the simulation engine as the load at the specific location. An eventual HD inflow source at the boundary location is hence not used to determine the mass inflow to the model.



Input type

Here the boundary can be defined as a constant value during the simulation or a time varying value.

Scale

The boundary condition value (constant or time series) will be multiplied by the specified Scale factor. This factor may be used to edit the boundary conditions without changing the input values.

Constant

The boundary value must be specified here when the Input type is set to 'Constant'.

AD file name

When the Input type is set to 'Time varying', the time series containing the boundary values must be selected here.

AD file item

This field shows the selected item in the time series, when the Input type is set to 'Time varying'.

Boundary condition buttons

Create AD/WQ Boundaries

Pressing this button will automatically create a boundary for all AD/WQ components on all existing boundaries.

For each AD/WQ boundary, the attribute 'Include boundary' is automatically set to true. The table is also filled with all components, with default parameters. Further adjustments of parameters may then be applied manually.

If AD/WQ boundaries already exist before pressing the button, a pop-up window asks whether existing AD/WQ boundaries should be overwritten or not. Pressing 'Yes' resets existing boundaries to default values, whereas pressing 'No' only inserts boundaries at locations where they don't already exist.

View all AD/WQ Boundaries

This button opens a new window which gives an overview of all existing AD/WQ boundaries, without having to navigate between each HD boundaries.

Data may be edited from this table, and sorted by clicking in the headers of the table.



12.1.3 ST Boundary

Include boundary

A boundary condition is only applied in the simulation if this option is checked. Note, that this option is not available for open boundaries, where the ST boundary condition is mandatory.

ST boundary type

Three boundary types are available:

- **Fractional sediment transport:** the load or concentration at the boundary for each sediment fraction must be specified.
- **Bed level change:** the relative change of bed level at the boundary must be specified.
- **Bed level:** the elevation of the bed at the boundary must be specified.

The ST boundary type can be controlled only for open boundaries. For sources along the river network, the ST boundary type is always a fractional sediment transport.

Input type

A *Bed level* or *Bed level change* boundary type can be set to either use or constant or time-varying value.

ST value

When the input type is set to Constant, the constant value for the bed level or bed level change is specified here.

File

When the input type is set to Time series, the time series used for the bed level or bed level change is specified here. The button to the right may be used to either browse, create, edit or plot the time series.

Item

This field shows the selected item in the time series.

Scale

The boundary condition value (constant or time series) will be multiplied by the specified Scale factor. This factor may be used to edit the boundary conditions without changing the input values, e.g. when performing a sensitivity analysis.

Fractional sediment transport table

For a *Fractional sediment transport* boundary type, the boundary condition for each fraction has to be specified in the table. For each fraction, the following data must be specified:



Fraction

The name of the fraction. Each fraction is described with its own row in the table.

Type

The fraction type (cohesive or non-cohesive) cannot be changed and is inherited from the *Sediment fractions definitions* menu.

Boundary type

The boundary condition for a cohesive fraction can be expressed using one of the two types below:

- **Concentration:** boundary input is specified as a concentration (mass/volume) and the simulation engine will multiply the HD inflow at the boundary location to define the total mass entering the river at the boundary location.
- **Load:** boundary input is specified as a specific load (mass) which is directly applied in the simulation engine as the load at the specific location. An eventual HD inflow source at the boundary location is hence not used to determine the mass inflow to the model.

A non-cohesive fraction is always expressed with a load, if only the bed load is included in the calculation for this fraction in the *Sediment fractions definitions* menu.

Input type

Here the boundary can be defined as a constant value during the simulation or a time varying value.

Scale

The boundary condition value (constant or time series) will be multiplied by the specified *Scale* factor. This factor may be used to edit the boundary conditions without changing the input values.

Constant

The boundary value must be specified here when the input type is set to *Constant*.

File name

When the input type is set to *Time varying*, the time series containing the boundary values must be selected here.

Item

This field shows the selected item in the time series, when the input type is set to *Time varying*.

Boundary condition buttons

View all ST boundaries

This button opens a new window which gives an overview of all fractions from all existing ST boundary conditions with type *Fractional sediment transport*, without having to navigate between each ST boundary. Data may be edited from this table and sorted by clicking in the headers of the table.

Copy to selected boundaries

This button may be used to copy the content of the *Fractional sediment transport* table from the active boundary to the selected boundaries.

12.2 Wind Friction

Wind friction on the water surface can be accounted for by including wind shear stress in the simulation. This is done by enabling 'Include wind shear stress' under the *Wind friction* section.

The wind shear stress is included in simulations by combining the wind scaling factor(s) with a wind field defined by wind velocity and wind direction. Wind fields and scaling factors are defined in the two separate tabs described below.

Wind field definition

The wind field definition must be specified here with two components: the wind velocity and the wind direction. The direction is expressed in degrees in clockwise direction from north,

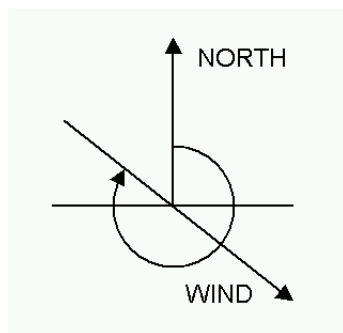


Figure 12.3 Definition of Wind field direction

Global value

A global wind field is applied throughout the network, unless local values have been defined. For both the wind velocity and wind direction, the options below are specified.



- **Input type.** Here the global value type can be set to either Constant or Time varying.
- **Value.** When the global value type is set to Constant, the constant value is specified here.
- **File.** When the global value type is set to Time varying, the time series used for the global wind field is specified here. The button to the right may be used to either browse, create, edit or plot the time series.
- **Item.** This field shows the selected item in the time series.

Distributed values

By using distributed wind field values it is possible to adjust the wind field to reflect local variations within the domain. Distributed values overwrite global values.

Distributed values are applied to specific reaches, defined by Branch name, Upstream chainage and Downstream chainage. Distributed values are added or deleted using the Append '+' or Delete '-' buttons above the table. As for global values, distributed values may have either a Constant or Time varying type.

Wind scaling factors

The wind scaling factor is a multiplication factor which is multiplied to the wind velocity. Wind scaling factors may be defined both as a global value and as distributed values at user-defined locations (chainage points).

Global value

A global wind scaling factor is applied throughout the network, unless distributed values have been defined.

Distributed values

By using distributed wind scaling factors it is possible to adjust the wind shear stress to reflect local variations within the domain. Distributed values overwrite the global value.

Distributed values are applied at specific locations defined by Branch name and Chainage. Distributed values are added or deleted using the Append '+' or Delete '-' buttons above the table. Alternatively, distributed values can be digitised in the Map view: in Map view, open the Spatial data ribbon and select 'Wind scaling factor' from the 'Type' drop-down menu (see Figure 12.4)

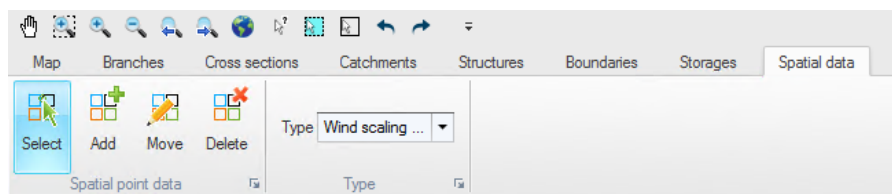


Figure 12.4 Map view ribbon for editing local Wind scaling factors

Click on the 'Add' button to activate creation of new scaling factor locations and click on the desired location in the river network for the local wind scaling factor. The location will automatically be added to the table of local values in the tabular view and a value may be entered. The 'Move' and 'Delete' buttons may also be used to edit the location on the map or to remove a distributed value.



Note: If two or more local values are defined in the same river branch, the intermediate value(s) will be calculated by linear interpolation.

12.3 Groundwater leakage

The Groundwater leakage page defines the groundwater head and leakage coefficients such that an additional loss of water from the river (to the groundwater) can be included in the simulation. Include the leakage in the simulation by activating the 'Include groundwater leakage' tick-mark on the top of the page.

Leakage coefficients may be defined by an overall global leakage coefficient or by local coefficients that overrides the global value at specific locations. Hence, if leakage is to be included only at some locations, then a global value of 0 must be defined and the local reaches where groundwater leakage takes place must be defined in the Local values table.

Groundwater head

Input type

Here you select from the drop-down menu whether the head elevation is constant or varies in time during the simulations. The respective options are:

- Constant
- Time varying

This choice also determines how the input value or files are entered.

Value

Head elevation, used for Constant input type.

For Time varying input type, a time series file must be specified. The button to the right may be used to either browse, create, edit or plot the time series.



Item

This field shows the selected item in the time series.

Leakage coefficient

Global value

The global value of the leakage coefficient is defined here.

Local values

By using local values of the leakage coefficient it is possible to adjust the leakage to reflect local variations of soil characteristics within the domain. Local values overwrite the global value.

Local values are applied at specific locations defined by Branch name and Chainage. Distributed values are added or deleted using the Append '+' or Delete '-' buttons above the table. Alternatively, distributed values can be digitised in the Map view: in Map view, open the Spatial data ribbon and select 'Groundwater leakage' from the 'Type' drop-down menu (see Figure 12.5).

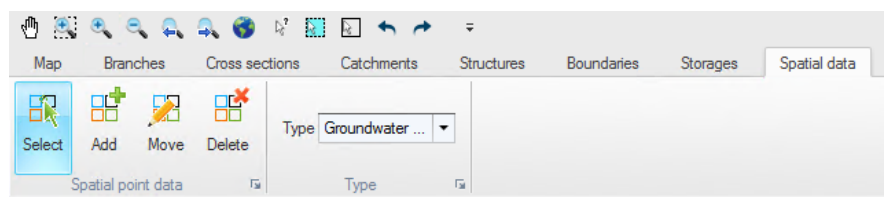


Figure 12.5 Map view ribbon for editing local Groundwater leakage

Click on the 'Add' button to activate creation of new locations of groundwater leakage coefficients and click on the desired location in the river network. The location will automatically be added to the table of local values in the tabular view and a value may be entered. The 'Move' and 'Delete' buttons may also be used to edit the location on the map or to remove a local value.



Note: If two or more local values are defined in the same river branch, the intermediate value(s) will be calculated by linear interpolation.

12.4 MIKE SHE Couplings

The *MIKE SHE Couplings* dialogue allows the coupling of all or some of the river branches included in the MIKE HYDRO River set-up to the MIKE SHE groundwater model. MIKE SHE will only exchange water with those branches that are listed in the *MIKE SHE Couplings* dialogue.

To enable coupling to MIKE SHE, tick the 'Include MIKE SHE coupling' check-box at the top of the *MIKE SHE Couplings* dialogue.



The river branches included in the coupling are those listed in the overview table at the bottom of the page. Branches are added or removed from the MIKE SHE coupling by using the Append '+' and Delete '-' buttons. The upper part of the dialogue displays the properties of the active branch in the overview table.

The properties of the MIKE SHE linkage of a specific location can be displayed in the *Property view* if the checkbox of that item is checked in the table. When several table items are checked, it is possible to unselect all of them at the same time by clicking on the Clear selection button above the overview table.

Include all branches

If this button is pressed all branches included in the MIKE HYDRO River set-up are copied to the MIKE SHE coupling page. Branches that should not be in the coupling can subsequently be deleted manually and remaining specifications completed. Thus you may have a large and complex hydraulic model, but couple only the main branches (or part of them) to MIKE SHE.

Note that the *Include all branches* feature will overwrite existing specifications.

Location

Branch name

Name of the branch where the MIKE SHE coupling should be applied. One branch can be sub-divided into several reaches. A reason for doing so could be to allow different riverbed leakage coefficients for different reaches of the river.

Upstream chainage

Chainage of the upstream extent of a MIKE SHE coupling.

Downstream chainage

Chainage of the downstream extent of a MIKE SHE coupling. If the downstream chainage is set equal to the upstream chainage, the coupling will be performed in a single point on the river branch.

River-aquifer exchange

Conductance

The conductance can be defined through three different types of exchange between surface water and aquifer (described in the technical documentation of the MIKE SHE User Manual). The available options are:

- Aquifer only
- River bed only
- Aquifer + river bed



Leakage coeff.

Leakage coefficient for the riverbed lining (see MIKE SHE documentation). The leakage coefficient is relevant only if the conductance type is either 'Aquifer + river bed' or 'River bed only'.

Linear reservoir exchange

When applying the Linear Reservoir mode for Saturated Flow in MIKE SHE, exchange between the river and the baseflow can be defined individually in coupling reaches as either a Gaining river or a Losing river. That is, only one directional exchange is possible for this option.

- **Gaining river.** In Gaining river, the river receives water from the baseflow reservoirs in MIKE SHE
- **Losing river.** In Losing river, the river loses water to the baseflow reservoirs in MIKE SHE as a function of the leakage coefficient, water depth in river, bank width and length of the coupling reach.

Overland-river exchange

When the Weir formula option has been selected in MIKE SHE for Overland-River exchange calculation in MIKE SHE, the overland spilling will be calculated using the 'Villemonthe Formula' weir equation. The parameters below, required for this equation, must be defined for each coupling reach.

Weir coefficient

This parameter refers to the weir coefficient in the Villemonthe Formula.

Weir exponential coeff

This parameter refers to the weir exponent in the Villemonthe Formula.

Min. upstream height above bank for full weir width

In the Villemonthe Formula, when upstream water depth above the weir approaches zero, the flow over the weir becomes undefined. To prevent numerical problems, the flow is reduced linearly to zero when the water depth is below the minimum upstream height threshold.

Allow overbank spilling from river to overland domain

Activate this checkbox to allow overbank spilling from the river to the overland grid cells. If left unchecked, the exchange of overland flow is only one-directional. That is, from the overland model to the river.

Min. flow area for overbank spilling

The minimum flow area threshold prevents overbank spilling when the river is nearly dry. The flow area is calculated by dividing the volume of water in a coupling reach by the length of the reach. If the calculated flow area is less than the specified threshold value, overbank spilling from the river to the overland flow is excluded.



Inundation

Flood area option

The Flood area option allows a number of cells from the MIKE SHE model to be flooded (being part of a river, lake, reservoir etc.) based on the water level from the river model. When this option is active, a river or a lake (with wide cross sections) may cause flooding of a number of cells in MIKE SHE. A correspondence / mapping is established between MIKE HYDRO h-points and individual cells in MIKE SHE. Subsequently a simple flood-mapping procedure is adopted to calculate water depths on the ground in MIKE SHE. The flood mapping procedure simply compares simulated water levels (in the corresponding h-point) with the ground level in the MIKE SHE cells. If the water level is higher than the ground level, flooding occurs.

Three options are available:

- No flooding
- Automatic
- Manual

The selected option may differ for each coupled reach. These options are described below.

No flooding

If the No flooding option is adopted, rivers are considered lines located between adjacent cells from the MIKE SHE model. No flooding can occur and overbank spilling is not possible, even if the water level in the river rises above the elevation of the adjacent cells in MIKE SHE.

River-Overland exchange is always one way with this option, namely overland to river. If this option is used, one of the three river-aquifer exchange formulations will be adopted.

Manual

The manual option allows the user to delineate the potentially flooded areas, using a T2 grid code file - the flood code file specified in MIKE SHE's user interface. If the river system considered is a very complex system with a looped or meandering network, generating a complex geometry, the best result will be obtained by creating a flood code file manually by digitizing the floodplain / lake, and using this option.

Each (potentially flooded) MIKE SHE cell is linked to the nearest MIKE HYDRO h-point on a coupling reach with the same flood code value. The flood code specified for each coupling reach must match the grid code in the flood code file. The flood code must be an integer.



Automatic

The automatic option is often useful if the geometry of rivers, lakes etc. is not too complex. This option may, for instance, be applied for wide rivers without too much meandering.

When the automatic option is chosen, MIKE SHE will automatically generate the potentially flooded areas (flood grid code map) depending on the location of the river branches and on the width and location of their cross-sections. The specified flood code is used as grid code, and the flood-mapping procedure described above is applied. Thus it is important to use unique flood codes to ensure correct mapping between MIKE HYDRO and MIKE SHE.

Flood code

As described above the flood code is used for mapping MIKE SHE cells to MIKE HYDRO h-points, and for the automatic option also for generating the flood grid codes of the coupling reach. It is important to use unique flood codes to ensure correct flood-mapping.

Bed topography

This option controls the source of the topography used in the flooded area. It should be emphasized that the flood mapping requires a good consistency between the MIKE HYDRO cross-sections and the ground levels in MIKE SHE in order to provide relevant results.

The options are:

- **Use cross sections**
When this option is used, the ground levels of the flooded cells are substituted with values directly interpolated from the MIKE HYDRO cross-sections. Notes on this option:
 - This interpolation uses an inverse-distance-weighted method, using points (elevations) on the MIKE HYDRO cross-sections as discrete input points. When the distance between MIKE HYDRO cross-sections is higher than half the cell size, extra cross sections are interpolated before the cells' ground levels are interpolated.
 - Please note that the ground levels are only interpolated inside the area delineated by the MIKE HYDRO cross-sections.
 - When the Manual option is used, the user defined flood area does not have to be identical to the area covered by the MIKE HYDRO cross sections. If the Automatic option is used, the area covered by the MIKE HYDRO cross-sections and the flood area will always be consistent, as the flood-area is generated (automatically) based on the MIKE HYDRO cross-sections.



- In principle the Use cross-sections option ensures a good consistency between MIKE SHE ground levels and MIKE HYDRO cross-sections. There will, however, often be interpolation problems related to river meandering, tributary connections, etc. where wide cross-sections of separate coupling reaches overlap. Thus it is recommended to make the initial model set-up using the Use cross-section option and then subsequently retrieve and check the resulting ground levels (using the MIKE SHE Input Retrieval tool). If needed the retrieved ground levels (T2 file) can be modified (MIKE SHE Graphical Editor) and then used as input for a new set-up, now using the Use grid data option described below.
- **Use Grid Data**

MIKE SHE grid data is used instead of MIKE HYDRO cross-sections. This option checks whether the optional bed elevation file has been specified in MIKE SHE's user interface:

 - If the bed elevation file has been specified, the ground levels of the flooded cells are substituted with values from the specified T2 file. The option is useful when the surface elevation data of the flood areas is more detailed than the regional terrain model.
 - If the bed elevation file has NOT been specified the regional MIKE SHE surface topography is also used in flood areas. As described above the specified T2 file will often be a retrieved and modified surface topography from a previous set-up with the Use cross sections option.

Bed leakage

As described in the technical documentation of MIKE SHE, the infiltration/seepage of MIKE SHE is calculated as ordinary exchange from the overland domain to the saturated or unsaturated zone, either using full contact or reduced contact with a specified leakage coefficient.

The bed leakage option tells whether the overland-groundwater exchange option and leakage coefficient specified in MIKE SHE's user interface should also be used in the actual flood area, or substituted by the corresponding river-aquifer conductance type and Leakage coefficient specified for the coupling reach.

Options are:

- **Use grid data**

The overland-groundwater exchange option and leakage coefficient specified in MIKE SHE's user interface is used. Both can be single value or distributed (T2 file).



- **Use river data**

The MIKE SHE overland-groundwater exchange option and leakage coefficient in flood areas are substituted with the corresponding river-aquifer conductance type and leakage coefficient specified for the coupling reach. Please note that the two reduced contact options (exchange types B and C) lead to the same overland-groundwater exchange option.

The substitution is made in all cells from the flood area of the coupling reach.





13 Initial Conditions

Initial conditions are required by most numerical models to define the initial state of hydrodynamic and other module specific variables at the start of simulation. Initial conditions for water levels or water depths and discharge are for example required for hydrodynamic river module simulations.

The initial conditions may be specified from three different options:

- Steady state (specific for the Hydrodynamic module)
- User-defined values for state variable
- Hotstart files (use of existing result file from a previous simulation)

Initial conditions can be defined by a combination of user defined values and Hotstart files. In case both options are selected, the prioritization is as follows:

- Priority 1: Hotstart files
- Priority 2: User-defined initial conditions

Values from the Hotstart file are used where available, but where computational points are not included in the Hotstart file, user-defined values are automatically applied.

13.1 State File

Initial conditions for all modules included in the simulation may be obtained from a state file. A state file is an output from a previous simulation, which may store results from multiple modules (Hydrodynamic, Advection-Dispersion, etc.), for a given time step.

The state file has the highest priority amongst the various types of initial conditions definitions. So, when a state file is used, all calculation points described by the state file will use the available values from this file, even if a hotstart file or a user-defined local value is defined. If no state file is used, or if the river network has been modified and some calculation points do not exist in the state file, the initial conditions defined in the other menus, respectively for HD, AD, or ST conditions, are used for the relevant points.

To use a state file for initial conditions, tick the *Use state file* check box at the top of the dialogue.

It is required to provide a path to the folder containing the state files created from a previous simulation, along with a date and time controlling which time step will be used as initial conditions. The appropriate state file from the selected folder, with the date and time the closest to the specified instant, will be selected automatically during the simulation.



Use simulation start time

If Use simulation start time is active, the initial condition will automatically be obtained from the available state file with the date and time the closest to the simulation start date, as specified in the Simulation period page. If inactive, a user-defined date and time is used.

Date and Time

The date and time at which the initial conditions are loaded from the state files, when Use simulation start time is inactive.

13.2 HD Initial Conditions

Global values

The required type of Initial Condition to apply in the HD simulation is specified in the Global values section. Options available are:

- Steady State
- User defined

When 'Steady State' is selected, initial conditions will be based on a steady state calculation for all river branches in the model, except for river reaches and locations where local, user-defined values are defined.

When 'User defined' is selected, global values for initial conditions are applied as default value for computational points which are not included in eventual Hotstart files, and for which no local values have been defined.

Level type

Water level may be specified either as Water level or Water depth.

Discharge type

Discharge may be specified either as User defined flow values or to be calculated as Natural flow.

Natural flow will calculate the discharge according to a manning formula

$$M = R^{2/3} \cdot A \cdot M \cdot \sqrt{I} \quad (13.1)$$

Where:

R: Hydraulic radius

A: Flow Area

M: Manning's number (*)

I: Water surface slope



(*): Natural flow calculation can utilise Resistance formulations from both Manning's M or Manning's n depending on the selected option in the '*Hydrodynamic Parameters* -> *Bed resistance*' dialogue.

Local values

Local values are applied at specific locations defined by Branch name and Chainage.

Local HD initial conditions may be added to river branches by digitised in Map view: In *Map view*, open the *Spatial data* ribbon and select '*HD Initial condition*' from the '*Type*' drop-down menu.

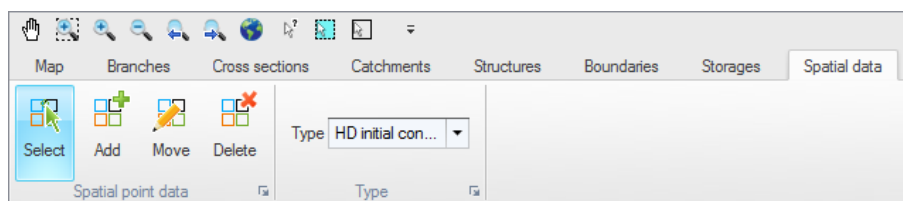


Figure 13.1 Ribbon bar for adding and editing local Initial condition locations

To insert a new location, click the '*Add*' button and click on the desired location of the local initial condition value. The location will automatically be added to the '*Local branch values*' table in the dialogue and the local HD initial conditions can be manually defined in the table. Initial conditions for Local values offer same options as described above for Global values.



Note: If two or more local values are defined within the same river reach, the intermediate values will be calculated by linear interpolation.

If only one local value location is defined in a river, then this value will be applied at the specific location and all other values in the river will be assigned with Global values.

Local initial water level / depth for storages can also be added, with the '*Local storage values*' table. These local initial conditions to storages cannot be added from the map.

Hotstart files

Hotstart files can be used to define initial conditions when the Global option for initial conditions has been defined as '*User Defined*'.

The HD initial conditions may be loaded from an existing result file by enabling '*Hotstart*' and clicking on the '*Append*' button. This will add a new row to the list of Hotstart files. Click on the '*Browse*' button and select the required hotstart file.



Note: It is possible to define multiple hotstart files covering different or overlapping areas. If a part of the network is covered by more than one hotstart file the first hotstart file will be used. If a part of the network is not covered by the specified hotstart files the global / local values will apply.

If the network in a hotstart file differs from the current network e.g. the number of gridpoints in a branch have changed then an interpolation is used.

Use simulation start time

If 'Use simulation start time' is enabled, the initial condition will automatically be extracted from the Hotstart file using the simulation start date and time as specified in the 'Simulation Period' page

Hotstart Date and Time

The date and time at which the initial conditions are loaded from the hotstart file. If 'Use simulation start time' is enabled the hotstart date and time will be taken as the simulation start.

13.3 AD Initial Conditions

Initial component concentrations are defined on this property page. If an initial concentration is not specified a default value of zero will be applied throughout the model. Global and local values of initial concentrations can be specified for each component. Local values are specified by entering the river branch name, chainage and concentration in the local values table. Initial concentrations are not used if the AD simulation is started with a hotstart file.

Global values

Component

Here the component in question is selected. It is possible to choose between the components defined in the Advection-Dispersion parameters Components property page by selecting from the drop-down selection box present in the 'Component' column.

Initial value

The initial value for the specific component is defined here. The initial value units are those defined in the Advection-Dispersion parameters Components property page for the advection-dispersion component.

Local values

MIKE HYDRO will use the initial values specified under global values except for those locations where local values have been specified. To add or remove locations with local initial values use the Append and Delete buttons.

Branch name

The name of the river branch with the local initial value.



Component

Here the component with a local initial value is selected. It is possible to choose between the components defined in the Advection-Dispersion parameters Components property page by selecting from the drop-down selection box present in the 'Component' column.

Chainage

The chainage in the river branch with the local value.

Initial value

The local initial value for the specific component is defined here. The initial value units are those defined in the Advection-Dispersion parameters Components property page for the Advection-Dispersion component.

13.4 ST Initial Conditions

13.4.1 Concentration

A global initial concentration must be specified for each cohesive fraction and is applied throughout all calculation points in the river network, unless local values have been defined. Local values may be applied at specific locations defined by Branch name and Chainage, and for a specified fraction.

For non-cohesive fractions with suspended load, the initial concentration always equals equilibrium: this is calculated automatically during the simulation and doesn't require any input in MIKE HYDRO.

13.4.2 Thickness and distribution

Global values

Global values must be specified and applied throughout all calculation points in the river network, unless local values have been defined.

Active layer thickness

The initial thickness of the active layer.

Passive layer thickness

The initial thickness of the passive layer. This field may be set to zero: in this case, the active layer will be set to the *Active layer target thickness* by exchanging with the passive layer.

Percentage, active layer

The initial percentage of each fraction within the active layer. The sum of the initial percentages should equal 100%, otherwise the percentage values will be scaled so that the sum equal 100%.



Percentage, passive layer

The initial percentage of each fraction within the passive layer. The sum of the initial percentages should equal 100%, otherwise the percentage values will be scaled so that the sum equal 100%.

Local values

Local values of initial thickness, for both active and passive layers, may be applied at specific locations defined by Branch name and Chainage, in the left table. The secondary table on the right-hand side contains the initial fractions distributions for the current local value in the left table.



14 Data Assimilation

Data assimilation enable processing of model results during simulation with options for model state updating using measurements as input as well as methods for calculating uncertainty predictions during a simulation.

The data assimilation method required is defined in the 'General parameters' page and method specific parameters are defined in method specific data assimilation dialogues within the Data Assimilation group.

14.1 General parameters

General parameter definition include the selection of data assimilation simulation mode as well as definition of a few general, mode-specific parameters.

Mode selection

Three data assimilation modes are available:

- **Updating with weighting function:** this mode performs a state updating of model parameters applying user-defined weighting functions to estimate the updates applied to model simulation result during simulation up to the Time of Forecast. Additionally, an 'Error forecast' correction option may be applied after Time of Forecast, if selected.
- **Updating with Kalman filter:** this mode updates the model with the ensemble Kalman filter based on Monte Carlo simulation techniques.
- **Uncertainty prediction:** uses Monte Carlo simulation to propagate uncertainties in the boundary conditions, and assess uncertainties in the simulated model outputs.

Basic parameters

First updating time step. The time step number at which the model starts to be updated. Parameter is only available for updating using either the weighting function or the Kalman filter.

A value of 0 corresponds to the first time step of the simulation and e.g. a value of 100 corresponds to the updating only starting at simulation time step number 100 and forward. Starting the update later than time step 0 may ensure that the stochastic process is sufficiently evolved to give good estimates of the uncertainty before the first update, thereby getting rid of potential instabilities occurring at the beginning of the simulation.

Ensemble size. Number of simultaneous runs that are to be carried out to evaluate the statistical properties needed for the uncertainty assessment output and also for determining the updating parameters applied in the Kalman filter. Parameter is hence only available for Kalman filter or uncertainty prediction simulation mode.



The quality of the statistical estimates are strongly dependent on the ensemble size: the larger the ensemble size the higher the confidence in the results. On the other hand, the ensemble size has a linear effect on the run time, that is, when the ensemble size is doubled the run-time is also doubled. Recommended values are 50-200.

If only reliable estimates of standard deviations are of importance, a smaller value may be chosen. When producing confidence intervals an estimate of the full uncertainty distribution is needed, thus an increase in the ensemble size is recommended for such cases.

Forecast

Time of forecast. Date and time at which the model switches to forecasting mode. Parameter is available when performing updating either with the Weighting function or the Kalman filter. Model state updating is only performed till the Time of Forecast.

Forecast mode. This option is only available for the Kalman filter option and gives a choice of applying deterministic or stochastic forecasting. At time of forecast, 'Deterministic' forecasting runs a single ensemble member based on the ensemble mean state at time of forecast. The 'Stochastic' type continues evaluating all ensembles after time of forecast, applying boundary perturbations similarly to an uncertainty prediction run.

14.2 Observations definitions

The Observations definitions menu is present data assimilation mode is set to updating with either the Weighting function or the Kalman filter method.

14.2.1 Update parameters

The update parameters page defines model locations where model updating is to be done. Additionally, specific parameters are to be defined in this page is the Weighting updating mode has been selected.

Update definitions (i.e. updated points in the model) are added or deleted using the Append '+' or Delete '-' buttons above the overview table at the bottom of the dialog.

Note that update points can also be added, edited and deleted from the map, using the 'Updating' type of data in the *Spatial data* ribbon in *Map view* (see Figure 14.1).

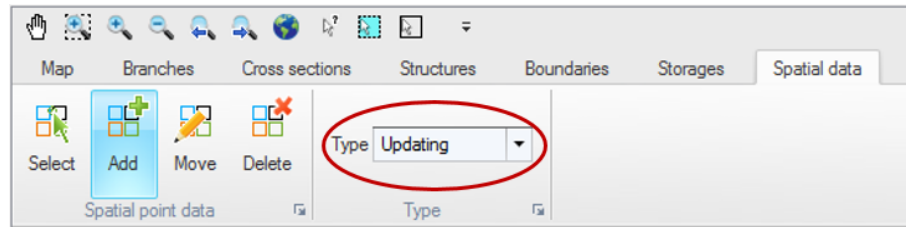


Figure 14.1 Map view ribbon for inserting and editing updates

Each update definition represents a location where observed data are available and where the model is going to be updated. To achieve this, the parameters below have to be specified.

Location

ID. String identification of the update definition.

Branch name. The name of the branch where observation data is located. It can be chosen from the drop down list.

Chainage. Chainage within the branch where the observed data are located. The chainage does not have to coincide with the location of a calculation point since MIKE HYDRO uses a linear interpolation for determining the simulated value at the appropriate measurement location.

Updated item. The type of variable on which the update is performed. Depending on the modules included in the simulation (in the Modules page), the type can be either a water level, a discharge or an AD/WQ component.

AD / WQ component. The component to be updated must be selected from the list of components either defined in the Advection-Dispersion module or derived from the list of state variables from the Water Quality module.

Observed data

Time series file. The time series file containing the observed data. File selection button '...' may be used to either browse, create, edit or plot the time series.

Item. This field will present the name of the item selected in the time series.

Weighting function parameters

The weighting function parameters are visible when data assimilation mode is 'updating with the weighting function'.

A weighting function definition must be specified for each update definition with the main purpose of defining how model corrections at the measurement location are distributed to neighbouring calculation points in the river network.



Distribution type. Three different types of weighting functions are available:

- **Constant:** in this case the error correction is distributed evenly over the grid points between the specified upstream and downstream chainages.
- **Triangular:** in this case the error correction is linearly decreasing from a maximum correction at the measurement location (specified in the 'Location' part) to 0 at the upstream and downstream chainages.
- **Mixed exponential:** in this case the error correction is decreasing according to an exponential function from a maximum correction at the measurement location to the upstream and downstream chainages, where the correction is about 0.01 times the correction at the measurement location.

Amplitude. The amplitude specifies the fraction of the observed error at the measurement location that should be applied as error correction at that point. The amplitude should reflect the confidence in the observation as compared to the model forecast. That is, if the amplitude equals one, the measurement is assumed to be perfect, whereas for smaller amplitudes, less emphasis is put on the measurement as compared to the model forecast.

Upstream chainage. It represents the upstream extent of the reach where the weighting function is applied (in the branch where the measurement point is located).

Downstream chainage. It represents the downstream extent of the reach where the weighting function is applied (in the branch where the measurement point is located).

Include connected branches. If this option is selected, the correction is also applied to tributaries (if any) which are connected to the main river between the specified upstream and downstream chainages. The correction is applied within the same extent and with the same amplitude and distribution as for the branch where the measurement is located.

Soft start. Number of time steps used to fade up the correction initially, starting from zero correction at the first updated time step up to full correction after the fade-up period. This function helps avoiding model instabilities since the error correction value is applied gradually rather than abruptly.

Error forecast

The error forecast parameters are visible only when performing updating with the weighting function. The error forecast definition include two parameters:

Include. The weighting function update procedure up to Time of Forecast may be combined with error forecasting at updating points, in the forecast period (i.e. after the time of forecast). This option is applied when the 'Include' box is checked.



In that case, an equation defining how the calculated error function is forecasted over time must be selected in the drop-down list. The error function is derived during simulation in the updating period (up to Time of Forecast). Forecast Equations are defined in the 'Equation' tab.

The forecasted error correction is also distributed to the neighbouring points. Thus, by applying error forecasting, the model is updated also in the forecast period, with respect to the selected equation.

Standard deviation

A Standard deviation parameters group box is visible when performing updating with the Kalman filter only. The standard deviation must be specified for each update definition.

Type. The standard deviation may be of three different types:

- **Constant.** The user specifies a constant value.
- **Relative.** The standard deviation is taken as a relative value of the measurement. Note that the percentage is applied to the absolute value, when the updated item is a discharge or component. When updating on water level, the percentage is interpreted as being with respect to the water depth.
- **Time varying.** The standard deviation may vary with time with the variation defined in a time series file.

Value. The value used as constant or relative standard deviation.

File. The time series containing the time varying standard deviation. The button to the right may be used to either browse, create, edit or plot the time series.

Item. This field shows the name of the item selected in the time series.

Apply lower limit. When applying a relative or time varying standard deviation, the user may apply a lower bound on the standard deviation, by activating this option. The lower limit must then be specified in the Value field to the right.

Apply upper limit. When applying a relative or time varying standard deviation, the user may apply an upper bound on the standard deviation, by activating this option. The upper limit must then be specified in the Value field to the right.



Important: When updating is applied, the model state is updated using observed data up to the time of forecast. After Time of Forecast, forecasting is performed either using the error prediction model (if an equation has been defined), or for kalman filter updating either using a stochastic or deterministic forecast.



This functionality therefore assumes that observed data are available up to the time of forecast. In case observed data are missing before the time of forecast, the simulation will still run, but the following rules will apply:

- If observed data are missing at the beginning of the simulation, the updating will only start at the time when observed data become available
- If the model is being updated and if some observed data are missing before the time of forecast, the error prediction model will be applied
- If some observed data are available before the time of forecast but not sufficiently to compute the error prediction, or if no observed data at all is available before the time of forecast, then the update location is fully disabled during the simulation. That is, no update or error prediction is performed at this location.

14.2.2 Equation

The Equation tab is only visible when performing updating with weighting function.

Equations are used to specify how error correction functions derived during the update period (simulation time up to Time of Forecast) is applied during the forecast period.

New Equations are added or deleted using the Append '+' or Delete '-' buttons above the overview table located in the bottom of the dialog.

Equation parameter definitions

Parameters to be used in the Equation definition are specified here.

Parameter name. Identification name of an equation parameter.

Type. Equation variables can be defined as one out of three types:

- **Constant value:** the parameter is assigned a constant value.
- **Time series:** the parameter is assigned a time-varying value specified in an external time series.
- **State variable:** the parameter is a state variable, i.e. a computed water level or discharge at a grid point in the river network.

Depending on the type definition selection for the equation parameter, specific parameters must be defined



Specific parameters for '**Constant value**' type include:

- **Estimated.** For a parameter defined as a 'Constant' type, an automatic parameter estimation can be performed to estimate the value from the computed time-varying error. Select this option to enable the parameter estimation.
- **Value.** Value assigned to the parameter. Must be defined for Constant type parameters for which the 'Estimated' option is not selected.
- **Minimum value.** Lower bound of the parameter when automatic parameter estimation is applied.
- **Maximum value.** Upper bound of the parameter when automatic parameter estimation is applied.

Specific parameters for '**Timeseries**' type include:

- **Time series file.** The time series containing the values, for time-varying parameters. The button to the right may be used to either browse, create, edit or plot the time series.
- **Item.** This field shows the name of the item selected in the time series.

Specific parameters for '**State variable**' type include:

- **Branch name.** The branch name where the state variable is located.
- **Chainage.** The chainage at which the state variable is located.
- **Item.** State variable type. Depending on the modules included in the simulation (in the Modules page), it can either be water level, discharge, or AD/WQ component.
- **AD / WQ component.** The component from the Advection-Dispersion module or the state variable from the Water Quality module, when the updated item is set to *AD/WQ component*.

Equation definitions

Equations are specified here, using the parameters below.

Equation name. Identification name of the equation. This name is shown in the drop-down list of available equations, when selecting an error forecast model in the 'Update parameters' tab.

Equation. The equation is defined here. The 'Edit' button gives access to an equation editor, from which it is possible to select the available parameters as well as a number of operators and functions.

A 'Validate' button evaluates whether the syntax of the equation is correct or not.



As an example, suppose the error forecast model is to be expressed as a function of the variables X1, X2 and X3 as 5 times variable X1 minus the square of variable X2 plus 2 times the natural logarithm of X3, the Equation field should be written:

$$5*[X1] - \text{Power}([X2], 2) + 2*\text{Log}([X3])$$

Where:

X1, X2 and X3 are defined equation parameters.

Another example: assuming that the error forecast model is based on a second order auto regressive model, the Equation field should be written:

$$[A] * E(-1) + [B] * E(-2)$$

Where:

A and B are the auto regressive equation parameters.

E(L) is Error function with time lag; L (L<0). That is, E(-1) is the error function value at the previous time step.

Estimation period. For parameters defined as 'constant values', automatic parameter estimations can be applied based on the record of computed errors. The period of the record to be used for the parameter estimation can be specified relative to the time of forecast.

This option allows parameters of the error forecast models to be updated continuously, and hence the error forecast models to adapt from one forecast to the next to the prevailing conditions at the time of forecast. For instance, the error forecast models can be adapted to the structural differences in the model errors that are often seen for different flow regimes.



Note: If the amount of observed data available before the time of forecast is not enough to compute the error prediction, the updating location may be disabled. Be aware that computing the error on a long period will therefore increase the risk that the update is disabled, when only a limited set of observed data is available.

14.3 Input perturbations

The Input perturbations page is visible only when performing updating with Kalman filter or uncertainty prediction.



14.3.1 Standard deviation

The dialogue enables definitions of standard deviation specifications, which are used in the specific perturbation definitions at boundary locations in the model.

Standard deviations are added or deleted using the Append '+' or Delete '-' buttons above the overview table.

Standard deviation

ID. Identification name of the standard deviation.

Data type. The data type to which the standard deviation may be applied. Only the standard deviations with the appropriate data type can be selected for a given perturbation, depending on the data type the perturbation is applied to.

Type. The standard deviation may be of three different types:

- **Constant.** The user specifies a constant value.
- **Relative.** The standard deviation is taken as a relative value of the boundary condition. Note that the percentage is applied to the absolute value, when the data type is discharge, wind velocity, wind direction, precipitation, temperature, salinity or concentration. When updating on water level, the percentage is interpreted as being with respect to the water depth.
- **Time varying.** The standard deviation may vary with time with the variation defined in a time series file.

Value. The value used as constant or relative standard deviation.

File. The time series containing the time varying standard deviation. The button to the right may be used to either browse, create, edit or plot the time series.

Item. This field shows the name of the item selected in the time series.

Time constant before TOF. The time constant should be interpreted as the time it takes for the correction to drop to half the initial magnitude (exponential decay). The errors applied at the boundaries may be described through a first order auto regressive process:

$$\xi_{n+1} = \phi \xi_n + \varepsilon \quad (14.1)$$

where

ξ_n the error at time step n

ϕ the regression coefficient

ϵ white noise

The regression coefficient should be interpreted as the 'memory' of the model error. To ensure that this 'memory' is independent of the time step used the user is required to specify a time constant instead. The relation between the regression coefficient and the time constant is:

$$\phi = \exp\left(-\frac{\Delta t \ln 2}{T_{1/2}}\right) \quad (14.2)$$

where

Δt the simulation time step

\ln the logarithm with base e.

$T_{1/2}$ the time constant TC.

The numerical value of the regression coefficient must be less than unity to ensure that the variance of the model error is limited. A negative time constant results in a regression coefficient which is greater than unity.

The model errors being described through the auto regressive process are used in the forecast period to generate errors to be added to the boundaries. If the forecast is deterministic, the model errors are described through the auto regressive process without adding white noise. The model errors fade out according to an exponential decay.

Time constant after TOF. See description above for Time constant before TOF.

Apply lower limit. When applying a relative or time varying standard deviation, the user may apply a lower bound on the standard deviation, by activating this option. The lower limit must then be specified in the Value field to the right.

Apply upper limit. When applying a relative or time varying standard deviation, the user may apply an upper bound on the standard deviation, by activating this option. The upper limit must then be specified in the Value field to the right.

14.3.2 Perturbations parameters

Perturbation parameters are added or deleted using the Append '+' or Delete '-' buttons above the overview table.



Perturbation parameters

ID. Identification name of the perturbation.

Apply perturbation to. The type of boundary item to which the perturbation is applied. Depending on the modules included in the simulation (in the Modules page), the types may be:

- Water level
- Discharge
- Q/h relation
- Global wind field
- Distributed wind velocity
- Distributed wind direction
- Input to catchment
- Output from catchment
- AD/WQ component

AD / WQ component. The component from the Advection-Dispersion module or the state variable from the Water Quality module, when the perturbation is applied to AD/WQ component.

Item. Depending on the type of boundary item to which the perturbation is applied, the selected item can be:

- Choice between wind velocity or wind direction, when the perturbation is applied to the global wind field
- Choice between rainfall, evaporation or temperature, when the perturbation is applied to an input to catchment
- Choice between runoff or net precipitation, when the perturbation is applied to an output from catchment

Boundary ID. The ID of the standard boundary condition, or the ID of the distributed wind velocity or direction, to which the perturbation is applied. Boundary ID's are extracted from the list of defined boundaries and it is therefore required to defined boundary conditions prior to defining perturbations.

Catchment name. The name of the catchment to which the perturbation is applied, when it's applied to an input to catchment or output from catchment.

Standard deviation. The standard deviation which is applied for this perturbation. This list is populated with items specified in the Standard deviation tab, and filtered on Data type. Note that when the perturbation is applied to a Q/h relation, the corresponding standard deviation must have a data Water level.



Update in rainfall-runoff model. Activating this option will also update the rainfall-runoff simulation.

14.4 HD additional results

Data assimilation

The data assimilation group on the additional results page defines which additional output items will be saved to the additional result file from an Uncertainty assessment analysis or a Kalman filter update simulation.

The following outputs may be selected:

- Standard deviation
- Correction (only available when updating with Kalman filter)
- Gain (only available when updating with Kalman filter)
- Confidence intervals

Confidence interval outputs require specification of a percentage value for each confidence interval. Percentage values are defined in the table appearing once the 'Confidence intervals' check box is activated. Press the '+' button to insert lines in the table and the '-' to delete lines that are not needed.

Additional outputs from Data Assimilation are stored in a separate result file with 'HDStat' included in the result file name.

14.5 AD additional results

Data assimilation

The data assimilation group on the additional results page defines which additional output items will be saved to the additional result file from an Uncertainty assessment analysis or a Kalman filter update simulation.

The following outputs may be selected:

- Standard deviation
- Correction (only available when updating with Kalman filter)
- Gain (only available when updating with Kalman filter)
- Confidence intervals

Confidence interval outputs require specification of a percentage value for each confidence interval. Percentage values are defined in the table appearing once the 'Confidence intervals' check box is activated. Press the '+' button to insert lines in the table and the '-' to delete lines that are not needed.

Additional outputs from Data Assimilation are stored in a separate result file with 'ADStat' included in the result file name.







15 Result Specifications

MIKE HYDRO simulations creates results from the selected type of model executed. Results are saved in result files of a specific type depending on the type of simulation.

The *Results specifications* dialogue enables the definition of location and names of the result files created during the simulations.

MIKE HYDRO suggests a default location and file names for simulation results, and it is therefore not strictly required to edit the result specifications by the user prior to running a simulation.

However, the *Results specifications* dialogue also enables the user to define a user-specified location as well as file names for results produced from the MIKE HYDRO simulation.

15.1 Standard Results

Simulation Results are stored in result files and the location as well as file names may be defined either using default definitions from MIKE HYDRO or using user defined folders and file names.

Overall, three settings can be made:

- Definition of Result folder path
- Definition of Result file names
- Definition of storing frequency of results


Details on result folder path specification

The 'Use default result folder' checkbox will determine whether you will use the default defined result folder path or wish to define a result folder path manually.

The default suggested result folder path will appear only after the MIKE HYDRO file has been saved initially. Default folder path is then created as a sub-folder to the folder where the .mhydro file is located. The name of the sub-folder will be a mix of the mhydro file folder name and path as indicated in the following example.



Example: A MIKE HYDRO setup file with the name 'WB-sc1.mhydro' has been saved to the location; 'C:\projects\WB-CC\Models'. In this case, the proposed default folder will be:
'C:\projects\WB-CC\Models\WB-sc1.mhydro - Result Files'

To make a user-defined file path definition it is required to deactivate the checkbox and then use the  button to browse for required folder. Press



'OK' when the correct folder has been located and result files will then be saved to this specific folder during simulation.

Details on result file name specifications

The 'Use default names' checkbox defines whether the default proposed file names or user-defined file names for results are used.

The default proposed result file name is created from the user-specified string in the 'Title' text-field in the 'Simulation specifications -> Description' page of the tree view, and the default result file name can hence be adjusted by editing the 'Title' definition of the project file.

Additionally, the specific type of simulation is included in the default file name to distinguish between multiple result files created in one simulation (see example below)



Example: The user has defined the following 'Title' in the mhydro file: 'CC scenario 1-A'. This will result in the following default result file names:

River module result file names:

- HD results: 'HD_CC scenario 1-A.res1d'

Result file-types for the MIKE HYDRO modules is:

- River module: '.res1d'



Note that simulation results with a title already used will overwrite the existing results unless a new result folder/file has been specified.

To make a user-defined result file name definition, simply deactivate the checkbox specifying the requested result file name in the file name edit field.

Storing frequency definition

River module simulations allow the user to define a frequency for results stored during simulation.

The storing frequency is typically of concern when a simulation includes a large number of time steps which are not all relevant to present and analyse in a post-processing session following the simulation.

To limit the size of the result files the user can specify an interval for saving of results to the result files. The storing frequency may be specified either as the number of time step intervals between each saving of the results or as a specific time. The latter, however, demands that the specified storing time frequency is a multiple of the time step.



Note: When the Advection-Dispersion simulation is decoupled from the Hydrodynamic simulation, the reference time step used for the storing fre-



quency of the AD result file is the time step of the input Hydrodynamic result file. For example, if this Hydrodynamic result file stores one time step every hour, and if the AD storing frequency is set to 2 time steps, it will save AD results every 2 hours.

15.2 HD Additional Results

A number of simulated parameters can be selected for storage in an additional HD output result file (with the file name extension '.res1d').

The parameters are saved for each save step at h/Q points of the river system, depending on each parameter. Time series and longitudinal profiles of the parameters can be viewed in the same way as normal MIKE 1D result files.

Hydraulic variables

Following hydraulic variables can be stored in the additional HD output file:

Velocity

Velocities are calculated as the discharge divided by the flow area.

Discharge

The discharge calculated at h-points is a weighting of up- and downstream discharges calculated at Q-points.

Flow area.

The area of flow in the cross section. At computational h-points where no cross section is present the area is linearly interpolated from upstream and downstream h-points with cross section.

Flow width

The channel width at the free surface level.

Radius

The radius (resistance or hydraulic radius, according to the selected option for each cross section).

Resistance number

The cross-sectional resistance (resistance number multiplied by the resistance factor).

Conveyance

The conveyance.



Froude number

The Froude number is defined as:

$$F = \frac{Q}{A \sqrt{g \frac{A}{b_s}}} \quad (15.1)$$

Where F is the Froude number, Q the discharge, A the flow area, g the acceleration due to gravity and b_s the flow width.

Flooded Area

At h-grid points: The flooded area of the water surface between two neighbouring Q-points. This area is equal to the flow width at the h-point multiplied by the distance between the closest upstream and downstream Q-points.

Total: The total surface water area for the river system.

Volume

At H-grid points: water volume calculated around the H-grid points.

Total: total water volume for the river system.

Simulation

Time step

The time step applied during simulation can also be stored. This is in particular useful when *Adaptive time step* has been applied as the output will then present as a timeseries which time step has been applied throughout the simulation.

Derived variables

Mass Error

this will save four items per calculation points; inflow volume, outflow volume, water volume change and the mass error, where the inflow and outflow volumes are volume entering or leaving a calculation box. The mass error is defined as the difference between the volume calculated in the model and the true volume based on a step-wise integration of processed data for the actual h-point. At nodal points with more than two connections the mass error is distributed uniformly between each connection.

Total: The total mass error for the river system.

Accumulated Mass Error

The sum of the 'Mass error' in time and space. Generally, the mass error can be reduced by increasing the number of iterations per time step, reducing the time step, and or by increasing the resolution of the cross-sections.

Total: total accumulated mass error for the river system.



NOTE! Some cross-sections may cause mass-balance problems due to abrupt and large changes in cross section profile definition. These problematic cross-sections can be detected by selecting the mass error item calculated for each grid point.

Energy level

Energy level at water level points.

Energy level slope

Energy level slope at discharge points.

Bed shear stress

The bed shear stress τ at water level points given as

$$\tau = \rho g R \frac{V^2}{M^2 R^{4/3}} \quad (15.2)$$

where ρ is fluid density, g the acceleration of gravity, M the Manning number, R is the radius (approximated equal to the flow depth, h) and V is the flow velocity.

Structures

Some variables at grid points where structures are located can also be stored in the additional output file.

Flow Velocity

Velocity in each individual structure. Velocities are calculated as the discharge through the structure divided by the flow area

Discharge

Discharge in each individual structure

Flow area

Flow area in each individual structure



Note: when multiple structures are defined at the same location, results are presented for each of these. The additional results are therefore useful to analyse e.g. the flow distribution between the different structures.

Note: When a gate is modelled, the computed gate level is automatically saved along with the active control rule number. When a dambreak is modelled, the results in the breach are automatically saved in a text file named 'DAMBREAK_<Dambreak ID>.txt'.

Control

Variables

All user-defined variables specified for the Control module will be saved.



15.3 AD Additional Results

The AD additional results page contains check boxes which can be used to store internal model parameters in an additional result file automatically created (filename is: '<AD filename>ADAdd.res1d').

Mass

The mass in the system. Given in the units specified on the 'Components' property page. Total and grid points values can be selected.

Mass error

The mass balance is given in o/oo (per thousands). Total and total accumulated as well as grid points and grid points accumulated values can be selected.

1. order decay

The 1st order decay is given in the units specified in the 'Components' property page, per second. Total and total accumulated as well as grid points and grid points accumulated values can be selected.

15.4 ST Additional Results

The additional items from the sediment transport simulation to be saved in the result files are selected here. Some items may be irrelevant for some simulations, for instance for some layers layouts, and may therefore not be saved even when selected.

Note that the ST numerical scheme is different from the staggered HD numerical scheme and results from ST simulation is therefore only available at water level calculation points in the HD model.

General

Bed level change

Bed level change.

Total shear stress

Total shear stress.

Skin shear stress

Skin shear stress.

d50

d50 grain diameter.

d90

d90 grain diameter.



Geometric mean diameter

Geometric mean diameter.

Integrated over fractions

The following items provide the summed values from all fractions.

Bed load

Total bed load for all fractions.

Suspended load

Total suspended load fractions.

Total load

Total load (sum of Bed load and Suspended load).

Concentration

Sediment concentration.

Total layer thickness

Total layer thickness (sum of Active and Passive layers).

Layer thicknesses (all layers)

Active layer thickness and Passive layer thickness saved separately.

Fractional

The following items provide results for each fraction separately.

Fractional thickness

Fractional thickness for both active and passive layers.

Fractional mass

Fractional mass for both active and passive layers.

Active mass

Fractional mass for the active layer only.

Passive mass

Fractional mass for the passive layer only.

Surface distribution

Surface distribution for active and passive layers.

Sediment concentration

Sediment concentration for active and passive layers.

Bed load

Fractional Bed load for active and passive layers.



Suspended load

Fractional Suspended load for active and passive layers.

Total load

Total load (bed + suspended load) for active and passive layers.

Substrate (each layer and fraction)

The following items provide results for each fraction and each layer separately.

Fractional thickness for each substrate layer

Fractional thickness.

Fractional mass for each substrate layer

Fractional Mass.

15.5 State Files

This page may be used to produce state files. A state file stores results from all modules active in the simulation (Hydrodynamic, Advection-Dispersion, etc.), for a given time step. It is designed to be used as initial condition afterwards, for another simulation.

To save state files, tick the *Save state files* check box at the top of the dialogue.

Overall, three settings can be made:

- Definition of folder path
- Definition of file names
- Definition of storing frequency of results

Details on folder path specification

The *Use default state files folder* check box will determine whether you will use the default folder path or wish to define a folder path manually.

The default suggested folder path will appear only after the MIKE HYDRO file has been saved initially. Default folder path is then created as a sub-folder to the folder where the .mhydro file is located.

To make a user-defined file path it is required to deactivate the check box and then use the button to browse for required folder. Press *OK* when the correct folder has been located and state files will then be saved to this specific folder during simulation.



Details on file name specifications

The *Use default names* check box controls whether the default proposed file names or user-defined file names are used for the state files.

The default proposed file name is created from the user-specified *Title* text field in the *Description* page of the tree view, and the default file name can hence be adjusted by editing this *Title*.

To apply a user-defined file name, simply deactivate the check box and specify the requested file name.

Storing frequency definition

The storing frequency controls how often a state file is created. This is typically of concern when a simulation includes a large number of time steps because saving too many time steps would lead to a big amount of data in the state files folder.

To limit the size of this folder, it is possible to specify an interval for saving of state files. The storing frequency may be specified either as a number of time steps between each saving of a file, or as a specific time.

15.6 Decoupling

This page may be used to produce special hydrodynamic result files, to be used as input for a decoupled Advection-Dispersion simulation. The created special hydrodynamic result file stores the simulated water level and the average discharge per saved time step.

To save a result file for decoupling, tick the 'Save HD result file for decoupled simulation' check box at the top of the dialogue.

Overall, three settings can be made:


- Definition of folder path
- Definition of file names
- Definition of storing frequency of results

Details on folder path specification

The Use default result folder check box will determine whether you will use the default folder path or wish to define a folder path manually.

The default suggested folder path will appear only after the MIKE HYDRO file has been saved initially. Default folder path is then created as a sub-folder to the folder where the .mhydro file is located.



To make a user-defined file path it is required to deactivate the check box and then use the  button to browse for required folder. Press OK when the correct folder has been located and the result file will then be saved to this specific folder during simulation.

Details on file name specifications

The Use default names check box controls whether the default proposed file name or user-defined file name are used for the result file.

The default proposed file name is created from the user-specified Title text field in the Description page of the tree view, and the default file name can hence be adjusted by editing this Title.

To apply a user-defined file name, simply deactivate the check box and specify the requested file name.

Storing frequency definition

The storing frequency controls how often a result file is created. This is typically of concern when a simulation includes a large number of time steps because saving too many time steps would lead to a big amount of data in the result file. The frequency should however remain high enough, so that the Advection-Dispersion simulation can later correctly take the dynamic of the flow into account.

15.7 Maps

The 'Maps' page may be used to produce two dimensional maps based on the one-dimensional simulations from MIKE HYDRO River. The maps are saved in dfs2 files (rectangular grids) and constructed through interpolation in space of the grid point results. Thus the maps constructed in this way should be viewed as a two dimensional interpretation of results from a one -dimensional model.

To save a map, tik the 'Map generation' check-box at the top of the 'Maps' dialogue

The results are mapped only within the extent of cross sections as illustrated in Figure 15.1.

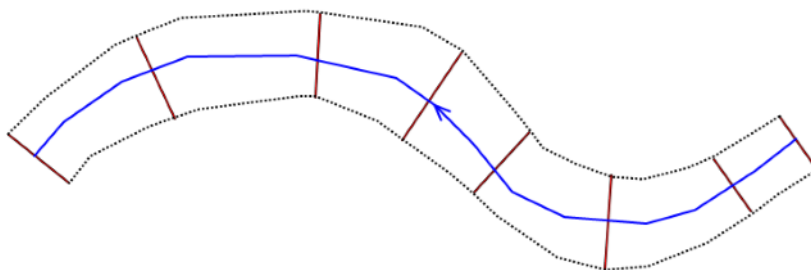


Figure 15.1 Illustration of area covered by the 2D map based on cross sections extent. Blue line represents the river branch, brown lines represent cross sections and the dashed black lines represent the area where results will be mapped.

No calculation of results will take place outside the cross sections extent, and the maps obtained, will therefore represent exactly what the model calculates during the actual simulation.

An example of a flood map produced through the map-feature is presented in Figure 15.2:



Figure 15.2 An example of a 2D flood inundation map presented in the Result Viewer

For each created map, the following parameters and options must be specified.



Name

The identification name of the map output.

File name

The path and the name of the dfs2 file storing the map results.

Item

The item being mapped in the result file. Only one item can be mapped in each file. Various results items may be mapped.

The complete list consists of:

- Water level: this option creates a map of water level. The water level is assumed to be constant along the cross sections, and is then interpolated in the longitudinal direction.
- Water depth: this option creates a map of water depth, being the difference between the above water level and the interpolated bed level.
- Velocity: with this option, the velocity is recomputed based on the water depth in each cell of the grid.
- Velocity * depth: this provides a derived result computed as velocity times depth
- Advection-Dispersion component: this option creates a map of the concentration of the selected component.
- DEM: this option generates a digital elevation model (DEM) for the river bed, based on the topography from the cross sections and the river branches. With this option, no period needs to be specified.
- (h, p, q): this option produces a file containing the three items Water depth, P-flux and Q-flux. This map type is primarily relevant for MIKE FLOOD simulations (coupled 2D- and 1D-simulations), as the map file can be burnt afterwards in the 2D result file in order to get a combined map, with the same result items on both the 1D and 2D domain.

Please refer to the MIKE 1D reference manual for details about the mapping procedure.

Type

The maps may be of three types:

- Maximum: the overall maximum value throughout the simulation period for each cell included in the map.
- Minimum: the overall minimum value throughout the simulation period for each cell included in the map.
- Dynamic: time-varying map able to animate the results.

Component

The selected Advection-Dispersion component to be mapped, when the selected Item is 'AD result'.



Cell size

The cell size of the dfs2 output file. The cell size controls the spatial resolution of the map.

Rotation

The rotation of the dfs2 output file. The rotation is defined as the angle between true north and the y-axis of the grid, measured clockwise. When a rotation is applied, the grid is rotated from its lower left corner, i.e. this corner remains at the same coordinates whereas the three other corners are moved.

Extent def.

This controls the way the extent of the map in the two spatial dimensions is defined. Two options are available:

- Lengths: with this option, you specify the width and height of the rectangle to define the dimensions along the two horizontal axes. MIKE HYDRO will derive the extent of the map from the origo and these lengths.
- No. of cells: with this option, you specify the number of cells of the grid in the two spatial dimensions, J being the number of cells eastward and K being the number of cells northward.

X0

The X coordinate of the lower left corner of the grid.

Y0

The Y coordinate of the lower left corner of the grid.

Width

The length of the grid in the eastward direction. This field can be edited only when the extent is defined using the option 'Lengths'. When the extent is defined using the option 'No. of cells', the field displays the width which is automatically derived from the number of J cells and the cell size.

Height

The length of the grid in the northward direction. This field can be edited only when the extent is defined using the option 'Lengths'. When the extent is defined using the option 'No. of cells', the field displays the height which is automatically derived from the number of K cells and the cell size.

No. of J cells

Number of cells of the grid in the eastward direction (note that this dimension is not strictly from West to East when the rotation is different than 0). This field can be edited only when the extent is defined using the option 'No. of cells'. When the extent is defined using the option 'Lengths', the field displays the number of cells which is automatically derived from the width and the cell size.



No. of K cells

Number of cells of the grid in the northward direction (note that this dimension is not strictly from South to North when the rotation is different than 0). This field can be edited only when the extent is defined using the option 'No. of cells'. When the extent is defined using the option 'Lengths', the field displays the number of cells which is automatically derived from the height and the cell size.

Storing frequency

The storing frequency controls the interval between two time steps saved in the output file. The unit used for the specified value is selected under 'Storing unit'. This parameter is only available for the 'Dynamic' type of map.

Storing unit

The unit in which the storing frequency is expressed. This parameter is only available for the 'Dynamic' type of map.

Period

This parameter controls how the period, for which the map is produced, is defined. Two options are available:

- Simulation period: with this option, the results are mapped for the full simulation period, as defined under the 'Simulation period' menu.
- Selected period: with this option, the results are mapped for a limited period specified manually. This option may be used in order to reduce the weight of the output file.

This parameter is only available for the 'Dynamic' type of map. When the type is either 'Minimum' or 'Maximum', the period is always the simulation period.

Start

The date and time starting from which the results are mapped. This date and time cannot be prior to the start date of the simulation.

End

The date and time at which the results stop to be mapped. This date and time cannot be posterior to the end date of the simulation.



Note: The extent of the map may be graphically defined by clicking 'Draw rectangle on the map' and dragging a rectangle on the map. This button is active only when the rotation is different than 0. Once defined, the rectangle may be edited (and edits must be saved) but it cannot be deleted.

15.8 Time series

This property page contains the specifications for time series output files to be generated during the simulation. These output files are created in addition to the regular and the additional .res1d output files. Time series output can be saved in .dfs0 or ASCII files. The purpose of this facility is to offer the possibil-



ity of getting time series files without manually extracting them from the .res1d file after the simulation has been completed. This is often useful for automatic or manual calibration or when running production simulations.

To request a time series output file to be generated during the simulation, a new item should be added to the overview table at the bottom of the page. Time series outputs are added or deleted from the table by use of Append '+' or Delete '-' buttons above the overview table.

For each row in this table, one time series output file will be generated. The upper part of the dialogue displays the properties of the active entry in the table, which are described below.

Time series

ID

String identification of the time series. ID will be used in the output results and hence makes it easier to identify each created time series.

Format

The format column gives three choices for the file format:

- dfs0. This will generate a .dfs0 file in the standard DHI file format for time series. Files can be loaded into for instance the time series editor, Plot Composer, MIKE View or used as simulated data for the auto calibration tool.
- Ascii – column based. This will generate an ASCII file with a column based format. There will be one time column and one column for grid point item selected for output. The number of lines will equal the number of saved time steps.
- Ascii – table based. This will generate an ASCII file with a table based format. There will be one table for each time step saved. Each table will have rows corresponding to the number of selected grid points, and columns corresponding to the number of selected items.

File name

Here the name of the output file should be specified together with the path where the file should be stored.

Text file settings

These check boxes only apply for the two ASCII file formats:

- Minimum: A separate table at the bottom of the file will show the minimum value of the output items.
- Maximum: A separate table at the bottom of the file will show the maximum value of the output items.
- Time of minimum: A separate table at the bottom of the file will show the time of minimum value of the output items.



- Time of maximum: A separate table at the bottom of the file will show the time of maximum value of the output items.

The following format specifications can also be expressed for the ascii output files:

- Delimiter: Between each column a special character like semicolon, comma etc. can be requested. This makes import to for instance Excel easier.
- Column width: The desired minimum width of each column can be specified.

Time period

In this section the desired period of the output time series can be specified.

Period

Here you can choose from the drop down menu whether the output time series will cover the full Simulation period or a Selected period.

Start

Start date of the output time series, when the type of period is 'Selected period'.

End

End date of the output time series, when the type of period is 'Selected period'.

Items

Time series output is produced for the selected items. These can be added or removed from the output file by using the Append '+' and Delete '-' buttons above the table. It is also possible to add to the output file all the available items by clicking on Add all items.

For each row in the table, the properties below must be specified.

Item

The items can be selected from the choices available in the drop down menu. Some of the items are dynamic and some are static. Dynamic items are items between "Water level" and "Bed shear stress". In the dfs0 files only dynamic items will be saved. In ASCII files both dynamic and static items will be saved.

The available items are:

- Water level
- Discharge
- Velocity
- Cross section area



- Flow width
- Radius
- Resistance number
- Conveyance
- Froude number
- Volume
- Flooded area
- Mass error
- Lateral inflows
- Water level slope
- Energy level slope
- Energy level
- Bed shear stress
- Lowest river bed level
- Left levee bank level
- Right levee bank level

Name

A user defined name for the item can be specified. If not specified, the default name will apply.

No decimals

The specification of the number of decimals will only apply for output in ASCII files.

Grid points

The calculation points which will be saved in the time series output are specified here. Using the 'Selection' drop-down list to the right, the user may choose to either save all points, only the selected ones from the table or all-but-selected ones.

The identification of grid points in the table is done through specifying a number of reaches, which are themselves defined by river name, upstream chainage and downstream chainage. Single grid points can be defined by setting the upstream chainage equal to the downstream chainage.



Note: Outputs are only saved at h-points locations. The discharge is however saved at h-points when saving to an ASCII file, or at Q-points when saving to a .dfs0 file.





16 Main menu bar features

There are a number of effective features and tools to assist in model creation which are accessible from options within the Main menu bar.

Specific main menu bar features described in this section include:

- File menu: Import and Export of MIKE HYDRO model data
- Tools menu: Cross sections processing tools and Load calculator

16.1 File: Import and Export of MIKE HYDRO data

MIKE HYDRO includes import and export features for model data components. The import and export option is activated from the main menu bar selecting either *Import* or *Export* from the File menu.

16.1.1 Import

Import from Shapefile

MIKE HYDRO include a comprehensive shapefile conversion feature, enabling import of most georeferenced model objects stored in Shapefiles in to a MIKE HYDRO model.

It is possible to import River branches, Alignment lines, Structures, Storages, boundary conditions as well as Catchment and Rainfall Runoff method from Shapefiles for the River module.

Figure 16.1 presents the dialogue for import of River model data from Shapefiles.



Figure 16.1 Import of River model features from Shapefile

To import model features from shapefile, do the following:

1. Activate the checkmark in front of the model features that should be imported (multiple features can be imported in one operation).
2. Browse and select the specific shape file using the file selection button '...'. .
3. Define feature specific attributes from the shape file using the dropdown menus. Each dropdown menu will present the shape file attributes that are possible to use for the specific MIKE HYDRO feature parameter.
Note: It is not required to specify all attributes in drop down menus. If some are left blank, the import utility will just assign default values for these features.

Some attributes in the shape files must be assigned specific numerical values to be recognized and imported correctly by MIKE HYDRO, as some of the features in MIKE HYDRO are defined by a numerical value rather than a string. The table below presents the attribute values for specific feature type definitions used during import from shapefile to MIKE HYDRO.

Table 16.1 Attribute values for specific feature type definitions

Feature attributes	Options	Values
Flow direction	Positive	0
	Negative	1
Branch Type	Regular	0
	Link Channel	1
	Kinematic Routing	2



Table 16.1 Attribute values for specific feature type definitions

Feature attributes	Options	Values
Alignment Type	Left Extent	0
	Left Bank	1
	Right Bank	2
	Right Extent	3
Structure Type	Weir	0
	Culvert	1
	Pump	2
	Gate	3
	Direct Discharge	4
	Dam break	5
	Energy Loss	6
Boundary Type	Tabulated	7
	Water Level	0
	Discharge	1
	QH Relation	2
	Free Outflow	3
	Closed	4
	Rainfall	5
Use Polyl. Area	Evaporation	6
	Runoff	7
RR method (Rainfall Runoff Model Type)	True	1
	False	0
	None	0
	NAM	1
	UHM	2
	Time Area	3
	Kinematic Wave	4

When selections and filename specifications are complete, press the 'Run' button to execute the import of model data from shapefiles. Press 'Close' to stop the conversion or to close the dialogue.



Note: Structures and boundaries (both point and distributed sources types) are attached to river branches in MIKE HYDRO, but imported points (for structures and point source boundaries) and polylines (for distributed source boundaries) must not necessarily be strictly superimposed to the imported branches. These objects will be inserted on the closest location on the river branch they belong to. Additionally, their chainages will be automatically computed based on this location.

Import from MIKE 11

MIKE 11 model data can be imported and applied in MIKE HYDRO River



Activate the MIKE 11 Import from the *Import from MIKE 11* option under *Import*. A selection dialogue as presented in Figure 16.2 will appear and the required MIKE 11 import option can be selected from this dialogue.

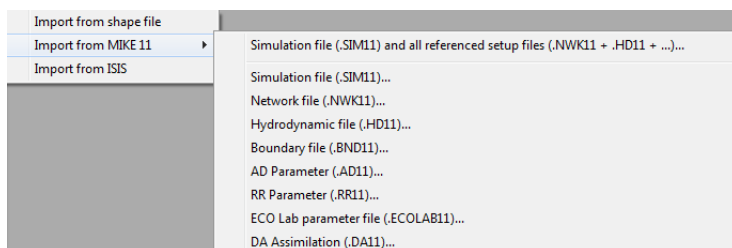


Figure 16.2 Import from MIKE 11 files to MIKE HYDRO River

Importing a complete MIKE 11 model can be made through the first option; '*Simulation file (.SIM11) and all referenced setup files...*'. This option import all model parameters defined in the MIKE 11 model setup.

Additionally, it is possible to import any individual MIKE 11 input parameter files by selecting the specific file type option in the import page.

Selection of any option listed in this page open a file selection dialogue that allows you to browse and select the specific MIKE 11 input file to import.



Important Notes:

- Importing any MIKE 11 file to MIKE HYDRO will automatically set the MIKE HYDRO calculation module to 'River'.
- Import of MIKE 11 files always add information to the actual MIKE HYDRO project. MIKE 11 model import can therefore either assist in creating a new River model or in adding additional data to an existing MIKE HYDRO project.
- When importing a HD11 file only data related to existing branches will be imported.
- Any data not supported by MIKE HYDRO will be ignored during the import. A message box will appear with errors and warnings related to the import.
- Imported data will not be re-projected. When importing data in an empty setup the feature coordinate system will be changed to the coordinate system of the imported data. When importing data in a setup with existing branches a warning is displayed.



Import from ISIS

The Isis to MIKE HYDRO converter is designed to convert the main features of an ISIS set-up to a MIKE HYDRO river set-up. It does not convert all features of ISIS and further there is not a one to one match between MIKE HYDRO river features and ISIS features. Hence the user must inspect the resulting MIKE HYDRO river set-up and further apply model refinements where needed.

Data requirements

The converter has the following data requirements

- The network file (*.xy) file holds the geospatial information for all the hydraulic elements that make up a river model set-up. Please note that the file does not carry any information with regards to which projection is applied
- The data file (*.dat) file is the main model set-up file. This file contains the various hydraulic parameters describing cross sections, structures, etc.
- The ISIS Execution file (*.ief) is used for setting the time step and the duration of the simulation.

Each of the above files may be selected by use of the browse buttons to the right of the file name field.

The converter requires as a minimum the ISIS data file is present. To generate a network topology and not just the cross sections the network file is also needed. The execution file may be omitted.

Simulation start time

ISIS works with a relative time axis i.e. time is measured relative to the start of the simulation date and a time must be provided to the converter. All resulting simulation input files will start at this time.

Simulation duration

The simulation duration will be filled if the *.ief file is present. The value is to be given in hours.

Simulation time step

If the *.ief file is provided this field will be filled with the time step from ISIS. Please note that the ISIS time step may be smaller than what is typically required by MIKE HYDRO.

Output folder name

The field holds name of the folder in which the resulting MIKE HYDRO set-up will be placed.



16.1.2 Export

Export to Shapefile

The majority of georeferenced model features defined in MIKE HYDRO River can be exported to Shapefiles.

The export option is activated through '*Export to Shape files*', which opens the dialogue as presented in Figure 16.3.

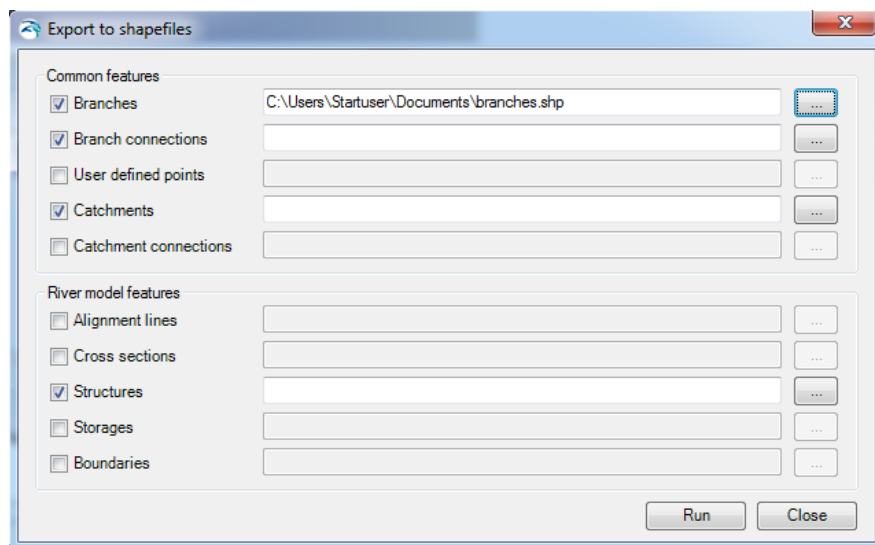


Figure 16.3 Export MIKE HYDRO Data to Shapefile dialogue

The dialogue contains two groups of data; 'Common features' which are features that are also used by other modules than River, and 'River Module Features' which are module specific features used only in the specific MIKE HYDRO module.

Export the requested model data by activating the checkbox in front features that should be processed and specify a shapefile names for the individual features. Use the file selection button '...' for specifying the output shapefile names.

After completing selections and filename specifications, press the 'Run' button to execute the export of model data to shapefiles. Press 'Close' to stop the conversion or to close the dialogue.



Shapefile content from Export of MIKE HYDRO features will include the following information:

Table 16.2 *Export from MIKE HYDRO; Shapefile content*

MIKE HYDRO features	Shape type	Key variables to export as shapefile attributes	Attribute name	Att
Branches	River shapes (Polylines)	Branch name Identifier Topo-ID Start Chainage End Chainage Flow direction Branch Type	BR_BrName BR_ID BR_TopoID BR_StartCh BR_EndCh BR_FlowDir BR_Type	Tex Tex Tex Do Do Sh Sh
Branch connections	Connecting lines (Polyline)	Branch name Upstr. Type Upstr. Branch name Upstr. Chainage Upstr. Storage ID Downstr. Type Downstr Branch name Downstr. Chainage Downstr. Storage ID	BC_BrName BC_UpsType BC_UpsName BC_UpsCh BC_UpsStID BC_DwsType BC_DwsName BC_DwsCh BC_DwsStID	Tex Sh Tex Do Tex Sh Tex Do Tex
User defined chainages	Points	Branch name Chainage	UDC_BrName UDC_Ch	Tex Do
Catchments	Catchment shapes (polygons)	Name Use Polygon Area Catchment Area Rainfall-Runoff model Branch Chainage	Cat_Name Cat_AreSel Cat_Area Cat_RRMod Cat_BrName Cat_Ch	Tex Sh Do Sh Tex Do
Catchment connections	Connection (Polyline)	Catchment name Branch name Chainage Upstream Ch Downstream ch	CC_CatName CC_BrName CC_Ch CC_UpsCh CC_DwsCh	Tex Tex Do Do Do
Alignment Lines	Lines (polylines)	Name Type	AL_Name AL_Type	Tex Sh



Table 16.2 Export from MIKE HYDRO; Shapefile content

MIKE HYDRO features	Shape type	Key variables to export as shapefile attributes	Attribute name	Attribute type
Cross Sections	Cross section lines (polylines)	Cross section ID Branch name Topo-ID Chainage Cross section type Radius type Datum Resistance distribution Uniform resistance Left high flow resistance Low flow resistance Right high flow resistance Resistance type	CS_ID CS_BrName CS_TopoID CS_Ch CS_Type CS_Radius CS_Datum CS_ResDist CS_Uniform CS_LeftHi CS_Low CS_RightHi CS_ResType	Text Text Text Double Short int Short int Double Short int Double Double Double Double Short int
Structures	Structure locations (Points)	ID Branch name Chainage Location Type Type of structure	Str_ID Str_BrName Str_Ch Str_Mode Str_Type	Text Text Double Short int Short int
Storages	Storage locations (Points)	ID Capacity type Initial Surface Area	ST_ID ST_CapType ST_Area	Text Short int Double
Boundaries (#1)	Boundary locations (Points)	Boundary ID Location Type Branch name Chainage Boundary Type	BC_ID BC_LocType BC_BrName BC_Ch BC_BndType	Text Short int Text Double Short int
Boundaries (#2)	Boundary locations (Polyline)	Boundary ID Branch name Chainage Downstr Chainage Boundary Type	BC_ID BC_BrName BC_Ch BC_DwsCh BC_BndType	Text Text Double Double Short int

Export to MIKE 11

MIKE HYDRO River models can be exported to MIKE 11 setup files which therefore, facilitate a compatibility between MIKE 11 and MIKE HYDRO River. Export of MIKE HYDRO river data can create a full MIKE 11 model setup including all required model parameter files, or individual MIKE 11 parameter files containing more specific model input data.

MIKE HYDRO River to MIKE 11 Exporter is activated from the '*Export to MIKE 11 files*' from *Export*. A selection dialogue as presented in Figure 16.4 appear from where the required MIKE 11 export options can be selected.

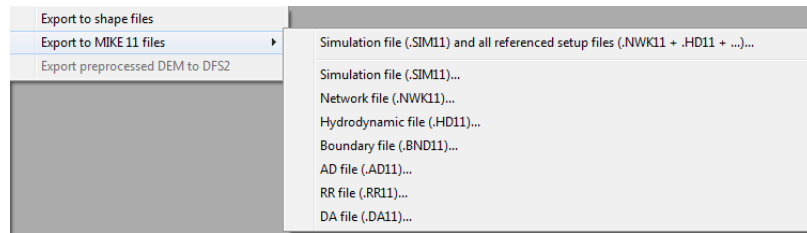


Figure 16.4 Export from MIKE HYDRO River to MIKE 11 files



Note: Some features in MIKE HYDRO River are new compared to MIKE 11 functionality can therefore not be exported to MIKE 11. Eventual unsupported features from the exported MIKE HYDRO River model will be listed in a status dialogue for the export to MIKE 11 operation.

Export Preprocessed DEM to dfs2

If Digital Elevation Model (DEM) is included in MIKE HYDRO for use in either catchment and river delineation or cross section extractions it is requirement to make a preprocessing of the DEM prior to these operations. This export function enable an option for saving the preprocessed data from this process in to a dfs2 file for general information about the preprocessing results.

16.2 Tools

The '*Tools*' menu in the MIKE Zero menu bar contains a set of tools used to help creating and modifying the setup.

The following tools are available:

- Auto Generate Cross Sections:
Create cross sections from a DEM or survey points.
- Trim or Extend Cross Sections
Modify cross sections based on alignment lines.
- Edit multiple cross sections
Apply changes for multiple cross sections
- Auto update cross sections
Update cross sections from a DEM
- Compute storage from DEM
Assign additional storage to cross sections
- Resistance values interpolation
Interpolate and assign resistance values / factor to cross sections



- Export embankment lines
Export polygons or polylines describing the extent of the river
- Importing cross sections from text file
- Exporting cross sections to text file

16.2.1 Auto Generate Cross Sections

The auto generate cross section tool offers options to define and extract cross sections either from a Digital Elevation Model (DEM) or from survey points.

Option

The elevation from the cross section can be taken either from a DEM or from survey points.

DEM

To generate cross sections from a DEM first activate the DEM radio-button. Cross sections will be generated along the selected branch. The following options are available:

- **Equidistant interval and Width:**

Interval: Equidistant distance between cross section to be generated.

Width: This defines the width of the generated cross sections. Cross section lines will be located with half of the width distance on each side of the river centre line.

The cross sections will be created with a resolution in each cross section corresponding to the options specified in the 'Digital Elevation Model' menu.

- **Location from shape file:** Use a shape file containing cross section polylines at the requested locations along the rivers in the model. Select the shape file to be used in the drop-down list of shape files.

Cross sections will only be generated where shape file geometries intersect with the selected branch. The cross section will be generated as a straight line from the first to the last point in the line shape.

It is possible to automatically assign an ID to each generated cross section, by activating the option 'Apply cross section ID'. This ID is obtained from the shape file's attribute selected in the 'Field' menu.



Note: Usage of shape files requires that they are specified in the 'Background layers' before opening the tool.



- **Use bilinear interpolation:** If '*Use bilinear interpolation*' is checked bilinear interpolation is used when extracting cross section z-values from the DEM cells. If unchecked the z-value in the cross section profile will be the exact value from the DEM cells that are intersected, whereas if bilinear interpolation is used, z-values will be the interpolated z-value from the 4 closest neighbouring DEM grid cells.

Survey points from shape file

To generate cross sections from a shape file with survey point first activate the '*Survey points from shape file*' radio-button.

Select a shape file with survey points from the drop-down menu and specify the following:

- **Z-value field:** Select the attribute in the shape file that contains the elevation.
- **Cross section ID field:** Select the attribute in the shape file that connects the individual survey points to a specific cross section. All survey points with the same ID field will be created as one cross section.



Note: Usage of shape files requires that they are specified in the '*Background layers*' before opening the tool.

The cross section will be created as a straight line through the connected survey points using a least square fit. The individual survey points are projected orthogonal on to the cross section line see Figure 16.5.

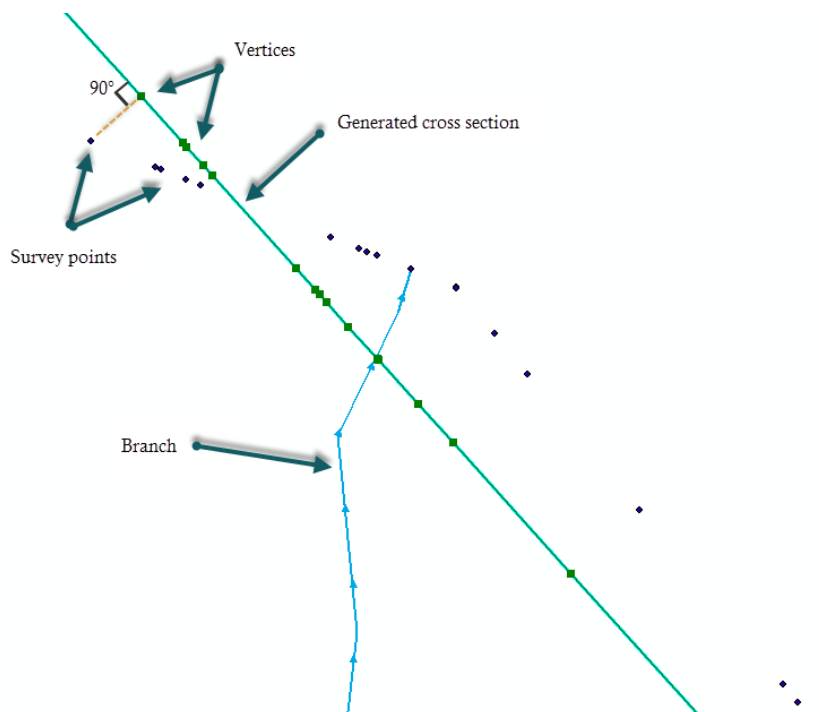


Figure 16.5 Survey points are projected to the cross section

Apply to

Options selected here control on which branches or parts of branches the new cross sections should be generated. Three options are available:

- **All branches:** New cross sections may be generated on all branches. In this case, it is not possible to control the range of chainages to be processed on the branches, i.e. the tool will process all branches in their entirety.
- **Selected branches:** New cross sections may be generated only on the selected branches. In this case, it is not possible to control the range of chainages to be processed on the branches, i.e. the tool will process selected branches in their entirety.
- **Table of reaches:** The locations where new cross sections may be generated are specified in a table, offering flexible options to control the branches to be included, and also the range of chainages for each branch, where the cross sections may be generated. Cross sections are generated only between the specified lowest and highest chainages, and if the box is checked in the 'Include' column. Furthermore, when using the 'DEM' option, the table allows to apply local 'Width' and 'Interval' values for each row of the table.



When the option 'Table of reaches' is used, the following buttons are available above the table:

- 'Import all': Imports all branches in the table, i.e. it creates one row for each branch in the model. Each branch is given its default lowest and highest chainages.
- 'Import from file': Imports the 'Branch name', 'Lowest chainage' and 'Highest chainage' from a text file. The file format is: three columns with lowest chainage, highest chainage and branch name (in this order), with one separate line for each row of the table. Columns must be space-separated or tab-separated.
- 'Include all': Checks all boxes in the 'Include' column of the table.
- 'Exclude all': Unchecks all boxes in the 'Include' column of the table.

Buttons

- **Generate:** Generates cross sections following the specifications in the dialogue window.
- **Save:** Saves the messages issued during the generation of a text file.
- **Clear log:** Clears the log-messages in the text box above the buttons.
- **Close:** Closes the '*Auto Generate Cross Sections*' tool.

16.2.2 Compute storage from DEM

The storage capacity (Elevation - Area relationship) can be computed for reservoirs, lakes, bays and small inlets, and be used as input definition for storages or as additional storage in cross sections. The storage capacity is calculated using elevations from a DEM and spatial extents from user-defined polygons. The tool will automatically extract the area at different elevations for each polygon, and assign the obtained relationships to the relevant storages or cross sections.

When the storage capacity is applied to cross sections, one or more polygons (representing extent of the storage area) can be assigned to the same cross section. For each level in the existing cross section's processed data, the surface area of the additional storage is calculated based on the DEM; see Figure 16.6.

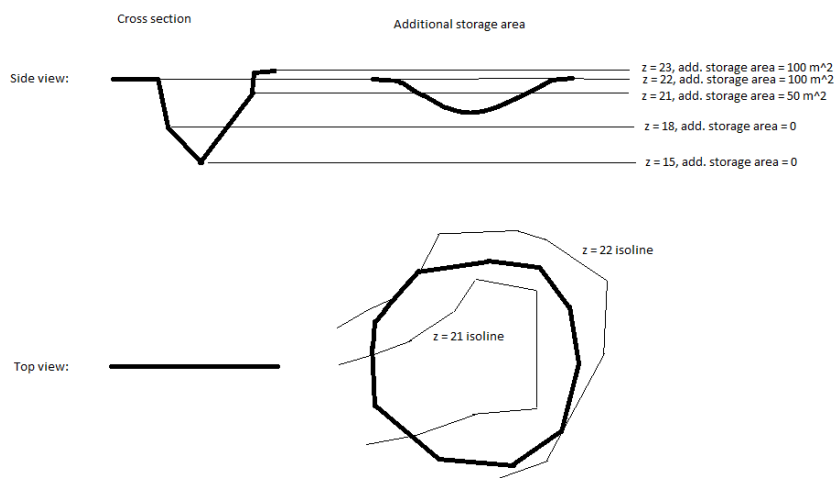


Figure 16.6 Example of assigning additional storage to a cross section

When the storage capacity is applied to storages, areas are calculated at regular intervals. The lowest and highest elevations correspond respectively to the lowest and highest elevations from the DEM within the polygon. The intermediate elevations are controlled by the user-defined number of elevations being calculated.

The 'Compute storage from DEM' tool requires some initial actions from the user to enable the functionality:

- Load a shape file through the '*Background layers*' page. The shape file must contain an attribute with an identifier corresponding to the storage ID, or cross section ID used in the cross section editor.
- Load a DEM through the '*Background layers*' page
- Select the DEM file through the '*Digital Elevation Model (DEM)*' page.

Apply elevation area relationships to

Select whether the computed relationships will be applied to storage definitions, or to additional storage area of cross sections.

Storage area shape file

Select the shape file that defines the additional storage area from the dropdown menu of polygon shape files included in the '*Background Layers*'.



Note: Usage of shape files requires that they are specified in the '*Background layers*' before opening the tool.



Attribute containing ID

Select the attribute in the shape file that connects the individual polygons to a specific storage or cross section. All polygons with the same ID will be used as additional storage to the cross section.

Number of elevations in relationships

This value specifies the number of equidistant elevations at which the storage areas are calculated. This value is only used for storages.

Create flood map helper file

In order to properly represent the additional storage in a flooding map a '*flood map helper file*' can be created. The file is a dfs2 file where areas with additional storage has the value of the cross section ID.



Note: The flood map helper file can only be created if cross section ID's are integers.

The flood map helper file can be used in connection with the 2D mapping feature in MIKE 11.

If the checkbox is enabled, it is required to specify a dfs2 file name either manually typing the entire file name or by using of the '...' file selection button.

Buttons

- **Close:** Closes the '*Additional storage area*' tool.
- **Clear log:** Clears the log-messages in the text box above the buttons.
- **Update:** Updates cross sections with additional storage area following the specifications in the dialogue window.

16.2.3 Trim or Extend Cross Sections

The '*Trim or extend cross sections*' tool can modify cross sections using alignment lines.

Additionally, the tool can also be used to change the position of markers (marker 1, 3, 4 and 5) in cross sections using alignment lines and thereby limit the part of the cross sections active in the river simulation or apply a distributed roughness based on markers positions.

Input

Two input types are available:



- Use alignment lines: this option will read the alignment lines from the data specified in the 'Alignment lines' menu.
- Use shape file: this option will read the alignment lines from a shape file.

Note: Usage of shape files requires that these files are previously loaded in the '*Background layers*' menu before opening the tool. The shape files must contain an attribute with an identifier corresponding to markers.

If the 'Use alignment lines' option is selected, no additional information is required in the Input menu, since the properties of alignment lines are specified in the 'Alignment lines' menu.

If the 'Use shape file' option is selected, the additional information below must be specified.

Alignment line shape file

Select the shape file that defines the alignment lines from the drop-down menu. This menu lists the polyline shape files included in the '*Background layers*' menu.

Alignment line marker field

Select the attribute that contains the identifier corresponding to markers.

Left extent marker

Activate the checkbox if the left extent line (corresponding to marker 1) should be used during trim, extend or update operation.

If activated, it is required also to select the value of the 'Alignment line marker field' attribute that represents the left extent line.

Left bank marker

Activate the checkbox if the left bank line (corresponding to marker 4) should be used during trim, extend or update operation.

If activated, it is required also to select the value of the 'Alignment line marker field' attribute that represents the left bank line.

Right bank marker

Activate the checkbox if the right bank line (corresponding to marker 5) should be used during trim, extend or update operation.

If activated, it is required also to select the value of the 'Alignment line marker field' attribute that represents the right bank line.

Right extent marker

Activate the checkbox if the right extent line (marker 3) should be used during trim, extend or update operation.

If activated, it is required also to select the value of the 'Alignment line marker field' attribute that represents the right extent line.



Option

The three operations below can be performed.

Set marker in cross sections

Activate the 'Set marker in cross section' option if the markers should be updated in all cross sections based on the alignment lines. If the input is read from the 'Alignment lines' menu, this option will update all relevant markers depending on the existing alignment lines types, whereas if the input is read from a shape file, only the selected markers will be updated.

This operation will not alter the geometry of cross sections, but only update the marker positions according to the alignment line locations. If the cross section does not contain a point at the marker position a new point is inserted with an elevation interpolated from the previous and next point in the cross section.

If a cross section does not intersect with an alignment line, the marker is not updated. If a cross section intersects with more than one alignment line of the same type the alignment line closest to the branch is used.

Trim cross sections

Activate the 'Trim cross sections' option if the alignment lines should be used to trim the width of all cross sections. Only the part of the cross sections which is between markers 1 and 3 will be kept in the updated cross sections.

If a cross section does not intersect with an alignment line the cross section is not trimmed. If a cross section intersects with more than one alignment line of the same type the alignment line closest to the branch is used.

Extend cross sections

Activate the 'Extend cross sections' option if alignment lines should be used to extend (enlarge) the width of all cross sections. Topography along the extended parts of cross sections is obtained from the underlying DEM.



Note: The 'Extend cross sections' option requires that options for cross sections creation are specified in the 'Digital Elevation Model (DEM)' menu before opening the tool. This requires that a DEM is selected, that the option for using the DEM for creating cross sections is active, and that the spacing between points as well as the maximum number of points are specified. When extending a cross section, the specified maximum number of points corresponds to the total number of points in the extended cross section, including original points.

If a cross section intersects with more than one alignment line of the same type the alignment line closest to the branch is used.

Two additional parameters should be specified:



- **Maximum length to extend per side:** The 'maximum length' defines the maximum distance a cross section can be extended. If an alignment line is not found within the given distance the cross section will not be updated.
- **Use bilinear interpolation:** If 'Use bilinear interpolation' is checked bilinear interpolation is used when extending the cross section while using a DEM. If unchecked the z-value in the cross section profile will be the exact value from the DEM cells that are intersected, whereas if bilinear interpolation is used, z-values will be the interpolated z-value from the 4 closest neighbouring DEM grid cells.

Apply to

Options selected here control on which branches or parts of branches the cross sections should be updated. Four options are available:

- **All cross sections:** All cross sections in the model will be updated.
- **Selected cross sections:** Only the selected cross sections will be updated.
- **Cross sections from selected branches:** Only the cross sections on the selected branches will be updated.
- **Cross sections from table of reaches:** The locations of cross sections to be updated are specified in a table, offering flexible options to control the branches to be included, and also the range of chainages for each branch, where cross sections are updated. Cross sections are updated only between the specified lowest and highest chainages, and if the box is checked in the 'Include' column.

When the option 'Cross sections from table of reaches' is used, the following buttons are available above the table:

- **'Import all':** Imports all branches in the table, i.e. it creates one row for each branch in the model. Each branch is given its default lowest and highest chainages.
- **'Import from file':** Imports the 'Branch name', 'Lowest chainage' and 'Highest chainage' from a text file. The file format is: three columns with lowest chainage, highest chainage and branch name (in this order), with one separate line for each row of the table. Columns must be space-separated or tab-separated.
- **'Include all':** Checks all boxes in the 'Include' column of the table.
- **'Exclude all':** Unchecks all boxes in the 'Include' column of the table.

Buttons

- **Update:** updates cross sections following the specifications in the dialogue window.



- **Save log:** Saves the messages issued during the generation to a text file.
- **Clear log:** Clears the log-messages in the text box above the buttons.
- **Close:** Closes the tool.

16.2.4 Edit multiple cross sections

The tool 'Edit multiple cross sections' may be used to perform multiple changes to raw data and/or processed data of cross sections, and apply them to multiple cross sections at the same time. To apply a change to a parameter, simply tick the corresponding option, and specify the new parameters.

Properties

Raw data - Datum

The datum of the cross section can be changed here.

Raw data - Resistance

A change of transversal distribution, resistance type definition and resistance values can be set here.

Raw data - Markers

Multiple options are available here to move markers. Markers 1 and 3 can be moved on river banks or to cross sections extents. Markers 4 and 5 can be moved on river banks. Marker 2 can be set either at the lowest point or at intersection with the branch.

The tick-boxes in front of the maker values determine which markers should be updated, as only selected marker values will be updated.

Additionally, when markers are moved on river banks, a minimum decrease of elevation must be specified to identify the location of the banks.

Processed data - Allow for recalculation

The option 'Allow for recalculation' of the processed data can be activated or deactivated here.

Sediment transport - Floodplain level

The floodplain level used in Sediment Transport calculations can be defined here. When it is set to User defined, the corresponding value must also be specified.

Processed data - Radius

The radius type can be changed here.

Processed data - Levels

The levels selection method, as well as the number of levels used for determining the processed data, can be changed here.



Processed data - Angle correction

Activating or deactivating the angle correction respectively checks or unchecks the 'Apply' angle correction in the processed data.

Computing the angle correction edits the angle value.

Location

Activating or deactivating the coordinates respectively checks or unchecks the 'Apply' option in the Coordinates tab.

Inverting left and right sides of the cross section swaps either the coordinates of the cross sections, the raw data or both. The option 'Invert only if coordinates are digitised from the right to the left side' allows to perform this change only in case the cross section's coordinates have been provided in the wrong direction.

Three options are available to edit the chainages or coordinates:

- Recomputing the chainage with the formula $\text{New Chainage} = a * \text{Original Chainage} + b$, where a and b can be changed manually.
- Updating chainages of cross sections by using the cross sections coordinates and the position of the branch to identify their intersection. This may be used after moving a cross section on the map, to reflect the change in the chainage and therefore in the simulation.
- Resetting the coordinates: this will fill in the Coordinates table, assuming that the cross section is perpendicular to the branch and that marker 2 is aligned with the branch.

Apply to

The selected changes can either be applied to all cross sections, or only to the selected ones.

If the option 'Only update cross sections from branches and Topo ID included in simulation' is checked, all cross sections defined on non-existing branches, or on a different Topo ID than the one used for the branch in the simulation, are ignored.

Buttons

- **Update:** Applies the selected changes to the cross sections.
- **Save log:** Saves the messages issued during the update to a text file.
- **Clear log:** Clears the log-messages in the text box above the buttons.
- **Close:** Closes the tool.



16.2.5 Auto update cross sections

The tool 'Auto update cross sections' may be used to update elevation data of existing cross sections using a DEM. Additionally, the tool may update only a part of the cross sections: this may be useful when a recent DEM is available but only covers floodplains, and in this case the DEM can be used to update the cross sections elevations in the floodplains while keeping the elevations in the riverbed unchanged.

Option

Two options are available to control the extent of the cross sections to be updated:

- **Update whole cross sections:** This updates the elevation data all along the cross sections.
- **Update parts of cross sections:** This updates the elevation data only along parts of the cross sections. When applying this option, it is possible to select the two markers defining the part of the cross section which is NOT updated.

When applying the option 'Update parts of cross sections', and additional option 'Update levels for these markers' is also available. When it's checked, levels of the selected markers are updated, whereas levels of the selected markers are unchanged when the option is unchecked. As an example, it is possible to update the elevation data only in the floodplain by use of markers 4 and 5, which may be located on the banks. If banks levels are obtained from a survey which is assumed to be more accurate than the DEM used to update the floodplain, then the option should remain unchecked in order not to update the bank levels. On the opposite, if the new DEM is assumed to be more accurate than original data describing the banks, the option should be checked to update the bank levels as well.

Regardless of the selected option, the tool updates elevation data (Z-values in the raw data table) but keeps the following data unchanged:

- Coordinates
- S-values of the start and end points of the cross section
- S-values of the markers.

The update is controlled by the options specified in the 'Digital Elevation Model (DEM)' page, which control e.g. the spacing between points extracted from the DEM. Therefore a different number of raw data points may be obtained in the updated cross sections, compared to the original cross sections. This may be useful to best benefit from a new DEM with a higher resolution than the data used to create the original cross sections.



Apply to

Options selected here control on which branches or parts of branches the cross sections should be updated. Four options are available:

- **All cross sections:** All cross sections in the model will be updated.
- **Selected cross sections:** Only the selected cross sections will be updated.
- **Cross sections from selected branches:** Only the cross sections on the selected branches will be updated.
- **Cross sections from table of reaches:** The locations of cross sections to be updated are specified in a table, offering flexible options to control the branches to be included, and also the range of chainages for each branch, where cross sections are updated. Cross sections are updated only between the specified lowest and highest chainages, and if the box is checked in the 'Include' column.

When the option 'Cross sections from table of reaches' is used, the following buttons are available above the table:

- **'Import all':** Imports all branches in the table, i.e. it creates one row for each branch in the model. Each branch is given its default lowest and highest chainages.
- **'Import from file':** Imports the 'Branch name', 'Lowest chainage' and 'Highest chainage' from a text file. The file format is: three columns with lowest chainage, highest chainage and branch name (in this order), with one separate line for each row of the table. Columns must be space-separated or tab-separated.
- **'Include all':** Checks all boxes in the 'Include' column of the table.
- **'Exclude all':** Unchecks all boxes in the 'Include' column of the table.

Buttons

- **Update:** updates cross sections following the specifications in the dialogue window.
- **Save log:** Saves the messages issued during the generation to a text file.
- **Clear log:** Clears the log-messages in the text box above the buttons.
- **Close:** Closes the tool.

16.2.6 Resistance values interpolation

The tool 'Resistance values interpolation' may be used to perform a linear interpolation of resistance values, and assign them to cross sections.



Note that the tool doesn't change the resistance distribution type for the cross sections, i.e. it may for example change the 'Uniform' value for a cross section which uses the type 'High / low flow zones' but in this case the cross section won't use the modified value in the simulation until its distribution type is changed.

Option

It is possible to select between two different types of values to be interpolated:

- 'Uniform': this is used to change the 'Uniform' resistance value of the cross sections
- 'High/low flow zones': this is used to change the 'Left high flow', 'Low flow' and 'Right high flow' values of the cross sections.

The input resistance values to be interpolated are entered in the table. Each row contains a location (branch and chainage) and resistance value(s). The locations specified in this table don't have to match the cross sections locations. When the cross sections are updated, values from this table are interpolated linearly as a function of the chainage, and are then assigned to cross sections found within the range of chainages defined between the smallest and the highest chainage specified for each branch. When a cross section's chainage is out of the specified ranges, it is not processed.

Two buttons are available above the table:

- 'Import all sections': adds one row for each cross section's location existing in the model definition. Only the cross sections from the active Topo ID (Topo ID selected in the 'Branches' menu) are imported.
- 'Import selected sections': adds only the cross sections being selected.

Apply to

The selected changes can either be applied to all cross sections, or only to the selected ones. Regardless of the selected option, only the cross sections lying within the range of chainages specified in the table are updated.

If the option 'Only update cross sections from branches and Topo ID included in simulation' is checked, all cross sections defined on non-existing branches, or on a different Topo ID than the one used for the branch in the simulation, are ignored.

Buttons

- **Update:** Applies the selected changes to the cross sections.
- **Save log:** Saves the messages issued during the update to a text file.
- **Clear log:** Clears the log-messages in the text box above the buttons.



- **Close:** Closes the tool.

16.2.7 Export embankment lines

The Export embankment lines tool offers options to export polygons or polylines describing the extent of the river, either in a shape file or in a text file format. The latter can be used to exclude the river extent from a 2D flexible mesh, when coupling the MIKE HYDRO River model to a 2D model with MIKE FLOOD. Exported files can also be used to generate lateral links in MIKE FLOOD.

Export option

The method used to define the lateral extents as well as the output format can be selected here.

Embankment lines definition

Two options are available to define how the lateral extents are defined:

- **Cross sections and river branch alignment:** With this method, lateral extents are defined by the extents of the cross sections, but also by the river alignment between consecutive cross sections. That is, the lateral extent polyline / polygon is also defined with extra vertices, every time the branch changes direction. This is the recommended method per default, so that the lateral extent follows the branch alignment between cross sections.
- **Cross sections only:** With this method, branch alignment is ignored, and the lateral extent polyline / polygon is defined by straight lines between the ends of consecutive cross sections. This method is to be used when cross sections are sufficiently close to each other's to accurately describe the location of embankments, even without taking into account the branch alignment. It avoids getting intermixed polylines when cross sections are close, at locations where the branch changes direction.

Output

Two output types are available. For each option, it is possible to select whether the output file is a text file (with file extension *.xyz) or a shape file (with file extension *.shp).

- **Export surrounding polygons with one file:** This option creates a single file, containing one or more polygon. The exported text file can easily be imported afterwards in the Mesh Generator as boundaries, to exclude the polygons from the mesh.



- **Export left and right alignment lines in individual files:** This option creates one file per side of each exported branch, containing polylines. Hence each branch gets one file for the left extent polyline, and one for the right extent polyline. These polylines can be used to create lateral links in MIKE FLOOD. When using this option, each generated file is named with the specified output file name, but with a suffix containing the branch name, the start and end chainages, and the side.

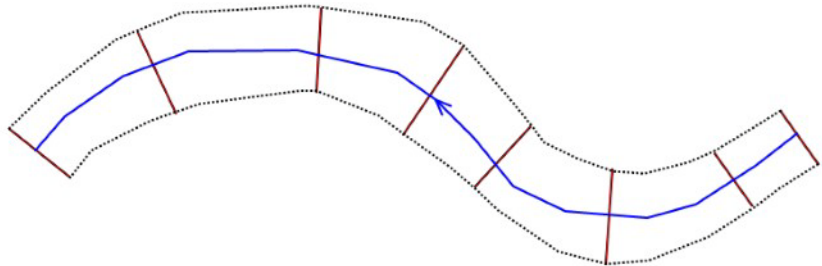


Figure 16.7 Illustration of the river extent. Blue line represents the river branch, brown lines represent cross sections and the dashed black lines represent the left and right lateral extent polylines. The polygon is defined with the two united lateral polylines.



Note: When exporting to text file, the file contains four columns with X and Y coordinates, followed by the embankment level Z and the connectivity. Units are always expressed in meters.

Apply to

Options selected here control which branches or parts of branches should be exported. Three options are available:

- **All branches:** All branches will be exported. In this case, it is not possible to control the range of chainages to be exported on the branches, i.e. the tool will process all branches in their entirety.
- **Selected branches:** Only on the selected branches will be exported. In this case, it is not possible to control the range of chainages to be exported on the branches, i.e. the tool will process selected branches in their entirety.
- **Table of reaches:** The list of reaches to be exported may be specified in a table, offering flexible options to control the branches to be included, and also the range of chainages for each branch. Only the parts of branches between the specified lowest and highest chainages will be exported, and if the box is checked in the 'Include' column.



When the option 'Table of reaches' is used, the following buttons are available above the table:

- 'Import all': Imports all branches in the table, i.e. it creates one row for each branch in the model. Each branch is given its default lowest and highest chainages.
- 'Import from file': Imports the 'Branch name', 'Lowest chainage' and 'Highest chainage' from a text file. The file format is: three columns with lowest chainage, highest chainage and branch name (in this order), with one separate line for each row of the table. Columns must be space-separated or tab-separated.
- 'Include all': Checks all boxes in the 'Include' column of the table.
- 'Exclude all': Unchecks all boxes in the 'Include' column of the table.

Buttons

- Export: Generates the shape files or text files following the specifications in the dialogue window.
- Close: Closes the 'Export embankment lines' tool.

16.2.8 Importing cross sections from text file

Via Tools and plugins \ Import from text files, it is possible to read and import cross section data from a text file into MIKE HYDRO River. This facility reads raw topographical data as well as additional cross sections' properties, which may be used by MIKE HYDRO River.

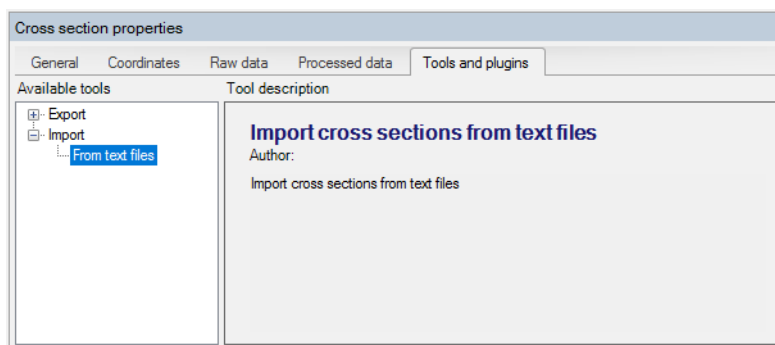


Figure 16.8 Import tool menu in the cross section editor

Executing the tool 'From text files' opens the dialog in Figure 16.9 below, where it is possible to select the text file to be imported. This dialog also contains an option for overwriting existing cross sections, when a cross section already exists at the location of an imported cross section. When this option is left unchecked, cross sections in the text file will not be imported when



other cross sections already exist at the same location. When 'Recompute processed data after importing' is selected, the table with processed data is automatically recomputed for each imported cross section

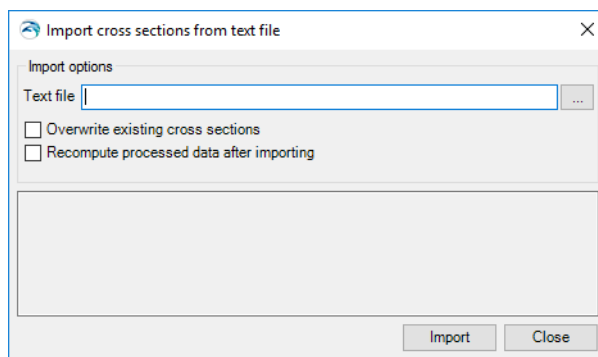


Figure 16.9 Import from text file dialog

Text file format

As a minimum, each cross section must contain the following information, in this order:

Topo ID
Branch name
Chainage

In order to also import the topographical data, a cross section also needs a profile section, which first row is PROFILE *N*, where *N* is the number of rows below, containing the S-Z data.

Each cross section data must end with a row containing stars:

*****.

There are no limits to the number of cross sections which can be imported from a text file.

An example of text file to import two cross sections is provided below:

```
Topo-Current
MainRiver
150
PROFILE 5
0 80.34
3.54 77.89
7.73 76.46
10.91 77.15
```



```
13.61 80.39
*****
Topo-Future
Tributary
0
PROFILE 6
0 75.16
2.15 74.89
3.52 73.21
5.69 73.36
8.46 73.54
11.94 74.79
*****
```

Note that columns can be either space-delimited or tab-delimited. Any white space or tab at the beginning of a row will be ignored during the import process. All numerical values should be entered in the default international units.

Additional cross sections properties, used in MIKE HYDRO River, can also be imported from the text file, by use of additional and optional sections in the text file. Except the location (Topo ID, Branch name and chainage) which must be the first information for each cross section, all other sections can be added in any order in the text file, for each cross section.

Usually these additional sections consist of a first row identifying the MIKE HYDRO River parameter, and one or more rows below with the corresponding values to be imported. Note that it is possible to use the other tool for exporting cross sections, to get an example of text file containing all properties that the importer can deal with. A list of additional properties is provided below:

- PROFILE: additional columns can be added to the right of the S and Z columns, to import distributed resistance numbers (third column) and markers (fourth column). Markers are identified with the following tags:
 - <#1> marker 1
 - <#2> marker 2
 - <#4> marker 3
 - <#8> marker 4
 - <#16> marker 5
 - <#32> marker 6
 - <#64> marker 7

When multiple markers are added to the same point of the cross section, the number to be inserted in the text file corresponds to the sum of num-



bers associated to these markers. For instance, <#6> will import markers 2 and 3 to the same point.

Additionally, the following data may optionally be added:

- User marker, in fifth column
 - X and Y coordinates for each point of the cross section, respectively in eighth and ninth columns. Sixth and seventh columns are not used in MIKE HYDRO River, and should be set to 0 for backward compatibility reason, in order to add coordinates. Note that when importing X and Y coordinates, it is important that specified S values are consistent with distances computed from these coordinates.
- COORDINATES: the line with numerical values first contains an indicator, which can usually be:
 - 0: The coordinates are not applied (disabled in MIKE HYDRO River). In this case, no coordinates values have to be specified afterwards
 - 3: The coordinates are applied. In this case, this indicator is followed by another value on the same line, providing the number of sets of coordinates to be specified. Each set of coordinates is then provided as a separate line below, containing successively S, X, and Y values.
- CLOSED SECTION: This section can be added to import the cross section as a closed section. In this case, a value of 1 has to be added in the row below.
- PROTECT DATA: when this parameter is set to 0, the 'Allow for recalculation' checkmark in MIKE HYDRO River is checked. When the value is 1, it's unchecked.
- DATUM: datum value of the cross section, to be entered in meters.
- RADIUS TYPE : numerical code indicating the radius type of the cross section, which can be defined as:
 - 0 : Resistance radius
 - 1 : Hydraulic radius using effective area
 - 2 : Hydraulic radius using total area.

When this section is not inserted, the default option (Resistance radius) is applied, except for closed sections, where the default is 2.

- SECTION ID: The optional identifier of the cross section.
- INTERPOLATED: the 'Interpolated' flag, shown with a star in the cross sections tree. It is set either to 0 (Not interpolated) or 1 (Interpolated)
- ANGLE: the correction angle used in the processed data. It is followed by a line with two numerical values:
 - The angle value, at first



- A value 0 when the angle correction is not applied, or 1 when applied.
- RESISTANCE NUMBERS: This section contains seven numerical values, as described below and in this order:
 - Transversal distribution, with a value of 0 for Uniform, 1 for High / low flow zones, and 2 for Distributed
 - Resistance type, with a value of 0 for Relative, 1 for Manning's n, 2 for Manning's M, 3 for Chezy, and 4 for Darcy-Weisbach
 - Uniform resistance value
 - Resistance value for left high flow zone
 - Resistance value for low flow zone
 - Resistance value for right high flow zone
 - Resistance value for a vegetation zone, old parameter used by MIKE 11.
- PROCESSED DATA: processed data of cross sections can also be imported, in case you don't want to apply the values which will be recomputed by MIKE HYDRO River. The next two lines are text lines containing headings. Each row of the processed data table is then added as a separate line below, and with the following columns (in this order):
 - Level
 - Cross section area
 - Hydraulic radius
 - Width
 - Additional flooded area
 - Resistance factor.

16.2.9 Exporting cross sections to text file

Via Tools and plugins \ Export to text files, it is possible to export cross section data from MIKE HYDRO River to a text file.

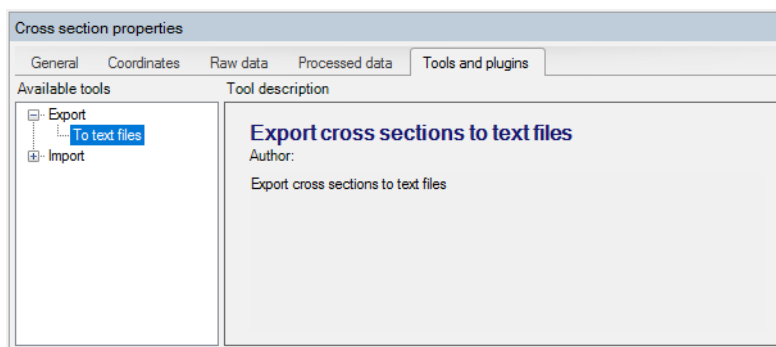


Figure 16.10 Export tool in the cross section editor

Executing the tool 'To text files' opens the dialog in below, where it is possible to select the text file to be created.

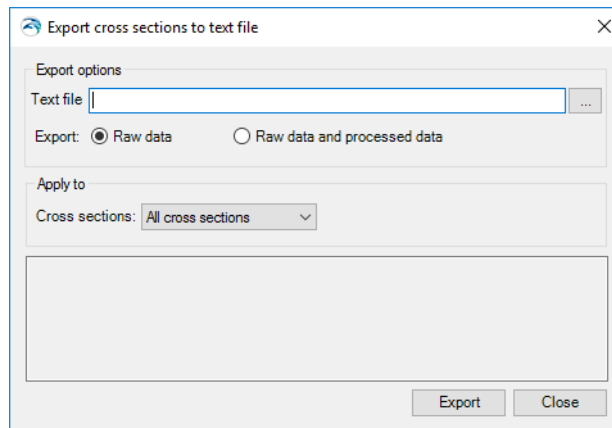


Figure 16.11 Export to text file dialog

In this dialog, you can choose between exporting the raw data only, or the raw data and the processed data of the cross sections. In both cases, most of the properties of the cross sections will be exported (including those used in the Processed data tab), but the only difference is that the table of processed data will not be exported with the first option.

Finally, using 'Apply to', you can choose between exporting all cross sections or only the currently selected ones.





APPENDIX A





A.1 Control Functions

"SimStartTime"

Returns the simulation start time as a DateTime value.

"SimTime"

Returns the current time of the simulation as a DateTime value.

Example: To get the month of the simulation time

```
"Month (SimTime ()) "
```

"SimTimeStep"

Returns the current time step size as a TimeSpan value.

Examples: To get the time step in seconds or hours

```
"TotalSeconds (SimTimeStep ()) "
```

```
"TotalHourss (SimTimeStep ()) "
```

"SimTimeSpan"

Returns the elapsed simulation time, i.e. the time since simulation start time, as a TimeSpan value.

Examples: To get the elapsed simulation time in seconds or hours

```
"TotalSeconds (SimTimeSpan ()) "
```

```
"TotalHourss (SimTimeSpan ()) "
```

"TableLookup"

Based on an input value it will do lookup in a table and return the looked-up value.

Usage:

```
"TableLookup('table-id', input) "
```

Arguments:

- The table-id is a string that identifies the table to use. The table must be supplied to the Control Module, added to the set of TableInfos. Values in the input column of the table must be strictly monotonically increasing.
- The input expression must return a double value. It is usually a sensor, but any expression returning a double value can be used.

Example:

```
"TableLookup('table1', [dischargeSensor]) "
```

**Remark:**

In case the input value is outside the range of the input column in the table, the first/last value in the output column is used.

"TSLookup"

Based on the current simulation time it will do lookup in a time series.

Usage:

```
"TSLookup('ts-id') "  
"TSLookup('ts-id', phaselag) "
```

Arguments:

- The ts-id identifies the time series to use, by a file and an item name/number. It must be supplied to the Control Module, added to the set of TimeSeriesInfos.
- The phaselag expression is a TimeSpan value used as offset from the current simulation time, to look forwards/backwards in time. If the phaselag is a negative time span, it will look backwards in time.

Example:

```
"TSLookup('ts1') "
```

To lookup in the time series 1½ hour back in time from simulation time, use

```
"TSLookup('ts1', #-0.1:30:00#) "  
"TSLookup('ts1', TimeSpanFromHours(-1.5)) "
```

"TSFileLookup"

Based on the current simulation time it will do lookup in an external time series (.dfs0 file).

Usage:

```
"TSFileLookup('tsfilepath', 'tsItemName') "  
"TSFileLookup('tsfilepath', 'tsItemName', #+PhaseLagExpr#) "
```

Arguments:

- The tsfilepath is the full path to the time series. Note that the syntax requires to use double \ symbols, as shown in the examples below.
- The tsItemName identifies the item name/number from the selected file.
- The phaselag expression is an optional TimeSpan value used as offset from the current simulation time, to look forward/backward in time. If the phaselag is a negative time span, it will look backwards in time.



Example:

```
"TSFileLookup('C:\\Data\\USDischarge.dfs0','Qobs')"
```

To lookup in the time series 1½ hour back in time from simulation time, use:

```
"TSFileLookup('C:\\Data\\USDischarge.dfs0', #-0.1:30:00#)"
```

```
"TSFileLookup('C:\\Data\\USDischarge.dfs0', TimeSpanFrom-  
Hours(-1.5))"
```

"TSTableLookup"

Based on the current simulation time it will do lookup in a table containing times in the input column

Usage:

```
"TSTableLookup('table-id')"
```

```
"TSTableLookup('table-id', phaselag)"
```

Arguments:

- The table-id identifies the table, which must be supplied to the Control Module, added to the set of TableInfos. Values in the input column of the table must be strictly monotonically increasing. The input column can contain DateTime values representing absolute time values, or double values representing the number of seconds since simulation start.
- The phaselag expression is a TimeSpan value used as offset from the simulation time, to look forwards/backwards in time. If the phaselag is a negative time span, it will look backwards in time.

Example:

```
"TSTableLookup('tsTable1')"
```

To lookup in the time series 1½ hour back in time from simulation time, use

```
"TSTableLookup('tsTable1', #-0.1:30:00#)"
```

```
"TSTableLookup('tsTable1', TimeSpanFromHours(-1.5))"
```

Remark:

In case the table input values do not include the current simulation time, the first/last value in the output column is used.

"PreviousInTime"

Based on an input value it will return the value as it was some time back.

Usage:

```
"PreviousInTime(input, timeBack)"
```



```
"PreviousInTime(input, timeBack, 'nointerp')"
```

Arguments:

- The input expression must return a double value. It is often a sensor, but any ex-pression returning a double can be used.
- The timeBack expression must be either a double or a TimeSpan value. A double value is interpreted as a number of hours.
- The option nointerp will imply that time interpolation will not take place, but the first value just after the requested time will be used.

Examples: To get the value 1½ hour back in time

```
"PreviousInTime([dischargeSensor], 1.5)"
```

```
"PreviousInTime([dischargeSensor], #+0.1:30:00#)"
```

"TimeSinceChange"

Based on an input value it will record the time of the last change and return the time since that change. The return value is a TimeSpan value

Usage:

```
" TimeSinceChange(input) "
```

Arguments:

- The input expression must return a double value. It is usually a sensor, but any expression returning a double value can be used.

Example:

```
"TimeSinceChange([GateLevelSensor])"
```

To get the number of hours since the gate changed:

```
"TotalHours(TimeSinceChange([GateLevelSensor]))"
```

Remarks:

This is relevant for gate levels or similar. For sensors that change every time step this function will always return zero.

"MinInTime"

Based on an input value it will return the minimum value within a specified time interval back in time from the current simulation time.

Usage:

```
"MinInTime(input, startTime, endTime)"
```



Arguments:

- The input expression must return a double value. It is usually a sensor, but any expression returning a double value can be used.
- The startTime expression must be either a double or a TimeSpan value. A double value is interpreted as a number of hours.
- The endTime expression must be either a double or a TimeSpan value. A double value is interpreted as a number of hours.

The startTime must return a larger value than the end time.

Examples: To get the minimum value between 1½ and ½ hour from the current simulation time

```
"MinInTime([dischargeSensor], 1.5, 0.5)"
```

```
"MinInTime([dischargeSensor], #+0.1:30:00#, #+0.0:30:00#)"
```

"MaxInTime"

Based on an input value it will return the maximum value within a specified time interval back in time from the current simulation time.

Usage:

```
"MaxInTime(input, startTime, endTime)"
```

Arguments:

- The input expression must return a double value. It is usually a sensor, but any expression returning a double value can be used.
- The startTime expression must be either a double or a TimeSpan value. A double value is interpreted as a number of hours.
- The endTime expression must be either a double or a TimeSpan value. A double value is interpreted as a number of hours.

The startTime must return a larger value than the end time.

Examples: To get the maximum value between 1½ and ½ hour from the current simulation time

```
"MaxInTime([dischargeSensor], 1.5, 0.5)"
```

```
"MaxInTime([dischargeSensor], #+0.1:30:00#, #+0.0:30:00#)"
```

"DiffInTime"

Based on an input value it will return the difference between the value at two specified times back in time from the current simulation time.

Usage:



```
"DiffInTime(input, startTime, endTime)"
```

Arguments:

- The input expression must return a double value. It is usually a sensor, but any expression returning a double value can be used.
- The startTime expression must be either a double or a TimeSpan value. A double value is interpreted as a number of hours.
- The endTime expression must be either a double or a TimeSpan value. A double value is interpreted as a number of hours.

The startTime must return a larger value than the end time.

Examples: To get the difference in the value between 1½ and ½ hour from the current simulation time

```
"DiffInTime([dischargeSensor], 1.5, 0.5)"
```

```
"DiffInTime([dischargeSensor], #+0.1:30:00#, #+0.0:30:00#)"
```

"TimeDerivative"

Based on an input value it will return the time derivative over a specified time interval back in time from the current simulation time.

Usage:

```
"TimeDerivative(input, startTime, endTime)"
```

Arguments:

- The input expression must return a double value. It is usually a sensor, but any expression returning a double value can be used.
- The startTime expression must be either a double or a TimeSpan value. A double value is interpreted as a number of hours.
- The endTime expression must be either a double or a TimeSpan value. A double value is interpreted as a number of hours.

The startTime must return a larger value than the end time.

Examples: To get the time derivative over one hour, from 1½ to ½ hour back in time from the current simulation time

```
"TimeDerivative ([dischargeSensor], 1.5, 0.5)"
```

```
"TimeDerivative ([dischargeSensor], #+0.1:30:00#,  
#+0.0:30:00#)"
```



"Average"

Based on an input value it will return the average value within a specified time interval back in time from the current simulation time.

Usage:

```
"Average(input, startTime, endTime)"
```

```
"Average(input, startTime, endTime, 'inside')"
```

Arguments:

- The input expression must return a double value. It is usually a sensor, but any expression returning a double value can be used.
- The startTime expression must be either a double or a TimeSpan value. A double value is interpreted as a number of hours.
- The endTime expression must be either a double or a TimeSpan value. A double value is interpreted as a number of hours.
- The option inside implies that only timesteps fully inside the time interval is included in the average.

The startTime must return a larger value than the end time.

Examples: To get the average value between 1½ and ½ hour from the current simulation time

```
"Average([dischargeSensor], 1.5, 0.5)"
```

```
"Average([dischargeSensor], #+0.1:30:00#, #+0.0:30:00#)"
```

"AverageIf"

Based on an input value it will return the average value within a specified time interval back in time from the current simulation time, including the value in the average if some condition is fulfilled.

Usage:

```
"AverageIf(input, condition, startTime, endTime)"
```

```
"AverageIf(input, condition, startTime, endTime, 'inside')"
```

Arguments:

- The input expression must return a double value. It is usually a sensor, but any expression returning a double value can be used.
- The condition expression must return a boolean value.



- The `startTime` expression must be either a double or a `TimeSpan` value. A double value is interpreted as a number of hours.
- The `endTime` expression must be either a double or a `TimeSpan` value. A double value is interpreted as a number of hours.
- The option `inside` implies that only timesteps fully inside the time interval is in-cluded in the average.

The `startTime` must return a larger value than the end time.

Example: To get the minimum value between 1½ and ½ hour from the current simulation time

```
"AverageIf([dischargeSensor], [dischargeSensor] > 0, 1.5, 0.5) "
```

Remarks:

The condition is evaluated together with the input expression at every time step, and the input expression is only stored for processing if the condition at the current simulation time evaluates to true.

"Accumulate" and "TimeIntegrate"

Based on an input value it will accumulate/time integrate the value over time, either since start of simulation or within a specified time interval back in time from the current simulation time.

Usage:

```
"TimeIntegrate(input) "  
"TimeIntegrate(input, startTime, endTime) "  
"TimeIntegrate(input, startTime, endTime, 'inside') "
```

Arguments:

- The input expression must return a double value. It is usually a sensor, but any expression returning a double value can be used.
- The `startTime` expression must be either a double or a `TimeSpan` value. A double value is interpreted as a number of hours.
- The `endTime` expression must be either a double or a `TimeSpan` value. A double value is interpreted as a number of hours.
- The option `inside` implies that only timesteps fully inside the time interval is in-cluded in the average.

If the start/end times are not provided, accumulation/integration from simulation start time till current time is performed.



The `startTime` must return a larger value than the end time.

Examples: To get the minimum value between 1½ and ½ hour from the current simulation time

```
"TimeIntegrate([dischargeSensor], 1.5, 0.5) "  
  
"TimeIntegrate([dischargeSensor], #+0.1:30:00#,  
#+0.0:30:00#) "
```

Remarks:

Accumulation and time-integration are two terms covering the same procedure. The term `Accumulate` is usually used for discharge type inputs, accumulating the total volume passing through some point. Time integration is used more generally

"AccumulateIf" and "TimeIntegrateIf"

Based on an input value it will accumulate/time integrate the value within a specified time interval back in time from the current simulation time, if some condition is fulfilled.

Usage:

```
"TimeIntegrateIf(input, condition, startTime, endTime) "  
  
"TimeIntegrateIf(input, condition, startTime, endTime,  
'inside') "
```

Arguments:

- The input expression must return a double value. It is usually a sensor, but any expression returning a double value can be used.
- The condition expression must return a boolean value.
- The `startTime` expression must be either a double or a `TimeSpan` value. A double value is interpreted as a number of hours.
- The `endTime` expression must be either a double or a `TimeSpan` value. A double value is interpreted as a number of hours.
- The option `inside` implies that only timesteps fully inside the time interval is included in the average.

The `startTime` must return a larger value than the end time.

Example: To get the minimum value between 1½ and ½ hour from the current simulation time

```
"TimeIntegrateIf([dischargeSensor], [dischargeSensor] > 0,  
1.5, 0.5) "
```

**Remarks:**

Accumulation and time-integration are two terms covering the same procedure. The term Accumulate is usually used for discharge type inputs, accumulating the total volume passing through some point. Time integration is used more generally.

The condition is evaluated together with the input expression at every time step, and the input value is only stored for processing if the condition at the current simulation time evaluates to true.



INDEX



A	
AD / WQ boundary	235
Advection-Dispersion Parameters	205
Autocalibration	172
B	
Background layers	66
Background maps	66
C	
Calibration parameters	172
Calibration plot	162
Catchment area	162
Catchment properties	162
Catchments	159
Computational parameters	62
Controls	37
Coordinate system	65
Create a new document	17, 43
Create Catchment	159
Create new dfs0 file	37
Create new table file	39
Create new time series file	37
Create River network	72
D	
Delineated catchments	160
Derive the River network from a DEM	73
Digitising a River network	72
E	
Elevation zones	168
G	
General tab	74, 162
Groundwater	172
I	
Importing a River network	73
Initial conditions	171
Irrigation	170
M	
Manning formula	82
Map view	22
N	
NAM	54
NAM model	163
O	
Objective functions	173
Open Examples	34, 41
Output window	25
P	
Properties view	22
Properties window	24
R	
Rainfall-runoff	54, 163
Rating Curve approach	82
Result file-types	272
Results specifications	271
Results tree view	22
River modules	54
River network	71
Routing	80
Running the autocalibration	174
Runoff parameters	164
S	
Seasonal variation	174
Setup tree view	19
Simulation description	56
Simulation period	56
Simulation specifications	53
Simulation tab	25
Snow Storage	172
Storages	164
Storing frequency	272
Storing Results	271
Surface and rootzone	171
Surface-rootzone	164
T	
Time series	175
Time series tab	26
Time Step Control	57
Tree view	18
U	
UHM	55, 176
UHM (Unit Hydrograph) model	176
UHM parameters	177
User Interface	18



Index

V

Validation tab 25

W

Wind Friction 240

Working area 69

