MIKE+

Collection System

User Guide

PLEASE NOTE

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

1 General Settings

The General Settings section contains the ‘Modules’ and ‘Description’ editors.

1.1 Model Type

The ‘Model type’ editor provides an ‘at a glance’ view of which MIKE+ features and modules are active. Active features and modules control the menus being accessible from the Setup tree view.

Within the "Rivers, collection system and overland flows" working mode, you can choose to have the following features activated:

- Catchments
- Collection system network
- River network
- 2D overland

Catchments can be connected to both Collection system and River networks. Collection system and River networks can also be connected together. When 2D overland is enabled with Collection system network and/or river network, couplings between the various features are enabled automatically.

You can then activate the following modules, which will apply to all selected features whenever relevant:

- Rainfall Runoff (RR): The RR module simulates hydrological processes on catchments. Enabling the Catchments feature but without including the RR module allows applying catchment discharge to the collection system and river networks.
- Hydrodynamic (HD): The HD module simulates the flows in collection system networks, river networks and/or 2D overland, depending on the selected features. The HD module includes the two sub-modules Control Rules (only apply applicable to Collection system and River networks) and Long Term Statistics (LTS), only applicable to the Collection system network.
- Transport (AD, SWQ): The transport module simulates the Advection-Dispersion processes (for e.g. pollution transport) in collection system networks, river networks and/or 2D overland, depending on the selected features. The transport option also enables stormwater quality (SWQ) simulations on catchments, when Catchment feature and Rainfall-Runoff module are active.
- Water quality (MIKE ECO Lab): The MIKE ECO Lab module simulates the advanced water quality processes (e.g. biological processes) in collection system networks, river networks and/or 2D overland, depending on the selected features.

- Sediment transport (ST): The ST module simulates transport of sediments in collection system and river networks. It is not available for 2D overland.
- Data assimilation (DA): The DA module includes 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 real-time forecasting where the forecasted values of network state conditions (water levels, flows, etc.) are much more accurate and reliable when update and error estimation are included. It is designed for river networks.

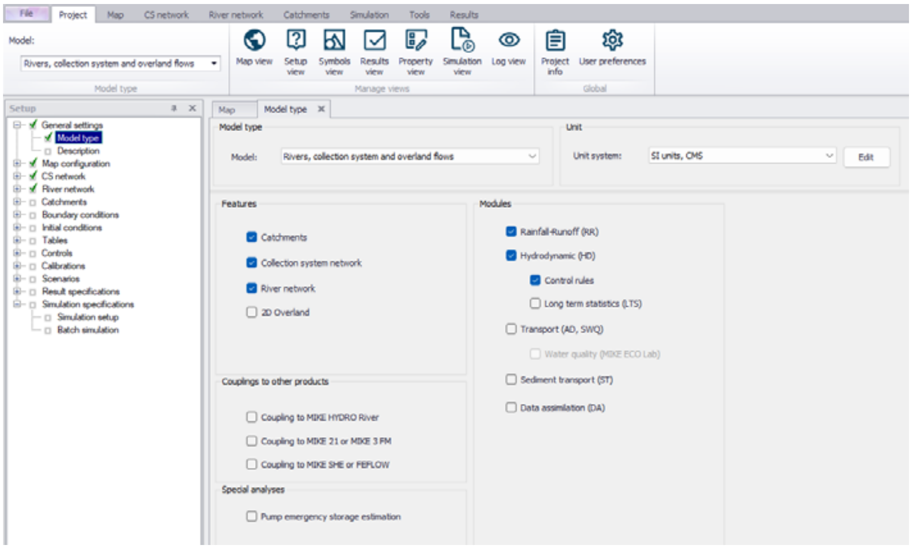


Figure 1.1 Model type and module selection

The following couplings to other MIKE products are also available:

- Coupling to MIKE HYDRO River: When this is active, the desired MIKE HYDRO model setup file must be selected in the 'River model' page. Note that coupling AD module with MIKE HYDRO is allowed and automatically enabled when the 'Transport (AD, SWQ)' module is also included in the simulation setup. AD components' names in MIKE HYDRO must, however, be identical to the components' names in MIKE+.
- Coupling to MIKE 21 or MIKE 3 Flow Model FM: When this is active, the desired MIKE 21 FM or MIKE 3 FM model setup file must be selected in the 2D / 3D overland model' page. Note that this coupling cannot be enabled at the same time as the 2D overland feature, since only one source of overland model can be used at a time.

- Coupling to MIKE SHE or FEFLOW: When this is active, it is possible to define parts of the river network to be coupled to a MIKE SHE or FEFLOW model. The network simulation file (*.m1dx file) must then be exported from MIKE+ for use in MIKE SHE or FEFLOW. This also allows coupling the collection system network to MIKE SHE: in that case, the corresponding coupling file (*.couple) must also be exported from MIKE+ for use in MIKE SHE.

When the different features and modules are activated, the corresponding menus will be added in the Setup view. Note that the features and modules are actually included in a simulation only if they are also included the simulation definition, from the 'Simulation setup' editor.

The 'Pump emergency storage estimation' option enables a special type of analysis for collection system networks, enabling extra simulation options under the 'Special analysis' group in the Setup tree view. It estimates the time and volume available prior to an overflow at the wastewater pump station assets in the event of system failure.

1.2 Description

The General Settings section contains the Description editor. This editor allows addition of information about the project and a free text description of the model. It may also be used as a model build log to make notes on updates and amendments.

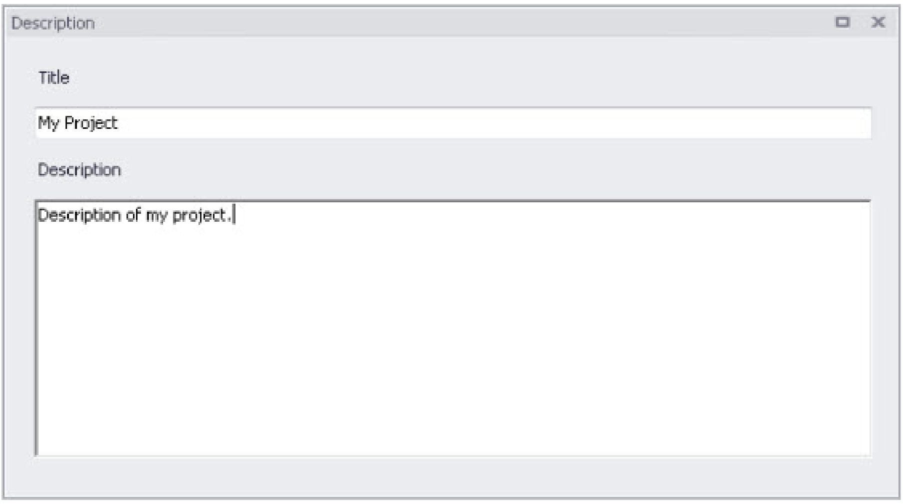


Figure 1.2 The Description Editor

2 Map Configuration

The Map Configuration section contains information on the coordinate system used in the MIKE+ project and presents options for customising the background image.

2.1 Coordinate System

The Coordinate System dialog (Figure 2.1) displays the Projection system used in the project.

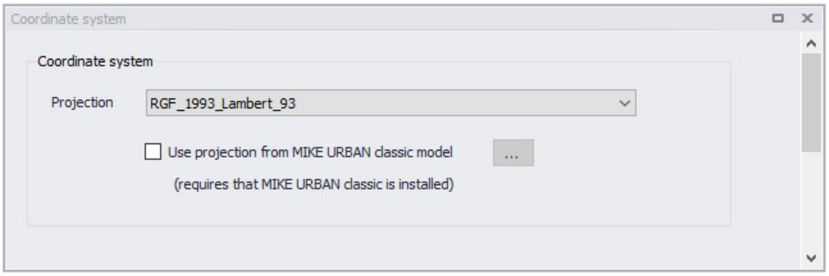


Figure 2.1 The Coordinate System dialog showing information on the projection system used in the project

The Projection can be selected from the short list, or by searching the projection amongst all the map projections available in MIKE+. The latter is achieved by selecting the <Browse…> option at the bMIKE+ottom of the list: this will open a window listing the available projections, and where it is also possible to import new projections from a projection file (*.prj file).

Alternatively, the map projection may be read from a MIKE URBAN classic file. If MIKE URBAN classic is installed on your computer, you can tick the corresponding option, select a MIKE URBAN classic file and the same projection will be used afterwards in MIKE+.

When changing the map projection, it is possible to reproject geographical data in the project, for example to convert the coordinates of the network and catchments data, or mesh arcs used for the creation of the 2D domain. Some data files used as input for the simulation can however not be re-projected: this is especially the case for an external 2D domain file (*.mesh or *.dfs2) or external 2D data file used to map input parameters (e.g. *.dfsu or *.dfs2 file used to map the 2D surface roughness).

The same options for selecting the Projection are also used in the 'New Module Setup' window when a new MIKE+ project is created (Figure 2.2).

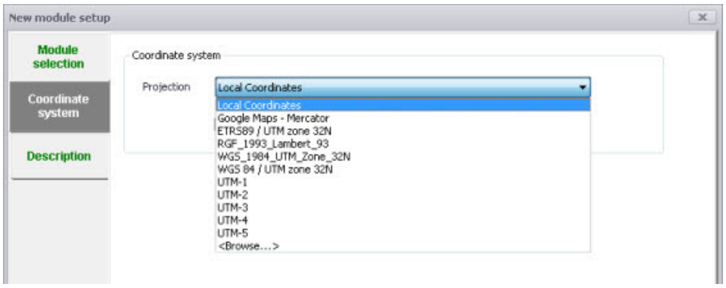


Figure 2.2 Specify the projection system for a new MIKE+ project on the New Module Setup window

2.2 Background Map

The Background Map editor allows the user to select a background image to show on the Map View in MIKE+ (Figure 2.3).

Activate a background map overlay by ticking the ‘Background map Visible’ checkbox on the editor. When a background map is visible, increasing the 'Transparency' with the corresponding cursor fades out the background map, e.g. to increase the visibility of model data.

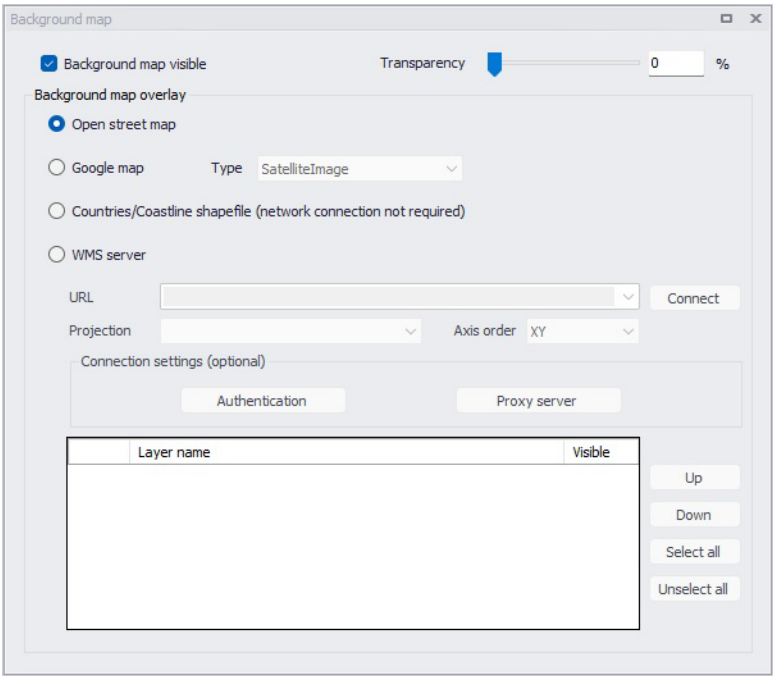


Figure 2.3 The Background Map Editor

The following background map overlay options are available:

- **Open Street Map**
- **Google Map**. Select the Google map type to display (i.e. Street map, Satellite image, Terrain, or Hybrid).
- **Countries/Coastline Shapefile**. Polygon feature showing coastlines and demarcating oceans and inland areas.
- **WMS server**. Background maps obtained from a remote server. Enter the URL of the server or select it from the list of previously used URLs, and click 'Connect'. If the server is a private server, you will need to supply the user name and password by using the ‘Authentication’ button, and you may optionally tick 'Save password' in order not to enter it again the next time you open MIKE+. If your network uses a proxy server, you will need to press the 'Proxy server' button to provide the server address (which should include the port number, if any) as well as optional user name and password for this proxy server. When the connection is established, the table will provide the list of layers available on the WMS server, and it is possible to select which layers to display in MIKE+ using the 'Visible' box. The list of projections will show the map projection(s) supported by the WMS server, and the one used for the model data in MIKE+ will be selected if possible. Note that displaying layers from a WMS server requires that the MIKE+ project uses the same map projection as the WMS layers: if the projection used in MIKE+ doesn't match any of the projections supported by the WMS server, you will be asked to update the map projection in MIKE+. Also note that it is only possible to connect to WMS servers using projected map projections (geographical coordinate systems not supported). An axis order also needs to be specified, defining the format of the coordinates on the WMS server: most of the servers provide coordinates in the XY order, but some servers provide coordinates in the opposite order and in this case the option must be changed to 'YX' otherwise the layers won't be displayed on the map.

An internet connection must be available for Open Street Map, Google Map and WMS server overlays (Figure 2.4).

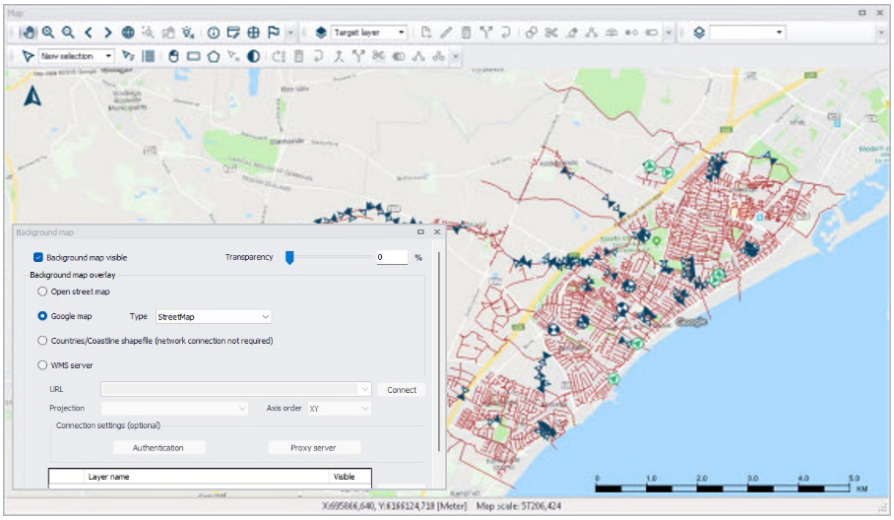


Figure 2.4 An example Google Map background on the Map View in MIKE+

Note: Because the proxy server settings are related to your local network and not to the model database, these settings are stored on the local machine instead of the MIKE+ project files. That means that the specified settings will then apply to other MIKE+ projects opened on the same computer. On the opposite, the proxy settings will have to be supplied again if the same model database is later opened from another computer connected to the same network.

Note: While connecting to WMS servers, in case of connection errors, it is possible to save the connection information to a log file. To enable this, start by locating the file named DHI.MIKEPlus.Shell.exe.config in the installation folder. In this file, search for the option 'SaveWMSLog' and set it to true. The information from the following connections will be saved in a log file named with the MIKE+ database name, in the folder "%appdata%\DHI\MIKE+\BackgroundMapCache\WmsRasterLayer"

3 Hydraulic Network Modelling

3.1 Introduction

MIKE 1D allows for the hydrodynamic simulation of flows and water levels in urban storm drainage and wastewater collection networks, thus providing an accurate information about the network functionality under a variety of boundary conditions. The hydrodynamic simulations can be extended with pollution, sediment transport and water-quality simulations. The model can also be enhanced by the variety of real-time control functions. The simulations can be carried out for single events or as efficient long-term simulations for longer historical periods.

This chapter provides a comprehensive guideline for the preparation of the basic MIKE 1D hydrodynamic simulation models. Information related to Control, Long Term Statistics, Water Quality etc. can be found in respective chapters of this manual.

Modelling of network hydrodynamics in MIKE 1D requires understanding of the information requirements. On the other hand, detailed knowledge of the computational theory is not essential.

The modelling process consists of the following distinct steps:

- Definition of the network data
- Specification of the boundary conditions
- Adjustment of the computation parameters and running the simulations
- Result analysis.

Furthermore, an important part of successful modelling is related to the model calibration and verification, which must ensure that the computed results fit reasonably well with the flow observations. These are important engineering activities in the modelling process.

3.2 Definition of a MIKE 1D Network

A MIKE 1D network within MIKE+ can be defined in one of the following ways:

- Import of existing MIKE+ CS Project
- Import of a backed up MIKE URBAN Classic project saved as a ".mdb" file
- Import of external data (e.g. GIS) into MIKE+ CS network
- Graphically digitizing and manual data typing within MIKE+

The last option is frequently used in a combination with one of the previous options as means for achieving a full consistency of the MIKE+ model.

The following paragraphs provide a comprehensive information on the MIKE 1D network data model and the associated editors.

A model consists of the following hydraulic elements:

- Nodes and Structures
- Pipes and Canals
- Weirs
- Orifices
- Curb Inlets
- Pumps
- Valves

3.2.1 Modelling real network elements

When setting up a model some knowledge of the principles used in the numerical solution of the flow equations is useful. This section will provide some information, for further please refer to the “MIKE 1D Reference Manual”.

In all pipes and canals the computational grid is set up in an alternating sequence of h- and Q-points. In these grid points the discharge Q and water level h, respectively, are computed at each time step. The links (pipes and canals) will always be setup with h-grid points at each end where the link connects to nodes in the network. This means that links will always have an odd number of computational grid points with three points (h - Q - h) as the minimum configuration.

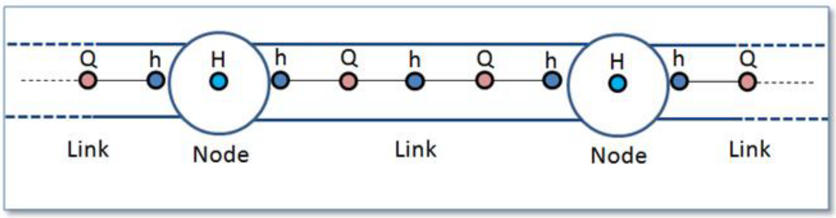


Figure 3.1 The computational grid

The nodes will only have a single computational point where the water level H is computed. The nodes are typically circular manholes in the sewer network. But it can also be basins or tanks with a significant volume. Still only a single water level computational point is located at the node. Based on the computed water level and the description of the geometry of the node the computation keeps track of the volume of water stored in the node.

It is of importance to notice that only a water level is computed at the nodes. In the simple case with one incoming pipe to a node and one outgoing pipe it may seem simple to compute a "flow through" the node. But think of the more complex situations with more than two pipes connected and also external flow entering the node. Defining a "flow through the node" is in the general situation not possible.

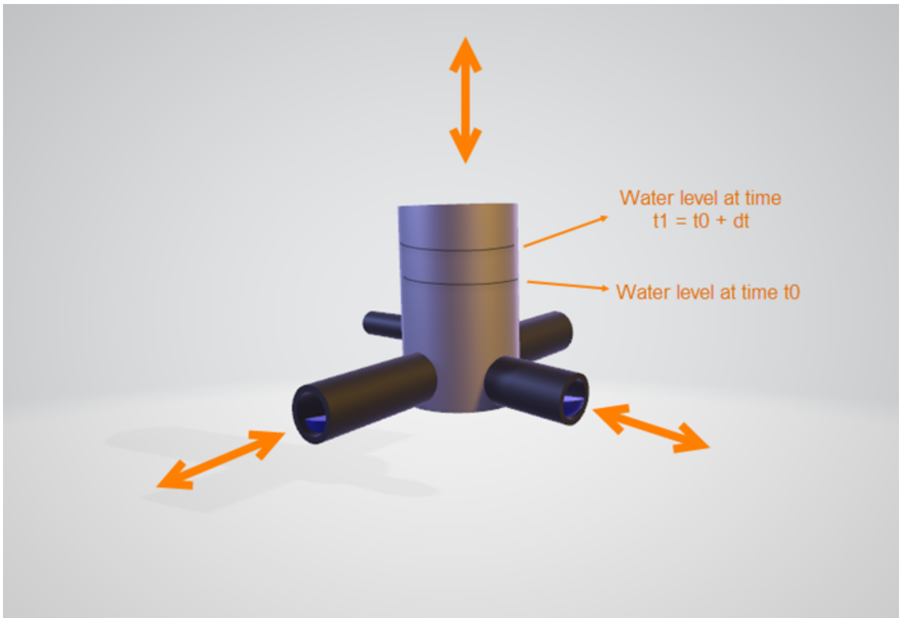


Figure 3.2 Water flowing through a node

At the nodes the water level is computed based on the water level at the previous time step and the flow contributions during the time step from each connected pipe and external connected flow like a catchment runoff discharge. When the computational grid is set up for a network of links and nodes it will end up like shown in Figure 3.3.

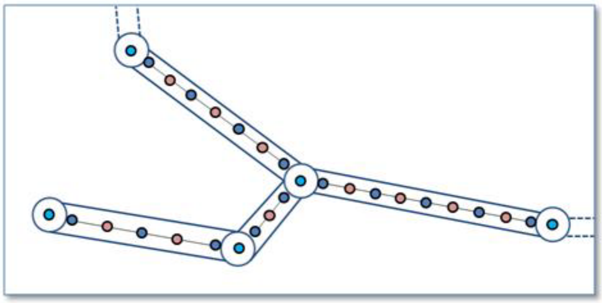


Figure 3.3 The computational grid for a given network

MIKE1D is able to handle various “devices” which basically are related to manholes, basins, soakaways or other constructions in the sewer network. These devices are: pumps, weirs, orifices, valves and storm water inlets. Typically these elements are placed at locations which in the real system could be manholes, basins or other structures. It is also characteristic for all mentioned elements that there will be a discharge computed for the device: pump discharge, discharge over weir, flow through orifice and flow through valve.

The main point to realize is the conflict between computing a discharge for these elements and the fact that only a water level is computed at nodes.

This is why the pump, weir, orifice, valve and storm water inlet elements from the computational and numerical point of view are links and not an element placed in one node. All the elements are links forming a connection between two nodes.

In MIKE+ we have five functional elements which from the model building point of view are related to nodes like manholes or basins. These are pumps, weirs, valves, orifices and storm-water inlets. The concept of elements related to nodes is reflected in the design of the dialog for editing the parameters for these elements. Here you find a field named "From" for all of the elements. The field takes the ID of a node as input. All elements also have a field for "To:" which also takes a node ID as input.

Seen from the computational solution point of view the five elements are actually connections from one node to another node. This is similar to how pipes are defining the link for flow between nodes as reflected in the dialog where you find fields for entering "From node:" and "To node:".

3.3 Nodes

This editor allows defining node elements such as manholes, basins, soakaways or outlets.

The editor organizes the related input data into the following groups:

- Identification: ID and location of the node
- Geometry: nodes's type and shape
- Cover: cover type for manhole
- Flow regulation: maximum inflow; in/exfiltration;
- Head loss: information on the head loss approach and coefficients
- Pressure node: used for tail nodes of pressurized branches.
- Soakaway: infiltration parameters
- Description: descriptive fields

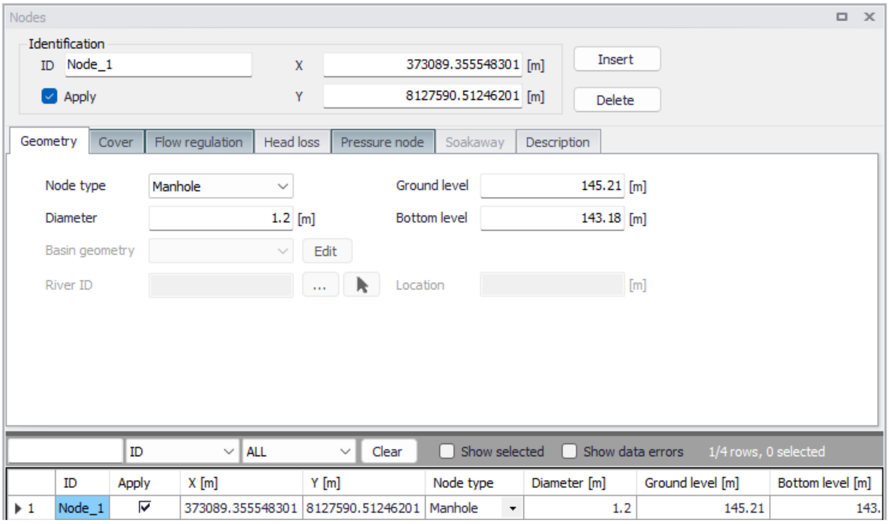


Figure 3.4 Nodes and Structures editor

The buttons Insert and Delete allows addition and deletion of nodes in the editor.

3.3.1 Identification

General properties for nodes are specified under the Identification group:

- **ID**: Unique identifier for the node.
- **X and Y coordinates**: The X and Y coordinates are used to define the physical (map) location of the node. When defining the node graphically on the Map using the drawing tools, the X and Y coordinates are automatically filled.
- **Apply**: This check box allows to toggle the active status of the node on and off. The simulations will omit all nodes that are not active.

Table 3.1 The edit fields in the Identification group (Table msm_Node)

Edit field	Description	Used or required by simulations	Field name in data structure
NodeID	A unique name for the node. Up to 40 characters (letters, numbers, blank spaces and underscore characters). Identical names with different case are not allowed	Yes	MUID
X coordinate	X coordinate of the node position	Yes	GeomX
Y coordinate	Y coordinate of the node position	Yes	GeomY
Apply	This check box allows to toggle the active status of the node on and off. The simulations will omit all nodes that are not active.	Yes	Enabled

3.3.2 Geometry

MIKE+ supports six types of nodes: circular manholes, basins, soakaway, outlets, junctions and river junctions. The same editor is used for all node types, but it adapts according to the selected node type.

Junction nodes are nodes without storage.

River junction nodes are similar to outlets, but they connect to a river, modelled using the 'river network' module. For this type of node, the connected River ID and the chainage of the connection must be specified.

For basin and soakaway nodes, a geometry table must be selected from the list. Geometry tables are defined in the 'Curves and relations' editor, and must be defined with the curve type 'Basin geometry'.

Soakaway nodes are used for hydraulic modelling of the green solutions. A soakaway represents a generic type of LID control as it can represent a number of different WSUD controls.

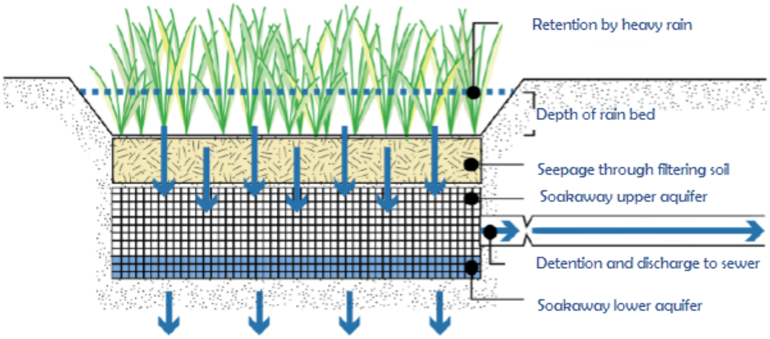


Figure 3.5 Conceptual drawing of a soakaway

A schematic drawing of a soakaway (Bioretention cell) is illustrated in Figure 3.5. The stormwater drains from the surface and enters the soakaway at the upper vegetated layer. Then the stormwater infiltrates vertically through the soakaway and infiltrates out of the sides and bottom of the soakaway. In some cases the soakaway is not connected to any drainage network and captured runoff to the soakaway is infiltrated and in case of extreme rainfall and exceedance of its infiltration and storage capacity, storm water is surcharged to the surface. In other cases the soakaway is connected to the drainage network by a flow controlled outlet pipe as illustrated in Figure 3.6. During extreme rainfall causing exceedance of its infiltration, storage and outlet flow capacity, the soakaway also surcharges to the surface.

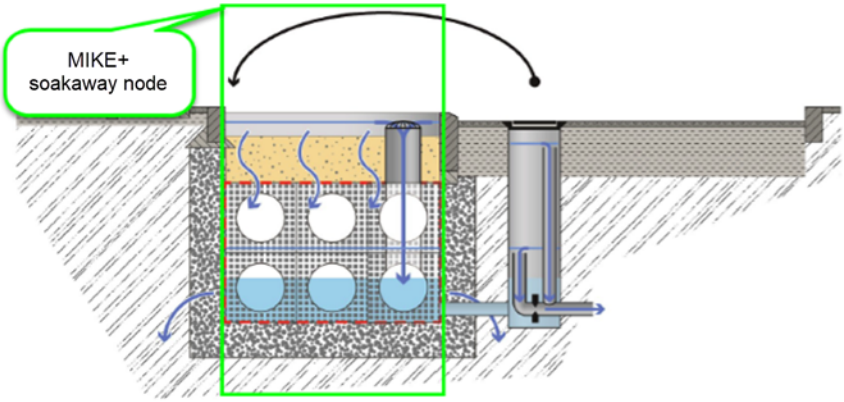


Figure 3.6 Soakaway concept in MIKE+

Table 3.2 The edit fields in the Geometry tab (Table msm_Node)

Edit field	Description	Used or required by simulations	Field name in data structure
Node type	Node Types: 1. Manhole - node with shaft and chamber storage 2. Basin - node with a more complex geometry 3. Outlet - node where water leaves the system (no storage) 4. Junction - node without storage 5. Soakaway - node with a similar geometric complexity as a basin but with the features of infiltration and internal porosity. 6. River junction - node connecting one end of a collection system network to a river.	Yes	TypeNo
Diameter	Diameter of the manhole - not enabled for any other node types	Yes	Diameter
Ground level	Ground level of the node	Yes	Groundlevel
Bottom level	Bottom level of the manhole	Yes	InvertLevel

Table 3.2 The edit fields in the Geometry tab (Table msm_Node)

Edit field	Description	Used or required by simulations	Field name in data structure
Basin geometry	Reference to a tabulated area-elevation function for the basin or soakaway geometry. The H-column for the basin geometry can start at any value, e.g. 0.0 for interpretation of H as depth in the basin. The MIKE 1D Engine will associate the first H-value to the bottom level of the node.	Yes	GeometryID
RiverID	The connected River ID, for a river junction node	Yes	BranchID
Location	The chainage location along the river, where the river junction node will discharge into the river.	Yes	BranchChainage

3.3.3 Cover

Manholes and basins are per default considered open at the top (Cover type equal to 'Normal'). This means, that when the water level in a node reaches the ground level, the water spills on the ground surface. In that case, MIKE 1D introduces an artificial basin on the top of the node, with a surface area 1000x larger (by default) than the node's surface. The surcharged water is stored in the basin, to be returned back into the sewer. Note that this 1000 factor can be customized in the 'MIKE 1D engine configuration' dialog.

Alternatively, a ‘Sealed’ cover type can be defined, i.e. a node with a fixed lid on the top - at the ground level - so water cannot escape although the pressure still builds up inside.

The node can also be specified as a 'spilling' node (Cover type equal to 'Spilling'). In a spilling node, water escapes irreversibly from the model, if the water level reaches and exceeds the node's ground level (optionally set off by a 'buffer pressure level). The rate of spill is approximated as a free overflow over the crest at a given level and with a “conceptual” crest length. For further details, see the MIKE 1D Reference Manual.

Table 3.3 The edit fields in the Cover tab (Table msm_Node)

Edit field	Description	Used or required by simulations	Field name in data structure
Cover type	Choose among available types: 1. Normal 2. Sealed 3. Spilling	Yes	CoverTypeNo
Buffer pressure	Buffer pressure is only active for type = spilling. Equal to the pressure above the ground level needed to cause spills from the manhole.	Yes	BufferPressure
Spill coefficient	Spill coefficient is only active for type = spilling. Controls the spill capacity.	Yes	SpillCoeff

3.3.4 Flow regulation

The use of a Q-H relation is only available for outlet nodes. Specifying a Q-H relation for an outlet controls the flow at the outlet. The flow (Q) value in the Q-H relation should be given as a positive value when water enters the node and a negative value for specifying a loss of water from the network model. The Q-H relation must be defined in the 'Curves and relations' editor, and with curve type 'QH relation'.

Table 3.4 The edit fields in the Flow regulation tab (Table msm_Node)

Edit field	Description	Used or required by simulations	Field name in data structure
Use maximum flow	Activates the inlet delimiter function	Yes	InletControlNo
Max Inflow value	Value of maximum possible inflow into the node from runoff	Yes	MaxInlet

Table 3.4 The edit fields in the Flow regulation tab (Table msm_Node)

Edit field	Description	Used or required by simulations	Field name in data structure
Use QH relation	When toggled on, uses Q-H tabulated function, toggled off calculates on the basis of water level.	Yes	QHTypeNo
QH relation ID	Reference to a tabu-lated QH relation	Yes	OutletQHID

3.3.5 Head loss

From the 'Head loss' tab, it is possible to select which numerical method is used to compute the outlet head loss in the node. The method can be selected from the list defined in the 'Tables | Outlet Head Loss' editor, where it is also possible to edit their properties. Please refer to the 'Outlet Head Loss' chapter for more information on the individual methods.

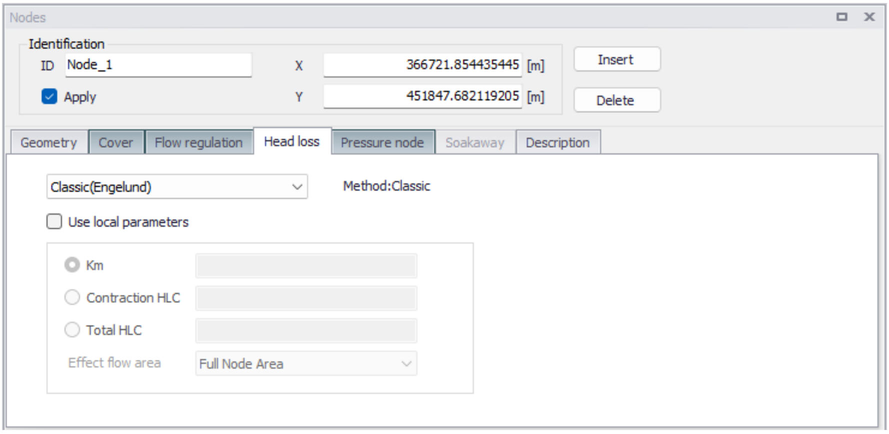


Figure 3.7 The Head Loss settings in the Nodes editor

When activating the 'Use local parameters' option, it is possible to override the settings of the selected method with values defined for the current node. When doing so, the radio button selection allows to select the loss coefficient type and value, and the 'Effect flow area' allows selecting the method for computing the effective node area.

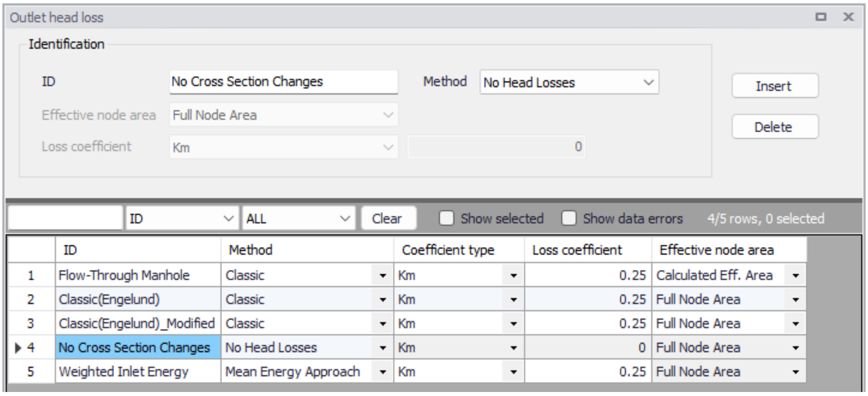


Figure 3.8 Outlet head loss editor

.

Table 3.5 The edit fields in the Head Loss tab (Table msm_Node)

Edit field	Description	Used or required by simulations	Field name in data structure
Drop-down list	Selection of the head loss calculation solution for the node	Yes	LossParID
Use local parameters	Enable local parameters in the node, overriding the default parameters of the selected method	Yes	LossParNo
Loss coefficient type radio buttons	Locally defined interpretation of head-loss coefficient. Km = "shape coefficient", Contraction HLC = outlet contraction head loss coeff. (relative to velocity head), Total HLC = outlet total head loss (relative to velocity head)	Yes	LossTypeNo

Table 3.5 The edit fields in the Head Loss tab (Table msm_Node)

Edit field	Description	Used or required by simulations	Field name in data structure
Coefficient value fields	Local value of loss coefficient	Yes	LossCoeffKm LossCoeffContrac-tion LossCoeffTotal
Effect. flow area	Locally defined choice of method for the cal-culation of wetted area 1.Full Node Area 2. Calculated Effective Area 3. Reduced calcu-lated Effective Area	Yes	EffAreaNo

3.3.6 Pressure node

This tab allows to declare the node as the downstream point of a pressure main's connection to the network. Manholes and basins can be declared as "Tail Node". When a node is defined as tail node, a water level (absolute elevation) needs to be specified for the "receiving node", used as lower boundary for permanently pressurized parts of the system. Please refer to the MIKE 1D Reference Manual for further information on pressure mains.

Table 3.6 The edit fields in the Pressure Node tab (Table msm_Node)

Edit field	Description	Used or required by simulations	Field name in data structure
Pressurized tail node	When selected, defines the node in the pressure main as the down-stream point of the pressure main's connection to the network.	Yes	PMTTypeNo
Tail level	Water level in the "receiving node".	Yes	PMLevel

3.3.7 Soakaway

A soakaway is modelled as a regular basin in MIKE+, but with the difference that it includes infiltration. Its infiltration settings are defined in the 'Soakaway' tab. Since a soakaway is a regular node, it can be used in various configuration as with other node types: it can e.g. receive inflow from connected upstream pipes or discharge into downstream pipes, with or without flow regulation, be connected to weirs or orifices, etc. Soakaway nodes can also be coupled in series to support the modelling of constructed infiltration trenches. When the soakaway is connected to a downstream pipe or structure, make sure that the upstream level of this link is set correctly, and not at the bottom of the basin which is the default if no link invert level is specified. If the link is connected at the bottom, then make sure that flow regulation is applied to the out-going pipe.

The inflow to the soakaway can also be defined as for any node. It can be received from an upstream pipe, from a boundary condition to the node, or

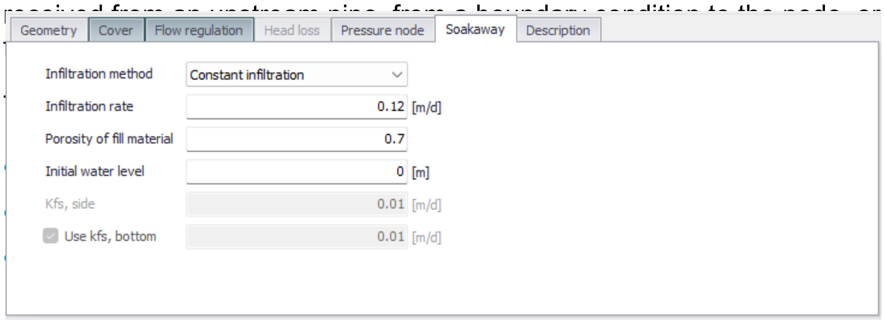


Figure 3.9 Soakaway settings in the Nodes Editor

The option 'No Infiltration' is used in cases when there is no infiltration out of the soakaway. The initial water level can be set.

The 'Constant Infiltration' option provides the functionality of defining a constant infiltration rate out of the soakaway. The input required for this option is the Infiltration rate, the porosity of the fill material and the initial water level.

The 'Infiltration' option provides the functionality of having a variation in the infiltration based on the water level in the soakaway.

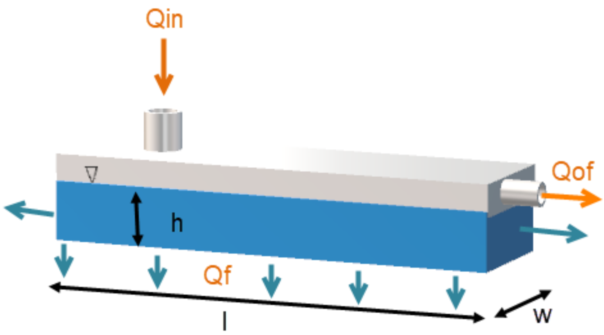


Figure 3.10 Schematic of the soakaway model

The 'Infiltration' option is based on the infiltration rate calculated by Equation (3.1) and Equation (3.2). Equation (3.1) describes the water balance of the model. Equation (3.2) and Equation (3.3) describe how the infiltration rate is calculated:

$$\frac{dh}{dt} = \frac{1}{l \cdot w \cdot \theta}(Q_{in}(t) - Q_f(t) - Q_{of}(t)) \tag{3.1}$$

$$Q_f = K(l \cdot w + 2h(l + w)) \tag{3.2}$$

where θ is the soakaway porosity and h the calculated water level.

In MIKE 1D the infiltration rate calculated by Equation (3.2) is rewritten to Equation (3.3) to be based on the basin geometry definition in MIKE+ as well as to support different hydraulic conductivity at the side and at the bottom.

$$Q_f = K_{fs,\,bottom} \cdot A_{s,\,h\,=\,0} + K_{fs,\,side} \cdot \left(2A_c + 2\frac{Vol}{A_c}d\right) \tag{3.3}$$

where $K_{fs,bottom}$ is the field-saturated hydraulic conductivity at the bottom, $K_{fs,side}$ is the field-saturated hydraulic conductivity at the side, A_s is the surface area and A_c the cross-sectional area.

The infiltration from the bottom can be turned off by a flag. However the infiltration from both side and bottom can be shut off by setting the field-saturated hydraulic conductivity to zero.

The porosity of the fill material is used to calculate the water level in the soakaway and the initial water level is used to set the initial water level in the soakaway. Table values of hydraulic conductivity, K_{fs}, for different soil classes are provided in Table 3.7. Within each soil type the hydraulic conductivity varies significantly, therefore it is important to measure the hydraulic conductivity at the site.

Table 3.7 Hydraulic conductivity for different soil classes

Soil classification	Hydraulic Conductivity [m/s]
Gravel	0.001 to 0.1
Sand	10^{-5} to 10^{-2}
Silt	10^{-9} to 10^{-5}
Clay	Below 10^{-9} to 10^{-2}
Moraine	10^{-10} to 10^{-6}

Table 3.8 The edit fields in the Soakaway tab (Table msm_Node)

Edit field	Description	Used or required by simulations	Field name in data structure
Infiltration method	Method of infiltration	Yes	InfiltrationNo
Infiltration rate	Defines the infiltration rate	Yes	InfConstValue
Porosity of fill material	Porosity of filling material	Yes	PorosityFill
Initial water level	Initial water level in soakaway	Yes	InitialWL
Kfs, side	Conductivity of soil on the sides of the soakaway	Yes	KfsSide
[Tickmark]	Activates the bottom conductivity function	Yes	KfsBottomNo
Kfs, bottom	Conductivity of soil on the sides of the soakaway	Yes	KfsBottom

3.3.8 Description

The Description tab is where free text descriptions for nodes data may be added. It offers options for providing asset and model data management information, as well as attributes for quick model data query.

Table 3.9 The edit fields in the Description tab (Table msm_Node)

Edit field	Description	Used or required by simulations	Field name in data structure
Description	User's descriptive information related to the node	No	Description
Data source	Reference to an external data source (e.g. table ID) where the record has been imported from	No	DataSource
Asset ID	Reference to an ID used in external data sources	No	AssesName

Table 3.9 The edit fields in the Description tab (Table msm_Node)

Edit field	Description	Used or required by simulations	Field name in data structure
Status	Data status for the entire record, serves for keeping track on the source of information	Mp	Element_S
Network type	Associates the node to a certain type of network. Used in cases when two or more different networks are included in the same project	No	NetTypeNo
Critical level	User defined critical level. Used in result presentations and in the Pump Emergency Storage Estimation analysis	Yes	CriticalLevel
Model	Associates the current node to a specified submodel	No	SubModelNo

3.4 Pipes and Canals

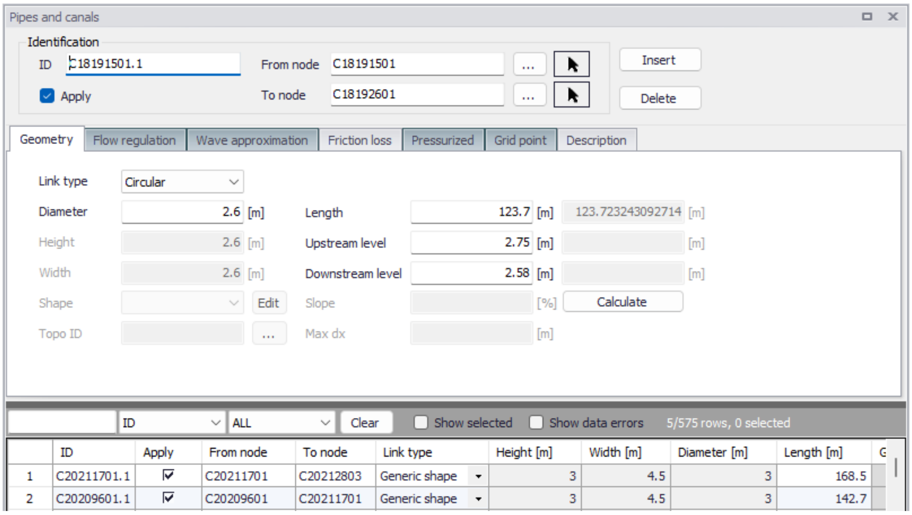


Figure 3.11 Pipes and Canals editor

A link (pipe or canal) is specified as a conduit between two nodes. A link is considered as either a straight line or a drawn polyline between two nodes and per default is assumed to connect the adjacent nodes at bottom levels.

3.4.1 Identification

General properties for pipes and canals are specified under the Identification group:

- **ID**: Unique identifier for a link.
- **From node**: ID of the connected node at the start of the link. This ID can be typed in manually, selected from a list using the corresponding ellipsis button or picked on the map using the arrow button.
- **To node**: ID of the connected node at the end of the link. This ID can be typed in manually, selected from a list using the corresponding ellipsis button or picked on the map using the arrow button.

In the computations the flow is considered positive when the water flows from the 'From node' to the 'To node'. Therefore, it is recommended to specify the nodes in the direction of predominant flows.

Table 3.10 The edit fields in the Identification group (Table msm_Link)

Edit field	Description	Used or required by simulations	Field name in data structure
Link ID	A unique name for the node. Up to 40 characters (letters, numbers, blank spaces and underscore characters)	Yes	MUID
From node	Upstream Node	Yes	FromNodeID
To node	Downstream Node	Yes	ToNodeID
Apply	This check box allows to toggle the Active status of the link on and off. The simulations will omit all links that are not active.	Yes	Enabled

3.4.2 Geometry

Depending on the selected type, a link may take the form of one of the 'standard' pipes (Circular, Rectangular, O Shaped, Egg-Shaped), or any closed or open generic shape (same cross section all along the pipe) or a natural channel (varying shape along the channel described with multiple cross sections).

Standard pipes are defined by diameter (or width and height for non-circular pipes).

For a pipe with a generic shape, the topography must be defined in the 'Generic shapes' editor, and the relevant shape ID must be selected in the 'Shape' list from the 'Pipes and canals' editor.

For a natural channel, the topography is defined by multiple cross sections, in the 'Cross sections' editor. In this cross sections editor, data are saved to an external cross sections file (*.xns11), and multiple 'Topo ID' can be defined for each natural channel. A Topo ID is a set of cross sections, containing all the cross sections along the natural channel. The possibility of storing multiple Topo IDs allows to define different topographies, for example before and after the construction of infrastructures in the channel. From the 'Pipes and canals' editor, the relevant Topo ID is selected in the corresponding field: pressing the '…' button will show the list of Topo IDs for the active natural channel, so it is important that the same natural channel ID is applied in both the 'Pipes and canals' editor and in the 'Cross sections' editor.

The length of a link is calculated from the shape of the line in MIKE+. The length is displayed in the 'Geometric length' field, and it is updated when the geometric reference is modified. If a user defined length is specified in the 'Length' field, this will overwrite the calculated one during simulation.

Per default, a link is assumed to connect the adjacent nodes at their bottom levels, which are shown for information in the fields 'UpLevel_C' and 'DwLevel_C', respectively.In case of a step-wise connection to the nodes (but not allowed below node bottom level), the elevations of both the upstream and downstream inverts of the link must be specified in the editable "'UpLevel'" and "'DwLevel'" fields.

Table 3.11 Geometry (Table msm_Link)

Edt field	Description	Used or required by simulations	Field name in data structure
Link type	Shape of pipe	Yes	TypeNo
Diameter	Nominel size of pipe (diameter of circular pipe, height of Egg-shape pipe and width for O-shaped)	Yes, if Link type = Circular, Egg-Shape and O-Shaped	Diameter
Width	Width of rectangular shape	Yes, if Link type = Rectangular	Width
Height	Height of rectangular shape	Yes, if Link type = Rectangular	Height

Table 3.11 Geometry (Table msm_Link)

Edt field	Description	Used or required by simulations	Field name in data structure
Shape	ID of the generic shape	Yes, if Link type = Generic shape	CrsID
Topo ID	ID of topography, i.e. ID of the set of cross-sections as defined in the Cross sections editor	Yes, if Link type = natural channel	TopographyID
Max Dx	Max distance between gridpoints	Yes, if Link type = natural channel	Maxdx
Length	User-defined length of link	Yes The geometric length will be used if left empty	Length
UpLevel	Upstream invert level of link	Yes	UpLevel
DwLevel	Downstream invert level of link	Yes	DwLevel

3.4.3 Flow Regulation

The 'Flow regulation' tab provides access to inserting a regulation in the selected link. This regulation does not require the Control module. The regulation can be either a maximum discharge as a function of the water level in a user specified node (Ctrl. Node A) or a maximum discharge as a function of the water level difference between two user specified nodes (Ctrl. Node A and Ctrl. Node B).

The option to apply a non-return valve to allow the water flowing in only one direction is also available.

Table 3.12 Flow regulation (Table msm_Link)

Edit field	Description	Used or required by simulations	Field name in data structures
Use flow regulation [Tickmark]	Allows the option to regulate flow to a given link	Yes, if flow regulation is chosen	FlowRegNo
Regulation type	Select the type of function desired to regulate the flow	Yes, if flow regulation is chosen	RegulaitonTypeNo

Table 3.12 Flow regulation (Table msm_Link)

Edit field	Description	Used or required by simulations	Field name in data structures
Function ID	Select the function that regulates the flow	Yes, if flow regulation is chosen	FunctionID
Non return valve [Tickmark]	Allows flow through the link in one direction only.	No	NonReturnNo

3.4.4 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.

By default, the wave approximation is selected for the entire CS network in the MIKE 1D engine configuration dialog. But it is possible to apply a different wave approximation for some pipes and canals. When the option 'Use specified local wave approximation' is selected, then it is possible to select the wave approximation for the current pipe / canal, which will take priority on the global wave approximation.

Table 3.13 Local wave approximation (Table msm_Link)

Edit field	Description	Used or required by simulations	Field name in data structures
Use specified local wave approximation [Tickmark]	Replaces the global wave approximation type by a local type for the pipe	No	SpecLocalWaveNo
Wave approximation type	Selects the type of wave approximation desired for the current pipe	Yes, if specified local wave approximation is chosen	WaveApproximationTypeNo

When the 'Resistance formulation' is set to Colebrook-White or Hazen-Williams in the friction loss settings, then the wave approximation cannot be selected and is always set to 'FullyDynamicImplicitFriction'.

3.4.5 Friction losses

The resistance in the link can be computed using the following formulations:

- Manning (M): the resistance is characterized by a Manning (M) value.
- Manning (n): the resistance is characterized by a Manning (n) value.

- Colebrook-White: the resistance is characterized by an Equivalent roughness value.
- Hazen-Williams: the resistance is characterized by a Hazen-Williams coefficient.

Once the resistance formulation is selected, three definition types are available:

- Use material: select the material of the link from the list. The corresponding resistance value is shown on the left of the list for information. Specification of the different kinds of materials and their roughness coefficients is done through the 'Tables | Materials' editor. Changing the resistance value for a material affects all links defined with this material.
- Use local parameters: the resistance is defined specifically for the current link.
- Depth-dependent friction loss: the resistance value varies with an exponential low from the bottom to the top of the link.

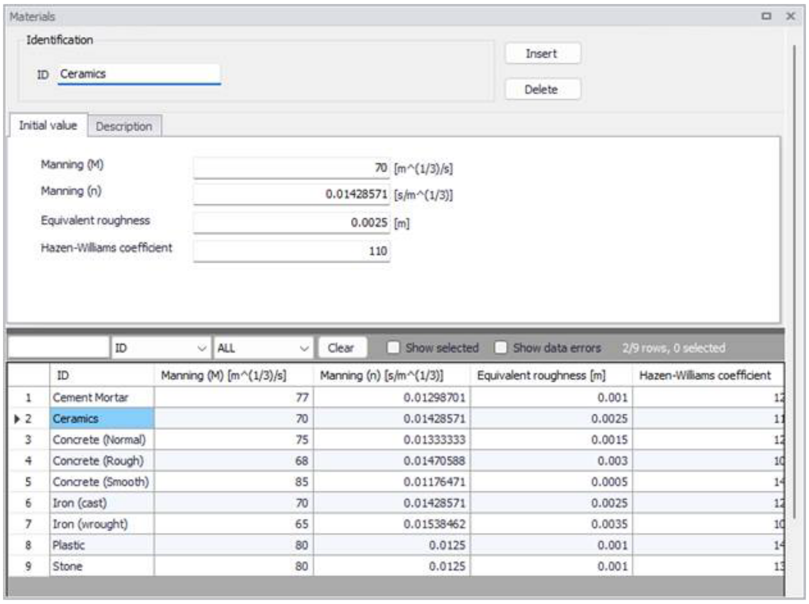


Figure 3.12 Materials Editor

Table 3.14 Hydraulic friction losses (Table msm_Link)

Edit field	Description	Used or required by simulations	Field name in data structure
Material	Material of link	Yes	MaterialID
Resistance formulation	Formula for calculation of the friction loss (Manning (M), Manning (n), Colebrook White, Hazen-Williams)	Yes	FricTypeNo
Friction loss type (radio buttons)	Determines if roughness values are defined from the material, from a local value or using a depth-dependent law	Yes	FricNo
Manning	Manning roughness value	Yes, if 'Manning (M)' or 'Manning (n)' is chosen	Manning
Eq. roughness	Equivalent roughness	Yes, if 'Colebrook White' formulation is chosen	Rough
H-W coef	Hazen-Williams roughness coefficient	Yes, if 'Hazen-Williams' is chosen	HWCoef
Top	Depth-variable Manning number on Top	Yes, if depth-dependent friction is selected	ManTop
Bottom	Depth-variable Manning number on Bottom	Yes, if depth-dependent friction is selected	ManBott
Exponent		Yes, if depth-dependent friction is selected	ManExp

3.4.6 Pressurized

Pipes permanently running under pressure are specified by setting the tick-mark in "Pressure main" in the "Pressurized" tab. Please refer to the MIKE1D

Reference Manual (Pipe Flow) for further information on pressure main branches. This option is not supported for link types defined as Generic shapes or Natural channels.

The option to control the Preissmann slot ratio is also available.

Table 3.15 The edit fields in the Pressurized tab (Table msm_Link)

Edit field	Description	Used or required by simulations	Field name in data structures
Pressure main	Defines a link as pressure main. A link connected to a manhole or basin, can only constitute a pressure main if the manhole/basin is declared to be "tail node". Please refer to the MIKE1D Reference Manual (Pipe Flow) for further information on pressure mains.	No	PMApprNo
Use speci-fied Pre-iss-mann slot ratio	Enable the use of a user-defined Preiss-mann slot ratio	No	SlotNo
Slot ratio value	Rising main conver-sion factor	Yes, if 'Use speci-fied Pre-iss-mann slot ratio' is chosen	Slot

3.4.7 Grid point

Links will often be simulated with three calculation points: a h-point at the start and end of the link (where e.g. water level is computed) and one Q-point in the middle (where e.g. discharge and velocity are computed). When the link is long enough, intermediate calculation points are automatically added.

By activating the option 'Use specified grid points number', it is possible to force using a specific number of calculation points.

Table 3.16 The edit fields in the Grid points tab (Table msm_Link)

Edit field	Description	Used or required by simulations	Field name in data structures
Use specified grid points number	Activates the use of a user-defined number of grid points	No	GridNo
Number of grid points	The specified number of grid points for the link	Yes, if 'Use specified grid points number' is chosen	Grid

3.4.8 Description

The Description tab page is where free text descriptions for links data may be added. It offers options for providing asset and model data management, as well as attributes for quick model data query.

Table 3.17 The edit fields in the Description tab (Table msm_Link)

Edit field	Description	Used or required by simulations	Field name in data structures
Asset ID	Reference to an ID used in external data sources	No	AssetName
Data source	Reference to an external data source (table ID) where the record has been imported from	No	DataSource
Element status	Data status for the entire record, serves for keeping track on the source of information	No	Element_S
Description	User's descriptive information related to the link	No	Description
Network type	Attributes the link to a certain type of network. Used in cases when two or more different networks are included in the same project	No	NetTypeNo

3.5 Weirs

A weir is actually a functional relation, which connects two nodes of a MIKE 1D urban network (two-directional flow and submerged flow possible), or is associated with only one node (free flow 'out of the system'). The latter case is achieved if the 'To' field is left empty.

Real urban sewer systems configure a weir to be located in a manhole or a similar construction which you normally would define as a node in the digital model. The numerical solutions for the flow equations, however, need a model configuration with two nodes where the weir is defined as the connection between the nodes. The weir will then be placed between the two nodes as the flow connection.

It is possible to define several weirs between the same two nodes if this is required. This is similar to the possibility of having more than one pipe as the link between nodes. The generation of the computational grid shown in Figure 3.14 for the orifice is also applied for pumps, weirs and valves. The numerical solution of the flow equations will depend on the selected device. Please refer to the reference manual on more on this.

It is recommended not to place the two nodes in the same spot, instead place the nodes a short distance apart. The reason is that the node head loss computation will have a component from change of flow direction. If the two nodes surrounding the device are placed exactly at the same location then the computational engine cannot determine the direction of the flow from the coordinates of the nodes and a default direction will be applied. This may unintentionally introduce a change in direction and therefore also an unexpected head loss.

By using a small displacement of the nodes the change in flow direction will be determined based on the coordinates and angles between the connected pipes. Therefore consider carefully the placement of the nodes with respect to the actual construction.

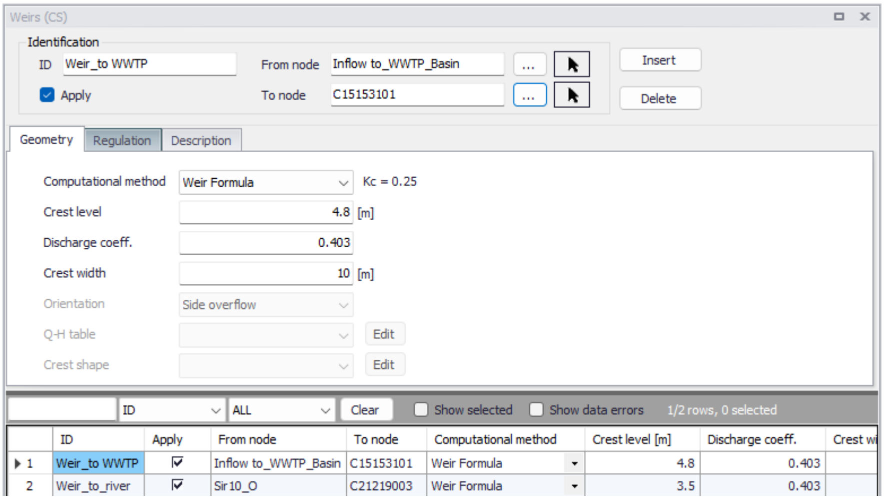


Figure 3.13 Weirs editor

A weir is primarily characterised by its computational method, i.e. equation type, which can be:

- Weir formula: with this method, the weir is described with a crest level, a discharge coefficient and a crest width, using the Overflow formula (broad crested weir).
- Energy loss coefficient: with this method, the weir is described with a crest level, a crest width and an orientation, using the Overflow formula (broad crested weir). The orientation plays an important role, since depending on the specified orientation, kinetic energy of the flow is included (90o) or excluded (0o) in calculations of the weir flows.
- Q-H: with this method, the weir is described with a crest level and a Q-H table, using the QH structure formula.
- CRS weir formula: with this method, the weir is described with a crest level, a discharge coefficient and a crest geometry, using an orifice formula.

Weirs are per default static (No control) but can be controlled using control rules. When 'Controlled by control rules' is selected, the weir crest level may be controlled during the simulation using control rules (unless the weir is a Q-H type weir).

There are no limitations on the number of weirs specified at one location.

3.5.1 Identification and connectivity

Table 3.18 Description (Table msm_Weir)

Edit field	Description	Used or required by simulations	Field name in data structure
Description	User's descriptive information related to the weir	No	Description
Data source	Reference to an external data source (table ID) where the record has been imported from	No	DataSource
Asset ID	Reference to an ID used in external data sources	No	AssetName
Status	Data status for the entire record, serves for keeping track on the source of information	No	Element_S
Network type	Attributes the weir to a certain type of network. Used in cases when two or more different networks are included in the same project	No	NetTypeNo
Weir ID	A unique name for the weir. Up to 40 characters (letters, numbers, blank spaces and underscore characters)	Yes	MUID
From node	ID of Node where Weir is located	Yes	MUID
To node	ID of Node where Weir is discharging to. If field left empty, then water is discharging out of the system	Yes	MUID
Apply	This check box allows to toggle the Active status of the weir on and off. The simulations will omit all weirs that are not active.	Yes	Enabled

3.5.2 Model data

Table 3.19 Weir Geometry and Regulation (Table msm_Weir)

Edit field	Description	Used or required by simulations	Field name in data structure
Computa-tional method	Selection of computation Method	Yes	MethodNo
Crest level	Crest level of weir	Yes	CrestLevel
Discharge coeff.	Discharge coefficient	Yes, if weir formula is chosen	Coeff
Crest width	Width of rectangular weir	Yes, if weir formula is chosen	CrestWidth
Orientation	Weir orientation relative to the main flow direction. “0” is Side weir, “90” is a transversal weir	Yes, if dis-charge coeff. is not speci-fied	AngleNo
Q-H table	Reference to tabulated Q-H funtion	Yes, if Q-H is chosen	QHID
Crest Geometry	Reference to tabulated variation of the weir crest along the weir, CRS geome-try	Yes	WeirCrestID
Non return flap	Flap indicating a flap-gate built-in weir (i.e. no return flow possible)	Yes	FlapNo
Controlled by control rules	If selected, the weir is controlled using control rules as defined in the ‘Control rules’ editor. Control rules will apply only when both this check box and the ‘Apply’ check box in the control rule definition are ticked. Note that the regulation with control rules is always disabled, when the 'Control rules' module is unselected in the 'Model type' editor.	Yes, if control rules apply	Con-trolTypeNo
Max level	The maximum elevation of a movable weir crest	Yes, if control rules apply	Max-CrestLevel
Min level	The minimum elevation of a movable weir crest. The fixed weir is not used with control rules	Yes, if control rules apply	Min-CrestLevel

Table 3.19 Weir Geometry and Regulation (Table msm_Weir)

Edit field	Description	Used or required by simulations	Field name in data structure
Max speed up	The maximum velocity for movement of the weir in upward direction	Yes, if control rules apply	Max-CrestLevelIn-creaseSpeed
Max speed down	The maximum velocity for movement of the weir in downward direction	Yes, if control rules apply	Max-CrestLevel-decreaseSpeed

3.6 Orifices

An orifice is actually a functional relation, which connects two nodes of a MIKE 1D urban network or is associated with only one node (free flow 'out of the system'). The latter case is achieved if the 'To node' field is left empty.

In real urban systems a flow restriction in the form of an orifice may be located in a manhole or a similar construction which you normally would define as a node in the model configuration. The numerical solutions for the flow equations, however, need a model configuration with two nodes where the orifice is defined as the connection between the nodes. The orifice will then be placed between the two nodes as the flow connection.

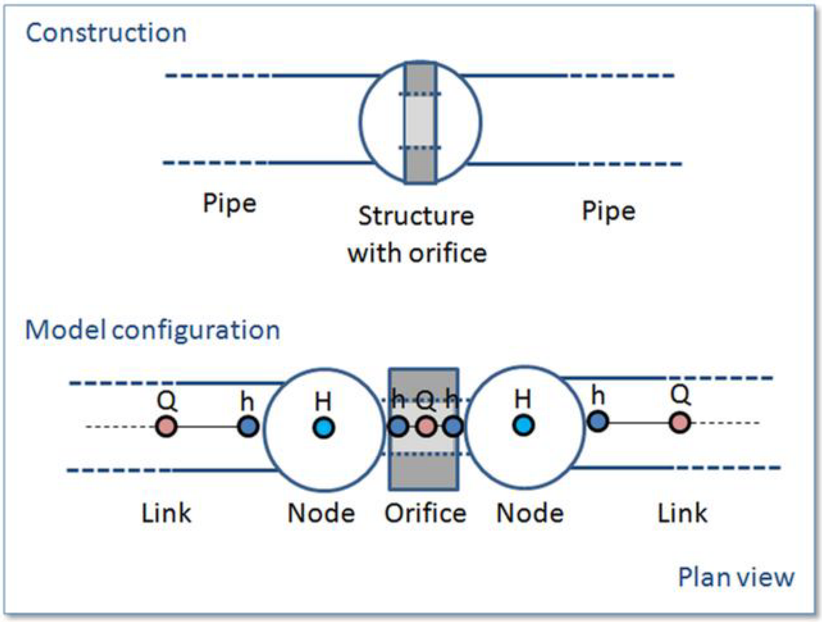


Figure 3.14 The difference between real world orifice and model configuration of orifice

It is possible to define several orifices between the same two nodes if this is required. This is similar to the possibility of having more than one pipe as the link between nodes. The generation of the computational grid shown in Figure 3.14 for the orifice is also applied for pumps, weirs and valves. The numerical solution of the flow equations will depend on the selected device. Please refer to the reference manual on more on this.

It is recommended not to place the two nodes in the same spot, instead place the nodes a short distance apart. The reason is that the node head loss computation will have a component from change of flow direction. If the two nodes surrounding the device are placed exactly at the same location then the computational engine cannot determine the direction of the flow from the coordinates of the nodes and a default direction will be applied. This may unintentionally introduce a change in direction and therefore also an unexpected head loss.

By using a small displacement of the nodes the change in flow direction will be determined based on the coordinates and angles between the connected pipes. Therefore consider carefully the placement of the nodes with respect to the actual construction.

An orifice is specified by a type; circular, CRS or rectangular, and the corresponding diameter, height and width.

A discharge coefficient can be specified (default = 1.0) and a non-return flap can be specified.

Orifices are per default static (No Control) but an orifice can be controlled by control rules. Selecting the 'Controlled by control rules option', the opening of weirs / gates in orifices can be controlled during the simulations

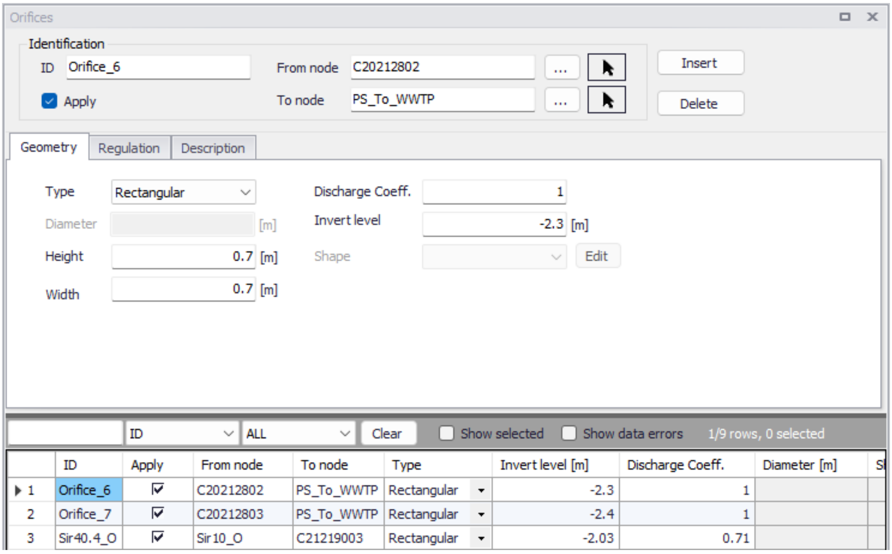


Figure 3.15 Orifice editor

3.6.1 Identification and connectivity

Table 3.20 Description (Table msm_Orifice)

Edit field	Description	Used or required by simulations	Field name in data structure
Description	User's descriptive information related to the orifice	No	Description
Data source	Reference to an external data source (table ID) where the record has been imported from	No	DataSource
Asset ID	Reference to an ID used in external data sources	No	AssetName
Status	Data status for the entire record, serves for keeping track on the source of information	No	Element_S

Table 3.20 Description (Table msm_Orifice)

Edit field	Description	Used or required by simulations	Field name in data structure
Network type	Attributes the link to a certain type of network. Used in cases when two or more different networks are included in the same project	No	NetTypeNo
Orifice ID	A unique name for the orifice. Up to 40 characters (letters, numbers, blank spaces and underscore characters)	Yes	MUID
From node	ID of Node where orifice is located	Yes	FromNodeID
To node	ID of Node where orifice is discharg-ing to. If field left empty, then water is discharging out of the system	Yes	ToNodeID
Apply	This check box allows to toggle the Active status of the orifice on and off. The simulations will omit all orifices that are not active.	Yes	Enabled

3.6.2 Model data

Table 3.21 Geometry (Table msm_Orifice)

Edit field	Description	Used or required by simulations	Field name in data structure
Type	Type of orifice according to shape, being rectangular, circular or CRS	Yes	TypeNo
Diameter	Diameter of circular orifice	Yes	Diameter
Height	Height of rectangular orifice	Yes	Height
Width	Width of rectangular orifice	Yes	Width
Discharge coeff	Calibration coefficient. Value = 1 results in the flow as determined by orifice algorithm	Yes	Dis-chargeCoeff
Invert level	Absolute elevation of the orifice invert	Yes	InvertLevel
CRS Geometry ID	Reference of a cross-section ID for irregularly-shaped orifice	Yes, if CRS type is cho-sen	CrsID

Table 3.21 Geometry (Table msm_Orifice)

Edit field	Description	Used or required by simulations	Field name in data structure
Non return flap	Flap indicating a flap-gate built-in (i.e. no return flow possible)	Yes	FlapNo
Controlled by control rules	If selected, the orifice is controlled using control rules, as defined in the 'Control rules' editor. Control rules will apply only when both this check box and the 'Apply' check box in the control rule definition are ticked. Note that the regulation with control rules is always disabled, when the 'Control rules' module is unselected in the 'Model type' editor.	Yes	ControlTypeNo

3.6.3 Defining a gate or a weir in an orifice

The orifice itself is just an opening with a static shape. In real constructions orifices are often equipped with a controlled gate or weir which can be used in real time control for regulating the flow through the orifice. The gate device will move from the top of the orifice opening and downwards until the orifice is fully closed. The weir moves from the bottom of the orifice upwards and closes fully when the weir crest reaches the top of the orifice opening (see Figure 3.16 for an illustration). It is possible to apply both types of movable devices in the computations. In both cases the device is “added” to a defined orifice. This is done from the “Control Rules” dialog.

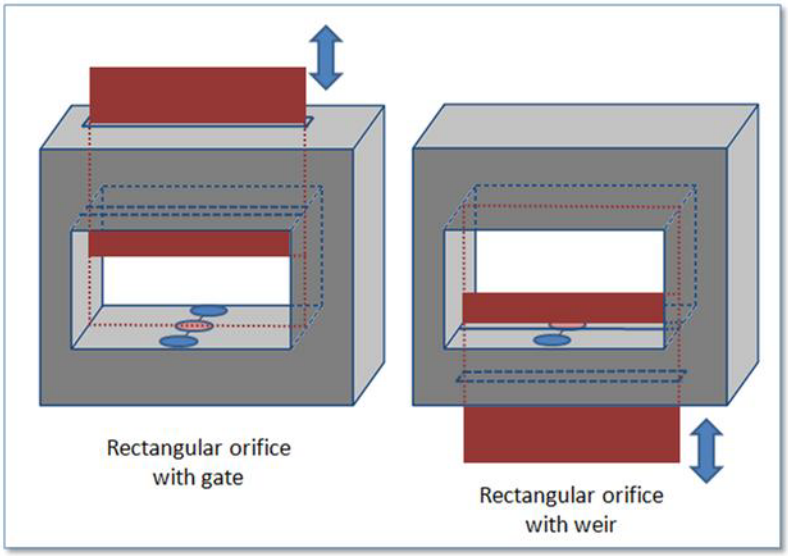


Figure 3.16 Examples on a rectangular orifice with a gate and a weir

3.7 Valves

A valve is a functional relation which connects two nodes of a MIKE 1D urban network.

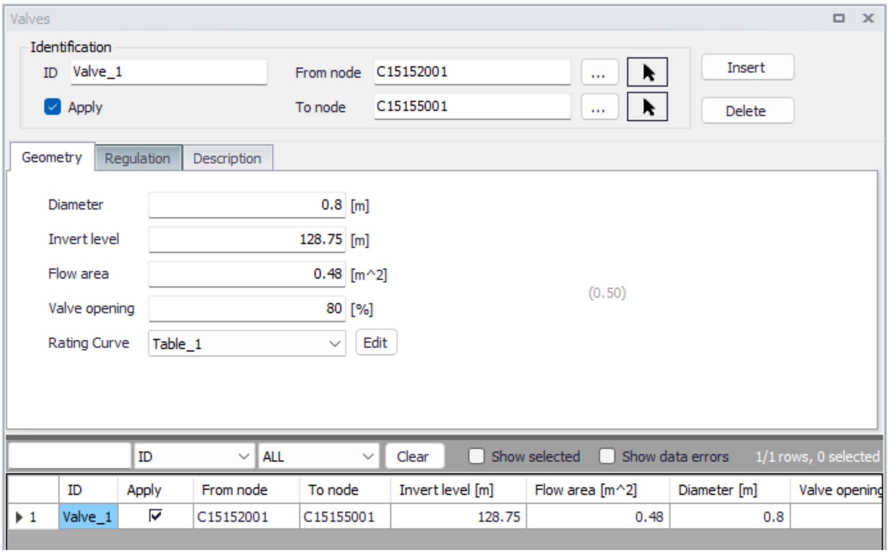


Figure 3.17 Valve dialog

In the real world a valve may be located in a manhole or a similar construction which you normally would define as a node in the model configuration.

The numerical solutions for the flow equations, however, need a model configuration with two nodes where the valve is defined as the connection between the nodes. The valve will then be placed between the two nodes as the flow connection.

It is possible to define several valves between the same two nodes if this is required. This is similar to the possibility of having more than one pipe as the link between nodes. The generation of the computational grid shown in Figure 3.14 for the orifice is also applied for pumps, weirs and valves. The numerical solution of the flow equations will depend on the selected device. Please refer to the MIKE 1D Reference Manual for more details about this.

It is recommended not to place the two nodes in the same spot, instead place the nodes a short distance apart. The reason is that the node head loss computation will have a component from change of flow direction. If the two nodes surrounding the device are placed exactly at the same location then the computational engine cannot determine the direction of the flow from the coordinates of the nodes and a default direction will be applied. This may unintentionally introduce a change in direction and therefore also an unexpected head loss.

By using a small displacement of the nodes the change in flow direction will be determined based on the coordinates and angles between the connected pipes. Therefore consider carefully the placement of the nodes with respect to the actual construction.

A valve is specified by a diameter, flow area (by default calculated on the basis of the diameter, but it is possible to overwrite this value) and an invert level.

It is possible to specify a valve to be a non-return valve and thereby preventing flow in the negative flow direction. A rating curve is specified to define the relation between the valve opening (%) and resistance (k). The rating curve is specified under “Tables| Curves & Relations”.

The valve is by default static, in which case the valve opening must be specified. It is also possible to define a valve to be regulated by control rules and then the valve opening may be controlled during the simulation using control rules.

Table 3.22 Identification and connectivity edit fields of the MIKE+ valve editor (Table msm_Valve)

Edit field	Description	Used or required by simulations	Field name in data structure
Description	User's descriptive information related to the valve	No	Description
Data source	Reference to an external data source (table ID) where the record has been imported from	No	DataSource
Asset ID	Reference to an ID used in external data sources	No	AssetName
Status	Data status for the entire record, serves for keeping track on the source of information	No	Elements
Network Type	Attributes the valve to a certain type of network. Used in case where two or more different networks are included in the same project	No	NetTypeNo
Valve ID	A unique name for the valve. Up to 40 characters (letters, numbers, blank spaces and underscore characters)	Yes	MUID
From node	ID of Node where valve is located	Yes	FromNodeID
To node	ID of Node where valve is discharging to	Yes	ToNodeID
Apply	This check box allows to toggle the Active status of the valve on and off. The simulations will omit all valves that are not active.	Yes	Enabled

Table 3.23 Geometrical and hydraulic properties, edit fields of the MIKE+ valve editor (Table msm_Valve)

Edit field	Description	Used or required by simulations	Field name in data structure
Diameter	The default value of the area (the field “Flow Area”) is calculated on the assumption of a circular valve, with the diameter specified in this field. Furthermore, the transition to a pressurized valve is defined by the invert level plus the diameter.	Yes	Diameter
Invert Level	The invert level defines the minimum water level, which generates flow through the valve	yes	InvertLevel
Flow Area	A user specified flow area overwrites the default valve area computed on the basis of a circular cross section.	Yes	Area
Valve Opening	Defines the opening of the valve in percentages (value between 0 and 100). When applying control rules, this value is not applied, and the opening value is controlled by the control rules	Yes	Opening
Rating Curve	Reference to the tabulated k-opening function	Yes	RatingCurveID
Non return flap	Indicating a flap-gate built-in valve (i.e. no return flow possible)	Yes	FlapNo
Controlled by control rules	If selected, the valve is controlled using control rules, as defined in the 'Control rules' editor. Control rules will apply only when both this check box and the 'Apply' check box in the control rule definition are ticked. Note that the regulation with control rules is always disabled, when the 'Control rules' module is unselected in the 'Model type' editor.	Yes	ControlTypeNo
Max opening	Maximum opening condition of the valve when controlled by control rules	Yes, if control rules apply	
Min opening	Minimum opening condition of the valve when controlled by control rules	Yes, if control rules apply	
Max speed	The maximum velocity for movement of the flap	Yes, if control rules apply	

3.8 Curb Inlets

The connections between pipe systems and overland flow networks to simulate the capture capacity (and surcharge) of side inlet pits and grates can be approximated in MIKE 1D using a combination of orifices and weir geometry. However, a method has been developed to incorporate the geometry of the inlet structure (Curb Inlet) via a network element which allows user input of the empirical relationship governing the structure capacity.

A typical Curb Inlet/grate configuration is shown below. Flow into the pit chamber is via both a grate and side weir (operates as an orifice for deeper flow depths).

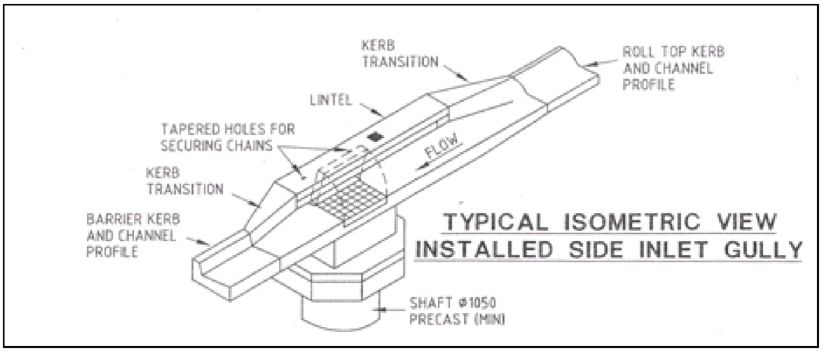


Figure 3.18 A typical curb inlet configuration

Standard curves have been developed in Australia for "ON-GRADE" type (using a $Q_{approach}/Q_{capture}$ relationship where flow can bypass the structure) and "SAG" type (using a Depth/Q relationship at locations/low points where water collects). However the formulation with MIKE 1D allows for non-specific and user defined relationships. An example of the empirical curves developed for the ON-GRADE type is shown below, with the flow captured represented as a proportion of the approach flow, and varying with approach slope.

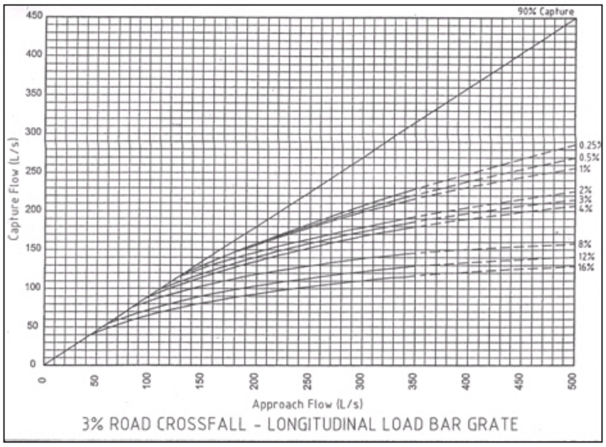


Figure 3.19 Example of empirical curves for On-Grade type

A Curb Inlet (Lintel) is a connection between two nodes of a MIKE 1D urban network (two-directional flow and submerged flow possible), describing the transfer of flow at a grate or inlet from an overland flow network to the sub-surface pipe network. The Curb Inlet dialog is accessed via the “TOC | Network | Curb Inlets” menu.

There are two types of Curb Inlet:

- SAG Type, where the connection node on the overland flow network is located at a sag or low point where water will collect. Transfer capacity of the connection is specified as a DQ-relation (tabular data type).
- ON-GRADE Type, where flow in the overland flow network can continue past the connection node. Transfer capacity of the connection is dependent on the slope of the overland flow network, and specified as a Capture ID (collective of QQ-relations defining the capture rate as a proportion of approach flow).

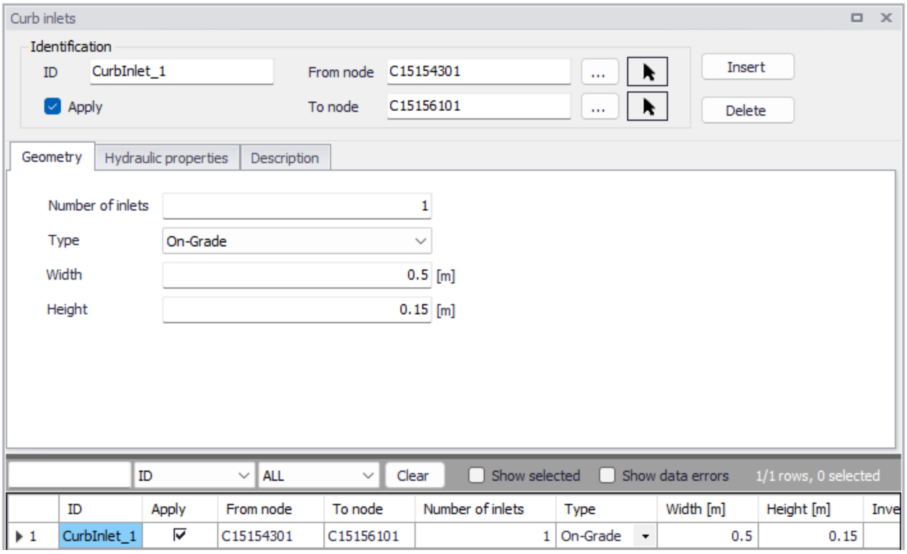


Figure 3.20 The Curb Inlet data dialog

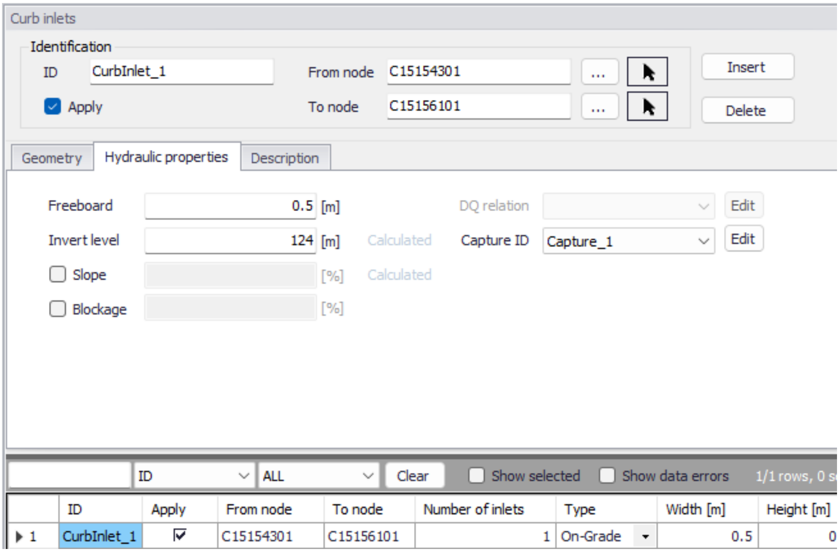


Figure 3.21 The Curb Inlet Hydraulic Parameters data

User defined parameters in the Curb Inlet dialog include:

- Invert level (m) defining the point at which spilling starts (similar to weir crest level). The user is shown a system calculated invert level which is the same as the invert of the connection node in the overland flow network. As with weir flow, a crest level at least 0.01 m higher than the connection node invert level is recommended for initial condition stability.

- Freeboard (m), defining a critical water level (Ground level - Freeboard) at the connection node in the pipe network below which the defined DQ and QQ-relations apply. For submerged and reverse flow (surcharge), the transfer capacity of the connection reverts to a standard orifice relationship.
- Slope (%), representing the slope of the steepest link in the overland flow network entering the connection node (only applies to ON-GRADE Type). The system calculated slope is used in the calculation unless a user defined slope is specified.
- Blockage factor (%) which can be used to account for debris blockage at the grate/inlet. This linear factor is applied to the tabular data sets defining the transfer capacity of the connection.
- Number of Curb Inlets, allowing multiple curb inlets of the same specified geometry (transfer capacity) applied at the same location within a single connection.
- Default rectangular orifice geometry, applies to those flow cases (submerged and reverse flow) were the defined DQ and QQ-relations do not apply. This generally applies when water levels at the connection node in the pipe network exceed the critical level defined by the Freeboard, including reverse flow (surcharge).

There are no limitations on the number of curb inlets specified at one location; however, the connectivity **must** be ‘From’ a node in the overland flow network ‘To’ a node in the pipe network, for correct automatic calculation of slope. *Note: Link slopes must be calculated in the link dialog for automatic calculation of slope to operate.*

The On-grade Capture dialog allows the user to group together QQ-relations (tabular data) that comprise a single On-grade Curb Inlet geometry (similar in function to the Topography dialog). As the transfer capacity for an On-grade Curb Inlet is dependent on the slope in the overland flow network, a number of QQ-relations can apply.

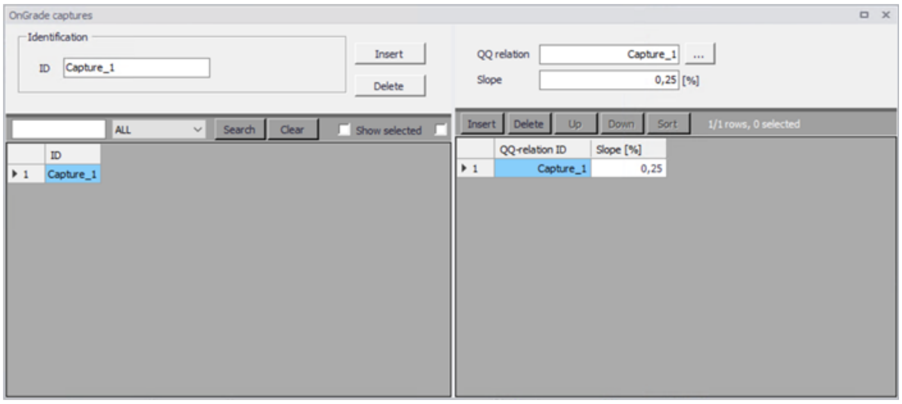


Figure 3.22 The On-Grade Capture data dialog (On-Grade Type)

For calculated or user defined slopes in the Curb Inlet dialog that are outside the range of slopes specified in the On-grade Capture dialog, the closest slope curve will be used. For intermediate calculated or user defined slopes (lying between slope curves in the On-grade Capture dialog), linear interpolation is applied.

In the case of an On-grade Curb Inlet capacity that is not dependent on slope of the overland flow network, the user needs to define the On-grade Capture with a single QQ-relation. *Note: In this case, the calculated or user defined slope in the Curb Inlet dialog for ON-GRADE Type will be ignored.*

Capacity curves

Two curve types specified in the tabular data (Curves & Relations) can be used with the two different types of Curb Inlets.

- Capacity Curve, DQ (depth/discharge relation specified in the Curb Inlets dialog)
- Capacity Curve, QQ ($Q_{approach}$,$Q_{capture}$ relation specified in the On-grade capture dialog).

The DQ relation specifies the depth based capacity curve for a SAG Type Curb Inlet. Values must be monotonously increasing in depth and discharge and starting at (0,0). For depths in excess of the maximum value specified in the last row of the table, the last corresponding discharge value is used.

The QQ relation specifies the relationship between approach flow in the overland flow network (Q_{app}) and the captured flow at the connection node for an ON-GRADE Type Curb Inlet (Q_{cap}). Values must be monotonously increasing and starting at (0,0). For approach discharges in excess of the maximum value specified in the last row of the table, the last corresponding capture discharge value is used.

3.9 Pumps

A pump is a functional relation, which connects two nodes of a MIKE 1D urban network or is associated with only one node (free flow 'out of the system'). The latter case is achieved if the 'To' field is left empty.

In the collector systems a pump may be located in a manhole or a similar construction which you normally would define as a node in the model configuration. The numerical solutions for the flow equations, however, need a model configuration with two nodes where the pump is defined as the connection between the nodes. The pump will then be placed between the two nodes as the flow connection.

It is possible to define several pumps between the same two nodes if this is required. This is similar to the possibility of having more than one pipe as the link between nodes. The generation of the computational grid shown in Figure 3.14 for the orifice is also applied for pumps, weirs and valves. The numerical solution of the flow equations will depend on the selected device. Please refer to the reference manual on more on this.

It is recommended not to place the two nodes in the same spot, instead place the nodes a short distance apart. The reason is that the node head loss computation will have a component from change of flow direction. If the two nodes surrounding the device are placed exactly at the same location then the computational engine cannot determine the direction of the flow from the coordinates of the nodes and a default direction will be applied. This may unintentionally introduce a change in direction and therefore also an unexpected head loss.

By using a small displacement of the nodes the change in flow direction will be determined based on the coordinates and angles between the connected pipes. Therefore consider carefully the placement of the nodes with respect to the actual construction. For pumps the distance between the nodes will often be larger.

3.9.1 Pump types

Several pump types can be specified in MIKE+.

Constant flow pumps

This is the simplest way of modelling pumps. In this case the pump will discharge the same constant flow Q at any time when the pump is switched on. Eventually with the variation as defined during acceleration or deceleration periods.

When specifying a constant flow pump, set the Speed to ‘Constant’ and choose ‘Constant’ in the pump type field. In the Capacity tab type the constant flow value set for

Notice: 'Constant flow' pump is different from 'Constant speed' pump. The later may have varying discharge.

Constant speed pumps

When defining pumps in MIKE+ models for sewer and drainage systems it will most commonly be pumps with a pump curve of type "Q, dH". The actual pump discharge Q will be a function of the actual pressure difference dH between the pump wet well and the receiving point in the model. The pump curves for this type of pumps will in general be as shown in the figure below.

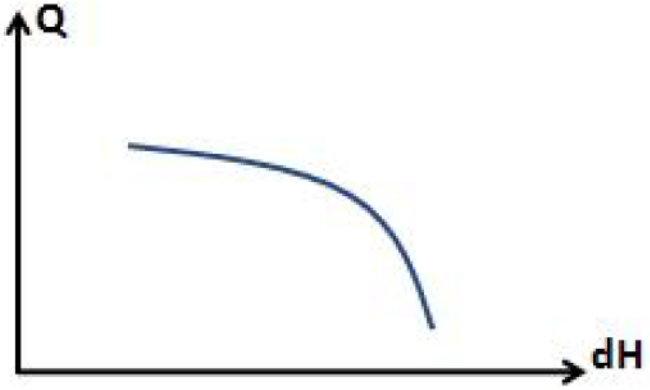


Figure 3.23 Typical example of pump curve

Pump curves are defined in MIKE+ by tabulating the curve under Curves and Relations. This can be done by selecting representative points on the pump curve as shown in the next figure. Information about pump curves are typically provide by the manufacturer of the pumps.

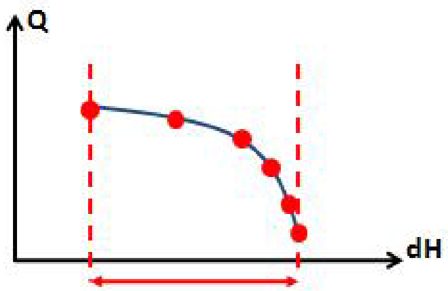


Figure 3.24 Tabulating the curve

When a pump curve is tabulated like this and used in the simulations executed with the MIKE 1D Engine, then the pump is allowed to operate within the range of the dH values in the table. At any time during the simulation the

MIKE 1D Engine will compute the actual dH and determine the corresponding Q value from the table, resulting in the actual duty point position for the pump.

As a standard feature the MIKE 1D Engine will stop the simulation with an error message if the hydrodynamic conditions result in an actual dH value outside the range of the pump curve table.

Variable speed pumps

When applying variable speed pumps the manufacturer typically provides a set of pump curves describing the pump capacity at various percentages of maximum rotation speed or maximum power input. You may have a set of curves available as show in the figure below.

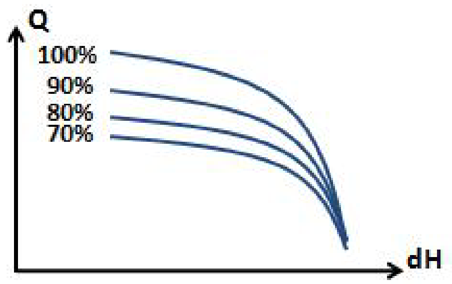


Figure 3.25 Pump curves at various percentages of maximum rotation speed or maximum power input

It is also found that variable speed pumps cannot be regulated over the full range between 0% and 100%. Instead the regulation may be in the range of 70% to 100% as indicated in the figure above. Search for specific information available for the pumps applied at the pumping stations being modelled.

For simplicity in the modelling the actual method of varying the pump capacity is not considered. It is not directly related to the actual number of rotations per minute (RPM) or the actual electrical power input. In the modelling we only consider the pump capacity as varying between a pump curve corresponding to the minimum speed and a pump curve corresponding to the maximum speed.

For variable speed pumps the two pump curves are defined as 'RPMmin' and 'RPMmax' pump curves. For constant speed pumps only the 'RPMmax' pump curve is applied.

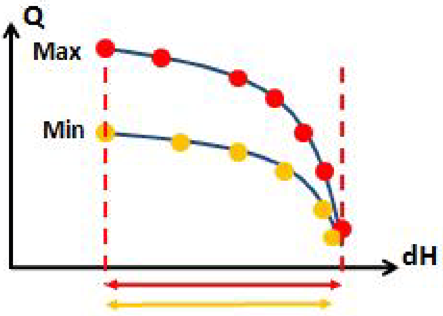


Figure 3.26 The RPMmax and RPMmin pump curves

When the MIKE 1D simulation is executed it is the standard condition that the simulation will stop and give an error message if the conditions exceed the range for the dH value in the pump curve table. This applies both for the 'RPMmax' and the 'RPMmin' pump curve.

Operation of variable speed pumps

In the Regulation tab it is possible to regulate the pump to maintain a water level in the wetwell. The corresponding 'Wetwell WL Setpoint' must be specified when using this option.

When a variable speed pump is in operation during the simulation with the MIKE 1D Engine the duty point will move between the two pump curves depending on the actual conditions. The actual flow is determined by using a PID function which will attempt to control the water level at the wet well to stay at the specified set point value.

The discharge Q determined by the PID function and the dH found by the actual hydraulic conditions defines the duty point. This may vary in the area between the two pump curves.

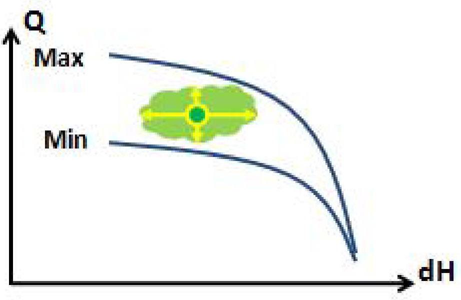


Figure 3.27

Eventually the conditions may result in the duty point ending up on one of the pump curves. If the PID regulation sets the discharge Q to a value higher than the limitation by the 'RPMmax' pump curve at the given dH condition, then the discharge will be defined by the pump curve. As a consequence the water level in the wet well will rise above the defined set-point value.

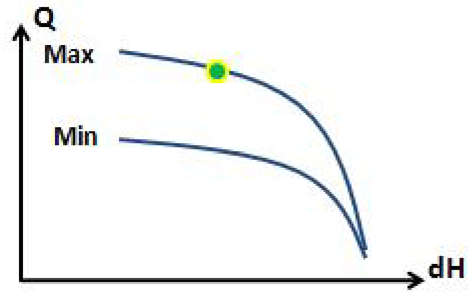


Figure 3.28

If the inflow to the pumping station is low, then the operation of the pump may result in an actual duty point located at the 'RPMmin' pump curve. In this case the water level at the wet well will drop below the defined set-point value. Eventually the water level will reach the stop level defined for the pump and the operation is switched off

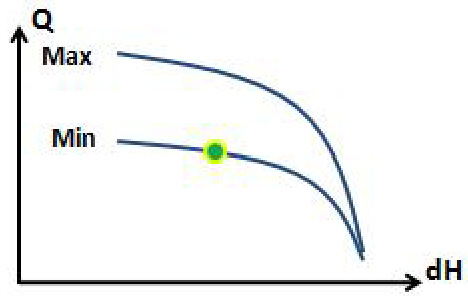


Figure 3.29

The MIKE 1D engine will pump as much flow available to discharge in the structure, which means that the pump works continuously during the simulation if there is lack of water in the pump wet well then the pump will discharge a flow of zero during those lapses but will not be forced stop.

The hydrodynamic network simulation provides a simulation summary report at the end of the computation. The summary for the pumps will show the number of pump starts during the simulation.

Pumps - Discharge

Pump ID	From Node	To Node	Minimum [m^3/s]	Maximum [m^3/s]	Total discharge [m^3]	Time of Minimum	Time of Maximum	Pump starts [count]	Speed	Activ [hh:r
Pump_2_3	Node_02	Node_3	0,000	1,775	141.876,6	16-05-1953 00:00:10	16-05-1953 06:39:59	1	Variable	2.05:
Pump_2a_3	Node_02A	Node_3	0,000	2,000	146.063,3	16-05-1953 00:00:10	16-05-1953 03:01:38	138	Variable	1.05:

Figure 3.30 Summary of pump starts during simulation

The pump flow continues during the specified deceleration period by a linear decrease to zero flow. During this time interval the water level in wet well may eventually drop to the bottom of the wet well during a single time step

The 'Pumps - Discharge' section is only shown in the summary report if 'pumps' has been selected in the “Network Summary” under Result Specifications.

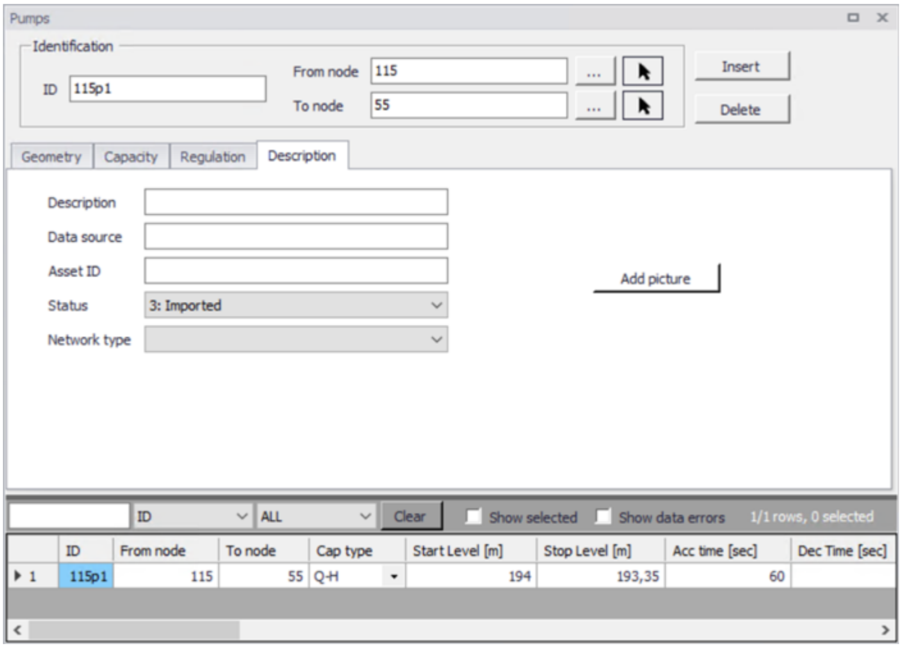


Figure 3.31 Pumps description tab

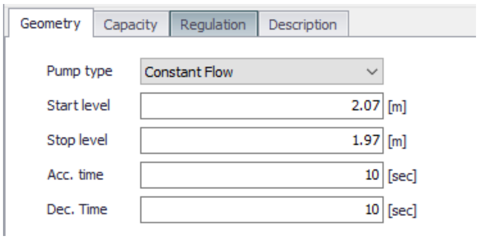


Figure 3.32 Pump geometry tab

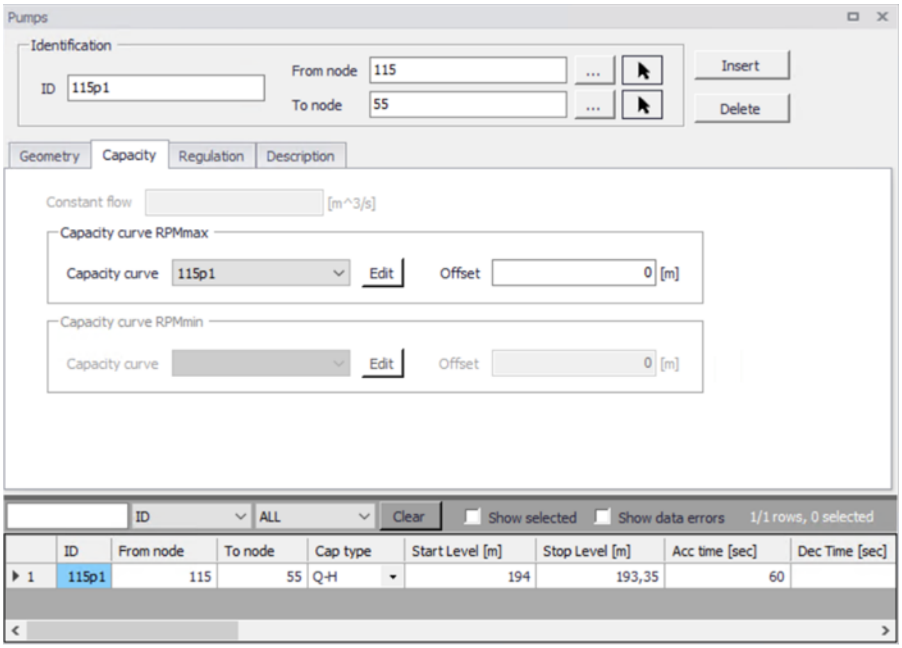


Figure 3.33 Pump Capacity tab

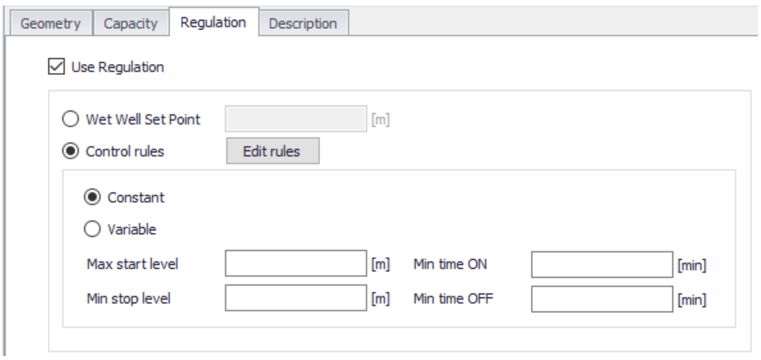


Figure 3.34 Pump regulation tab

A pump is characterised by the 'Start Level' and 'Stop Level', acceleration and deceleration time, an offset and a capacity curve. The capacity curve is specified in the Curves and Relations section. The capacity curve can be specified as a 'Capacity Curve QH' relation (for screw pumps) or as 'Capacity Curve QdH' relation (for differential head pumps), where 'H' is the absolute water level in the pump's wet well (at 'Location'), and 'dH' is the water level difference between the 'To node' and the 'From node' location.

A pump type with a 'Capacity Curve QH' relation is named a *screw pump*, while a pump type with a 'dH-Q' relation is named a *differential head pump*.

If an offset is specified this will be added to the capacity curve relation.

Pumps are per default static (No Control) but can be regulated through control rules. Selecting the ‘Control rules’ radio button enables the control of start and stop levels or pump flows during the simulation.

3.9.2 Identification and connectivity

Table 3.24 Identification & Description (Table msm_Pump)

Edit field	Description	Used or required by simulations	Field name in data structure
Pump ID	A unique name for the pump. Up to 40 characters (letters, numbers, blank spaces and underscore characters)	Yes	MUID
From node	ID of Node where pump is located	Yes	FromNodeID
To node	ID of Node where Pump is pumping to.	Yes	ToNodeID
Apply	This check box allows to toggle the Active status of the pump on and off. The simulations will omit all pumps that are not active.	Yes	Enabled
Description	User's descriptive information related to the pump	No	Description
Data source	Reference to an external data source (table ID) where the record has been imported from	No	DataSource
Asset ID	Reference to an ID used in external data sources	No	AssetName

Table 3.24 Identification & Description (Table msm_Pump)

Edit field	Description	Used or required by simulations	Field name in data structure
Status	Data status for the entire record, serves for keeping track on the source of information	No	Element_S
Network type	Attributes the pump to a certain type of network. Used in cases when two or more different networks are included in the same project	No	NetTypeNo

3.9.3 Model data

Table 3.25 Overview of the Pumps editor attributes (Table msm_Pump)

Edit field	Description	Used or required by simulations	Field name in data structure
Pump Type	Type of flow variation, constant or based on a capacity curve. “QH Curve” is applicable for archimed Screw, the “QdH Curve” is applicable for all rotodynamic pumps	Yes	CapTypeNo
Start level	Water level in pump sump which triggers the pump to start	Yes	StartLevel
Stop level	Water level in pump sump which triggers the pump to stop	Yes	StopLevel
Acc. time	Pump acceleration time. Used to dampen sudden increase of flow at the pumps START events	Yes	AccTime
Dec. time	Pump decceleration time. Used to dampen sudden decrease of flow at the pumps STOP events	Yes	DecTime
Constant flow	Regards to the pump capacity, flow set for a pump in steady state	Yes	DutyPoint
Capacity curve RPMmax	Capacity curve for the nominal (max) rotation speed	Yes	QMaxSetID
Capacity curve RPMmin	Capacity curve for the minimum speed. Applies only for variable speed pumps	Yes	QMinSetID

Table 3.25 Overview of the Pumps editor attributes (Table msm_Pump)

Edit field	Description	Used or required by simulations	Field name in data structure
Offset	Offset of the capacity curve. Applies to Q-H types, which may be specified relative to different datums	Yes. if Q-H type	Offset1
Offset	Offset of minimum capacity curve. Applies only for variable speed pumps	Yes, if variable speed pump	Offset2
Use regulation	Checkbox enabling the option to regulate the pump operation. Note that this check box is overruled by any rule defined in the 'Control rules' editor: if the control rule should be deactivated, use the 'Apply' check box in the control rule definition. Also note that the regulation is always disabled, when the 'Control rules' module is unselected in the 'Model type' editor.	Yes, if regulation is desired	RegNo
Wetwell WL set-point	Absolute water level in the pump sump (i.e. wet well) which the pump is supposed to maintain	Yes	WetWellSetPoint
Variable Speed	Allows the option to set variable speed pumps. Only variable speed pumps can be used for PID control	Yes	SpeedNo
Control rules	Activate operation using control rules. Control rules will apply only when both this 'Control rules' check box and the 'Apply' check box in the control rule definition are ticked. “Wet-well set point” is a special control rule, which is accessible also without control add-on module	Yes	ControlTypeNo

3.10 Generic CRS & Topography

The Generic CRS & Topography editors allows the definition of the conduit cross-sections.

Generic CRS support open and closed cross sections. The X, Z types are appropriate for irregular cross sections, while H, W are best for symmetric cross sections.

Cross sections are classified in seven types: three of them are closed cross sections, and four of them are open cross sections. Each of the types has three sub-types, defined by the way how the CRS geometry is described. Thus, the following CRS types are supported:

- X, Z open: The CRS geometry is described by points defined by co-ordinate pairs (x, z), where 'x' is a horizontal axis, and 'z' a vertical axis. The points are specified in a counter-clockwise direction.
- X, Z closed: The CRS geometry is described by points defined by co-ordinate pairs (x, z), where 'x' is a horizontal axis, and 'z' a vertical axis. The points are specified in a counter-clockwise direction. The first and last points are connected to close the cross section.
- H, W open: The CRS geometry is described by pairs (h, w), where 'h' is relative height, and 'w' is the corresponding cross section width. The pairs are specified in an upward direction.
- H, W closed: The CRS geometry is described by pairs (h, w), where 'h' is relative height, and 'w' is the corresponding cross section width. The pairs are specified in an upward direction. The last specified (h, w) pair defines the top of the closed cross section.
- X-Z-R-M open: The CRS geometry is described by points defined by co-ordinate pairs (x, z), where 'x' is a horizontal axis, 'z' a vertical axis, the relative resistance (R) and the marker (M). The points are specified in a counter-clockwise direction.

This cross section type allows to specify open channel systems to have variable roughness across a cross section (common in urban and drainage studies).

Cross sections can be edited and shown graphically in the *Map View* and in the *tabular view.* The data created will be stored as a external cross section file with the extension xns11.

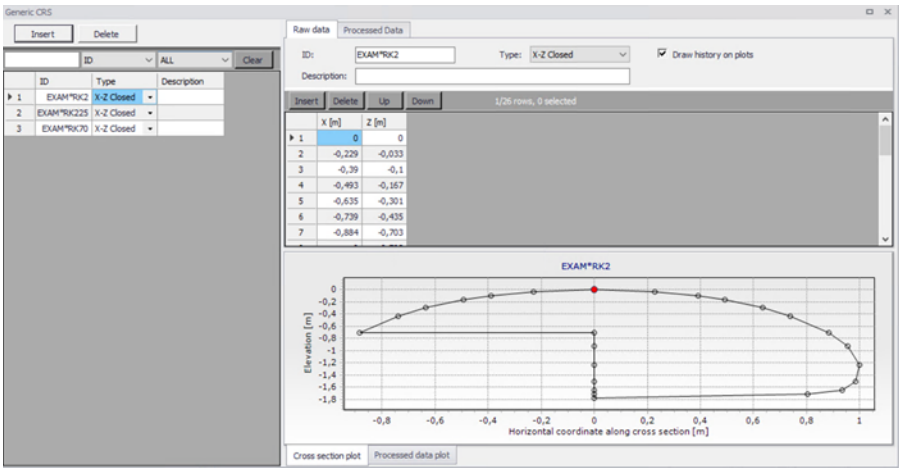


Figure 3.35 The Generic CRS Tabular view

Cross section tree

The cross section tree window in the tabular view shows the list of all cross sections in the setup. The data are stored in an external cross section file with the extension *xns11*.

In the tabular view it is possible to see all the cross section in the setup. In the first column, cross sections are identified by their branch name, topo ID and chainage.

Clicking a cross section in the 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. The options offered in the contextual menu depend on where you 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, a dialogue will be displayed. Where the “From” groupbox indicates with cross sections are being moved and where the “To” groupbox indicates the final destination which has been specified by the user.

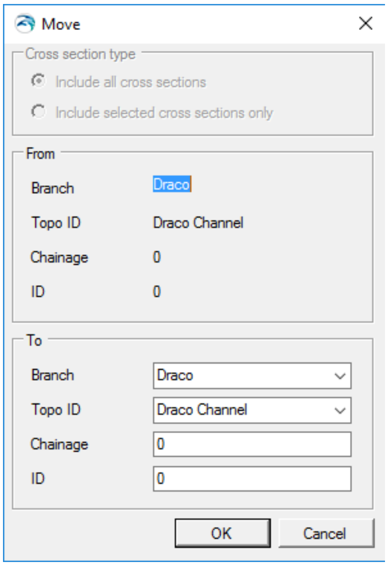


Figure 3.36 Dialogue for moving a cross section

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.

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 3.37

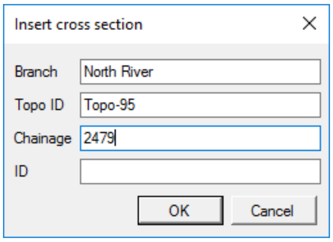


Figure 3.37 Insert a blank Cross Section

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 3.38

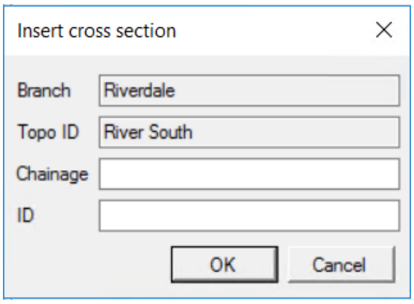


Figure 3.38 Dialogue for inserting a cross section in a selected topo ID

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 3.39, 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.

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’).

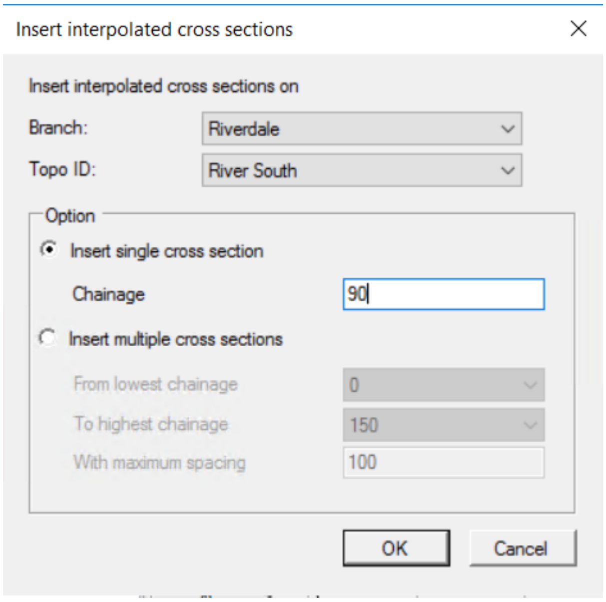


Figure 3.39 Dialogue for inserting an interpolated cross section

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’).

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.

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 sys tem 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.

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 3.40 from which a requested marker number can be assigned for the selected point.

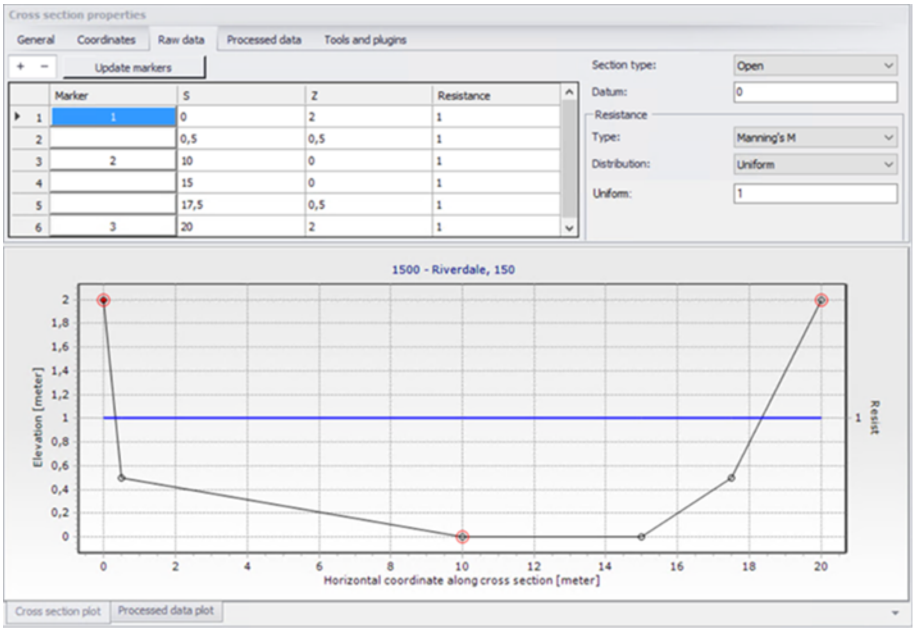


Figure 3.40 The marker options for the cross section type X-Z-R-M

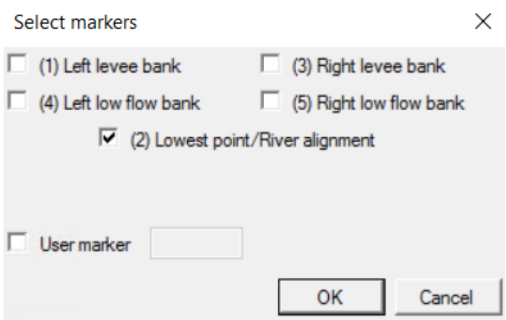


Figure 3.41 Selection of markers

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.

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 channel (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 s/m(1/3).
- Manning’s M: the resistance number is specified as Manning’s M in the unit m(1/3)/s (Manning’s M = 1/Manning’s n).
- Chezy number: the resistance number is specified as Chezy number in the unit m(1/2)/s.
- Darcy-Weisbach (k): the resistance is specified in the form of an equivalent grain diameter.

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.

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 situations 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.

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. 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.

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.

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

$$x_{cor} = x\cos\theta$$

Where θ is illustrated below

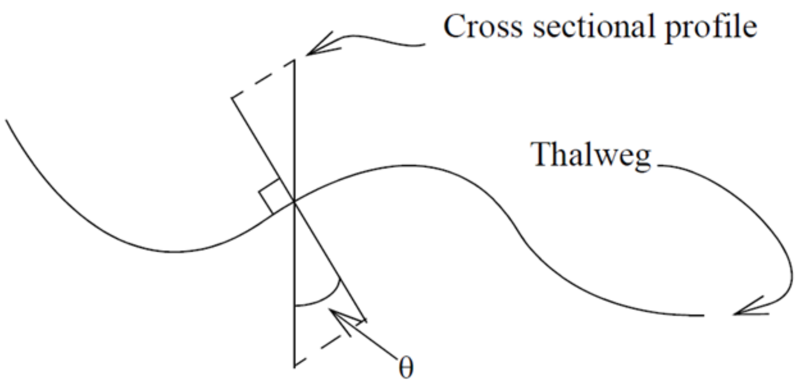


Figure 3.42 Definition sketch of the correction angle

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.

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.

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.

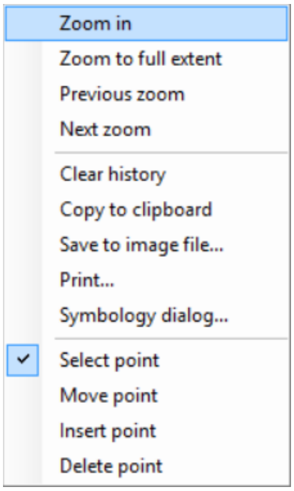


Figure 3.43 Contextual menu for the cross section plot

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.
  - 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.

4 Rainfall-Runoff Modelling

MIKE+ provides a versatile set of tools and computational models for modelling surface storm runoff and infiltration on urban and semi-rural catchments. The User can quickly prepare a precipitation-runoff model setup of desired level of detail (in terms of spatial discretization and input data) and use the computed runoff as a load to the collection network.

This Chapter provides a comprehensive guide for the preparation of hydrological models.

Modelling of urban storm runoff and infiltration requires understanding of information requirements and the involved processes. This understanding is supported by the illustration in Figure 4.1. Note that MIKE+ can model rainfall-dependent infiltration (RDI). This is specifically discussed within the Parameters RDI section, and is not visually represented in Figure 4.1.

Although runoff computation and its subsequent use as a network load are, in principle, two distinct steps in the modelling process, MIKE+ has the facility to simulate the two processes simultaneously.

Runoff modelling engages the following:

- Catchments
- Optionally (only if network computations will follow), definition of the catchment connection, i.e. specification of the point of runoff inflow into the network.
- Definition of the hydrological models
- Precipitation (optionally, temperature and evapotranspiration)
- Runoff computations

The runoff is typically used as a hydraulic load to the collection network. This requires:

- Declaration of the computed runoff as a network hydraulic load
- Execution of the network computation.

Runoff and hydrodynamic network computations are run simultaneously in MIKE+. Nevertheless, they can also be launched in two distinct steps, if needed.

Furthermore, an important part of successful modelling is related to model calibration and verification, which ensure that the computed results fit reasonably well with the flow observations. The calibration and verification are important engineering activities in the modelling process, and they must be paid due attention.

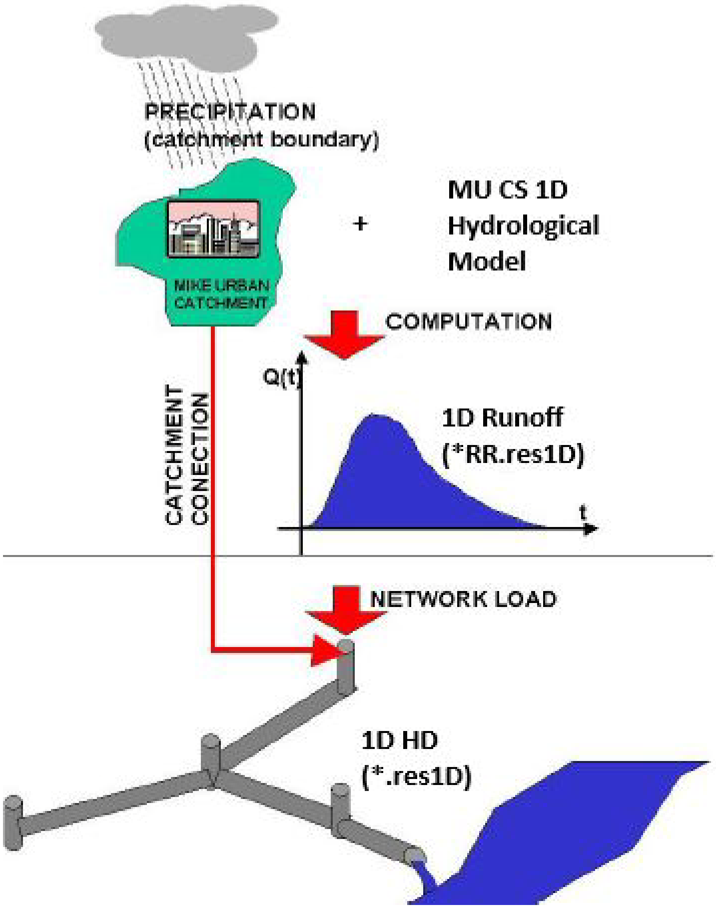


Figure 4.1 Illustrated flow of information in hydrological modelling

4.1 Catchments

Catchments are essential for any hydrological model. MIKE+ catchments are geographical polygon features which represent hydrological urban catchments or wastewater drainage areas. As such, catchments may be used for hydrological modeling or as wastewater sources.

In the context of hydrological modelling, MIKE+ catchments are treated as hydrological units where storm runoff and infiltration are generated based on a single set of model parameters and input data. Catchments represent the level of spatial discretization of the hydrological model.

Catchments are featured as a single data layer in MIKE+. After the definition of the catchments' connections to a model network, the catchments become a source of loads for a network model.

Catchments can be managed both graphically on the map and through the Catchments Editor. The two modes complement each other, and efficient management of catchment data can be achieved through joint application of both modes.

The graphical mode (i.e. Edit Features functionality) allows digitisation of catchment extent by tools like Create, Edit, Delete, and Split. Descriptions of graphical catchment tools are found in the Model Manager User Guide.

The Catchments Editor is used for:

- Editing catchment attributes. It is possible to insert catchments through the Editor; these are given a schematised quadratic shape.
- Editing connections to model networks and hydrological data for Rainfall-Runoff models.
- Editing catchment hydrological model attributes. It is possible to edit catchment attributes in relation to various hydrological runoff models available in MIKE+.

The Catchments Editor can be accessed via Catchments|Catchments.

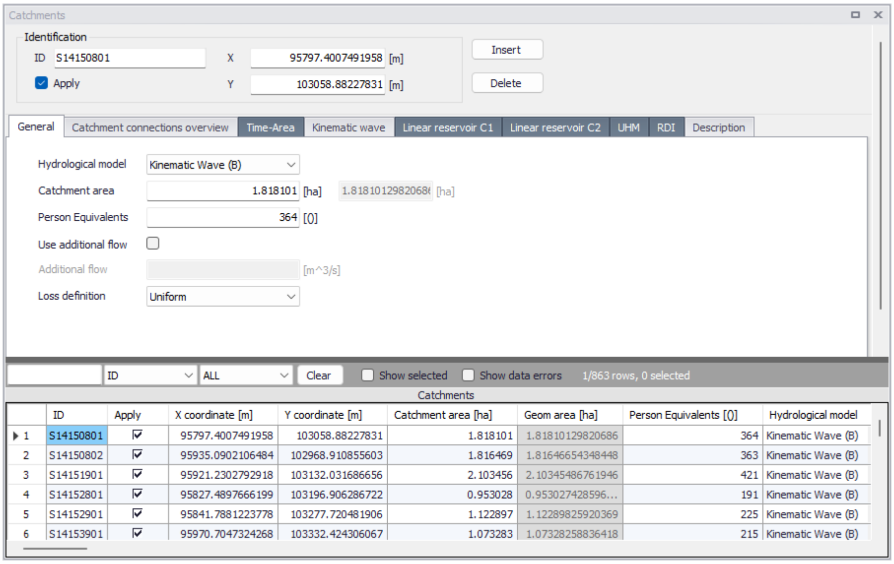


Figure 4.2 The Catchments Editor

The Catchments Editor has a general Identification Group containing information on catchment identifier and geographic location. The 'Apply' check box allows the user to toggle the Active status of the catchment on and off. The simulations will omit all catchments that are not active.

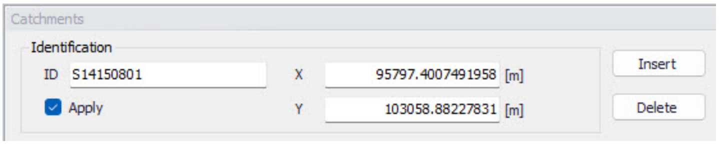


Figure 4.3 The Catchments Editor Identification Group

Table 4.1 Overview of the Catchments Editor Identification Group attributes (Table msm_Catchment)

Edit field	Description	Usage	Attribute Table Field
ID	Unique catchment identifier	Yes	MUID
X	X coordinate of the catchment geometry centroid	Calculated (read-only)	-Derived-
Y	Y coordinate of the catchment geometry centroid	Calculated (read-only)	-Derived-
Apply	This check box allows to toggle the Active status of the catchment on and off. The simulations will omit all catchments that are not active.	Yes	Enabled

Other catchment attributes related to hydrological modelling and connections to model networks are organized in Tabs within the Catchments Editor and are described in succeeding Sections.

4.1.1 General

General catchment attributes related to geometry and Person Equivalents are specified in the ‘General’ tab of the Catchments Editor. Optionally, land use of the catchment's surface can also be specified in this tab.

In MIKE+, the geographical extent of a catchment is determined by the catchment polygon perimeter. MIKE+ provides information on the total catchment area based on polygon geometry.

In some cases, the geographical boundaries of a catchment do not coincide with the actual drainage area. A catchment extent may be defined based on some administrative division, while the drainage network is present only in some parts of the catchment. In such cases, the User may specify the Catchment Area value, which overwrites the actual geometric area in all hydrological computations.

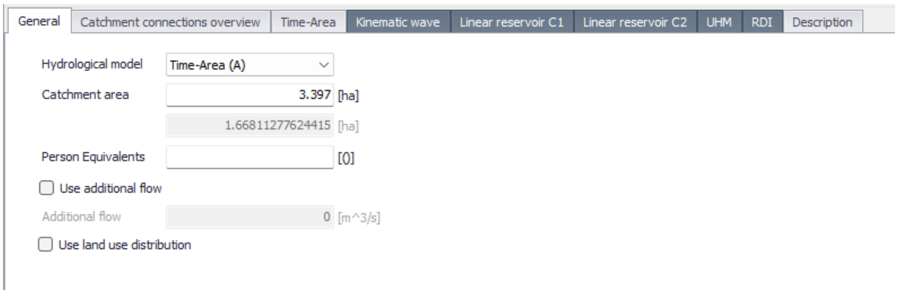


Figure 4.4 The General tab of the Catchments editor

A number of person equivalents may also be specified for the catchment, which will be used to compute the wastewater load on the catchment.

An optional additional flow can also be applied. It acts as a constant flow added to the computed catchment runoff during the simulation, if ‘Use additional flow’ is active.

Finally, the definition of land uses over the catchment may be activated from the 'Use land use distribution' option. This option allows defining local parameters within the catchment as a function of e.g. the surface coverage type. When this option is inactive, all parameters for the catchment simulation are defined in the tab corresponding to the selected hydrological model. When this option is active, an extra table must be filled with information about the description of each land use on the catchment. Land use definitions may be of two types:

- For hydrological models Time-Area (A), Kinematic wave (B), Linear Reservoir (C1) or Linear Reservoir (C2), the specification of the land use controls the local loss and infiltration settings, and is defined in the 'Land uses' editor. The computation of the runoff dynamics is instead performed at the catchment level, using the sum of runoffs from all land uses on the catchment, using the select hydrological model's method.
- For the RDI hydrological model, a land use is specified using a RDI parameter set (see Parameters RDI *(p. 115)*). Each RDI land use is simulated independently from other land uses: its resulting flow time series are simply summed up with flows from other land uses

The buttons next to the table are used to add or remove new land use types to the catchment. For each land use type, a contributing area (expressed as a percentage of the catchment area) must be specified. The fields on the right of the table present the land use settings, for information (except for RDI land use type).

The percentage of the catchment area not contributing to the runoff is presented below the table. It represents the remainder of the catchment, not covered by the land uses, and therefore not generating runoff.

The approach based on land use is not available for the UHM hydrological model.

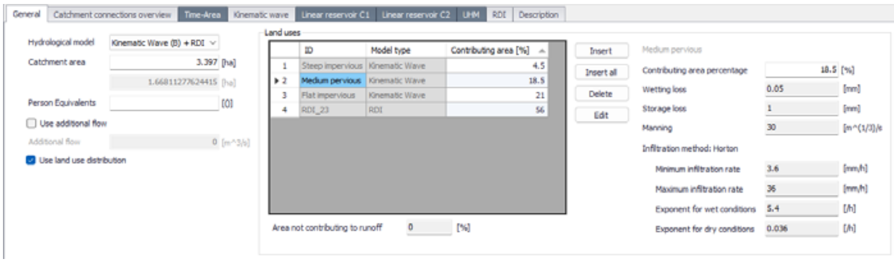


Figure 4.5 Definition of the land use on the catchment

Table 4.2 Overview of the Catchments Editor Geometry attributes (Table msm_-Catchment)

Edit field	Description	Usage	Attribute Table Field
Hydrological Model	Dropdown menu for selecting the hydrological runoff model to use for a catchment	Yes	HydrologicalModelNo
Catchment Area	The catchment area of relevance for the project	Optional	Area
(Geom. Area)	Program-calculated geometric area of the catchment polygon	Calculated (read-only)	-Derived-
Person Equivalents	Unit per capita loading for the catchment	Yes, Used as a method for wastewater load definition	Persons

Table 4.2 Overview of the Catchments Editor Geometry attributes (Table msm_-Catchment)

Edit field	Description	Usage	Attribute Table Field
Use additional flow	Option to include or exclude constant additional flow on the catchment	Optional	AddFlowNo
Additional Flow	Amount of constant additional flow	Optional	AddFlow
Use land use definition	Selection between uniform description of the catchment, or paramaters distributed depending on the land use.	Yes	LossDefinition
Land use ID	Reference to a land use ID, as defined in the 'Land uses' editor, or to a RDI parameters ID, depending on the selected type of hydrological model.	Yes, If applying land use definition	LandUseID
Land use contributing area	Percentage of the catchment area covered by the corresponding land use.	Yes, If applying land use based loss definition	LandUseContrib

4.1.2 Description

The Catchments Editor Description Tab allows the User to provide additional information for a catchment record.

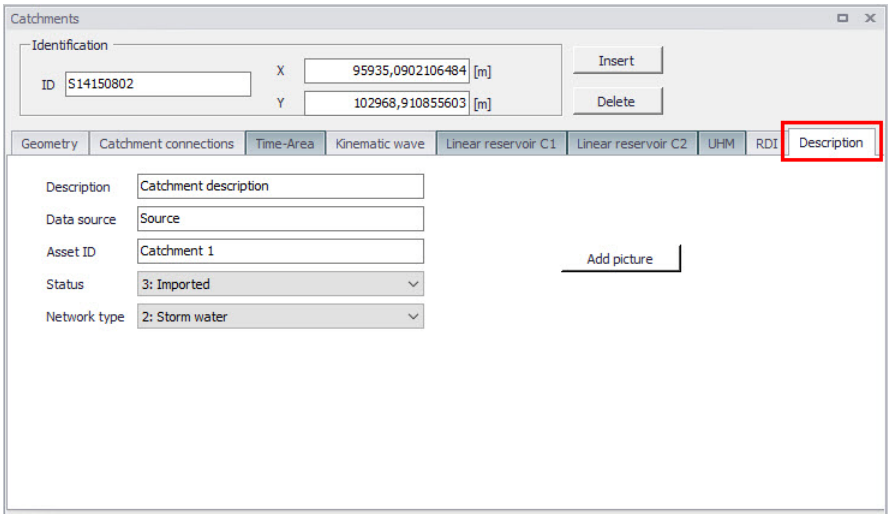


Figure 4.6 The Catchments Editor Description Tab

Table 4.3 The Catchments Editor Description Tab attributes (Table msm_Catchment)

Edit field	Description	Usage	Attribute Table Field
Description	Free text description related to the catchment	Optional	Description
Data source	Reference to an external data source (e.g. table ID) from where the record was taken	Optional	DataSource
Asset ID	Reference to an ID used in external data sources	Optional	AssetName
Status	Data status for the record for keeping track of information source	Optional	Element_S

Table 4.3 The Catchments Editor Description Tab attributes (Table msm_Catchment)

Edit field	Description	Usage	Attribute Table Field
Network Type	Attributes the catchment to a certain type of network. Used in cases where different network types are in the same project.	Optional	NetTypeNo
Add picture button	Facility for defining an image file for the catchment record. Accepts .PNG, .JPG, and .BMP image files.	Optional	-

4.2 Hydrological Models

Catchment records include information related to hydrological modelling. Such information is shown in separate tabs: one tab for each model.

Hydrological models for catchments include two distinct classes of models:

- Surface runoff model: These are the most common types in urban runoff analysis. The common characteristic of all the models in this class is that only surface runoff is computed. This implies discontinuous runoff hydrographs where flow starts as a result of rainfall and reduces to zero again after the end of rainfall. As such, these models are suitable for relatively densely urbanized catchments with dominant amount of runoff generated on impervious surfaces, and for single-event analyses (e.g. design rainfall of certain recurrence interval). These models fail to provide realistic results in dominantly rural catchments and for long-term analyses involving multi-event rainfall series.
- Continuous hydrological models: These models treat the precipitation volume balance without any truncation through complex concepts. As a result, the generated runoff includes both the overland and subsurface runoff components. Due to longer time scales involved, the runoff hydrographs appear practically continuous. An important property of continuous hydrological models is hydrological memory, i.e. the ability to simulate the catchment reaction to certain rainfall dependent on previous rainfalls. This type of model is essential for any long-term analysis and for dominantly rural catchments. On the other hand, these models are usually incapable of simulating extremely fast response of heavily impermeable urban catchments.

MIKE+ includes a series of surface runoff models and one continuous hydrological model. The surface runoff models available are:

- Time-Area Method (A)
- Kinematic Wave (B)
- Linear Reservoir (C1 and C2)
- Unit Hydrograph Method (UHM)

The continuous hydrological model included is Rainfall Dependent Infiltration (RDI).

Any of the surface runoff models can be used on their own, or in combination with RDI. When a surface runoff model is combined to RDI, the simulation will compute the runoff for each method (surface runoff model and RDI) separately as if they were different catchment, and will then sum the runoff of the surface runoff model and RDI model.

Combining different models for individual catchments in one runoff computation is also possible in MIKE+.

Detailed descriptions of the models are available in the following Sections and in the MIKE 1D Reference Manual.

4.2.1 Time-Area Method (A)

The Time-Area Method is a simple surface runoff model with minimum data requirements. The runoff computation is based on a simple treatment of hydrological losses and the runoff routing by the so called Time-Area curve. Technical details on the method can be found in the MIKE 1D Reference Manual.

Model Data

A full overview of the editor fields and corresponding attributes is provided in the Table 4.4.

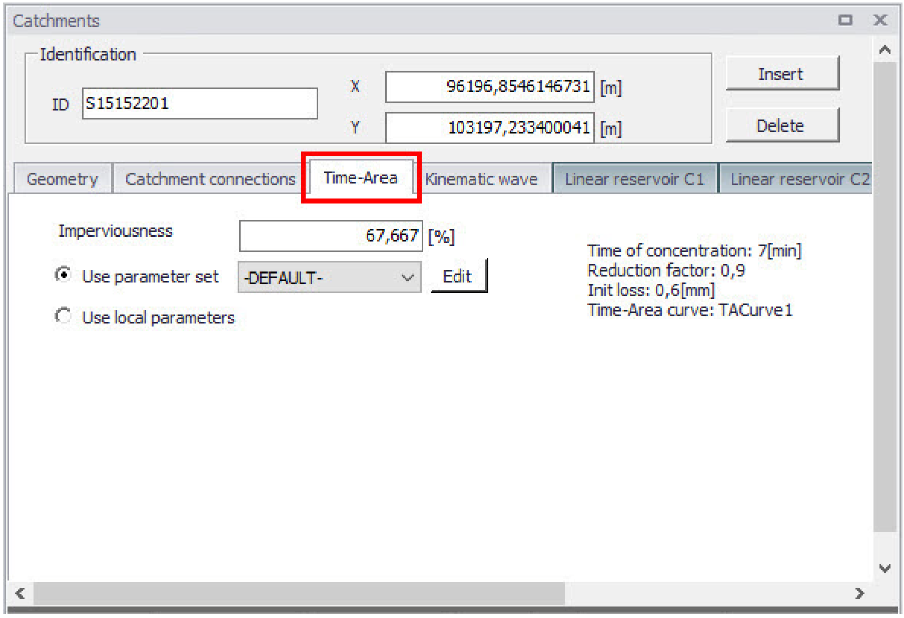


Figure 4.7 The Time-Area Model Tab

Table 4.4 Overview of the Time-Area Model attributes (Table msm_Catchment)

Edit field	Description	Used or required by simulations	Field name in datastructure
Imperviousness	Impervious catchment area, as percent of the actual model area	Yes	ModelAImpArea
Use Parameter Set	Reference to a set of model parameters to be used for the current catchment	Yes	ModelAParAID
Use Local Parameters	Allows for local use of individual parameters	Optional	ModelALocalNo

Parameters Time-Area

The Time-Area model uses several parameters. For practical reasons, these parameters have been grouped in parameter sets, which can be associated with certain catchments. This means the entire model setup can be established with a very small amount of information, while still allowing for full spatial variation of model parameters for individual catchments through the application of local values.

New parameter sets can be inserted and values of individual parameters can be edited in the Parameters Time-Area Editor.

MIKE+ comes with a Default parameter set (-DEFAULT-). The User can insert any number of parameter sets and edit them as needed.

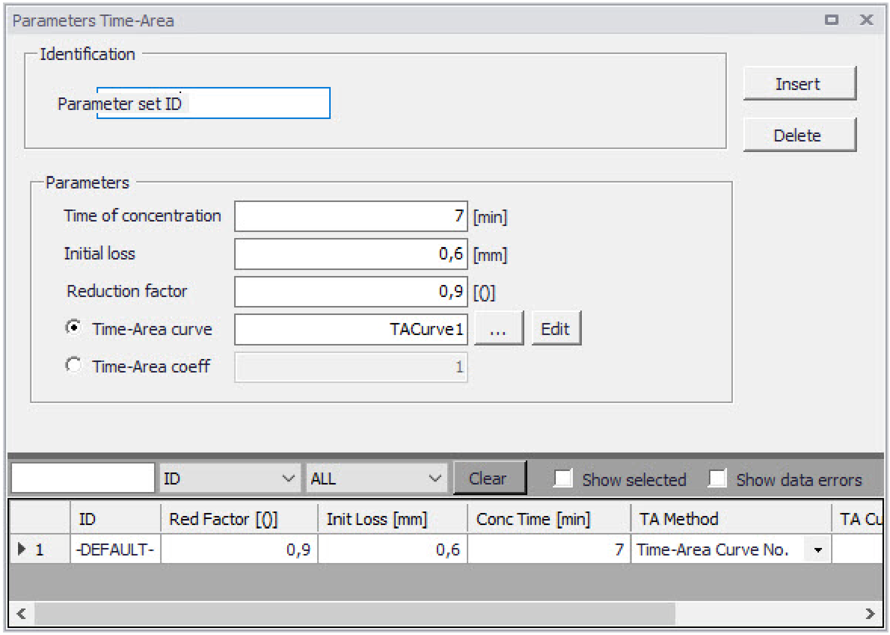


Figure 4.8 The Time-Area Parameter Sets Editor (Catchments|Parameters Time-Area)

A full overview of the editor fields and corresponding database attributes is provided in the Table 4.5.

Table 4.5 Overview of the Time-Area Parameter Set Database attributes (Table msm_HParA)

Edit field	Description	Used or required by simulations	Field name in datastructure
Parameter Set ID	Parameter set identifier	Yes	MUID
Time of Concentration	Concentration time	Yes	ConcTime
Initial Loss	Initial loss (wetting, interception, local depressions…)	Yes	InitLoss
Reduction Factor	Hydrological reduction factor	Yes	RedFactor
Time-Area curve/coefficient radio buttons	Switch for use of predefined tabulated T-A curves or analytically computed T-A relation	Yes	TAMethodNo
Time-Area Curve	Predefined T-A curve ID	Optional, alternates with TACoeff	TACurveID
Time-Area Coeff.	Value of the analytical T-A curve coefficient	Optional, alternates with TACurveID	TACoeff

Time-Area Curve Editor

The Time-Area model uses (except if analytical curve is applied) predefined tabulated time-area curves. A Time-Area curve represents the contributing part of the catchment surface as a function of time. Implicitly, the Time-Area curve accounts for the shape of the catchment in relation to the outlet point.

MIKE+ comes with three Default Time-Area curves: TACurve1, TACurve2, and TACurve3, applicable for rectangular, divergent and convergent catchments, respectively (see the figures below).

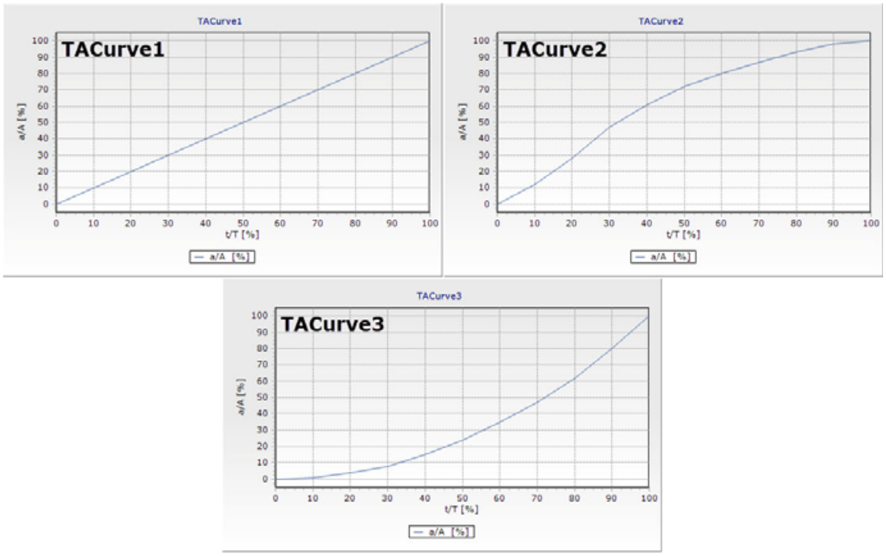


Figure 4.9 Three Default Time-Area curves

You can define any number of custom Time-Area curves. These can be inserted and edited in the editor for Curves and relations (Tables|Curves and Relations). Each Time-Area table must start with a pair of values (0,0) and must end with a pair of values representing the whole (per Default, MIKE+ maintains T-A curves in percent (%), i.e. the last pair of values in the table must be (100,100).

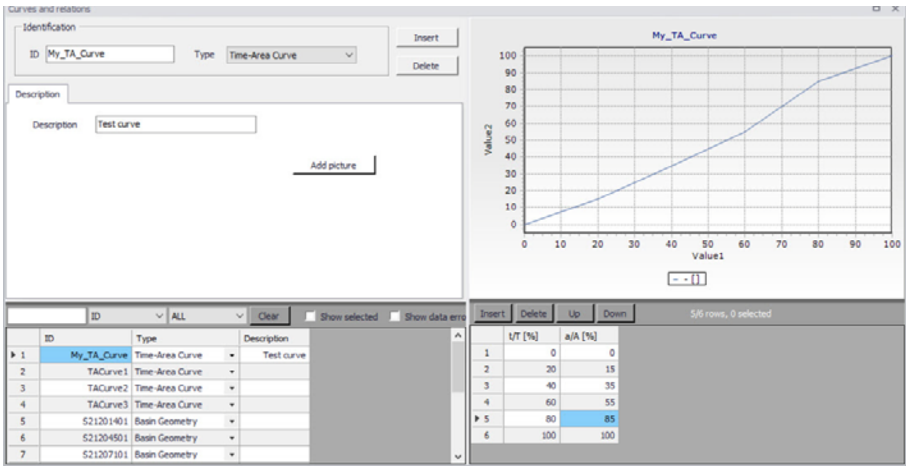


Figure 4.10 The Curves and Relations Editor with a user-defined Time-Area curve

4.2.2 Kinematic Wave (B)

The Kinematic Wave (Model B) is a surface runoff model with moderate data requirements. The runoff computation is based on a comprehensive treatment of hydrological losses (including infiltration) and the runoff routing by the Kinematic Wave (Manning) formula. Technical details can be found in the MIKE 1D Reference Manual.

Model Data

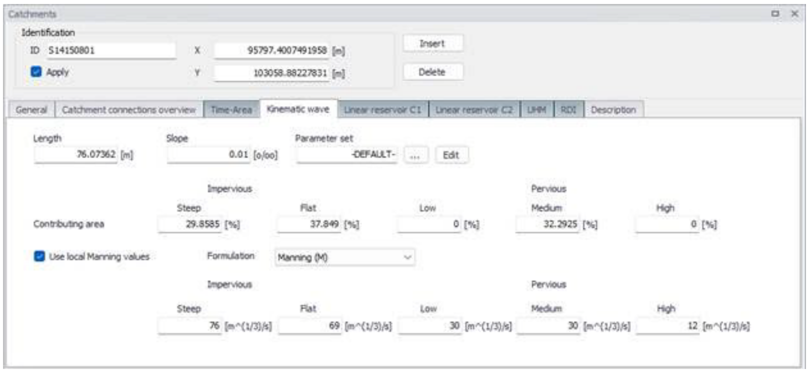


Figure 4.11 The Kinematic Wave Model Tab

A full overview of the editor fields and corresponding attributes is provided in the Table 4.6.

Table 4.6 Overview of the Kinematic Wave model attributes (Table ms_Catchment)

Edit field	Description	Used or required by simulations	Field name in datastructure
Length	Characteristic length of the catchment	Yes	ModelBLength
Slope	Representative slope of the catchment	Yes	ModelBSlope
Parameter Set	Reference to a set of model parameters to be used for the current catchment	Yes	ModelBParBID

Table 4.6 Overview of the Kinematic Wave model attributes (Table ms_Catchment)

Edit field	Description	Used or required by simulations	Field name in datastructure
Contributing Area (five fields)	Fraction of the actual model area for each surface category	Yes	ModelBAISteep, ModelBAIFlat, ModelBAPSmall, ModelBAPMedium, ModelBAPLarge
Use Local Manning values	Allows for local use of individual Manning coefficients	Optional	ModelBLocalNo
Formulation	Selects whether the local Manning values are expressed as Manning (M) or Manning (n)	Yes, If Use local Manning values is chosen	FricTypeNo
Manning Number (five fields)	Local value of the Manning number for various types of catchment surfaces	Optional	ModelBMISteep, ModelBMIFlat, ModelBMPSmall, ModelBMPMedium,ModelBMPLarge

Parameters Kinematic Wave

The Kinematic Wave model uses a relatively large number of parameters. For practical reasons, these parameters have been grouped in parameter sets, which, in turn, can be associated with certain catchments. The entire model setup can thus be established with a very small amount of information, while still allowing for full spatial variation of model parameters for individual catchments through the application of local values.

New parameter sets can be inserted and values of individual parameters can be edited in the Parameters Kinematic Wave Editor. This can be activated at Catchments|Parameters Kinematic Wave.

MIKE+ comes with a Default parameter set (-DEFAULT-). The User can insert any number of parameter sets and edit them as needed.

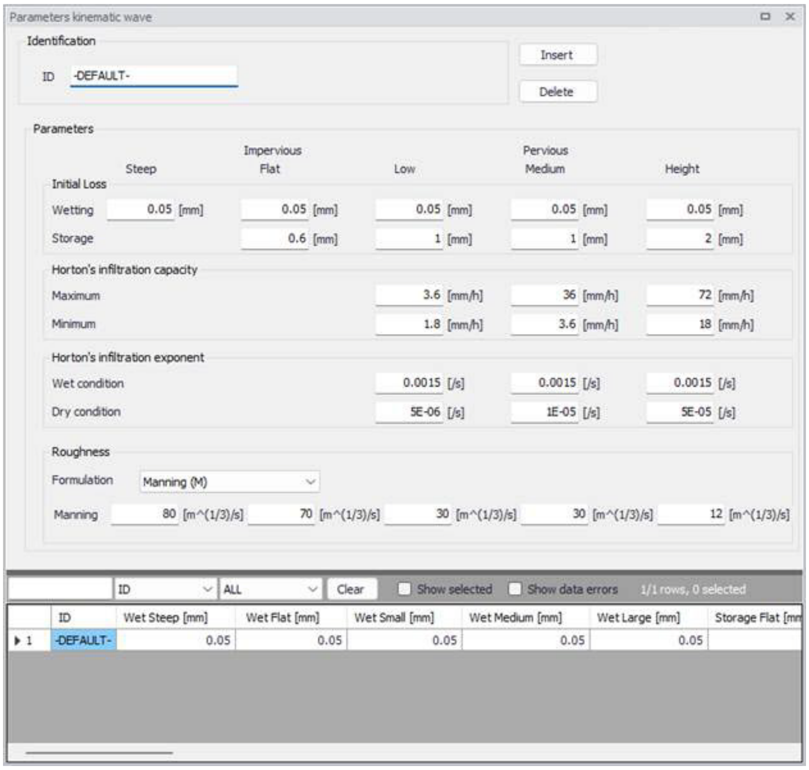


Figure 4.12 The Kinematic Wave Parameter Sets Editor (Catchments|Parameters Kinematic Wave)

A full overview of the editor fields and corresponding database attributes is provided in the Table 4.7.

Table 4.7 Overview of the Kinematic Wave Parameter Set Attributes (Table msm_HParB)

Edit field	Description	Used or required by simulations	Field name in datastructure
Parameter set ID	Parameter set identifier	Yes	MUID
Wetting (five fields)	Wetting initial loss on various types of surfaces	Yes	WetSteep, WetFlat, WetSmall, WetMedium, WetLarge
Storage (four fields)	Storage initial loss on various types of surfaces	Yes	StorageFlat, StorageSmall, StorageMedium, StorgaeLarge

Table 4.7 Overview of the Kinematic Wave Parameter Set Attributes (Table msm_HParB)

Edit field	Description	Used or required by simulations	Field name in datastructure
Maximum (three fields)	Horton's maximum infiltration capacity on various types of surfaces	Yes	InfMaxSmall, InfMaxMedium, InfMaxLarge
Minimum (three fields)	Horton's minimum infiltration capacity on various types of surfaces	Yes	InfMinSmall, InfMinMedium, InfMinLarge
Wet Condition (three fields)	Horton's exponent for wet conditions on various types of surfaces	Yes	InfExpWetSmall, InfExpWetMedium, InfExpWetLarge
Dry Condition (three fields)	Horton's exponent for dry conditions on various types of surfaces	Yes	InfExpDrySmall, InfExpDryMedium, InfExpDryLarge
Formulation	Selects whether the Manning values are expressed as Manning (M) or Manning (n)	Yes	FricTypeNo
Manning (five fields)	Manning number for various types of catchment surfaces	Yes	ManningSteep, ManningFlat, ManningSmall, ManningMedium, ManningLarge

4.2.3 Linear Reservoir (C1 and C2)

The Linear Reservoir model (Model C) is a surface runoff model with minimum data requirements. The runoff computation is based on a comprehensive treatment of hydrological losses (including infiltration) and runoff routing by the linear reservoir principle. Technical details can be found in the MIKE 1D Reference Manual.

MIKE+ includes this model in two versions: C1 and C2. These are variants of the same model, and are used as national standards in the Netherlands and France, respectively.

Linear Reservoir C1 Model Data

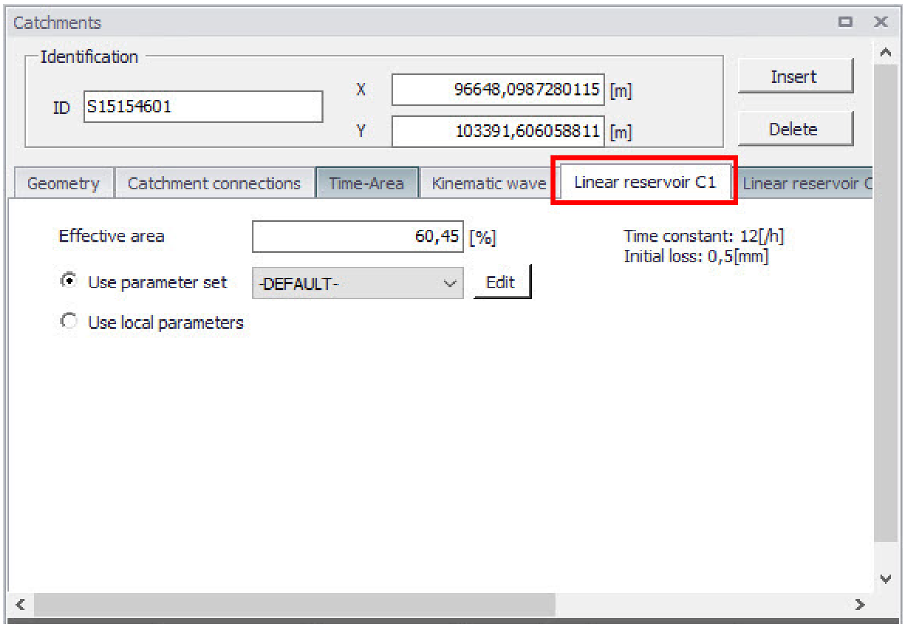


Figure 4.13 The Linear Reservoir C1 Model Tab

A full overview of the editor fields and corresponding attributes for the Linear Reservoir C1 model is provided in the Table 4.8.

Table 4.8 Overview of the Linear Reservoir C1 attributes (Table msm_Catchment)

Edit field	Description	Used or required by simulations	Field name in datastructure
Effective Area	Contributing area	Yes	ModelC1EffectiveArea
Use Parameter Set	Reference to a set of model parameters to be used for the current catchment	Yes	ModelC1ParCID
Use Local Parameters	Allows for local use of individual parameters	Optional	ModelC1LocalNo

Table 4.8 Overview of the Linear Reservoir C1 attributes (Table msm_Catchment)

Edit field	Description	Used or required by simulations	Field name in datastructure
Time Constant	Local value of the time constant	Optional	ModelC1TimeConst
Init. Loss	Local value of the initial loss	Optional	ModelC1Iloss

Linear Reservoir C2 Model Data

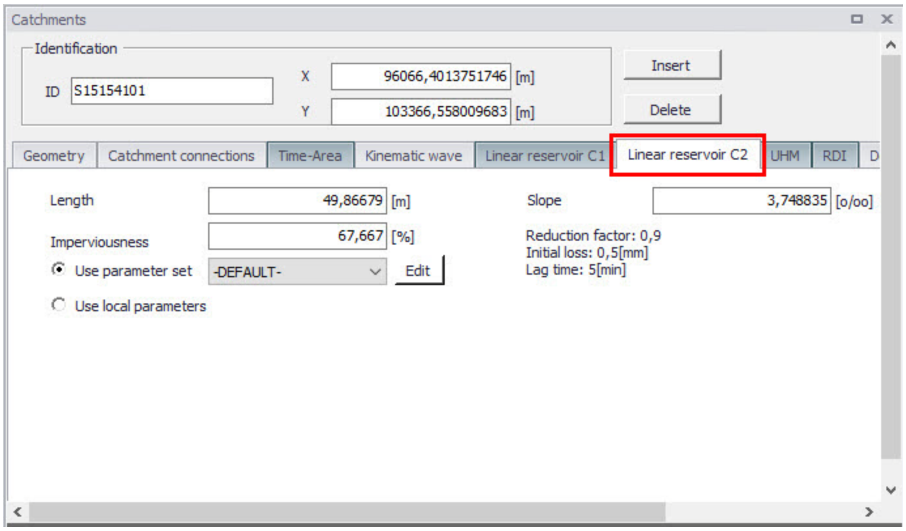


Figure 4.14 The Linear Reservoir C2 Model Tab

A full overview of the editor fields and corresponding Linear Reservoir C2 model attributes is provided in the Table 4.9.

Table 4.9 Overview of the Linear Reservoir C2 attributes (Table msm_Catchment)

Edit field	Description	Used or required by simulations	Field name in datastructure
Length	Characteristic length of the catch-ment	Yes	ModelCLength
Slope	Representative slope of the catch-ment	Yes	ModelCSlope

Table 4.9 Overview of the Linear Reservoir C2 attributes (Table msm_Catchment)

Edit field	Description	Used or required by simulations	Field name in datastructure
Imperviousness	Impervious area as a percentage of the actual model area. For values >= 20%, the catchment is an urban catchment. For values < 20%, the catchment is a rural one.	Yes	ModelC2ImperviousArea
Use Parameter Set	Reference to a set of model parameters to be used for the current catchment	Yes	ModelC2ParCID
Use Local Parameters	Allows for local use of individual parameters	Optional	ModelC2LocalNo
Reduction Factor	Local value of the hydrological reduction factor	Optional	ModelC2RedFactor
Init. Loss	Local value of the initial loss	Optional	ModelC2Iloss
Lag Time	Local value of the lag time	Optional	ModelC2LagTime

Parameters Linear Reservoir

The Linear Reservoir model uses a number of parameters. For practical reasons, these parameters have been grouped in parameter sets, which, in turn, can be associated with certain catchments. This means the entire model setup can be established with a very small amount of information, while still allowing for full spatial variation of model parameters for individual catchments through the application of local values.

New parameter sets can be inserted and values of individual parameters can be edited in the Parameters Linear Reservoir Editor. This can be activated via Catchments|Parameters Linear Reservoir.

MIKE+ comes with a Default parameter set (-DEFAULT-). The User can insert any number of parameter sets and edit them as needed.

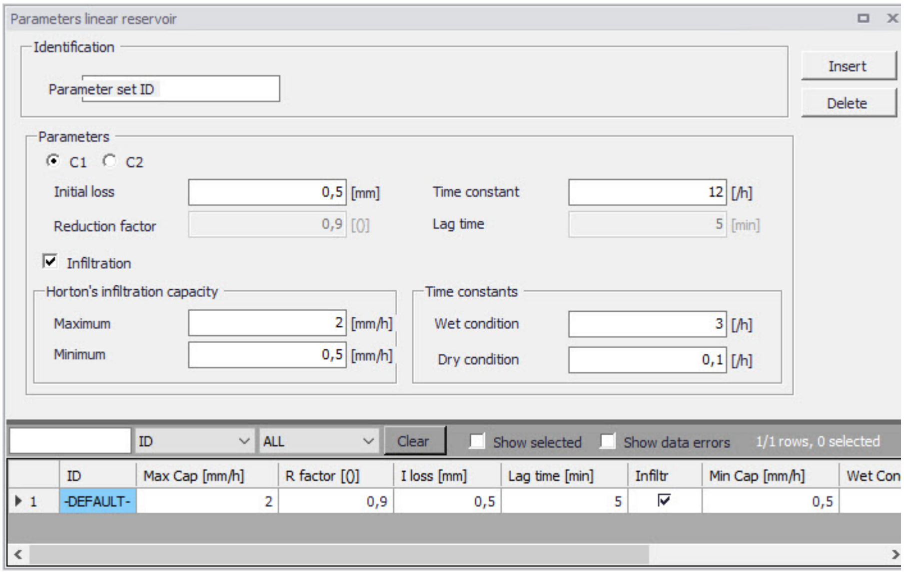


Figure 4.15 The Linear Reservoir C1 Parameter Sets Editor (Catchments|Parameters Linear Reservoir) - Dutch version

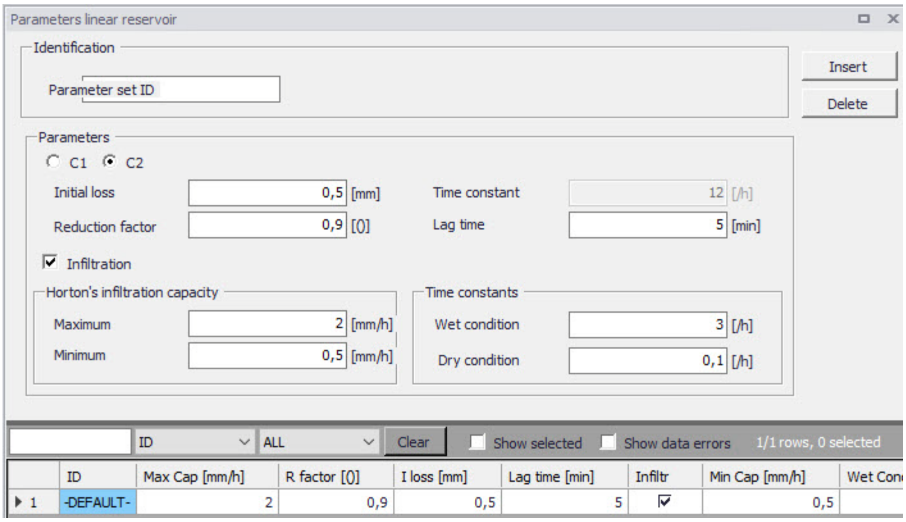


Figure 4.16 The Linear Reservoir C2 Parameter Sets Editor (Catchments|Parameters Linear Reservoir) - French version

A full overview of the editor fields and corresponding database attributes is provided in the Table 4.10.

Table 4.10 Overview of the Linear Reservoir Parameter Set attributes (Table msm_HParC)

Edit field	Description	Used or required by simulations	Field name in datastructure
Parameter Set ID	Parameter set identifier	Yes	MUID
C1/C2	Toggle for editor setting for model C1 and C2, respectively	Yes	-
Initial Loss	Local value of the initial loss	Yes	Iloss
Time Constant	Local value of the time constant	Optional, model C1 only	CTime
Reduction Factor	Local value of the hydrological reduction factor	Optional, model C2 only	RFactor
Lag Time	Local value of the lag time	Optional, model C2 only	LagTime
Infiltration	Toggle for switching ON and OFF calculation of infiltration	Optional	InfitrNo
Maximum Horton´s Infiltration Capacity	Maximum infiltration capacity	Optional, if infiltration included	MaxCap
Minimum Horton´s infiltration capacity	Minimum infiltration capacity	Optional, if infiltration included	MinCap
Wet Condition Time Constant	Infiltration time constant for wet conditions	Optional, if infiltration included	WetCond
Dry Condition Time Constant	Infiltration time constant for dry conditions	Optional, if infiltration included	DryCond

4.2.4 Unit Hydrograph Method (UHM)

The Unit Hydrograph Method (UHM) is a simple linear surface runoff model used to derive hydrographs for any amount of excess precipitation. The runoff computation includes a comprehensive treatment of hydrological losses (i.e. calculation of excess precipitation) and runoff routing through creation of a composite hydrograph. Technical details can be found in the MIKE 1D Reference Manual.

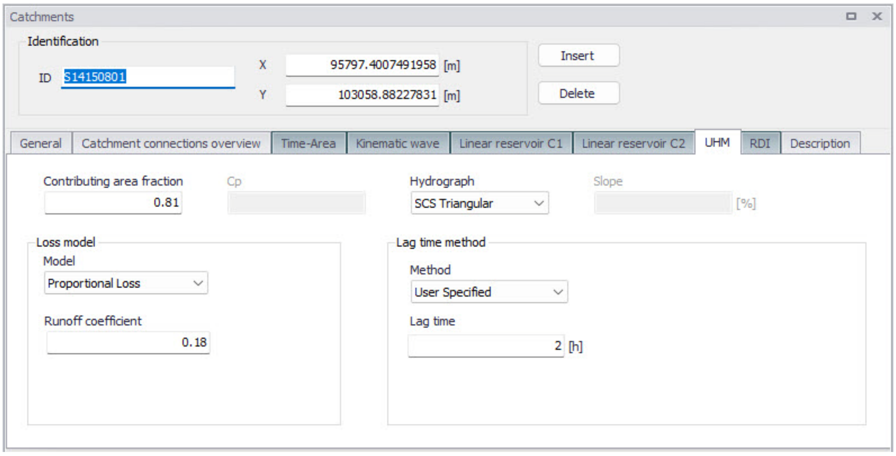


Figure 4.17 The Unit Hydrograph Model Tab

An overview of the editor fields and corresponding UHM Model attributes is provided in the Table 4.11.

Table 4.11 Overview of the UHM Model attributes (Table msm_Catchment)

Edit field	Description	Used or required by simulations	Field name in datastructure
Contributin area fraction	Relative size of contributing area	Yes	UHMAreaFactor
Cp	Hydrograph peak factor	Optional, SUH Standard method only	UHMCp
Hydrograph	Method for hydrograph computation	Yes	UHMMethodNo
Slope	Representative catchment slope	Optional, SUH Alameda method only	UHMSuhSlope
Loss Model	Method for computing hydrological losses	Yes	UHMLossModelNo
Initial Loss	Initial loss	Optional, Constant Loss method only	UHMInitLoss
Constant Loss	Constant loss	Optional, Constant Loss method only	UHMConstLoss
Runoff Coefficient	Runoff coefficient	Optional, Proportional Loss method only	UHMRunoffCoeff

Table 4.11 Overview of the UHM Model attributes (Table msm_Catchment)

Edit field	Description	Used or required by simulations	Field name in datastructure
Curve Number	Standard hydrograph curve number	Optional, SCS and SCS Generalized loss methods only	UHMCurveNum
Initial AMC	Antecedent moisture condition	Optional, SCS loss method only	UHMAMC
Initial Abstr. Depth	Initial abstraction depth	Optional, SCS Generalized loss method only	UHMInitAbstractDepth
Lag Time Method	Method for lag time computation	Yes	UHMLagTimeMethodNo
Lag Time	User-specified lag time	Optional, User specified lag time method only	UHMLagTime
Hydraulic Lengt	Hydraulic length of the catchment	Optional, SCS Formula lag time method only	UHMHydraulicLength
LT Curve No.	CSC Curve number used for computing lag time	Optional, SCS Formula lag time method only	UHMLagCurveNum
LT Slope	Average catchment slope	Optional, SCS Formula lag time method only	UHMSlope
L	Length of the main stream from outlet to the divide	Optional, SUH Standard and Alameda lag time methods only	UHMSuhL
Ct	Watershed (catchment) coefficient	Optional, SUH Standard lag time method only	UHMSuhCt
Lc	Length of the main stream from the outlet to the point closest to the catchment centroid	Optional, SUH Standard and Alameda lag time methods only	UHMSuhLc
Stream Slope	Average Overland Slope	Optional, SUH Alameda lag time method only	UHMStreamSlope
Basin Factor	Basin factor	Optional, SUH Alameda lag time method only	UHMBasFactor

4.2.5 RDI

Continuous runoff from MIKE+ catchments can be modelled using an RDI computation, a continuous hydrological model Rainfall Dependent Infiltration (RDI). RDI provides detailed, continuous modelling of the complete land phase of the hydrologic cycle, providing support for urban, rural, and mixed catchments analyses. Precipitation is routed through four different types of storage: snow, surface, unsaturated zone (root-zone) and groundwater. This enables continuous modelling of the runoff processes, which is particularly useful when long-term hydraulic and pollution load effects are analyzed.

Instead of only performing hydrological load analysis of the sewer system for short periods of high intensity rainstorms, a continuous, long-term analysis is applied to look at periods of both wet and dry weather, as well as inflows and infiltration to the sewer network. This provides a more accurate picture of actual loads on treatment plants and combined sewer overflows.

When studying the real flow conditions in sewer systems, flow peaks during rain events are often found to exceed the values that can be attributed to the contribution from participating impervious areas. This is a consequence of the phenomenon, usually named Rainfall Induced Infiltration. This differs from the Rainfall Induced Inflow by the fact that it does not only depend on the actual precipitation, but is heavily affected by the actual hydrological situation, i.e. the memory from earlier hydrological events. For a certain rainfall event, the increase in flow will therefore differ, depending on hydrological events during the previous period. The Rainfall Induced Infiltration is also distinguished by a slow flow response, which takes place during several days after the rainfall event.

From a hydrological point of view, parts of the infiltration behave in the same way as the inflow. Therefore, classification of total hydrological loads to infiltration and inflow is not suitable for modelling approach. Rather, to describe appropriately the constitutive components of flow hydrographs distinguished by their hydrological behaviour, the following concept is used instead:

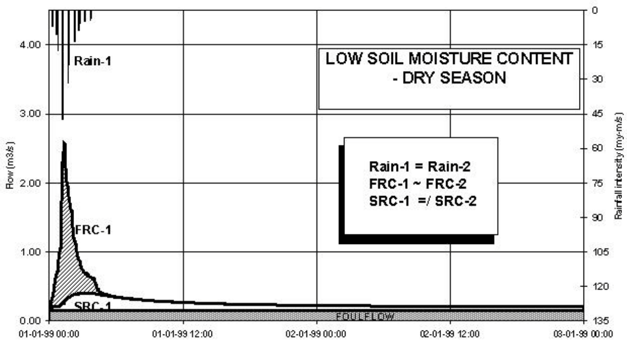
- FRC - Fast Response Component: comprises the rain induced inflow and fast infiltration component;
- SRC - Slow Response Component: comprises slow infiltration component.

Distinctive for the FRC component is that it is not influenced by the previous hydrological situation, i.e. high or low soil moisture content. It occurs as a direct consequence of a rainfall. The FRC component consists of the inflow to the sewer system and the fast flow component of the infiltration, not dependent on previous hydrological conditions.

On the other hand, characteristic of the SRC component is that it is highly dependent on previous hydrological conditions and usually responds slowly to a rainfall. The SRC component consists of the rest of the precipitation-induced infiltration and dry weather infiltration/inflow.

When combined with any of the surface runoff models, RDI provides a platform for accurate and reliable computation of runoff free from the limitations inherent to standard urban runoff modelling.

Figure 4.18 shows an example illustrating the influence of previous hydrological conditions for the two components and their response to a rainfall.



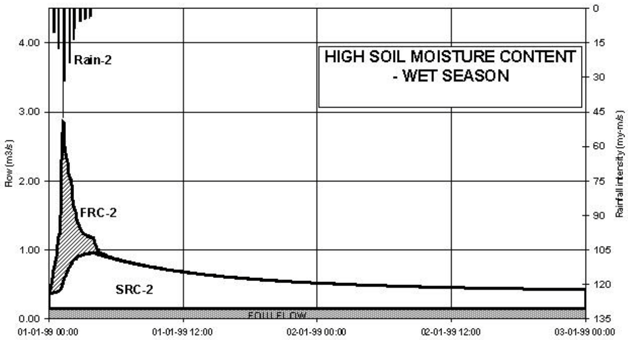


Figure 4.18 Different catchment response under the same rainfall due to different soil moisture conditions at the beginning of the rainfall

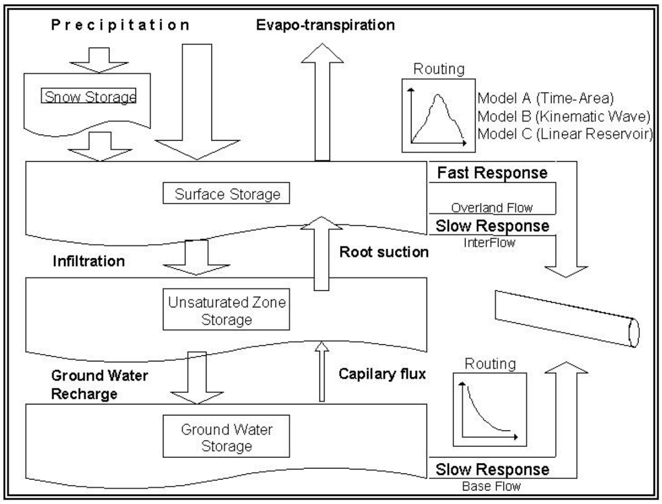


Figure 4.19 Schematics of the RDI Model

Model Data

The model data for additional catchment flow and RDI are in the database table msm_Catchment.

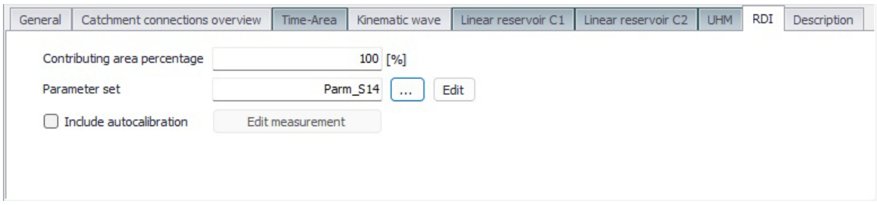


Figure 4.20 The RDI tab in the Catchments editor

The contributing area percentage is the fraction of the catchment area to which the RDI model is applied. The parameter set is selected from the list of sets, defined in the 'Parameters RDI' editor.

When 'Include autocalibration' is selected, the simulation will initially run iterations to calibrate the RDI parameters in the selected parameter set against a discharge measurement time series, in .dfs0 file format. This discharge measurement time series must be specified in a measurement station connected to the same catchment (in 'Calibrations | Measurement stations').

A full overview of the editor fields and corresponding database attributes is provided in the Table 4.12.

Table 4.12 Overview of the RDI attributes (Table msm_Catchment)

Edit field	Description	Used or required by simulations	Field name in datastructure
Contributing area percentage	Fraction of catchment area to which the RDI model is applied	Yes, if RDI activated	RdiiArea
Parameter Set	Reference to a set of RDI model parameters to be used for the current catchment	Yes, if RDI activated	ParRDiiID
Include autocalibration	Enables automatic calibration of the RDI parameters against measurement time series	Optional	IncludeAutoCalNo

Parameters RDI

The RDI Model uses a large number of parameters. For practical reasons, these parameters have been grouped in parameter sets, which can be associated with certain catchments. This means the entire model setup can be established with a very small amount of information.

New parameter sets can be inserted and values of individual parameters can be edited in the Parameters RDI Editor. This can be activated via Catchments|Parameters RDI.

MIKE+ comes with a Default RDI parameter set (-DEFAULT-). The User can insert any number of parameter sets and edit them as needed.

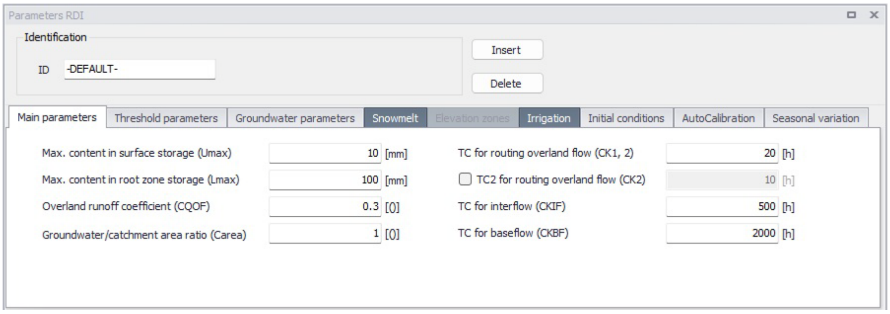


Figure 4.21 The Main Parameters tab

The RDI parameters are grouped into Tabs in the editor by:

- Main parameters
- Threshold parameters
- Groundwater parameters
- Snow melt
- Elevation zones
- Irrigation
- Initial conditions
- AutoCalibration
- Seasonal variation

A full overview of the editor fields and corresponding database attributes is provided in the talbles below.

Table 4.13 Overview of Parameters RDI Main Parameters Tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	Parameter set identifier	Yes	MUID
Max. content in surface storage (Umax)	Capacity of surface storage. 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.	Yes	Umax

Table 4.13 Overview of Parameters RDI Main Parameters Tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
Max. content in root zone storage (Lmax)	Capacity of root zone (lower) storage. 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.	Yes	Lmax
Overland runoff coefficient (CQOF)	Fraction of runoff going to overland flow. It determines the division of excess rainfall between overland flow and infiltration.	Yes	Cqof
Groundwater(catchment area ratio (Carea)	This parameter describes the ratio of the groundwater catchment area to the topographical surface water catchment area. 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.	Yes	GwCarea

Table 4.13 Overview of Parameters RDI Main Parameters Tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
TC for routing overland flow (CK1,2)	Time constant for overland runoff component. It determines 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.	Yes	Ck
TC2 for routing overland flow (CK2)	Enables specification of a second time constant for the second linear reservoir.	Optional	Ck2TypeNo
CK2 value	The time constant for the second linear reservoir, when 'TC2 overland flow (CK2)' is selected	Yes, if 'TC2 overland flow (CK2)' is selected	Ck2

Table 4.13 Overview of Parameters RDI Main Parameters Tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
TC for interflow (CKIF)	Time constant for interflow runoff component. It determines the amount of interflow, which decreases with larger time constants. Values in the range of 500-1000 hours are common.	Yes	Ckif
TC for baseflow (CKBF)	Time constant for baseflow runoff component. 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, in the 'Groundwater parameters' tab.	Yes	Ckbf

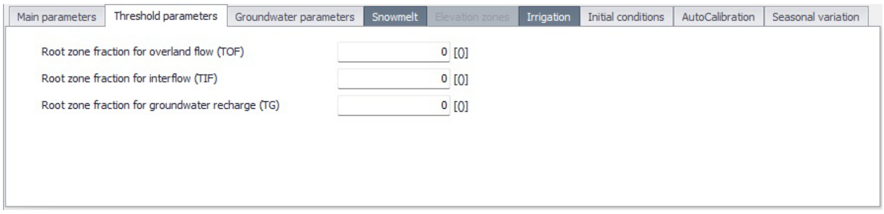


Figure 4.22 The Threshold Parameters Tab

Table 4.14 Overview of Parameters RDI Threshold Parameters Tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
Root zone fraction for overland flow (TOF)	Fraction of root zone capacity above which overland flow starts. 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 Lmax, and the maximum value allowed is 0.99.	Yes	Tof
Root zone fraction for interflow (TIF)	Fraction of root zone capacity above which interflow starts.	Yes	Tif
Root zone fraction for groundwater recharge (TG)	Fraction of root zone capacity above which ground water recharge starts. The main impact of increasing TG is less recharge to the groundwater storage. Threshold values range between 0 and 70% of Lmax and the maximum value allowed is 0.99.	Yes	Tg

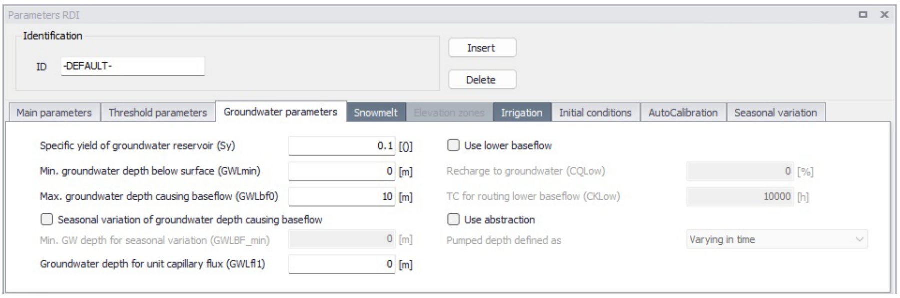


Figure 4.23 The Groundwater Parameters Tab

Table 4.15 Overview of Parameters RDI Groundwater Parameters Tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
Specific yield of groundwater reservoir (Sy)	Specific yield of the groundwater reservoir (porosity). This parameter should be kept at the default value except for the special cases in which the groundwater level is used for 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 GWLbf0, which may vary in time. The value of Sy depends on the soil type and may often be assessed from hydrogeological data, e.g. test pumping. Typically values of 0.01-0.10 for clay and 0.10-0.30 for sand are used.	Yes	GwSy
Min. groundwater depth below surface (GWLmin)	Top of the groundwater storage (depth from surface)	Yes	GwLmin

Table 4.15 Overview of Parameters RDI Groundwater Parameters Tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
Max. groundwater depth causing base-flow (GWLbf0)	Bottom of the ground-water storage (depth from surface)). This parameter should be kept at the default value except for the special cases in which the groundwater level is used for calibration, cf. Sy above.	Yes	GWLbf0
Seasonal variation of groundwater depth causing base-flow	In low-lying catch-ments the annual vari-ation of the maximum groundwater depth may be of importance. This variation relative to the difference between the maxi-mum and minimum groundwater depth can be entered if the option is ticked. The monthly values are given relative to the difference between GWLbf0 and GWLB-F_min [0-1]. This is done in the 'Seasonal variation' tab, in the column 'Variation of groundwater maximum water depth'.	Optional	GWSeasonTypeNo
Min. GW depth for seasonal variation (GWLBF_min)	If the Seasonal varia-tion of groundwater depth causing base-flow is selected, the minimum groundwater depth [m] has also to be entered for the cal-culations.	Optional	GWLBFMin

Table 4.15 Overview of Parameters RDI Groundwater Parameters Tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
Groundwater depth for unit capillary flux (GWLfl1)	Groundwater depth causing 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 applications. Keep the default value of 0.0 to disregard capillary flux.	Yes	Gwlfl1
Use lower baseflow	Activates the use of a second linear reservoir, which can have a larger time constant to better represent the groundwater recession.	Optional	LowerBFTypeNo
Recharge to groundwater (CQLow)	The recharge to the lower (second) groundwater reservoir is specified here as a percentage of the total recharge.	Yes, if 'Use lower baseflow' is selected	CQLow
TC for routing lower baseflow (CKLow)	A baseflow time constant, which is usually larger than the CKBF.	Yes, if 'Use lower baseflow' is selected	CKLow

Table 4.15 Overview of Parameters RDI Groundwater Parameters Tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
Use abstraction	If this option is selected, it is possible to specify the ground-water abstraction depth from the catch-ment.	Optional	AbstractionTypeNo
Pumped depth defined as	When abstraction is used, the pumped depth can be defined either as 'Varying in time' (in which case the time series is pro-vided as a catchment boundary condition), or using 'Seasonal var-iation' (in which case monthly values of abstraction depth are defined in the 'Sea-sonal variation' tab).	Optional	AbstractionSource-TypeNo

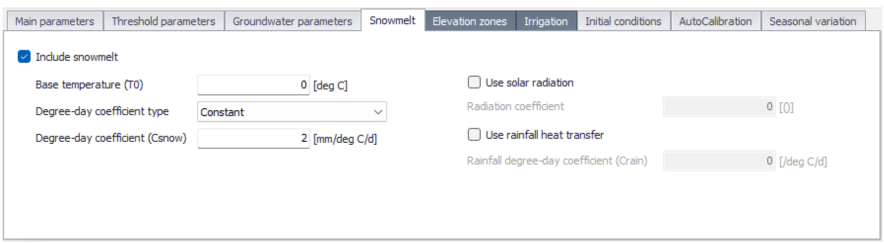


Figure 4.24 The 'Snowmelt' Tab

The snow module simulates the accumulation and melting of snow in a RDI catchment. It is included in the model when the 'Include snowmelt' checkbox is ticked.

The snow melt module uses a temperature input time series, usually mean daily temperature. This has to be specified as a boundary condition for the catchments.

Table 4.16 Overview of the 'Snowmelt' tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in data-structure
Include snowmelt	Switch for activation of the snow storage / snowmelt processes	Yes	SnowmeltNo
Base temperature (T0)	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.	Optional	T0
Degree-day coefficient type	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'. This degree-day coefficient can either be constant during the simulation, or varying in time (in which case a time series of degree-day coefficient has to be provided as boundary condition to the catchment), or using seasonal variation (in which case the monthly values of the coefficient have to be specified in the 'Seasonal variation' tab).	Optional	SnowmeltCTypeNo

Table 4.16 Overview of the 'Snowmelt' tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in data-structure
Degree-day coefficient (Csnow)	The constant value of the degree-day coefficient. Typical values for Csnow are 2-4 mm/day/C.	Optional	SnowmeltC
Use solar radiation	This option may be enabled when time series data for incoming radiation are available. The radiation time series must be specified as a boundary condition to the catchment. The total snow melt is calculated as a contribution from the traditional snow melt approach based on Csnow (representing the convective term) plus a term based on the radiation.	Optional	RadiationTypeNo
Radiation coefficient	The radiation coefficient value, when the effect of solar radiation is included.	Optional	RadiationCoef
Use rainfall heat transfer	This option may be enabled when the melting effect from the advective heat transferred to the snow pack by rainfall is significant.	Optional	RainfallCoefTypeNo
Rainfall degree-day coefficient (Crain)	This heat transfer from the rainfall is represented in the snow module as a linear function of the precipitation multiplied by the rainfall degree-day coefficient and the temperature deviation above the base temperature.	Optional	RainfallCoef

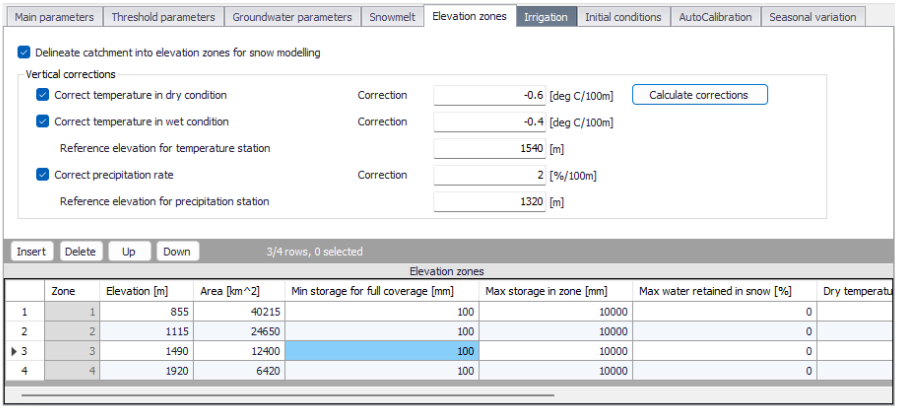


Figure 4.25 The ‘Elevation zones’ Tab

The 'Elevation zones' tab becomes available when snowmelt is included in the model. When the 'Delineate catchment into elevation zones for snow modelling' option is selected, the snowmelt 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.

Global parameters used with elevation zones are described in the table below.

Table 4.17 Overview of the global attributes in the 'Elevation zones' tab (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in data-structure
Delineate catchment into elevation zones for snow modelling	Switch to enable or disable the distributed approach for snowmelt modelling	Yes	ElevZonesNo
Correct temperature in dry condition	When this option is ticked, the temperature correction will be recomputed in the table of zones when pressing the 'Calculate corrections' button. The correction value specifies the vertical gradient [C/100 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 zone.	Optional	CorrectDryTempNo and CorrectDryTempCorrection

Table 4.17 Overview of the global attributes in the 'Elevation zones' tab (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in data-structure
Correct temperature in wet condition	When this option is ticked, the temperature correction will be recomputed in the table of zones when pressing the 'Calculate corrections' button. The correction value 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.	Optional	CorrectWetTempNo and CorrectWetTempCorrection
Reference elevation for temperature station	This parameter defines the elevation of the reference temperature station. This station is used as a reference for calculating the temperature correction within each elevation zone.	Optional	TempStationElev

Table 4.17 Overview of the global attributes in the 'Elevation zones' tab (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in data-structure
Correct precipitation rate	When this option is ticked, the precipitation correction will be recomputed in the table of zones when pressing the 'Calculate corrections' button. The correction value 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.	Optional	CorrectPrecNo and CorrectPrecCorrection
Reference elevation for precipitation station	This parameter defines the elevation of the reference precipitation station. This station is used as a reference for calculating the precipitation correction within each elevation zone.	Optional	PrecStationElev

Elevation zones are defined in the secondary table at the bottom of the tab. The number of zones is defined by the number of records in this table. Parameters used for each elevation zone are described in the table below.

Table 4.18 Overview of the zones' attributes in the 'Elevation zones' tab (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in data-structure
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).	Yes	Elevation
Area	The area of the zone. The total area of the elevation zones must equal the area of the catchment.	Yes	Area
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.	Yes	MinStorage
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.	Yes	MaxStorage

Table 4.18 Overview of the zones’ attributes in the 'Elevation zones' tab (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in data-structure
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 T0, the liquid water of the snow re-freezes with rate Csnow.	Yes	MaxWaterRetain
Dry temperature correction	The temperature correction for dry conditions to estimate actual temperature for the specific zone.	Yes	DryTempCorrection
Wet temperature correction	The temperature correction for wet conditions to estimate actual temperature for the specific zone.	Yes	WetTempCorrection
Precipitation correction	The relative correction for precipitation, expressed in percent, to estimate the precipitation for the specific zone.	Yes	PrecipTempCorrection

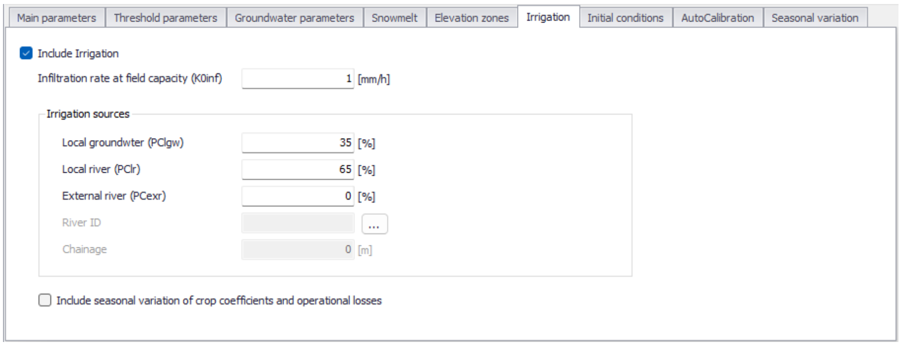


Figure 4.26 The 'Irrigation' Tab

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 RDI model. The purpose of the irrigation module is that of simulating the runoff and groundwater recharge/baseflow from the irrigated areas, so that RDI 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 RDI 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.

Table 4.19 Overview of the 'Irrigation' tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
Include irrigation	A switch to enable or disable the irrigation module.	Yes	IncludeIrrigationNo
Infiltration rate at field capacity (K0inf)	This parameter defines the rate of infiltration at field capacity	Optional	K0inf
Local groundwater (PClgw)	Percentage of water for irrigation supplied by groundwater sources.	Optional	PClgw
Local river (PClr)	Percentage of water used for irrigation supplied by a local river.	Optional	PClr
External river (PCexr)	Percentage of water for irrigation supplied by a river external from the catchment.	Optional	PCexr
River ID	When part of the irrigation source comes from an external river (PCexr > 0), the River ID field is used to specify the location on the river network where the water is taken from.	Optional	RiverID

Table 4.19 Overview of the 'Irrigation' tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
Chainage	When part of the irrigation source comes from an external river, the Chainage field is used to specify the location on the river where the water is taken from.	Optional	Chainage
Include seasonal variation of crop coefficients and operational losses	If this checkbox is ticked, 'Irrigation crop coefficient' and 'Irrigation operational and conveyance losses' can be specified as monthly values in the 'Seasonal variation' tab. The 'Irrigation crop coefficient', 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' represent the system losses of irrigation water for the different components / processes (Groundwater, Overland flow, Evaporation).	Optional	IncludeSeasonal-VarNo

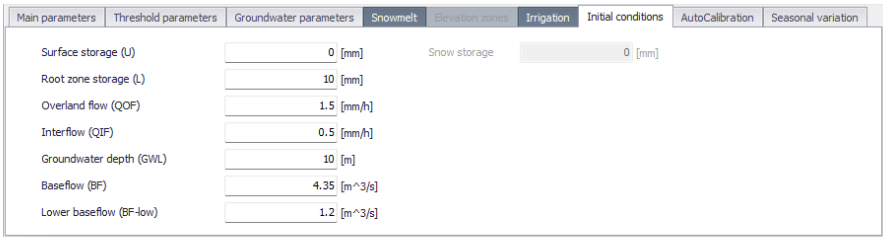


Figure 4.27 The ‘Initial Conditions’ Tab

Table 4.20 Overview of Parameters RDI Initial Conditions Tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
Surface Storage (U)	Water depth in the surface storage at the simulation start	Yes	InitU
Root zone storage (L)	Moisture contents in the root zone at the simulation start	Yes	InitL
Overland flow (QOF)	Overland flow intensity at the simulation start	Yes	InitOf
Interflow (QIF)	Interflow intensity at the simulation start	Yes	InitIf
Groundwater depth (GWL)	Groundwater depth at the simulation start	Yes	InitGwl
Baseflow (BF)	The baseflow at the beginning of the simulation, which is normally estimated from the hydrograph.	Yes	InitBF
Lower baseflow (BF-low)	The lower baseflow at the beginning of the simulation, which is normally estimated from the hydrograph.	Yes	InitBFLow

Table 4.20 Overview of Parameters RDI Initial Conditions Tab attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
Snow storage	If elevation zones are not used for snow melt modelling, a unique snow storage value describes the initial amount of snow over the entire catchment. If elevation zones are used, a snow storage value is specified in the table for each zone.	Optional	SnowStorage
Water in snow	If elevation zones are used, a value representing the water content in the snow pack is specified in the table for each zone.	Optional	WaterInSnow

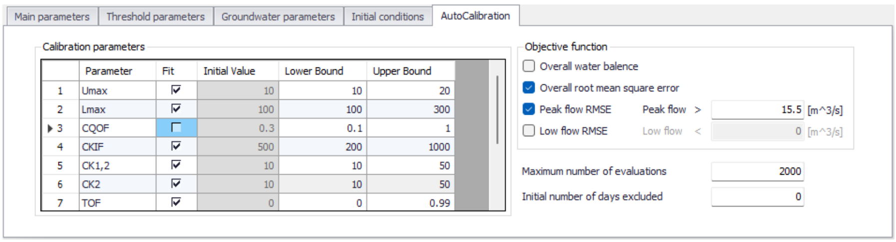


Figure 4.28 The AutoCalibration Tab

It is possible to use an automatic optimisation procedure to calibrate the 12 most important parameters in the RDI model. The calibration routine used in RDI is based on a multi-objective optimisation strategy, the SCE algorithm. The procedure implemented in RDI allows to simultaneously optimise four different calibration objectives or a combination of them. This automatic calibration is activated or deactivated from the 'RDI' tab in the Catchments editor.

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. These are selected and used by ticking the available check boxes:

- 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.

Table 4.21 Overview of Parameters RDI, AutoCalibration tab, attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
Fit	If the 'Fit' check-box is ticked, the parameter is included in the autocalibration	Yes	FitNo
Initial value	It is the initial value for the parameter. It always corresponds to the value specified in the corresponding tab.	Yes	*Read-only*
Lower bound	Minimum value that the parameter can get	Yes	LowerBound
Upper bound	Maximum value that the parameter can get	Yes	UpperBound

Table 4.21 Overview of Parameters RDI, AutoCalibration tab, attributes (Table 'msm_HParRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
Objective function	Selection of objective function for the calibration target.	Yes	ObjFuncWaterBalanceNo ObjFuncRmsErrorNo ObjFuncPeakflowNo ObjFuncLowflowNo
Maximum number of evaluations	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.Yes	Yes	ObjFuncMaxEvaluation
Initial number of days excluded	A warm up period that will be disregarded when calculating the objecting function.		ObjFuncInitDaysExcluded

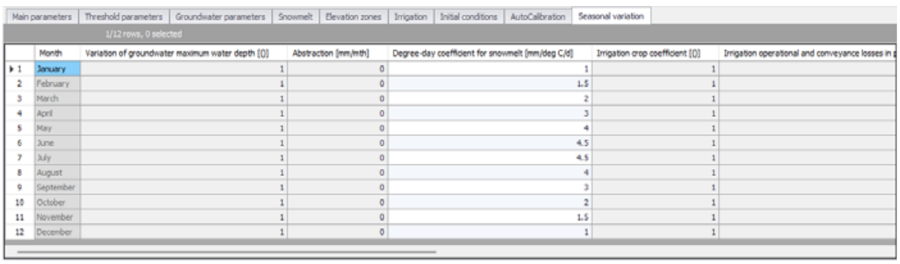


Figure 4.29 The 'Seasonal variation' Tab

In this tab, it is possible to enter the monthly values of some selected parameters. Whether a parameter uses a constant or a seasonal varying value is determined in the different tabs of the RDI parameters editor.

It is possible to specify a seasonal variation for the following parameters:

- Variation of groundwater maximum water depth: used for the seasonal variation of groundwater depth causing baseflow, when this option is active in the 'Groundwater parameters' tab.
- Abstraction: used for the seasonal variation of groundwater abstraction depth, when 'Use abstraction' is active in the 'Groundwater parameters' tab.
- Degree day coefficient for snow melt: used for the seasonal variation of the melting coefficient, when the 'Degree-day coefficient type' is defined with seasonal variation in the 'Snowmelt' tab.
- Irrigation crop coefficient: used for the seasonal variation of the parameter quantifying the amount of water required (and transpired) by the crop, when 'Include seasonal variation of crop coefficients and operational losses' is ticked in the 'Irrigation' tab.
- Irrigation operational and conveyance losses in percent of abstracted water: used for the seasonal variation of the system losses of irrigation water for the different components/processes (Groundwater, Overland flow and Evaporation), when 'Include seasonal variation of crop coefficients and operational losses' is ticked in the 'Irrigation' tab.

4.2.6 RDI - Guidelines for Application

Choice of calculation time step

When calculating with RDI, time steps are given separately for the Surface Runoff Model and for the rain dependent infiltration part.

The RDI calculation can often be performed with a relatively long time step (several hours), while calculation with the Surface Runoff Model is typically performed with a time step in the order of several minutes.

The time step for Surface Runoff computations is primarily about the sufficient resolution of the runoff process in time.

Generally, the RDI simulation time step should be chosen in accordance with the resolution of precipitation data, e.g. a time step of 24 hours could be suitable if only daily precipitation data is available. However, in cases when precipitation data with high resolution of e.g. few minutes are available, the RDI time step should be chosen in accordance with the response of the discharge when raining. E.g. an RDI time step of 2-4 hours should be chosen if the time constant CKOF is given a value of 8 hours.

To minimize the calculation time as well as the size of the result files, the RDI calculations are performed according to the following principle:

The RDI simulation is carried out continuously for the whole period specified. On the contrary, the Surface Runoff simulation is carried out only when raining. Thus, the start time for the Surface Runoff calculation is set as the start time for the rain hydrograph. The Surface Runoff calculation continues until all the surface runoff hydrographs are regressed.

The RDI hotstart

There is a HOTSTART facility for RDI, i.e. the initial conditions for the various storages can be automatically taken from a former result file at a simulation start time.

The structure and contents of the result file used as a HOTSTART file requires that the time series in the boundary connection start at least for the maximum specified concentration time Tc earlier than the start time for the HOTSTART is specified. This is required for the correct reconstruction of the surface runoff component (FRC).

RDI Validation

Some of the parameters in RDI (here meaning both for the rain dependent inflow and the infiltration part) are related to actual physical data. However, the final choice of parameter values must be based on a comparison with historical measured discharges since a number of the parameters have an empirical character.

The available period of the measured discharge data and its resolution in time are of major importance for the credibility of the obtained parameter values. Ideally, for good accuracy, a 3-5 years long time series of measured discharge data with daily values is required for the calibration of the RDI parameters. Several months long time series with higher resolution, i.e. minutes or hours, depending on the size of the area, are needed for the calibration of the surface runoff model. Measured time series with shorter duration are also useful, although not securing optimal parameter values. In such case it is important that the time series represents different hydrological situations, i.e. typical wet period or dry period.

An exact correspondence between simulations and measurements can however not be expected and for areas where precipitation data of worse quality is used, a less accurate calibration result must be accepted. In this case it may be preferable to recall the purpose of the actual model application and concentrate on calibrating yearly volumes, flow peaks or base flows, depending on what kind of analysis is to be performed with the model.

It must be remembered that RDI calculates the precipitation-dependent flow component. When comparing with measured discharge data the total measured discharge therefore has to be reduced with the flow components not being precipitation dependent, e.g. foul flow.

RDI calculates the total generated discharge from a catchment, i.e. overflow within the sub-catchment will also be included in the calculated discharge. Therefore, when comparing with measured peak flows and controlling the water balance (total volume) this has to be taken into consideration.

In principle, the model validation is concerned about comparison of the computed and measured hydrographs. As there are almost an infinite number of possibilities to describe level of agreement between two hydrographs, it is recommended to establish some validation criteria, i.e. a measure for accuracy of the model, relevant for the current application. There are several types of criteria, such as numeric criteria based on single values (e.g. peak discharge, volume, etc.), or more complex numeric criteria based on statistical analysis of the computed and calculated time series. Also, there are different types of visual criteria, based on visual inspection, e.g. comparison of graphic presentations of the calculated and measured duration curves. An important issue is to find the most appropriate criteria for the intended application of the model.

The choice of criteria is important since it may affect the final choice of parameter values and by that the behaviour of the calibrated model. Numerical criteria are, however, limited and therefore a visual comparison between the hydrographs is indispensable.

Comparison between the simulated runoff and the observed discharge time series may be obtained in the 'Calibrations | Plots and statistics' view.

Surface runoff model

When simulating storm sewer systems or fully combined systems, usually a good estimation of the area drained by the FRC component (impervious areas, etc.) can be obtained from physical data (maps, etc.). The final model verification of a FRC should however be based upon comparison with measured discharges during rainfall.

To separate the Afrc component (Surface Runoff Model) and the fast part of the SRC component (Surface Runoff Component in RDI), measured discharge data with fairly high resolution in time (hours) is required.

For calibration of the parameters describing the response of the discharge (e.g. tc and TAtype for model A, or M, L and S for model B), a very high resolution in time is usually required, from minutes to hours.

General hydrological model - RDI

It is not possible to determine the RDI parameters from geophysical measurements, since most of the parameters are of empirical nature. It is therefore necessary that measured discharge from the studied area is available, so that the RDI parameters can be determined by comparison between simulated and measured discharge through the calibration procedure.

The introductory calibration is performed visually by comparing simulated and measured discharge. The final optimization of the parameters is thereafter performed preferably using different numeric and graphical criteria.

The effects of changing each particular parameter are discussed below. Also, the most suitable hydrological periods for calibrating certain parameters are identified, which implies that a certain parameter affects the model behaviour more during periods with specific hydrological conditions. Usually, effects will also be obtained during other periods, why these should also be studied when adjusting a parameter.

The parameters are discussed in the preferable order of adjustment. However, it may be necessary to return to the previous calibration step, as well as repeating the whole process several times. It is recommended, especially for less experienced users, that only one parameter is changed at a time (i.e. for each calculation), so that the effect of the adjustment will appear clearly.

Sometimes, however, the effect of changing one parameter is not sufficient. Then, several parameters controlling similar phenomena can be adjusted together.

In some other cases, undesired secondary effects can be obtained when adjusting certain model parameter. These effects can often be eliminated by simultaneously adjusting other parameters, which do not influence the desired effects, but reduce secondary effects induced by the first parameter.

The following sequence of action is recommended:

- The first step in the RDI calibration is usually to adjust the water balance in the system, i.e. the accuracy between the calculated and measured total volume during the observed period. This is done by correcting the proportion of area, Asrc. An increase of Asrc proportionally increases every flow component at each time step.

  The total volume generally also contains the runoff from impervious areas (Surface Runoff Model).
- Next, the overland flow coefficient CQOF is adjusted to obtain a correct distribution of volume between overland flow (peak flows) and baseflow. This is done after wet periods and preferably for a period with low evaporation.

  A reduction of CQOF reduces the overland flow and increases the infiltration, i.e. induces increase in the baseflow.

  The measured flow peaks generally also contain the runoff from impervious areas (Surface Runoff Model).

- CKBF is adjusted against the response of the baseflow, i.e. the build-up and regression of the baseflow. Adjustment against the build-up of baseflow is done during and after wet periods with low evaporation. Adjustment against regression is done during the start of dry periods with high evaporation, preferably when baseflow is the only flow component.

  An adjustment of CKBF does not influence the size of the discharged volume studied for a longer period, but displaces the volumes in time.
- CKOF is adjusted against the response, i.e. the shape of the peak flows. This is done during periods with heavy rainfall, preferably after a wet period.

  The measured flow peaks generally also contain the runoff from impervious areas (Surface Runoff Model).
- A reduction of Umax reduces the actual evapotranspiration, the process responsible for reduced discharges during period with high potential evaporation. The effect of reducing Umax will be largest for periods preceded by a wet period. Additionally, an increased overland flow is obtained, as well as more water transported to the groundwater storage resulting in an delayed effect of increased baseflow, because of the long response time of baseflow.

  An important behaviour of the RDI model is that the surface storage must be filled-up before overland flow and infiltration, respectively, occur. Therefore, during dry periods with high potential evaporation, Umax can be estimated from how much rainfall is required for filling-up the surface storage, i.e. generating overland flow. The same methodology can also be used for the periods with low potential evaporation, but only if the rain event is preceded by a long dry period.
- CKIF is adjusted against the response of interflow during periods with low potential evaporation. A reduction of CKIF will result in a small increase in volume during these periods.
- The relative water content in the unsaturated zone (i.e. root-zone), L/Lmax controls several of the different water transports in the RDI model. Since the storage capacity, Lmax, influences the velocity of the filling of L towards Lmax, Lmax is adjusted during periods of heavy filling of the root zone storage. This usually occurs during periods with low potential evaporation preferably in combination with a wet period.

  A reduction of Lmax increases the discharge, but it may decrease a little during period with very high potential evaporation.

- The threshold values indicate at which relative water content in the root zone, L/Lmax, overland flow, interflow and baseflow respectively will be generated. Therefore, the threshold values can be estimated from the time of filling the root zone storage when each flow component starts discharging.

  The threshold values have no effect during periods when the root zone storage is full, L = Lmax.

  An increased threshold value reduces the discharge during dry periods and in the beginning of wet periods, i.e. periods with low relative water content in the root zone storage.

  TG is adjusted during periods with heavy filling of the root zone storage, preferably in combination with low potential evaporation and preceded by a dry period. TG is therefore an important parameter for adjusting the increase of the groundwater level in the beginning of wet periods.

  TOF is adjusted after a dry period at events with heavy filling of the root zone storage. For example adjustment can be done for events where even larger rainfall volumes does not generate overland flow.

  TIF is adjusted after a dry period when filling of the root zone storage, preferably in combination with low potential evaporation. However, TIF is one of the less important parameters.
- The degree-day-coefficient, Csnow can be estimated from analysis of the relation between temperature, water content in the snow storage and measured discharge. When temperature is below zero, the precipitation is stored in the snow storage. When temperature is above zero the content in the snow storage is emptied into the surface storage, where the velocity of emptying is controlled by Csnow. An increase of Csnow increases the emptying procedure.

  This process should be addressed now and then during the whole calibration procedure. Otherwise, there is a risk that a snow-melting phenomenon is attempted to be described through adjusting other parameters.

- The Carea coefficient establishes the ratio of groundwater catchment and surface catchment (per Default, the two surfaces are equal). By changing the ratio, the ratio between the baseflow and other runoff components is correspondingly changed.

  The Default values of the remaining RDI parameters: Sy (specific yield of the groundwater reservoir), GWLmin (minimum groundwater depth), GWLBF0 (maximum groundwater depth causing baseflow) and GWLFL1 (groundwater depth for unit capillary flux) are adjusted only in exceptional cases. Therefore, these parameters have been included into the RDI parameter set dialog in a separate box. The effects of changing the Default values should be well understood prior to adjustment.

  Since the variation of water contents in the surface and root zone storage controls many of the other processes, they should be studied continuously throughout the calibration procedure.

  Monthly and yearly values for the different processes, e.g. precipitation volume, real evaporation and total discharge, are written to an ASCII file, NAMSTAT.TXT after every RDI calculation. It is recommended that the content of this file is studied now and then during the calibration procedure.

Overflow within the model area

In cases when overflow occurs in the model area, e.g. when simulating the discharge to the treatment plant, this has to be considered when calibrating the peak flows during rainfall. RDI calculates the total generated discharge in the catchment area and is therefore not able to describe hydraulic processes like e.g. overflow (loss of water). Calibration of parameters affecting the volume in the peak flows should therefore be performed for rain events when overflow is unlikely to occur. Model parameters affecting the response of the discharge for rain events when overflow occur can be calibrated against the peak flows base or width.

A well-calibrated RDI model can therefore be used for a rough estimation of overflow volume by studying the difference between calculated and measured discharge for heavy peak flows. The credibility for such estimation is however strongly affected by the quality of measured precipitation and discharge time series.

Non-precipitation dependent flow components

RDI calculates the precipitation dependent flow component. Therefore, both for calibration and validation, other flow components should be treated outside RDI.

Examples of non-precipitation dependent flow components are foul flow and sea water leaking into the sewer system.

The foul flow is preferably estimated through daily values from produced water volumes weighted with yearly charged water volumes. This will however only give a rough estimate, and departure from this methodology may be necessary, e.g. for areas where a large amount of freshwater is used for irrigation.

The amount of leaking sea water is preferably estimated through an iterative procedure between RDI calculation and studies of the difference between the calculated and measured discharge. Only a rough estimation can be achieved, and less accurate calibration results may have to be accepted.

Specially, during the calibration procedure it is very important that non-hydrological errors are generally kept at the lowest level possible in the flow series used. Otherwise, there is a risk of hydrological interpretations of these errors, and the error transmitting in the model and increasing when simulating extreme hydrological situations. A typical example is a rough resolution in time for the foul flow component. The method described above should give a sufficiently correct description for most cases.

4.2.7 Land uses

Land uses can optionally be used to define loss and infiltration settings varying as a function of the land use distributions on the catchments. This method is only available in combination with the following hydrological models:

- Time-Area (A)
- Kinematic wave (B)
- Linear Reservoir (C1)
- Linear Reservoir (C2).

The 'Land uses' editor holds loss and infiltration properties only. Relevant land uses then need to be selected for each catchment individually, and the percentage of the catchment coverage for each land use also needs to be specified in the 'Catchments' editor.

New land uses can be added using the 'Insert' button, or removed using the 'Delete' button. After inserting a new land use, a description ID should be specified.

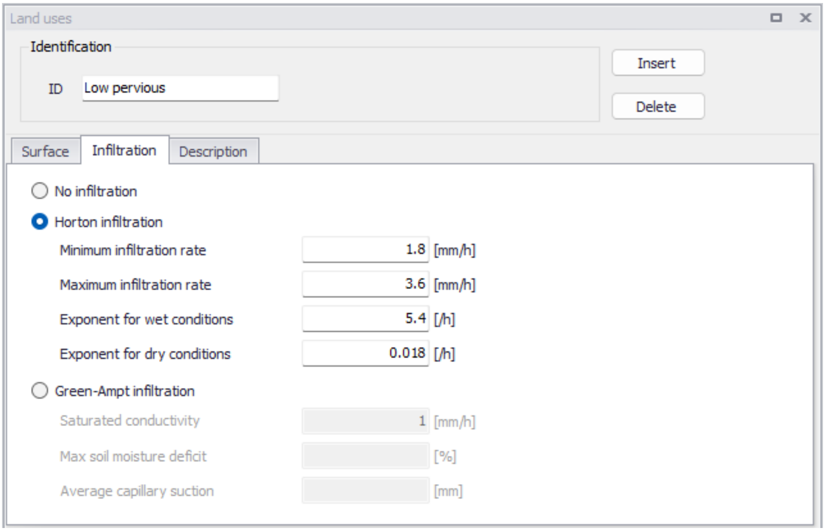


Figure 4.30 The ‘Infiltration’ tab in the ‘Land Uses’ editor

Note: Land uses on the catchment may also be modelled with the RDI hydrological model. In this specific case, the land use is described with a RDI parameter set instead.

Surface

In the 'Surface' tab, two types of initial losses can be applied to the land use:

- Wetting loss: this is a discontinuous loss, which happens as a first loss process. When precipitation starts, a part of the precipitation is used for wetting of the surface if the surface is initially dry. When wetting loss storage is filled, wetting loss process does not happen. It is not possible to infiltrate from wetting loss storage and the storage is recovered only by process of evaporation.
- Storage loss: this is the loss due to filling of 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 roughness of the surface must also be specified with a Manning coefficient, which is a routing parameter. This Manning coefficient is only used in combination with the Kinematic wave (B) hydrological model to calculate the runoff of every individual surface / land use.

Infiltration

In the 'Infiltration' tab, three methods are available to characterize the infiltration for the land use:

- No infiltration
- Horton infiltration: this method is parametrized using:
  - Minimum infiltration rate: infiltration rate at the end of the infiltration process.
  - Maximum infiltration rate: infiltration rate at the start of the infiltration process.
  - Exponent for wet conditions: a time factor that 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.
  - Exponent for dry conditions: a time factor defining the rate of the soil infiltration capacity recovery after a rainfall, i.e., in a drying period.
- Green-Ampt infiltration: this method is parametrized using:
  - Saturated conductivity: hydraulic conductivity of fully saturated soil.
  - Max soil moisture deficit: parameter describing how dry the soil can become.
  - Average capillary suction: parameter describing capillary suction force as a head.

Description

In this tab, a free description text can optionally be provided to describe the land use.

4.3 Catchment Connections

In order to transfer the runoff generated on catchment surfaces into the collection network, the model must include information about the connection of the catchment outlet to the collection network. One or multiple catchments can be connected to one node, and a catchment can be connected to multiple nodes.

In order to use Catchments in the context of network modeling, they have to be connected to the network.

MIKE+ supports the connection of catchments to multiple locations (i.e. nodes or links), as well as separately allocating runoff and catchment discharges to multiple locations.

4.3.1 Catchment Connections Overview

An overview of relevant connections to a pipe network for catchments is available in the Catchments Editor Catchment Connections Overview Tab (Catchments|Catchments). It shows a table summarizing the connections of the catchment to the network model. The data dynamically link and refer to records in the Catchment Connections Editor (Catchments|Catchment Connections).

The summary table shows information on the following:

- Location. To which type of network element the catchment is connected, and the ID of the element.
- Catchment Runoff. Percentage of the Catchment Runoff from the catchment entering a location.
- Catchment Discharge. Percentage of the Catchment Discharge from the catchment entering a location.
- Action. Offers options for editing or adding connections for the active catchment.

  Edit. Opens the Catchment Connections Editor, wherein attributes for the existing catchment connection entry can be modified.

  Add connection. Adds a connection for the active catchment. The new connection is reflected in the overview table and the Catchment Connections Editor.

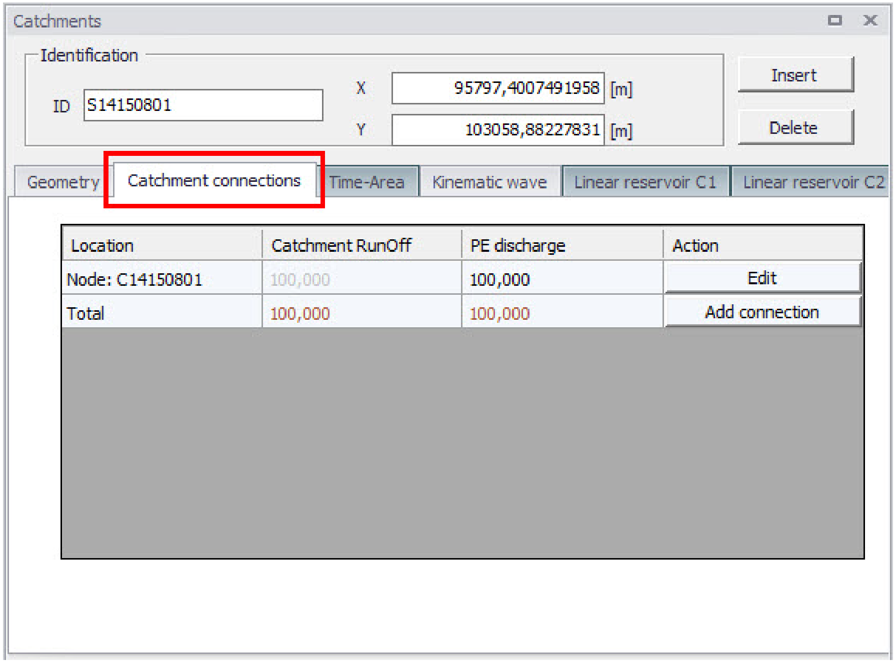


Figure 4.31 The Catchments Editor Catchment Connections Overview Tab

4.3.2 Catchment Connections Editor

Connect catchments to the pipe network via the Catchment Connections Editor (Catchments|Catchment Connections).

The definition and management of catchment connections is supported both through Editors and by a set of graphical catchment connection tools.

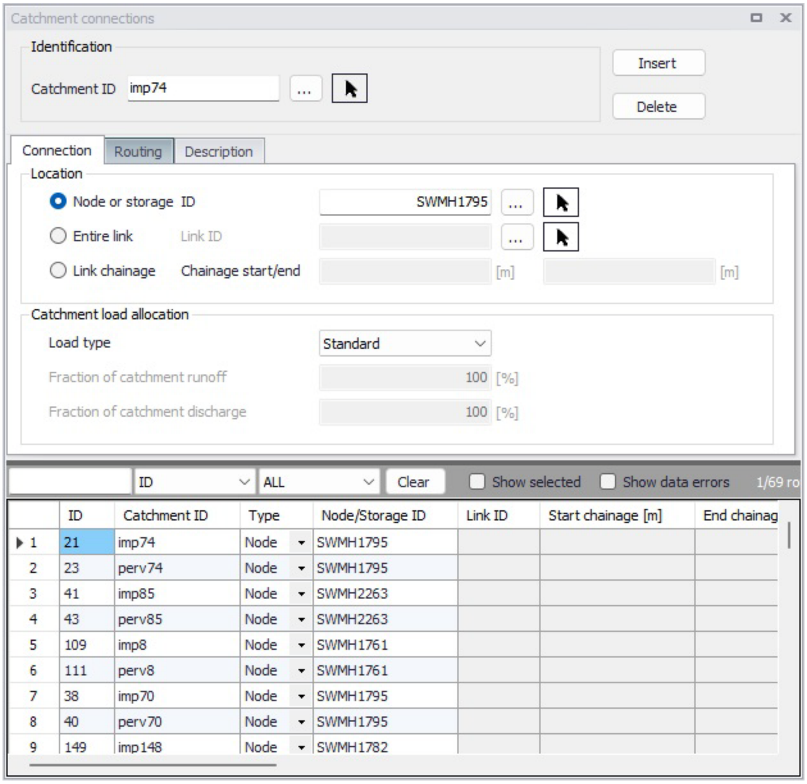


Figure 4.32 The Catchment Connections Editor

Create catchment connections through the ‘Insert’ button. Multiple connections for a single catchment can be set up.

Table 4.22 The Catchment Connections Editor attributes (Table msm_CatchCon)

Edit field	Description	Usage	Attribute Table Field
Catchment ID	Unique catchment identifier	Yes	CatchmentID
Location Type radio buttons	Specifies the type of network element to which the catchment is connected. Options are: Node Entire link, or Link chainage	Yes	TypeNo
Node ID	Unique identifier for the connected node	Yes, If Connection Type = Node	NodeID
Link ID	Unique identifier for the connected link	Yes, If Connection Type = Entire link or Link chainage	LinkID
Chainage start/end	Start and end chainages of the connected link	Yes, If Connection Type = Link chainage	StartChainage/EndCHainage
Load Type dropdown menu	Parameter that defines how the loads from the catchment are allocated to the pipe network for a connection. Options are: Standard, Wastewater Total, Stormwater Total, Combined Partial, Wastewater Partial, and Stormwater Partial. These different Load Types are further explained in the text below.	Yes	LoadTypeNo
Fraction of Catchment Runoff	Fraction of the catchment stormwater runoff to allocate for the connection	Optional, If Load Type = Combined Partial or Stormwater Partial	RRFraction
Fraction of Catchment Discharge	Fraction of the catchment discharge to allocate for the connection	Optional, If Load Type = Combined Partial and Wastewater Partial	PEFraction
Runoff routing method		Yes	RoutingTypeNo

Table 4.22 The Catchment Connections Editor attributes (Table msm_CatchCon)

Edit field	Description	Usage	Attribute Table Field
Delay parameter	Time for the runoff to travel to the network connection location.	Yes if Muskingum routing is chosen	RoutingDelay
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.	Yes if Muskingum routing is chosen	RoutingShape
Description	Optional description of the catchment connection	No	Description

As a catchment can be the source of multiple load types (i.e. stormwater and wastewater), and can be connected to multiple network elements and network types, qualifying a load connection type into clear categories according to pipe network type and connection options is important. These Load Types are described in more detail below:

- Standard: This type of load connection applies to combined systems where all the catchment output is connected to a single location. This is the Default type, which corresponds to the MIKE URBAN Classic Single Node connection type.
- Wastewater Total: This type of load connection applies to fully separated systems, where the catchment is connected to a single location in the wastewater network.
- Stormwater Total: This type of load connection applies to fully separated systems where the catchment is connected to a single location in the stormwater network.
- Combined Partial: This type of load connection applies to combined systems where the catchment is connected to multiple locations in a combined network. This is the fully versatile connection type.
- Wastewater Partial: This type of connection applies to fully separated systems, where the catchment is connected to multiple locations in a wastewater network.
- Stormwater Partial: This type of connection applies to fully separated systems where the catchment is connected to multiple locations in a stormwater network.

The User's choice of Load Type affects the Catchment load allocation Editor fields and the internal data validation.

A facility for data validation checks that for each catchment in the Catchment Connections Editor, the sum of the fractions for Catchment Discharge (i.e.

PEFraction) and Runoff Discharge (i.e.RRFraction) is close to 100 (99.9<sum<100.1).

For catchments where this sum is not found to be close to 100%, all specified connections will be reported as faulty and marked in red.

In the 'Routing' tab, it is possible to activate Muskingum routing along the catchment connection. When no runoff routing is applied, the runoff hydrograph entering the network matches the runoff hydrograph computed on the catchment. If Muskingum routing is applied, a change in hydrograph appearance occurs. This routing is designed to take into account the parts of the network between the catchment and the point of connection on the network, which are not included in the network model. This is especially relevant when the network is trimmed using the 'Network simplification' tool, in which case some parts of the network are removed and the neighboring catchments are reconnected to network locations further away, therefore neglecting parts of the flow path. The runoff routing in catchment connections can be activated automatically for reconnected catchments, while running the 'Network simplification' tool.

When the Muskingum method is selected, the two additional parameters below must also be specified:

- Delay parameter: time for the hydrograph to travel between the catchment and the connection location on the network. 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. If the shape parameter is set to zero, then the Muskingum routing will be very similar to linear reservoir routing.

4.4 Low Impact Development (LID)

Water sensitive urban design (WSUD) represents an approach to land development (or redevelopment) that works with nature to manage stormwater as close to its source as possible. It is also known as Low Impact Development (LID).

Low Impact Development employs principles such as preserving and recreating natural landscape features, minimizing effective imperviousness to create functional and appealing site drainage that treat stormwater as a resource rather than a waste product.

Many practices have been used to adhere to these principles such as bioretention facilities, rain gardens, vegetated rooftops, rainwater tanks, and permeable pavements. By implementing LID principles and practices, stormwater can be managed in a way that reduces the impact of impervious built-up areas and supports the natural movement of water within an urban ecosystem or catchment.

Applied on a broad scale, LID can maintain or restore a catchment's hydrologic and ecological functions. LID has been characterized as a sustainable stormwater practice.

The ability to assess the benefit of LID practices installed in urban catchments is required by Water Utilities and other stakeholders responsible for the urban drainage. An essential part of this requirement is the ability to model the various LID practices at both hydrological screening level and at a detailed hydraulic level in order to evaluate the effect of installing LID practices as parts of a stormwater drainage system.

This chapter describes the implementation and development of model support of various LID structures in MIKE+.

The modelling of LID practices is divided into 2 main approaches:

- Modelling of LID at screening level - catchment-based approach
- Detailed hydraulic modelling of individual LID structures - drainage network based approach.

  This method will provide the User with the option of detailed modelling of individual LID structures hydraulically connected to the stormwater pipe network. This approach is based on the concept of soakaway nodes.

In this Chapter, the MIKE+ implementation of LID at the screening level - the catchment-based approach - is explained.

The catchment-based methods implemented in MIKE+ are dominantly based on research published by US EPA, adapted appropriately to the MIKE+ modelling concept of urban hydrology. In the US, the term Low Impact Development Systems (LIDS) is used for WSUD and the LID concept provides the Users with an overall method for evaluating the various LID controls such as bioretention cells, rain gardens, green roofs, infiltration trenches, permeable pavements, rain barrels/rainwater tanks and vegetative swales.

A user-defined number of LID controls can be deployed and assessed for each individual catchment. This catchment-based approach is used to size the required infiltration or rainwater harvest by subtraction of flow from the calculated runoff within each catchment.

Please note that modelling LID practices in MIKE+ is only allowed for the Kinematic Wave runoff model. Therefore, catchments can either apply the plain Model B or the land use approach, provided that all land uses are based on the Kinematic Wave approach. For any other land use, the simulation engine will generate an error for that catchment.

LIDs are low impact development structures designed to capture or reduce surface runoff from the collecting area by means of a combination of detention, infiltration and evapotranspiration. Low Impact Development controls are conceptual objects that are not displayed on the map visualization of the urban catchment model. Once deployed, they are considered as properties of a given sub-catchment. MIKE+ can model seven types of LIDs:

1. Bioretention Cells
2. Infiltration Trenches
3. Porous Pavement
4. Rain Barrels
5. Vegetative Swales
6. Rain Garden
7. Green Roof

Bioretention cells, infiltration trenches, and porous pavement systems can have optional underdrain systems in their gravel storage beds to convey storage runoff off the site rather than infiltrate it all. They can also have an impermeable floor of liner that prevents infiltration into the native soil. Infiltration trenches and porous pavement systems can also be subject to a decrease in hydraulic conductivity over time due to clogging.

While some LID practices can also provide important pollutant reduction, the current LID implementation in MIKE+ can only be used to model the LIDS' quantitative hydrologic performance.

4.4.1 Bioretention Cells

Bioretention Cells are terrain depressions that comprise of selected types of vegetation, resistant to the extended periods of high moisture and extreme levels of nutrient concentrations (Nitrogen and Phosphorus) found in stormwater runoff, grown in an engineered soil mixture above a gravel drainage bed. They provide storage, infiltration and evaporation of both direct rainfall and runoff captured from the collecting area surrounding the cell. Rain gardens, street planters, and green roofs are all different types of bioretention cells.

The different structural layers of this feature are:

- Surface
- Soil
- Storage
- Drain (underdrain)

These are illustrated in Figure 4.33. There are one or several relevant hydrological processes associated with each layer.

In the surface occur surface storage, surface infiltration, the collecting area run-on, surface evaporation and overflow.

In the soil layer occur percolation (vertical water movement to the storage layer), evapotranspiration (loss due to the plant root action) and storage in the soil's voids.

In the storage layer, stormwater provided by the percolation through the soil layer is detained in the storage layer's voids. Infiltration (leakage) to the native soil through the storage bottom, controlled by the characteristics of the surrounding soil, restores the storage capacity.

An optional underdrain may be included to empty the storage. The underdrain is activated (with the specified capacity) when the water level in the storage reaches the offset level.

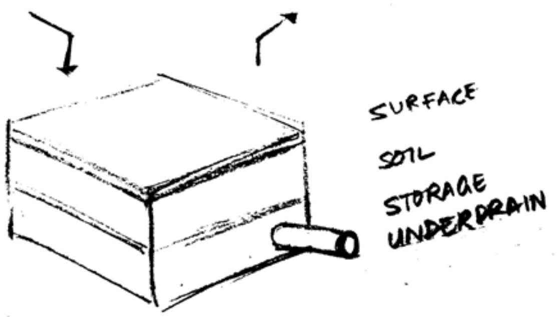


Figure 4.33 Bioretention cell structure layers

4.4.2 Infiltration Trenches

Infiltration Trenches are storage cells filled with gravel that capture runoff from upstream impervious areas. They provide storage capacity and the possibility for captured runoff to infiltrate the soil underneath.

Figure 4.34 illustrates an infiltration trench structure showing the following components:

- Surface
- Storage
- Drain (optional underdrain)

The processes simulated for the infiltration trench are similar as for bioretention cells, except for the missing soil layer; stormwater from the surface enters the storage directly, i.e. without detention in the soil layer.

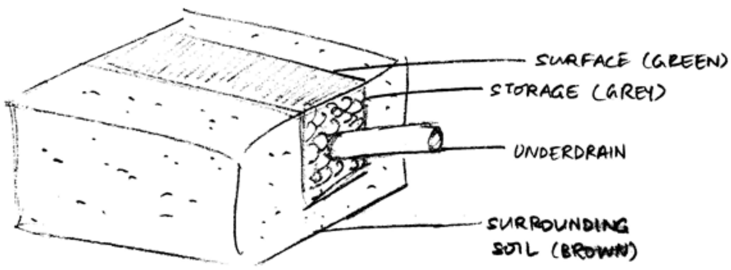


Figure 4.34 Infiltration trench structure layers

4.4.3 Porous Pavement

Porous Pavement systems are excavated areas filled with gravel and paved over with a porous concrete or asphalt mix. Porous pavements are implemented in places where the surface has to provide a firm support for vehicle traffic, such as streets and parking lots.

Normally, all rainfall will immediately pass through the pavement into the gravel storage layer below where it can infiltrate at natural rates into the native soil.

Figure 4.35 shows a porous pavement structure, which has the following components:

- Surface
- Pavement material
- Storage
- Drain (underdrain)

Essentially, hydraulic functionality of the porous pavement is similar to that of the bioretention cell, except that the soil layer (and vegetation) are replaced by some porous asphalt or concrete.

Some types of porous pavements may be subject to clogging by fine sediment particles, which reduces their infiltration capacity.

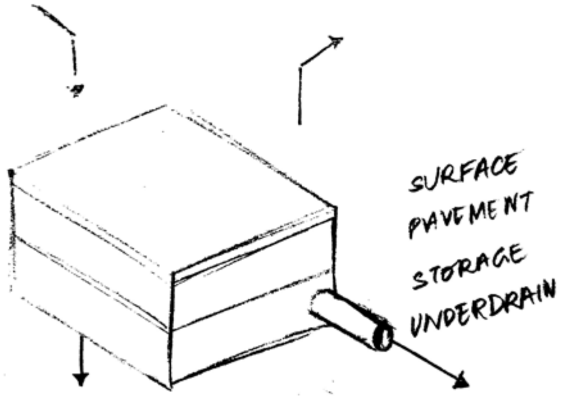


Figure 4.35 Porous Pavement structure layers

4.4.4 Rain Barrels

Rain Barrels are containers that collect roof runoff during storm events and can either release or re-use the rain water during dry periods.

A rain barrel includes:

- Storage (with total porosity, i.e. empty volume)
- Drain (underdrain)

The processes simulated include water detention in the storage, overflow and drainage (i.e. storage recovery). The action of the underdrain is timed; the time offset between the end of rain and the drain activation can be specified. This feature allows for the simulation of rainwater reuse for garden watering, which normally takes place several hours or days after the rain has stopped.

A conceptual sketch of a rain barrel is presented in Figure 4.36.

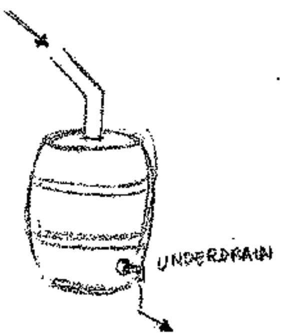


Figure 4.36 Rain Barrel

4.4.5 Vegetative Swales

Vegetative Swales are waterways or depressed areas with sloping sides covered with grass and other vegetation. They slow down the conveyance of collected runoff and allow it more time to infiltrate to inherent soil beneath it. The only relevant layers regarding vegetative swales are:

- Surface (and surrounding soil)

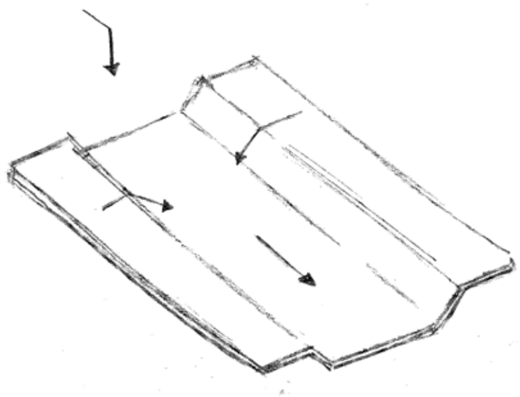


Figure 4.37 Vegetative Swale layer structure

4.4.6 Rain Garden

Rain Garden is a simplified form of bioretention cell, which takes advantage of rainfall and stormwater runoff in its design, simultaneously contributing to the reduction of runoff volume and pollutants released from the site. Typically, it is designed as a small garden with selected types of vegetation resistant to the extended periods of high moisture and extreme levels of nutrient concentrations (Nitrogen and Phosphorus) found in stormwater runoff.

The different layers comprising rain gardens are presented in Figure 4.38. These include:

- Surface
- Soil layers

Surface storage, surface infiltration, the collecting area run-on, surface evaporation, and overflow occur on the surface.

In the soil layer occur storage in the soil's voids, evapotranspiration (loss due to the plant root action) and infiltration (leakage) to the native surrounding soil through the bottom. The infiltration is controlled by the characteristics of the surrounding soil.

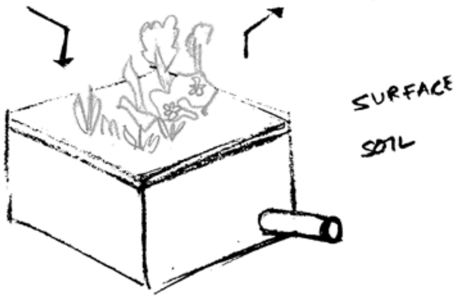


Figure 4.38 Rain Garden process layers

4.4.7 Green Roof

Green Roofs are structural components that reduce the negative effects of urbanization on water quality and rainfall runoff by absorbing or detaining, and filtering runoff. They are built of a planted lightweight soil media and a drainage layer, and have the following components:

- Surface
- Soil
- Drainage Mat

Green Roofs may be designed with intensive or extensive vegetation cover. Intensive green roofs are heavy, with soil media layers larger than 15 cm. Extensive green roofs may include a soil layer as thin as 3 cm, affecting the choice of vegetative cover.

Green roofs are equipped with a drainage layer called a drainage mat placed beneath the soil media. The purpose of the drainage mat is to conduct the surplus water percolated through the soil layer from the roof to the drainage system.

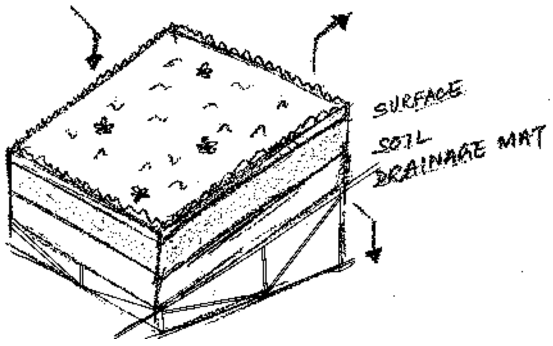


Figure 4.39 Green roof process layers

4.4.8 LID Properties Editor

The MIKE+ LID Properties editor organizes the data input for the different layers and functional elements of LID structures. The input data is organized into the following group and tabs:

- Identification: This group provides unique identification of the specified LID and definition of its type. Each individual LID is generic and is specified per unit area. As such, it can be replicated and placed in any sub-catchment of the study area (i.e. deployed) with different actual sizes and in any number of instances.
- Tabs: Surface, Soil, Pavement, Storage, Drain, and Drainage Mat. These tabs represent different layers and functional elements defining the structure of the LID. The properties of each layer and functional element, which constitute the actual LID type, are entered in these tabs. Only the relevant data fields for a LID type are activated and available for data input.

Identification

Table 4.23 The LID Properties Identification Group (Table msm_LIDcontrol)

Edit field	Description	Used or required by simulations	Field name in data structure
ID	ID of LID	Yes	MUID
Type	Type of LID 1:Bioretention Cell 2:Porous Pavement 3:Infiltration Trench 4:Rain Barrel 5: Vegetative Swale 6:Rain Garden 7:Green Roof	Yes	LIDTypeNo

LID Properties

Figure 4.40 through Figure 4.45 illustrate the six tabs containing the data for various functional elements of LID structures. The corresponding parameter attributes in the msm_LIDcontrol database table are summarised in subsequent tables divided by component following the tabs.

Surface

A Surface component is used for the following LID structures:

- Bioretention Cell

- Porous Pavement
- Infiltration Trench
- Vegetative Swale
- Rain Garden
- Green Roof

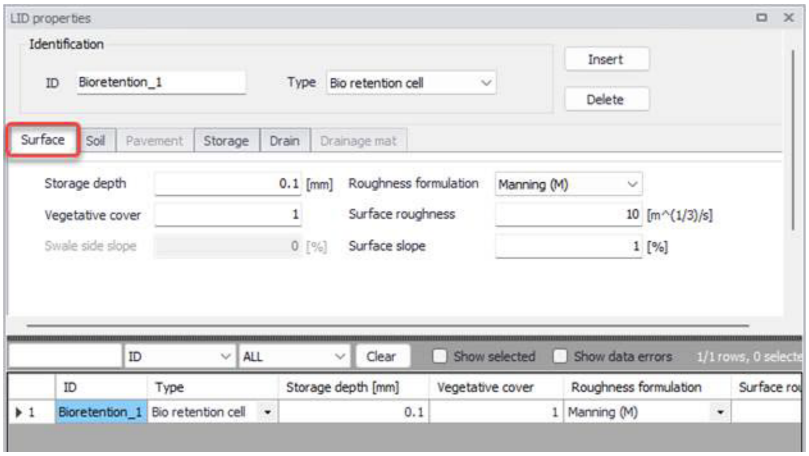


Figure 4.40 The LID Properties Editor Surface Tab

Table 4.24 LID Properties Surface tab attributes (Table msm_LIDcontrol)

Edit field	Description	Used or required by simulations	Field name in datastructure
Storage Depth	When confining walls or berms are present, this is the maximum depth to which water can pond above the surface before overflow occurs. For LIDs that experience ponding it is the height of any surface depression storage. For swales, it is the height of its trapezoidal cross section.	Yes If LIDTypeNo = (1,2,3,5,7)	StorHt
Vegetative Cover	The fraction of the storage area above the surface that is filled with vegetation (0 = no vegetation, 1 = no storage available). NOTE: for infiltration trench, vegetative cover is typically non-existent	Yes If LIDTypeNo = (1,2,3,5,7)	VegFrac
Swale Side Slope	Slope (run over rise) of the sidewalls of a vegetative swale´s cross section. Used for the calculation of the stored volume and wetted width. This value is ignored for other types of LIDs	Yes If LIDTypeNo=5	Xslope
Roughness formulation	Selects whether the roughness value is expressed as Manning (M) or Manning (n)	Yes	FricTypeNo

Table 4.24 LID Properties Surface tab attributes (Table msm_LIDcontrol)

Edit field	Description	Used or required by simulations	Field name in datastructure
Surface Roughness	Manning´s n or M (used in combination with surface slope and width) for routing of overflow from the surface of bioretention cell, rain garden, infiltration trench, porous pavement and for flow routing in vegetative swale (see Table 4.30 for typical values). If specified zero (for all types except vegetative swale), no routing of the overflow is applied	Yes If LIDTypeNo = (1,2,3,5,6,7)	Rough
Surface Slope	Slope (used in combination with surface roughness and width) for routing of overflow from the surface of bioretention cell, rain garden, infiltration trench, porous pavement and for flow routing in vegetative swale. If specified zero (for all types except vegetative swale), no routing of the overflow is applied	Yes If LIDTypeNo = (1,2,3,5,6,7)	Slope
Soil infiltration capacity	The rate of water moving from surface into the soil. It is characteristics of the surface and the uppermost soil layer (equivalent to Horton´s initial infiltration capacity). Infiltration takes place at all times during a rain event and after, as long as there is water available as rainfall, as run on or as water stored in the surface.	Yes If LIDTypeNo = 5	InfiltrationCapacity

Soil

A Soil component is used for the following LID structures:

- Bioretention Cell
- Rain Garden
- Green Roof

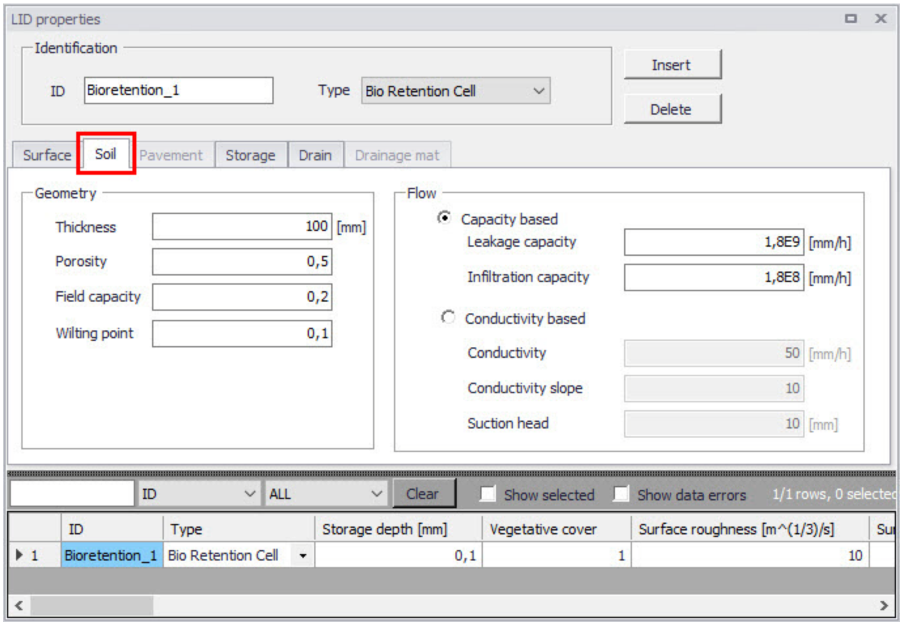


Figure 4.41 The LID Properties Editor Soil Tab

Table 4.25 LID Properties Soil tab attributes (Table msm_LIDcontrol)

Edit field	Description	Used or required by simulations	Field name in datastructure
Thickness	Thickness of the soil layer. Typical values range from 450 to 900 mm for rain gardens, street planters and other types of land-based bioretention units, but only 75 to 150 mm for green roofs	Yes If LIDTypeNo = (1, 5, 6, 7)	SThick
Porosity	Volume of pore space relative to total volume of soil (as a fraction)	Yes If LIDTypeNo = (1, 5, 6, 7)	Por
Field Capacity	Volume of pore water relative to total volume held in the soil after excess water has drained away and the rate of downward movement has decreased (as a fraction). Vertical percolation of water through the soil occurs only when field capacity level is reached or exceeded.	Yes If LIDTypeNo = (1, 5, 6, 7)	FC
Wilting Point	Volume of pore water relative to total volume for a well-dried soil where only bound water remains (as a fraction). The moisture content of the soil cannot fall below this limit and wilting point cannot be higher than the Field Capacity level.	Yes If LIDTypeNo = (1, 5, 6, 7)	WP

Table 4.25 LID Properties Soil tab attributes (Table msm_LIDcontrol)

Edit field	Description	Used or required by simulations	Field name in datastructure
Leakage Capacity	The rate of water leaving the soil layer into storage. It is characteristic of the soil layer. Leakage starts when soil storage exceeds field capacity, continues at all times while soil storage is above the field capacity.	Yes If LIDTypeNo = (1, 5, 6, 7) AND FlowMethod = 1 (Capacity-based)	LeakageCapacity
Infiltration Capacity	The rate of water moving from surface into the soil. It is characteristics of the surface and the uppermost soil layer (equivalent to Horton´s initial infiltration capacity). Infiltration takes place at all times during a rain event and after, as long as there is water available as rainfall, as run on or as water stored in the surface.	Yes If LIDTypeNo = (1, 5, 6, 7)	InfiltrationCapacity
Conductivity	Hydraulic conductivity for the fully saturated soil. This is equivalent to leakage capacity	Yes If LIDTypeNo = (1, 5, 6, 7) AND FlowMethod = 2 (conductivity based))	Ksat

Table 4.25 LID Properties Soil tab attributes (Table msm_LIDcontrol)

Edit field	Description	Used or required by simulations	Field name in datastructure
Conductivity Slope	Slope of the curve of log (conductivity) versus soil moisture content (dimensionless). Typical values range from 5 for sands to 15 for silty clay	Yes If LIDTypeNo = (1, 5, 6, 7) AND FlowMethod = 2 (Conductivity-based)	Kcoeff
Suction Head	The average value of soil capillary suction along the wetting front. This is the same parameter as used in the Green-Ampt infiltration model	Yes If LIDTypeNo = (1, 5, 6, 7) AND FlowMethod = 2 (Conductivity-based)	Suct

Pavement

A pavement component is used for the following LID type:

- Porous Pavement

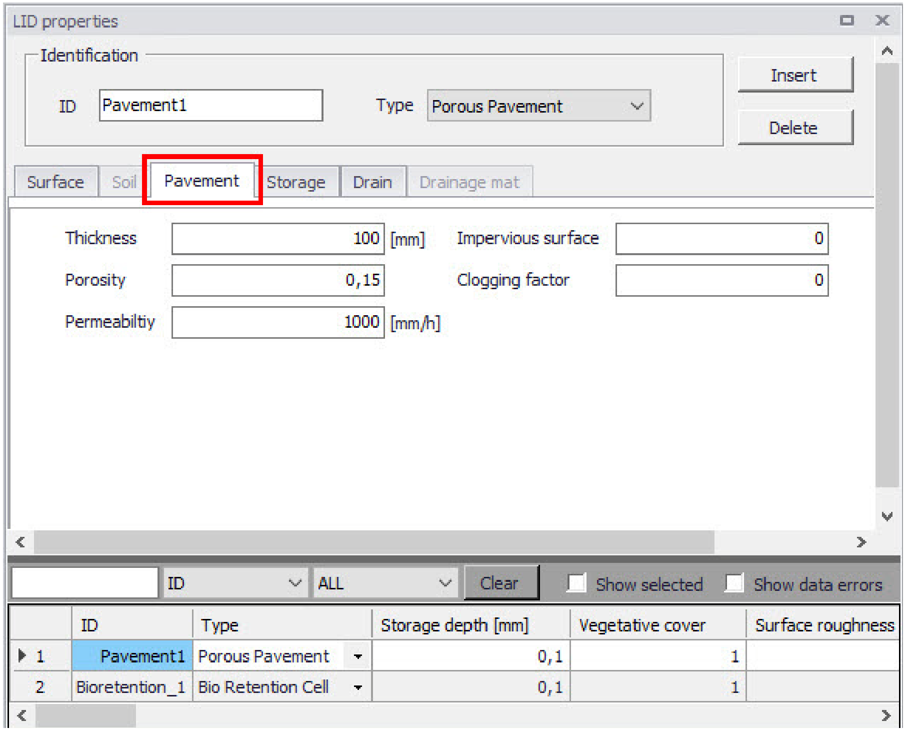


Figure 4.42 The LID Properties Editor Pavement Tab

Table 4.26 LID Properties Pavement tab attributes (Table msm_LIDcontrol)

Edit field	Description	Used or required by simulations	Field name in data-structure
Thickness	Thickness of the pavement. Typical values are 100 to 150 mm	Yes If LIDTypeNo = 2	PThick
Porosity	The ratio (expressed as a fraction) of the volume of the pores or interstices of a material to the total volume of the pavement. Typical values range from 0.11 to 0.17 for pavements Note that porosity = void ratio/(1 + void ratio).	Yes If LIDTypeNo =2	PVPorosity

Table 4.26 LID Properties Pavement tab attributes (Table msm_LIDcontrol)

Edit field	Description	Used or required by simulations	Field name in data-structure
Permeability	Permeability of concrete or asphalt used in continuous systems or hydraulic conductivity of the fill material (gravel or sand) used in modular systems. Permeability of new porous concrete or asphalt is high (>2450 mm/h), but over time the fine particles in the runoff tend to clog the pavement, reducing the permeability of the structure.	Yes If LIDTypeNo = 2	Perm
Impervious Surface	Ratio of impervious paver material to total area for modular systems; 0 for continuous porous pavement systems	Yes If LIDTypeNo = 2	FracImp
Clogging Factor	Voids that are clogged due to fine particles accumulation, as a fraction of total voids area. Use a value of 0 to ignore clogging. Max. value = 1.	Yes If LIDTypeNo = 2	PVClog

Storage

A Storage component is used for the following LID structures:

- Bioretention Cell
- Porous Pavement
- Infiltration Trench
- Rain Barrel

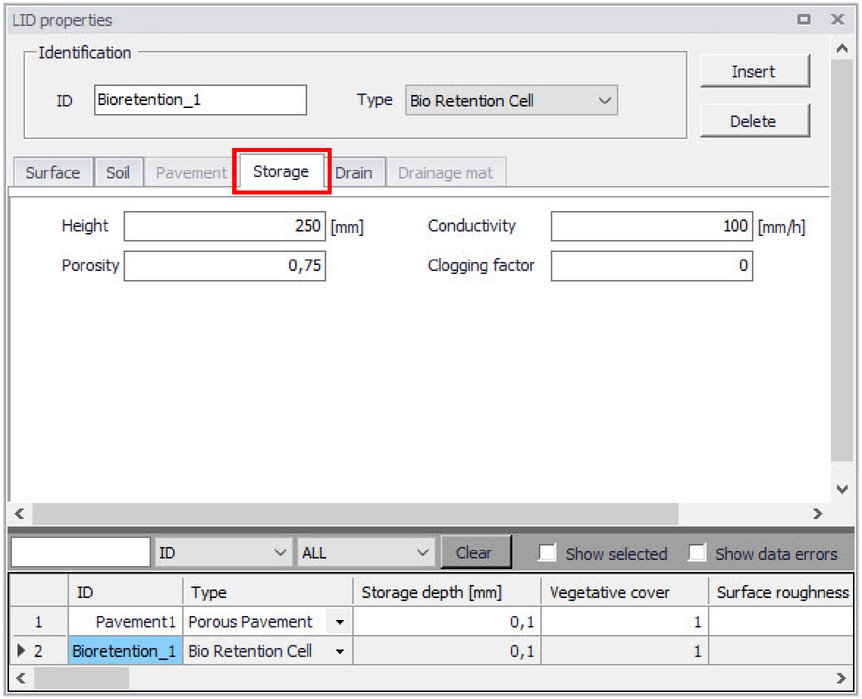


Figure 4.43 The LID Properties Editor Storage Tab

Table 4.27 The LID Properties Storage tab attributes (Table msm_LIDcontrol)

Edit field	Description	Used or required by simulations	Field name in datastructure
Height	The height of the storage layer in case of bioretention cell, porous pavement and infiltration trench; the height of a rain barrel (mm or inches). Crushed stone and gravel layers are usually 150 to 450 mm thick while rain barrels vary in height from 600 mm upwards.	Yes If LIDTypeNo = (1,2,3,4)	Height
Porosity	The ratio of the volume of the pores or interstices of a material to the total volume of the layer. Typical values range from 0.30 to 0.45 for gravel beds. Note that porosity = void ratio/(1 +void ratio).	Yes If LIDTypeNo = (1,2,3)	SVPorosity
Conductivity	The maximum rate at which water infiltrates to the surrounding soil through the bottom of the freshly constructed storage layer.	Yes If LIDTypeNo = (1, 2, 3)	Filt
Clogging Factor	Volume of voids that are clogged due to fine particles accumulation. Use a value of 0 to ignore clogging.	Yes If LIDTypeNo = (1,2,3)	SVclog

Drain

A Drain component is used for the following LID structures:

- Bioretention Cell
- Porous Pavement
- Infiltration Trench
- Rain Barrel

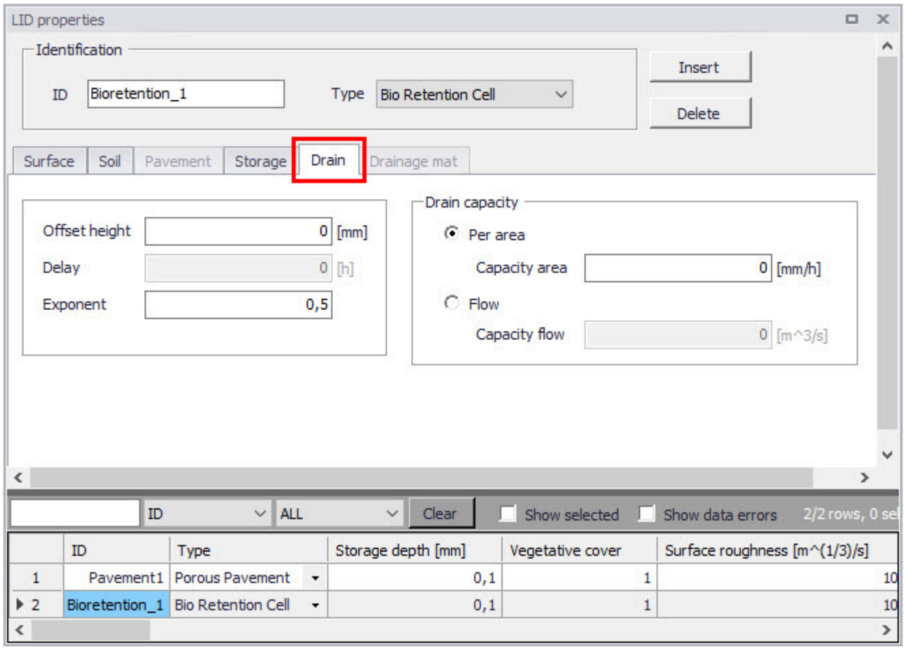


Figure 4.44 The LID Properties Editor Drain Tab

Table 4.28 The LID Properties Drain tab attributes (Table msm_LIDcontrol)

Edit field	Description	Used or required by simulations	Field name in datastructure
Offset Height	Height of any underdrain piping above the bottom of a storage layer or rain barrel	Yes If LIDTypeNo = (1,2,3,4)	HOffset
Delay	The number of dry weather hours that must elapse before the drain in a rain barrel is activated	Yes If LIDTypeNo = 4	Delay

Table 4.28 The LID Properties Drain tab attributes (Table msm_LIDcontrol)

Edit field	Description	Used or required by simulations	Field name in datastructure
Exponent	Exponent n that determines the rate of flow through the underdrain as a function of height of stored water above the drain height.	Yes If LIDTypeNo = (1,2,3,4)	Expon
Capacity Area	Coefficient that determines the rate of flow through the underdrain as a function of height of stored water above the drain bottom. Expressed in terms of flow capacity per area.	Yes If LIDTypeNo = (1,2,3,4) AND Drain Capacity = Per Area	DrainCapacityArea
Capacity Flow	Coefficient that determines the rate of flow through the underdrain as a function of height of stored water above the drain bottom. Expressed in terms of flow capacity.	Yes If LIDTypeNo = (1,2,3,4) AND Drain Capacity = Flow	DrainCapacityFlow

Drainage Mat

A Drainage Mat component is used for the following LID structure:

- Green Roof

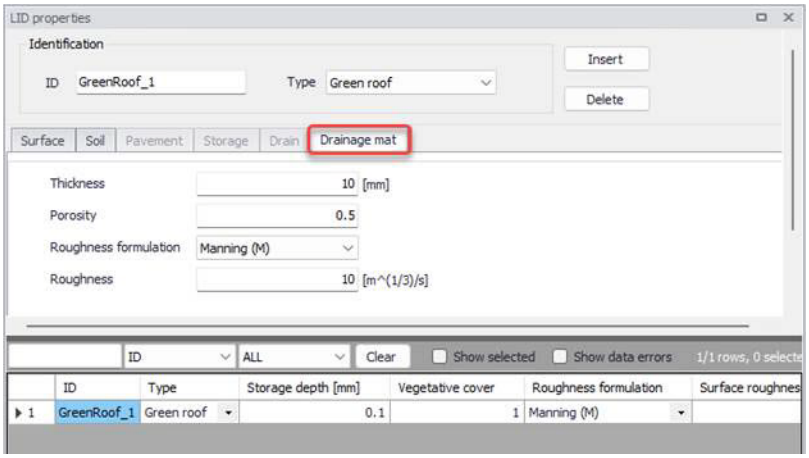


Figure 4.45 The LID Properties Editor Drainage Mat Tab

Table 4.29 The LID Properties Drainage Mat tab attributes (Table msm_LIDcontrol)

Edit field	Description	Used or required by simulations	Field name in datastructure
Thickness	The thickness of the mat or plate. It typically ranges between 25 to 50 mm	Yes If LIDTypeNo = 7	DMThick
Porosity	The ratio of void volume to total volume in the mat. It typically ranges from 0.5 to 0.6	Yes If LIDTypeNo = 7	DMVPorosity

Table 4.29 The LID Properties Drainage Mat tab attributes (Table msm_LIDcontrol)

Edit field	Description	Used or required by simulations	Field name in datastructure
Roughness formulation	Selects whether the roughness value is expressed as Manning (M) or Manning (n)	Yes	DMFricTypeNo
Roughness	Manning´s number, used to compute the horizontal flow rate of drained water through the mat. In absence of standard product specifications provided by manufacturers, the roughness must be estimated. Use of n values from 0.1 to 0.4 (M = 2.5 - 10) is suggested.	Yes If LIDTypeNo = 7	DMRough

Table 4.30 shows examples of Manning M values for different types of surfaces. Table 4.31 shows hydraulic conductivity properties for various porous media.

Table 4.30 Manning´s M of surface for porous pavement or vegetative swale [a]

Surface type	Manning M
Smooth asphalt	91
Smooth concrete	83
Ordinary concrete lining	77
Good wood	71
Brick with cement mortar	71
Vitrified clay	67
Cast Iron	67
Corrugated metal pipes	42
Cement rubble surface	42
Fallow soils (no residue)	20
Cultivated soils	50 to 20
Residue cover < 20%	17

Table 4.30 Manning´s M of surface for porous pavement or vegetative swale [a]

Surface type	Manning M
Residue cover > 20%	6
Range (natural)	8
Short, prairie	7
Dense	4
Bermuda grass	2
Woods	10
Light underbrush	2,5
Dense underbrush	1,25

a.Source: McCuen, R. et al. (1996), Hydrology, FHWA-SA-96-067, Federal Highway Administration, Washington, DC

Table 4.31 Hydraulic conductivity and porosity of unconsolidated porous media [a]

Material	Hydraulic conductivity *K* (cm/s)	Porosity η (%)
Gravel	10^{-1} - 10^{2}	25 - 40
Sand	10^{-5} - 1	25 - 40
Silt	10^{-7} – 10^{-3}	35 - 50
Clay	10^{-9} – 10^{-5}	40 - 70

a. Source: Freeze, R.A., and Cherry, J.A., (1979), Groundwater, Prentice-Hall, Englewood Cliffs, NJ

4.4.9 LID Deployment

The LIDs are assigned to a catchment by means of the LID Deployment Editor. In this Editor, it is possible to specify the size (i.e. area) of the individual LID structure, the size of the catchment´s impervious area that is funnelled into it and the number of the current LID units.

The size properties can be specified either directly as surface area, or as a percentage of the catchment's area. The specified total collecting area must not exceed the size of the catchment's impervious area.

Furthermore, level of initial saturation and routing width for overland flow can be specified.

The input data is organized into the following groups:

- Identification
- Deployment Parameters

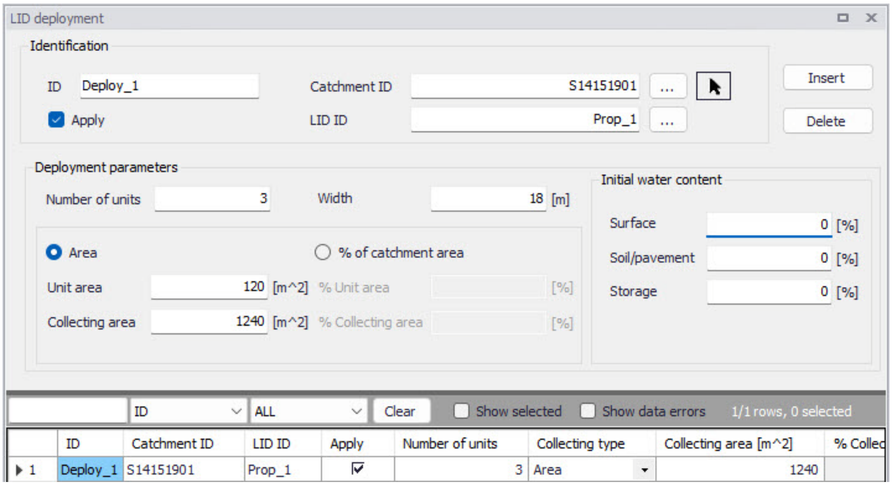


Figure 4.46 The LID Deployment Editor

Identification

Each deployed LID is linked to a LID Deployment ID. I.e. each deployment represents a LID deployed on a single catchment in a specified number of units. For each LID deployment, a result file can be generated containing time series of relevant variables (fluxes and storages). This result file is in the DFS0 time series format.

A full reference of the attributes related to LID deployment in shown in Table 4.32 and Table 4.33.

Table 4.32 The LID Deployment Identification Group (Table msm_LIDusage)

Edit Field	Description	Used or required by simulations	Field name in data structure
ID	Unique deployment ID	Yes	MUID
Catchment ID	Identification of the associated catchment	Yes	CatchID
LID ID	Identification of the LID structure to be deployed	Yes	LidID
Apply checkbox	Option for activating/deactivating deployment of a LID structure	Yes	IncludeNo

Including or excluding a LID is handled by the Active checkbox. This checkbox is checked by Default.

Deployment Parameters

Table 4.33 The LID Deployment Properties (Table msm_LIDusage)

Edit Field	Description	Used or required by simulations	Field name in datastructure
Number of Units	Number of replicate LID units deployed within the catchment	Yes	ReplicateNumber
Width	The width of the outflow face of each identical LID unit (meter or feet). This parameter is applied for all control units that use overland flow to transport surface runoff off the unit, being these roofs, pavement, trenches, and swales for the other control types this parameter can be set as 0	Yes (for porous pavement, swales, roofs and trenches)	Width
Area/% of Catchment Area options	Toggle for LID collecting area setting	Yes	CollectingNo
Unit Area	The surface area of each replicate LID unit	Yes If CollectingNo=Area	UnitArea
Collecting Area	A LID practice is connected to a tributary area whose runoff is treated by the unit. This area includes the LID practice area itself.	Yes If CollectingNo=Area	CollectingArea
% Unit Area	The surface area of each replicate LID unit, expressed as a percentage of the total catchment area	Yes If CollectingNo=% of Catchment Area	UnitAreaPercent
% Collecting Area	The tributary area connected to the LID unit presented as a percentage of the catchment	Yes If CollectingNo=% of Catchment Area	CollectingAreaPercent
Surface	Initial water content of the surface layer	Yes	InitSatSurface

Table 4.33 The LID Deployment Properties (Table msm_LIDusage)

Edit Field	Description	Used or required by simulations	Field name in datastructure
Soil/Pavement	The degree to which the unit´s soil is initially filled with water (0% saturation corresponds to the wilting point moisture.	Yes	InitSatSoil
Storage	Initial water content of the storage layer.	Yes	InitSatStorage

LID Deployment Result File

Optionally, for each LID structure deployment, a DFS0 time series file can be created. This file includes time series of relevant variables inside the LID structure in terms of inflow, flow between layers, storage levels in various layers and output from the structure to the native soil.

If the User does not activate this option, the only visible outputs from the runoff simulation including LIDs are the changed (reduced) runoff hydrographs, caused by infiltration loss and storage in the LID structure, and the LID summary table in the simulation summary file.

The contents of the DFS0 file depends on the actual LID type. In Figure 4.47, an example of the DFS0 file for porous pavement is presented. Thirteen columns contain time series for the processes occurring inside the porous pavement.

NOTE: The flow inside the LID structure and the drain flow are reported as intensities based on the LID area. In cases where the collecting area is bigger than the LID area, the reported intensities will not be comparable with rainfall and evapotranspiration intensities, which are given as the model boundary conditions. In order to make the comparison possible, the reported flow intensities must be scaled down by the ratio between the LID area and the collecting area.

	1	2	3	4	5	6	7	8	9	10	11	12	13
	Time	Inflow [mm/h]	Rain [mm/h]	Evaporation [mm/h]	Infiltration [mm/h]	SurfaceFlow [mm/h]	DrainFlow [mm/h]	SurfaceDepth [m]	PavementMoisture [()]	StorageDepth [m]	SurfaceToPavement [mm/h]	PavementToStorage [mm/h]	MassChecksum [m^3]
1	07-08-1994 16:36:00.000	0	1,2	0	0	0	0	0	0	4,444444e-005	1,2	1,2	1,734723e-018
2	07-08-1994 16:37:00.000	0	1,2	0	1,2	0	0	0	0	4,444444e-005	1,2	1,2	5,20417e-018
3	07-08-1994 16:38:00.000	0	1,2	0	1,2	0	0	0	0	4,444444e-005	1,2	1,2	5,20417e-018
4	07-08-1994 16:39:00.000	0	1,2	0	1,2	0	0	0	0	4,444444e-005	1,2	1,2	5,20417e-018
5	07-08-1994 16:40:00.000	0	1,2	0	1,2	0	0	0	0	4,444444e-005	1,2	1,2	5,20417e-018
6	07-08-1994 16:41:00.000	0	4,8	0	1,2	0	0	0	0	0,0001777778	4,8	4,8	1,214306e-017
7	07-08-1994 16:42:00.000	0	4,8	0	4,8	0	0	0	0	0,0001777778	4,8	4,8	1,908196e-017
8	07-08-1994 16:43:00.000	0	4,8	0	4,8	0	0	0	0	0,0001777778	4,8	4,8	1,908196e-017
9	07-08-1994 16:44:00.000	0	4,8	0	4,8	0	0	0	0	0,0001777778	4,8	4,8	1,908196e-017
10	07-08-1994 16:45:00.000	0	4,8	0	4,8	0	0	0	0	0,0001777778	4,8	4,8	1,908196e-017
11	07-08-1994 16:46:00.000	0	4,8	0	4,8	0	0	0	0	0,0001777778	4,8	4,8	1,908196e-017
12	07-08-1994 16:47:00.000	0,2571431	4,8	0	4,8	0	0	0	0	0,0001873016	5,057143	5,057143	2,602085e-017
13	07-08-1994 16:48:00.000	2,057143	4,8	0	5,057143	0	0	0	0	0,0002539683	6,857143	6,857143	3,989864e-017
14	07-08-1994 16:49:00.000	2,057143	4,8	0	6,857143	0	0	0	0	0,0002539683	6,857143	6,857143	3,989864e-017
15	07-08-1994 16:50:00.000	2,057143	4,8	0	6,857143	0	0	0	0	0,0002539683	6,857143	6,857143	6,765422e-017
16	07-08-1994 16:51:00.000	0,0857143	0,2	0	6,857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,93636e-017
17	07-08-1994 16:52:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
18	07-08-1994 16:53:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
19	07-08-1994 16:54:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
20	07-08-1994 16:55:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
21	07-08-1994 16:56:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
22	07-08-1994 16:57:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
23	07-08-1994 16:58:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
24	07-08-1994 16:59:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
25	07-08-1994 17:00:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
26	07-08-1994 17:01:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
27	07-08-1994 17:02:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
28	07-08-1994 17:03:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
29	07-08-1994 17:04:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
30	07-08-1994 17:05:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
31	07-08-1994 17:06:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
32	07-08-1994 17:07:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
33	07-08-1994 17:08:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
34	07-08-1994 17:09:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
35	07-08-1994 17:10:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
36	07-08-1994 17:11:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
37	07-08-1994 17:12:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
38	07-08-1994 17:13:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
39	07-08-1994 17:14:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
40	07-08-1994 17:15:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
41	07-08-1994 17:16:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
42	07-08-1994 17:17:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
43	07-08-1994 17:18:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
44	07-08-1994 17:19:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
45	07-08-1994 17:20:00.000	0,08571429	0,2	0	0,2857143	0	0	0	0	1,058201e-005	0,2857143	0,2857143	7,979728e-017
46	07-08-1994 17:21:00.000	17,48571	40,8	0	0,2857143	0	0	0	0	0,00215873	58,28571	58,28571	3,018419e-016

Figure 4.47 Example of a result report DFS0 file per deployment (a porous pavement control)

1. **Time:** calendar time for the simulated time steps.
2. **Inflow (mm/h):** inflow to the LID unit given as a multiple of the rain intensity and the collecting area outside LID/LID area. It is the run-on from the collecting area outside the LID and rain on top of the LID unit. The run on represents the net rain on the impervious collecting area, i.e. with initial losses subtracted.
3. **Rain (mm/h):** input rain for the catchment containing the LID unit. This rain loads the LID area directly.
4. **Evaporation (mm/h):** this is a given parameter specified by the User by means of boundary conditions valid for the catchment as well as for the LID unit.
5. **Infiltration (mm/h):** infiltration from the LID unit to the surrounding native soil.
6. **Surface flow (mm/h):** surface water flow. Represents overland flow for vegetative swale; overflow for other LID controls
7. **Drain flow (mm/h):** water flow through the drain.
8. **Surface depth (m):** accumulated water on the surface expressed as the water height on the surface. Maximum value is the specified surface storage height.
9. **Soil moisture ():** presented as a fraction, relative to the total volume of the soil layer. Corresponds to the water that is held in the spaces between soil particles. Maximum soil moisture (i.e. full saturation) is equivalent to the specified soil porosity.

10. **Pavement moisture():** presented as a fraction, relative to the total volume of the porous pavement layer. Corresponds to the water that is held in the pavement's voids. Maximum pavement moisture (i.e. full saturation) is equivalent to the specified pavement porosity.
11. **Storage depth (m):** accumulated water in the storage layer expressed as the water height in the storage. Maximum value is the storage height.
12. **Surface to soil (mm/h):** infiltration from the surface layer to the soil layer expressed in mm/h.
13. **Surface to pavement (mm/h):** infiltration from the surface layer to the pavement layer expressed in mm/h.
14. **Soil to storage (mm/h):**): flow from the soil layer to the storage layer expressed in mm/h.
15. **Pavement to storage (mm/h):** flow from the pavement layer to the storage layer mm/h.
16. **MassChecksum (m³):** regards to the error check in the mass water balance per time step computed, a low value in the order of 1*E-015 indicates an accurate internal flow estimation.

LID Simulation Summary

The LID simulation summary is provided as part of the overall simulation summary HTML file. The summary table reports the volume balance for each LID deployment (see example in Figure 4.48).

ID	Inflow/runon [m^3]	Rain [m^3]	Evaporation [m^3]	Infiltration [m^3]	Surface runoff [m^3]	Drain Outflow [m^3]	Initial Storage [m^3]	Final Storage [m^3]
Green_roof3	0,000	19,83	0,000	0,000	0,000	0,000	0,000	19,83
Vegetative_swale1	17,26	1,983	0,000	19,24	0,003	0,000	0,000	0,000
Vegetative_swale2	17,26	1,983	0,000	19,24	0,004	0,000	0,000	0,000
Vegetative_swale3	17,26	1,983	0,000	19,24	0,001	0,000	0,000	0,000
Bio_retention1	17,26	1,983	0,000	2,956	1,585	10,20	0,000	4,500
Bio_retention2	17,26	1,983	0,000	10,66	0,675	0,000	0,000	7,911
Bio_retention3	17,26	1,983	0,000	11,00	0,000	0,000	0,000	8,242
Bio_retention4	17,26	1,983	0,000	10,82	0,000	0,000	0,000	8,421
Bio_retention5	17,26	1,983	0,000	10,82	0,000	0,000	0,000	8,421
Rain_garden1	17,26	1,983	0,000	11,74	0,000	0,000	0,000	7,500
Rain_garden2	17,26	1,983	0,000	11,74	0,000	0,000	0,000	7,500
Rain_garden3	17,26	1,983	0,000	11,74	0,000	0,000	0,000	7,500
Rain_garden4	17,26	1,983	0,000	11,74	0,000	0,000	0,000	7,500
Porous_pavement1	0,000	19,83	0,000	19,83	0,000	0,000	0,000	0,000
Porous_pavement2	0,000	19,83	0,000	19,83	0,000	0,000	0,000	0,000
Popous_pavement4	0,000	19,83	0,000	19,83	0,000	0,000	0,000	0,000
Porous_pavement3	0,000	19,83	0,000	19,83	0,000	0,000	0,000	0,000
Rain_barrel2	19,18	0,000	0,000	0,000	9,177	9,000	0,000	1,000
Rain_barrel1	19,18	0,000	0,000	0,000	9,177	0,608	0,000	9,392
Infiltration_trench1	0,000	19,83	0,000	19,83	0,000	0,000	0,000	0,000
Rain_barrel3	19,18	0,000	0,000	0,000	9,177	0,608	0,000	9,392
Infiltration_trench3	0,000	19,83	0,000	19,83	0,000	0,000	0,000	0,000
Infiltration_trench2	0,000	19,83	0,000	19,83	0,000	0,000	0,000	0,000
Green_roof1	0,000	19,83	0,000	0,000	0,000	0,000	0,000	19,83
Green_roof2	0,000	19,83	0,000	0,000	0,000	0,000	0,000	19,83
Total	**264,6**	**222,1**	**0,000**	**209,7**	**29,80**	**20,42**	**0,000**	**146,8**

Figure 4.48 Example of a LID simulation summary

5 Modelling Stormwater Quality (SWQ)

5.1 Introduction

SWQ - A model associated with urban catchments and rainfall-runoff modelling

The importance of wide-ranging and yet efficient modelling of stormwater quality on urban catchments and the transport of polluted stormwater in the urban drainage networks is growing with increased focus on local handling of stormwater, drainage network separation, stormwater treatment prior to release to recipients, as well as untreated stormwater overflows.

Stormwater pollution associated with urban catchment surfaces includes dissolved matter and suspended particles originating from:

- Soil erosion
- Erosion of construction materials (roofs, roads)
- Air borne pollution (e.g. industrial emissions particles)
- Local biological pollution related to humans and wildlife (various waste, birds droppings)
- Traffic debris
- Etc.

MIKE+ SWQ simulates stormwater quality as a special model associated with urban catchments and stormwater runoff and infiltration. This is illustrated in Figure 5.1.

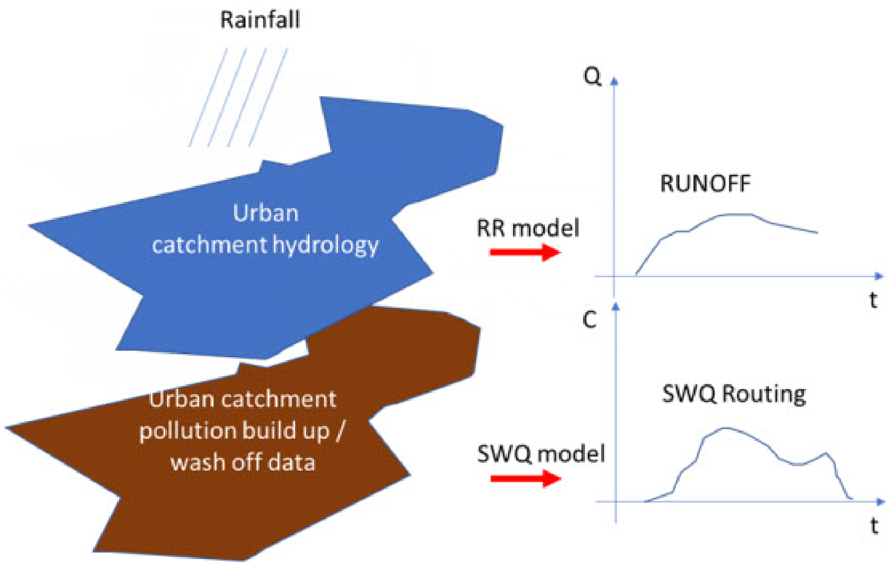


Figure 5.1 Rainfall-runoff and SWQ models act simultaneously on an urban catchment. Both models are driven by the rainfall and by the model parameters controlling the quantities and dynamics of runoff and surface pollution, respectively, being routed towards the catchment outlet.

Spatial and temporal variation of pollution in urban catchments

Model catchments vs. spatial distribution of surface pollution

The amount of surface pollution on urban catchments varies both in space and time, governed by types of pollutants, types of catchment surface and its uses. This variation needs to be captured as correctly as possible in the simulation model in order to achieve realistic model results.

The urban drainage model area is typically delineated into a number of sub-catchments, with delineation determined by the drainage network layout, topography or even administrative limits (parcels, urban districts, etc.).

Sometimes, the sub-catchment delineation is done by a simple division (e.g. Thiessen polygons), i.e. is based on purely geometrical reasoning. Only in rare cases, such delineations are appropriate for efficient characterisation of surface pollution loads generated on such sub-catchments.

Therefore, model sub-catchments would typically include different surfaces, such as building roofs made of different material, roads with various traffic intensity and with different paving, green areas with different vegetation, etc.).

Each of these surfaces may have different usage as well. This means that a single model catchment may be a source of various pollutants, differing by:

- Pollutant type:
  - Organic pollution
  - Chemicals
  - Microplastic
  - Heavy metals
  - Etc.
- Pollutant origin and build-up/wash-off mechanism:
  - Airborne (wind, smog)
  - Area use (traffic, people, animals, local industry)
  - Erosion of surface materials (metal roofs, asbestos, PAH…)
  - Soil erosion
- Attachment to surface sediments:
  - Particulate suspended, attached to surface sediments
  - Particulate suspended, on its own
  - Dissolved

These differences get manifested in various pollutants' quantities and their different behaviour on the catchment surface in dry weather (build-up) and when exposed to rainfall (wash-off). The following simple examples illustrates a typical situation.

A catchment has a total area of 2.1 ha. From the point of view of surface pollution, three types of surfaces may be distinguished: building roofs (15% or 0.315 ha), roads (10% or 0.21 ha) and the remaining 75% (or 1.575 ha) is green area. Each of these surfaces may be a source of various pollutants by itself and is exposed to build-up of exogenous pollution by different intensity and dynamics, depending on the surface exposure to external impacts. Further, these pollutants may behave differently in terms of wash-off by action of rainfall.

A MIKE+ model may contain many - often thousands - of such catchments.

Decomposing catchments to pollution-related layers

In order to capture such situation correctly and efficiently in the model, the catchment (i.e. each sub-catchment in the model) is decomposed to a number of stacked, geographically identical catchments. In the described case, the catchment is represented by three layers, each one representing one of the three surface types. This is illustrated in the following figure:

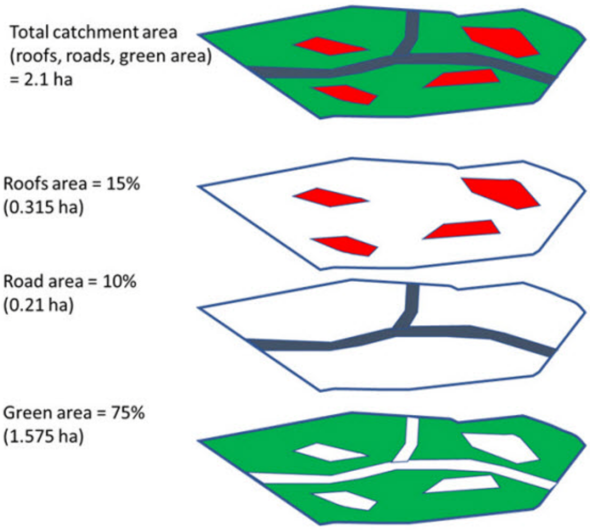


Figure 5.2 A catchment decomposed to three layers, each representing one type of catchment surface.

The upper-most polygon represents the actual catchment. In the model, this catchment is represented by the three polygons below, representing roofs, roads and green areas, respectively.

These three model catchments may inherit the hydrological parameters and connections to the network, but these can also be modified to facilitate a better description of behaviour of the various surface types in the actual catchment.

In any case, the layered catchments will differ by the actual drainage area, which corresponds to the actual area of respective type of surface in the given physical catchment. The sum of the contributing areas for the three sub-catchments must equal to the actual contributing area for the catchment.

Modelling SWQ from multiple surface types as special type of boundary conditions

Surface pollution loads are defined as special types of boundary conditions to the catchments model (SWQ boundary conditions), similarly as rainfall is specified as a boundary for a precipitation-runoff model. Specifying surface pollution loads as boundary conditions, independent of rainfall boundary conditions, is necessary because spatial distributions of rainfall and of surface pollution are fully independent. Treating rainfall and surface pollution through independent boundary conditions implies that the rainfall and surface pollution can each be modelled with its own geographical distribution.

Since every SWQ boundary condition is associated with one definition of geographical location (i.e. reference to one or more catchments: All, List or Individual), and with any number of WQ boundary properties - one for any given pollutant component, each type of catchment surface must be described by a separate SWQ boundary condition. I.e. as many surface types user wants to simulate, as many stacked catchment layers and SWQ boundary conditions shall be specified.

Each of these boundary conditions shall be connected to a sub-set of catchments representing certain type of surface and would include as many WQ boundary properties as there are pollutant components associated with that type of surface - each with its own quantities (i.e. temporal variation) method.

5.2 SWQ Model Setup

SWQ model setup includes the following components:

- Urban catchments with hydrological parameters for the applied precipitation-runoff model
- Rainfall boundary condition (any type of uniform or spatially distributed rain (dfs0 or dfs2)
- WQ component(s) and/or ST fraction(s)
- SWQ Global data
- SWQ local parameters (applies for Build-up/wash-off and EMC methods only)
- SWQ boundary conditions (Surface and/or RDI): multiple boundary conditions are specified in case when spatial differentiation of SWQ loads is analyzed, e.g. case of different catchment surface types or land uses
- SWQ Boundary properties: for each SWQ boundary condition properties have to be defined for each pollutant component and/or sediment fractions.
- SWQ Output definition
- Simulation setup

In the following sections, SWQ-specific model setup is presented in full detail.

Finally, the workflow sequence is briefly described in Section 5.7.

General issues (urban catchments, rainfall boundary conditions, WQ components, ST fractions, Output definition and Simulation setup) are described elsewhere in the MIKE+ Collection System User Guide.

5.3 SWQ Global Data

Model parameters included in SWQ Global data (see Figure 5.3) apply for entire model and for SWQ advanced methods only.

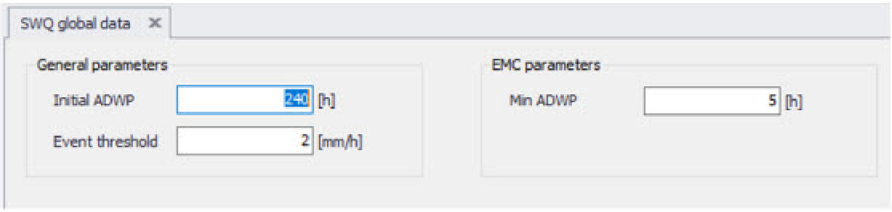


Figure 5.3 SWQ Global Data Editor

An overview of SWQ Global Data editor attributes is provided in Table 5.1.

Table 5.1 Overview of the SWQ Global Data editor attributes (Table msm_SWQGlobalData)

Edit Field	Description	Usage	Attribute Table Field
Initial ADWP	Duration of antecedent dry weather period before the start of the simulation [hours]	Build-up /Wash-off and EMC methods	ADWPIni
Event threshold	Peak rainfall intensity in a rainfall event, used as a minimum threshold to include the rainfall event in the SQW computation	EMC method	EventThreshold
Min ADWP	Any two rain intervals separated by a dry interval are considered as the continuous rain event if the dry interval between the two wet-weather periods is less than the specified ADWPmin	EMC method	ADWPMin

5.4 SWQ Methods

Similarly as hydrological processes and runoff from urban catchments can be simulated by various runoff models, SWQ supports several methods for simulating processes related to stormwater quality. The available methods are:

1. Simple concentration
2. Table concentration
3. Build-up/Wash-off
4. Event mean concentration (EMC)

Method 1 (Simple concentration) applies a pollutant concentration directly to the runoff from a catchment. The applied concentration can be specified as constant or based on an arbitrary time series.

Method 2 (Table concentration) applies a tabulated concentration of pollutant specified as a tabulated function, based on the runoff intensity (i.e. specific runoff).

Note that Method 1 and Method 2 do not consider availability of pollutants. In other words, these methods assume unlimited supply of pollutants on the catchment surface. As such, they should be applied with care and only in appropriate cases.

Method 3. (Build-up/Wash-off) simulates the build-up of pollutants or sediments on the catchment surface and wash-off of this pollution by mechanical action of rainfall events. This method is mass-conservative, i.e. the amount of pollutant washed-off from the catchment surface is limited by the build-up process. This method is appropriate for simulating first flush.

Method 4. (EMC) assumes that the concentration of pollutant in runoff is constant throughout rainfall event. The method simulates the build-up of pollutants or sediments on the catchment surface in the same way as Method 3, but the rain-dependent wash-off process is replaced by a calculated event-mean concentration. This method is mass-conservative, i.e. EMC or amount of pollutant washed-off from the catchment surface is limited by the build-up process.

Both Method 3 and Method 4 are appropriate for simulating multiple, consecutive rainfalls, e.g. in connection with LTS. These methods are also referenced as SWQ advanced methods.

SWQ Advanced Methods

SWQ advanced methods include Build-up/Wash-off and Event-mean Concentration (EMC) methods.

In addition to SWQ global data, computations in these methods are driven by user-specified sets of parameters (SWQ local data), which may be associated with each SWQ boundary property separately. This means that different behaviour of each pollutant and each catchment surface type, in terms of pollutant quantities and build-up/wash-off mechanisms, can be simulated.

Parameter set for modelling SWQ using advanced methods are specified through the SWQ Advanced Methods Editor (Figure 5.4).

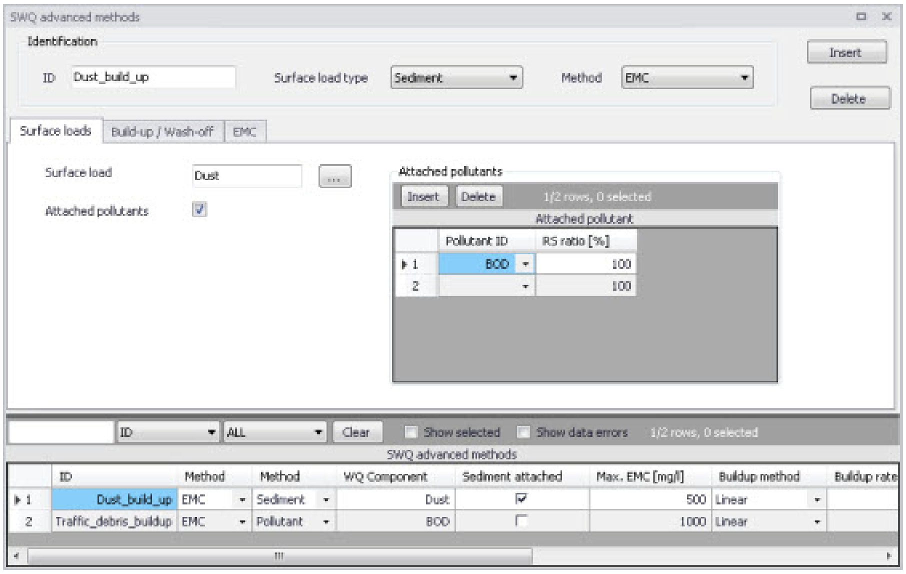


Figure 5.4 The SWQ Advanced Methods Editor

The different parts tabs of the editor are described in succeeding sections below.

5.4.1 Identification

The main properties of the SWQ parameter set are defined in the Identification box. Any SWQ parameter set may refer either to a pollutant component or to a sediment fraction (with or without pollutants attached). Also, the set will apply for one of the two SWQ advanced methods only (Build-up/Wash-off or EMC).

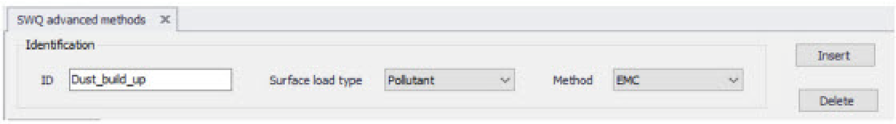


Figure 5.5 Identification box in the SWQ Advanced Methods Editor

An overview of database attributes and their usage is found in Table 5.2 below.

Table 5.2 SWQ Advanced Methods Identification box attributes (Table msm_SWQPollutant)

Edit Field	Description	Usage	Attribute Table Field
ID	Unique name (identifier) for SWQ local parameter set	Build-up/Wash-off and EMC methods	MUID
Surface load type	1 = Pollutant component 2 = Sediment fraction	Build-up/Wash-off and EMC methods	TypeNo
Method	Choice of SWQ computation method 1 = EMC 2 = Build-Up/Wash-Off	Build-up/Wash-off and EMC methods	MethodNo

5.4.2 Surface Loads

The Surface Loads tab (see Figure 5.6) specifies the pollutant component (or sediment fraction) associated with actual SWQ parameter set.

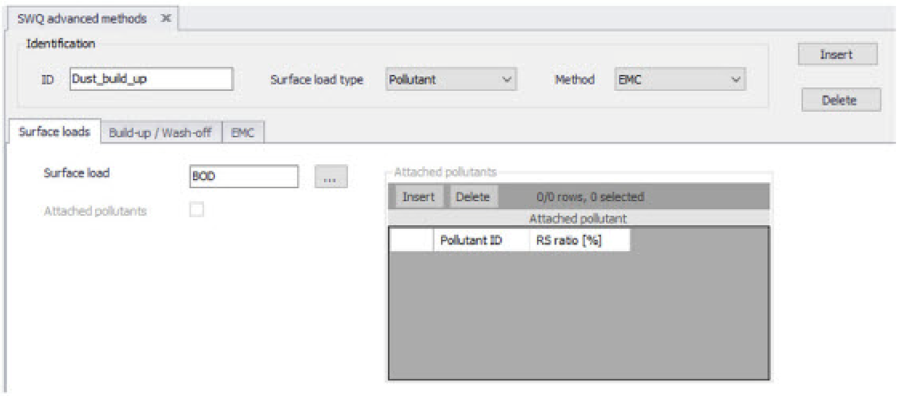


Figure 5.6 Surface Loads Tab in the SWQ Advanced Methods Editor

A description of attributes in the Surface loads tab and their usage is found in Table 5.3 below.

Table 5.3 SWQ Advanced Methods editor Surface Loads tab attributes (Table msm_SWQPollutant)

Edit Field	Description	Usage	Attribute Table Field
Surface Load	Choice of pollutant component (or sediment fraction)	Build-up/Wash-off and EMC methods	ComponentID
Attached pollutants	0 = No pollutants attached (default) 1 = Pollutants attached to sediments	Build-up/Wash-off and EMC methods, TypeNo = 2 (sediment fraction)	SedimentAttachNo

If the actual parameter set relates to a sediment fraction, pollutants may (optionally) be attached to the sediments though inputs in the editor's secondary grid (database table msm_SWQAttachedPollutant (see Figure 5.7).

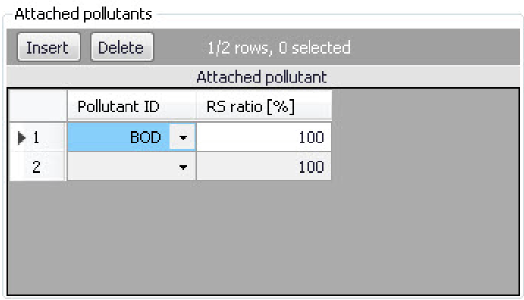


Figure 5.7 The Attached Pollutants secondary grid in the Surface Loads Tab

The amount of the sediment-attached pollutants is specified as a fraction (percentage) of the sediment mass. The total of the attached pollutants shall not exceed 100%.

When surface load is specified as a pollutant component, the model output is directed into the SWQ pollutants output, which may be used as input into the network advection-dispersion simulation (directly in case of simultaneous catchment and network simulation, or as a WQ boundary property to the run-off load into the network model).

When surface load is specified as a sediment fraction, the model output is directed into the SWQ sediments output, which may be used as input into the network sediment transport simulation (directly in case of simultaneous catchment and network simulation, or as an ST boundary property to the run-off load into the network model).

Attached pollutants (if any) are directed into the SWQ pollutants output, similarly as in case of a pollutant surface load. The mass of pollutants is calculated as a specified fraction of sediment mass.

A description of attributes in the Attached Pollutants secondary grid (table msm_SWQAttachedPollutant) and their usage is found in Table 5.4 below.

Table 5.4 Overview of the Attached Pollutants secondary grid attributes (Table msm_SWQAttachedPollutant)

Edit Field	Description	Usage	Attribute Table Field
PollutantID	Duration of antecedent dry weather period before the start of the simulation [hours]	Build-up/Wash-off and EMC methods, SedimentAttachNo =1	PollutantID
RSRatio	Peak rainfall intensity in a rainfall event, used as a minimum threshold to include the rainfall event in the SQW computation	Build-up/Wash-off and EMC methods, SedimentAttachNo =1	PSRatio

5.4.3 Build-Up/Wash-Off

The Build-up/Wash-off tab (see Figure 5.9) provides the local parameters for the Build-up/Wash-off method.

Build-up parameters are:

Method: Linear or Exponential. During dry weather periods pollution or sediments accumulate on the surface of urban catchments. The most common formulations for this process are to assume that the build-up is a linear or an exponential function of time. The choice between the two formulations is not straightforward due to insufficient experimental results.

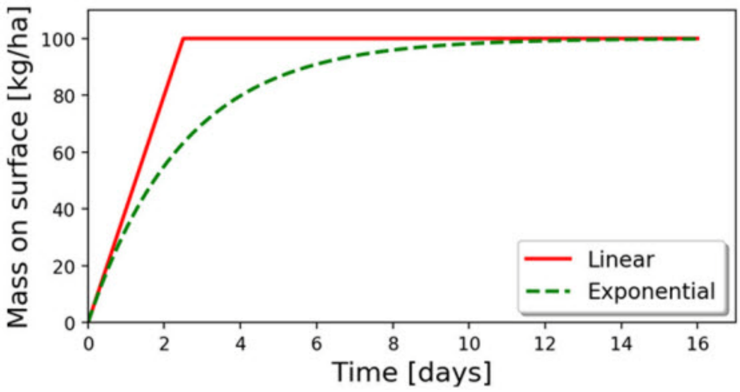


Figure 5.8 Illustration of the linear and the exponential build-up function. For details of mathematical formulation, see MIKE 1D Reference Manual.

Rate: Linear build-up rate of the pollutant component (or sediment fraction) on the catchment surface, given as mass per unit area per unit of time.

Max: Maximum possible amount of pollution (or sediment) on the catchment surface. This represents an equilibrium situation, when removal rate is equal to build-up rate.

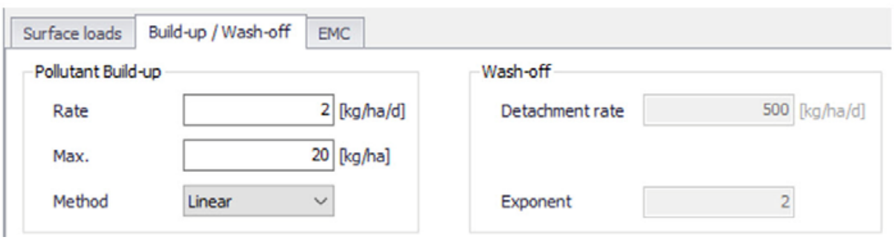


Figure 5.9 The Build-Up/Wash-Off Tab in the SWQ Advanced Methods Editor

Wash-off parameters are related to the applied wash-off formula, which is based on the assumption that the wash-off is exponentially related to the rain intensity.

Detachment rate: Wash-off (detachment) rate at rainfall intensity 25.4 mm/hour (1 inch/hour). This is a calibration factor for the wash-off process: smaller values apply for pollutants firmly attached the catchment surface, higher values apply for loose particles.

Note that the "Detachment rate" operates with sediment mass uniformly spread over entire contributing catchment area. The default value of 10000 kg/ha/day, assuming the sediment density of 2650 kg/m^3 and zero porosity, is equivalent to the velocity of rain penetration into the sediment layer of 16 mm/h. The recommended values are up to max. 30 mm/h.

Exponent: Default value is 2, which implies that the wash-off is proportional to the kinetic energy of the rain drops.

A description of attributes in the Build-up/Wash-off tab and their usage is found in Table 5.5 below.

Table 5.5 Overview of the SWQ Advanced Methods Build-up/Wash-off tab attributes (Table msm_SWQPollutant)

Edit Field	Description	Usage	Attribute Table Field
Pollutant Build-Up/ Rate	Linear build-up rate of pollutant (sediment) on the catchment surface ($ML^{-2}T^{-1}$)	Build-up /Wash-off and EMC methods	BuildUpRate
Pollutant Build-Up/ Max.	Maximum amount of pollutant (sediment) on the catchment surface per unit area (ML^{-2})	Build-up /Wash-off and EMC methods	MaxBuildUp
Pollutant Build-Up/ Method	1 = Linear 2 = Exponential	Build-up /Wash-off and EMC methods	BuildUpTypeNo
Wash-Off/ Detachment Rate	Pollutant (sediment) detachment rate at rainfall intensity 25.4 mm/hour (1 inch/ hour)	Build-up /Wash-off method	DetachRate
Wash-Off/ Exponent	Default value is 2. Larger values increase the wash off non-linear dependency on the rainfall intensity. Value 1 implies a linear dependency	Build-up /Wash-off method	WashOffExp

5.4.4 EMC

The EMC tab (see Figure 5.10) provides a single local parameter for the EMC (Event Mean Concentration) method.

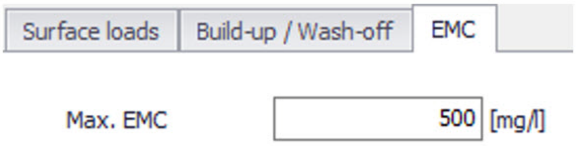


Figure 5.10 The EMC Tab in the SWQ Advanced Methods Editor

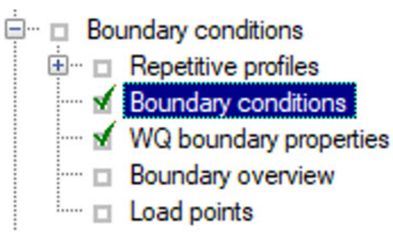
Max. EMC: Is the maximum possible concentration of the pollutant component (or sediment fraction) in the runoff leaving the catchment.

The primary role of this parameter is to prevent unrealistic results i.e. very high values of pollutant EMC in case of long ADWP and/or high build-up rates during small rain events. In such cases, Max. EMC will prevent the full removal of pollutants by a small rain.

Table 5.6 Overview of the SWQ Advanced Methods EMC tab attributes (Table msm_SWQPollutant)

Edit Field	Description	Usage	Attribute Table Field
Max. EMC	Maximum event mean concentration; used to prevent unrealistically high concentrations and full wash-off by a minor rainfall	EMC method	MaxEMC

5.5 Boundary Conditions for SWQ



SWQ boundary conditions define domain of validity for the actual boundary condition (surface runoff or RDII) and the boundary condition spatial extent.

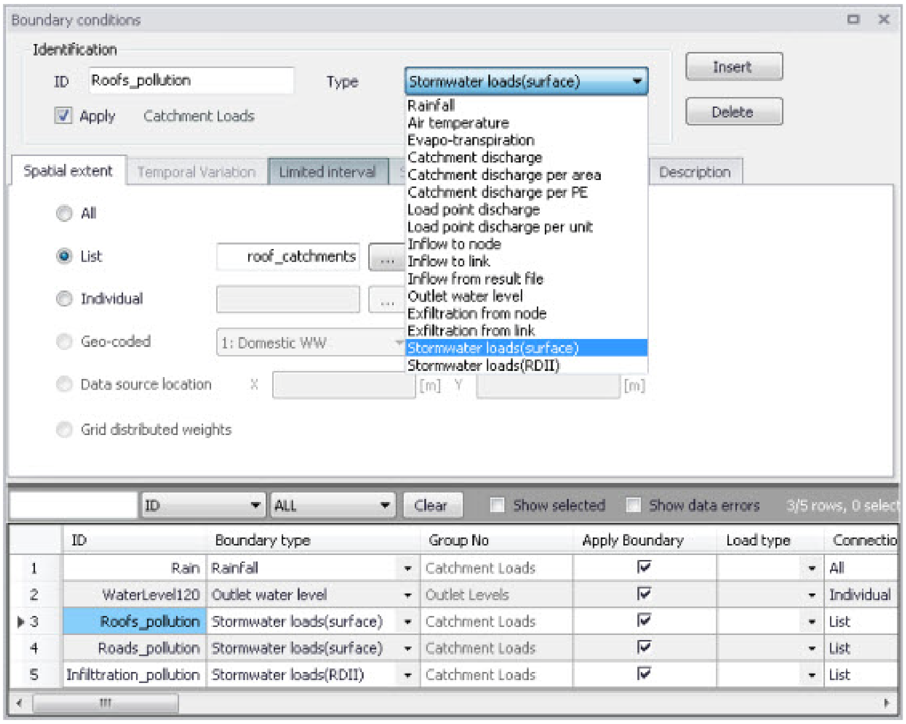


Figure 5.11 The Boundary Conditions Editor showing Types relevant for SWQ

5.5.1 SWQ Boundary Condition Types

SWQ boundary conditions are of the following types (see Figure 5.11):

- Stormwater Loads (Surface). This type applies for WQ in surface runoff.
- Stormwater Loads (RDII). This type applies for WQ in RDII flows.

Table 5.7 provides an overview of the SWQ methods that are applicable under the two types of stormwater loads (surface runoff and RDII), i.e. with the two types of SWQ boundary conditions.

Table 5.7 Overview of applicable SWQ boundary condition types and SWQ methods for various hydrological models and their combinations. All surface runoff models support all four SWQ methods, while RDI supports only Method 1 (Simple Concentration) and Method 2 (Tabulated concentration).

Hydrological model/combination	SWQ Boundary condition and SWQ methods	
	Stormwater loads (surface)	Stormwater loads (RDII)
Time-Area	Methods 1, 2, 3, 4	-
Time-Area + RDI	Methods 1, 2, 3, 4	-
Kinematic Wave	Methods 1, 2, 3, 4	-
Kinematic Wave + RDI	Methods 1, 2, 3, 4	Methods 1 ,2
Linear Reservoir (C1)	Methods 1, 2, 3, 4	-
Linear Reservoir (C1) + RDI	Methods 1, 2, 3, 4	Methods 1 ,2
Linear Reservoir (C2)	Methods 1, 2, 3, 4	-
Linear Reservoir (C2) + RDI	Methods 1, 2, 3, 4	Methods 1 ,2
UHM	Methods 1, 2, 3, 4	-
UHM + RDI	Methods 1, 2, 3, 4	Methods 1 ,2
RDI (solo)	-	Methods 1 ,2

5.5.2 Spatial Extent

As any other boundary condition, SWQ Boundary Conditions may be specified as uniform for the entire model (i.e. associated with all model catchments), or as spatially distributed through selections of catchments and/or individual catchments.

Figure 5.12 illustrates definition of spatial extent based on a list (selection). Note that selections may refer to different geographical areas as well as to different catchments layers.

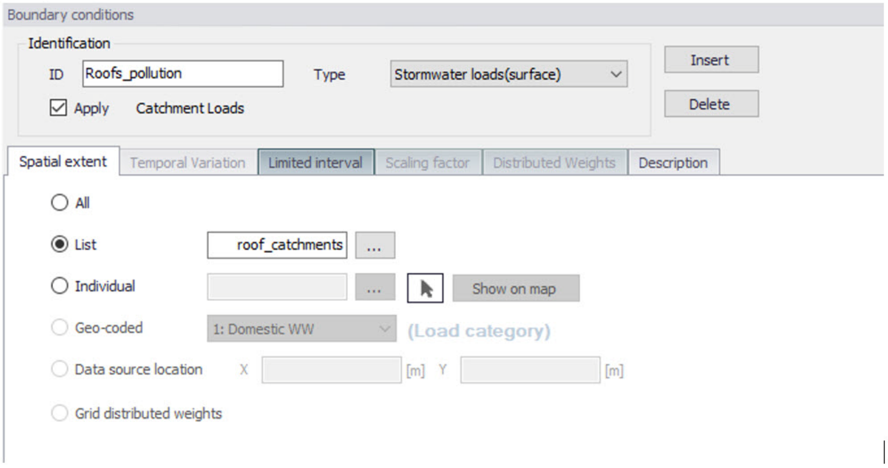
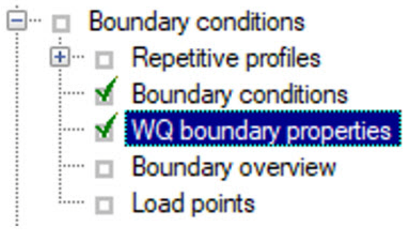


Figure 5.12 Example of a Stormwater Loads (Surface) boundary condition with spatial extent defined by a list (selection) of catchments (roof_catchments)

5.6 WQ Boundary Properties for SWQ

SWQ quantities and computation methods for various pollutants associated with an SWQ Boundary Condition are specified in the WQ Boundary Properties Editor.



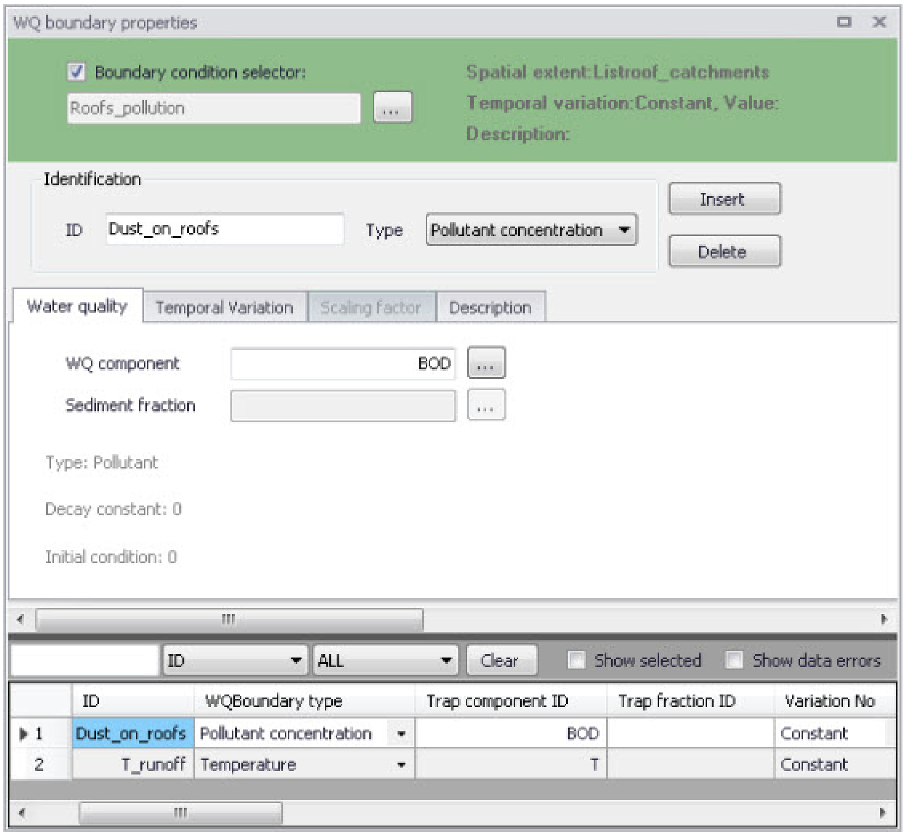


Figure 5.13 The WQ Boundary Properties Editor with an example SWQ boundary water quality item (i.e. Roofs pollution with dust)

Normally, one SWQ boundary property should be specified for each pollutant included in an SWQ boundary condition.

Definition of WQ boundary properties follows the same workflow as any other WQ boundary property. For details, see descriptions under the Boundary Conditions Chapter in the manual.

In addition, WQ boundary properties for SWQ include two special types of Temporal variations:

- Table concentration (supports application of SWQ table concentration method)
- SWQ advanced (supports application of SWQ advanced methods: Build-up/Wash-off and EMC).

Definition of WQ boundary properties in the WQ Boundary Properties Editor for all SWQ methods is described in the following sections.

5.6.1 Boundary Condition Selector

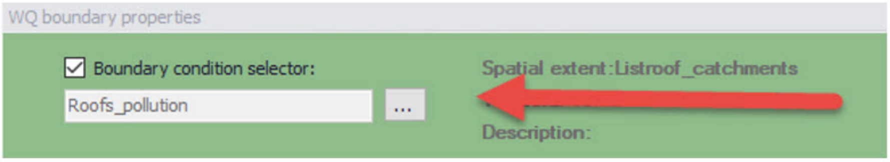


Figure 5.14 The Boundary Condition Selector is the step in specifying WQ boundary properties for SWQ

The wanted SWQ boundary condition item to be associated with a water quality boundary property is selected in the Boundary Condition Selector on top of WQ Boundary Properties editor.

5.6.2 Identification

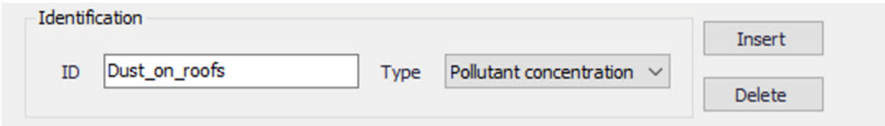


Figure 5.15 Identification information for the WQ boundary property for SWQ

After inserting a new WQ boundary property, it shall be given a proper name (identifier) and a correct (water quality) type shall be selected.

For easier identification, the property name should reflect its contents, i.e. which pollutant (or sediment) and which model area it describes.

Type is selected among available choices filtered according to currently specified WQ components.

5.6.3 Water Quality

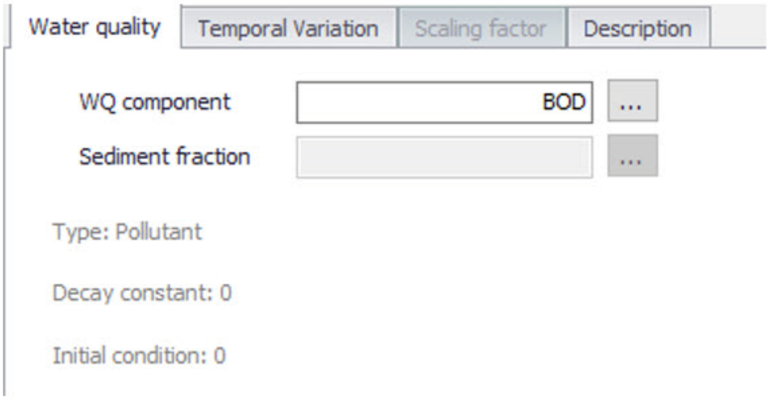


Figure 5.16 The Water Quality Tab in the WQ Boundary Properties Editor. Used to define pollutant component (or sediment fraction) associated with the current WQ boundary property.

Depending on the specified (water quality) type, the user selects the relevant WQ component or sediment fraction.

5.6.4 Temporal Variation

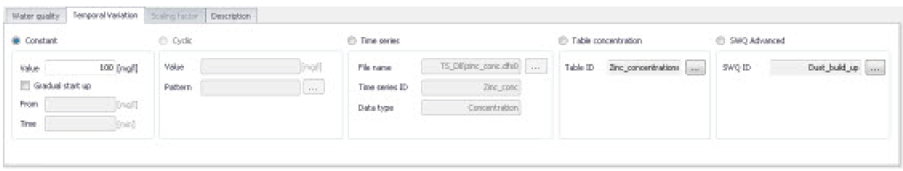


Figure 5.17 The Temporal Variation Tab in the WQ Boundary Properties Editor

In the context of SWQ, definition of temporal variation for WQ boundary property is closely connected to the choice of SWQ method, as summarized in the overview in Table 5.8.

Table 5.8 Overview of relation between Temporal Variation and SWQ Methods

Temporal Variation	SWQ Method			
	Method 1 (Simple concentration)	Method 2 (Table concentration)	Method 3 (Build-up/ Wash-off)	Method 4 (EMC)
Constant	Yes	-	-	-
Cyclic	-	-	-	-
Time series	Yes	-	-	-
Table concentration	-	Yes	-	-
SWQ advanced	-	-	Yes	Yes

Method 1: Simple Concentration (Constant)

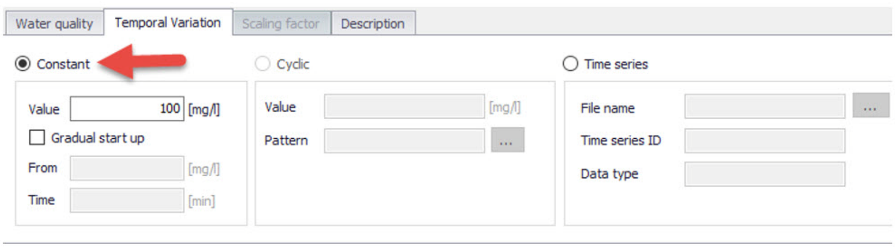


Figure 5.18 Definition of constant SWQ concentration

A constant value of any WQ component (e.g. concentration, temperature, etc.) is assigned to runoff.

Method 1: Simple Concentration (Time series)

Figure 5.19 Time series of values of any property is assigned to runoff. In absence of measured data, time series typically reflect an estimate of seasonal variations.

Application of time series values for SWQ boundary properties implies knowledge or estimates of historical variations of any WQ component, e.g. in the context of a WQ model calibration.

Method 2: Table Concentration

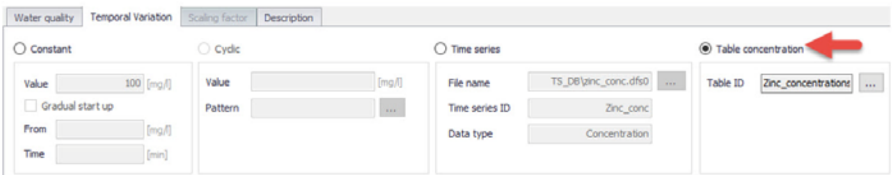


Figure 5.20 Table Concentration applies to the SWQ Table Concentration method

A tabular function of the type Runoff pollutants is selected and associated with the WQ boundary property. It establishes a fixed relation between the runoff intensity (specific runoff) and the value of WQ component, e.g. pollutant concentration.

Methods 3 (Build-up/Wash-off) and 4 (EMC): Advanced Methods

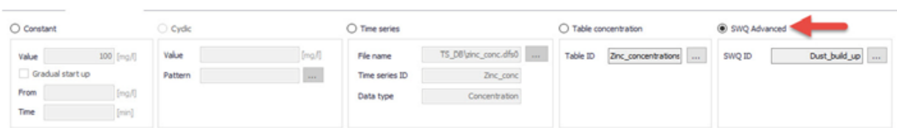


Figure 5.21 SWQ Advanced is chosen when SWQ Method 3 (Build-up/Wash-off) or Method 4 (Event Mean Concentration) is applied

After selecting SWQ advanced, the user must choose one among previously specified SWQ parameter sets for the pollutant or sediment fraction associated with the current WQ boundary property.

The SWQ parameter sets are related to one of the two SWQ advanced methods and contain a definition of the parameters for the chosen method. The actual parameters can be looked up in the SWQ Advanced Methods editor.

5.7 Workflow

Modelling of SWQ is summarised through description of a workflow.

At the outset, it is assumed that a functional MIKE+ model including one or more catchments is available.

Prepare layered catchments (optional)

If the analysis includes various types of surfaces, and if the catchments delineation does not reflect the surface types, the original sub-catchments shall be cloned into layers. All layers of catchments shall be connected to the network identically. Drainage area for all catchments representing one physical catchment shall be specified. The sum of the drainage area for a catchment must be equal to the full contributing catchment area.

Specify rainfall boundary

A rainfall boundary condition series shall be specified. A single rainfall time series may be associated with all catchments in the model, or the rainfall can be spatially distributed by specifying multiple rain gauges or RADAR rainfall data.

Specify WQ components and sediment fractions

To be included in SWQ computation, any WQ component and/or sediment fraction must be defined in respective editors (WQ Components editor or Sediment Fractions editor, respectively).

Activate SWQ module

To enable the Stormwater Runoff WQ (SWQ) module, activate the Catchment feature, and the Rainfall-Runoff (RR) and Transport (AD, SWQ) modules (in the ‘General Settings | Model type’ editor). This will provide access to all relevant editors.

Consider appropriate SWQ method

As shown in Table 5.7, the applicability of the SWQ methods depends on the runoff component, i.e. choice of the runoff model.

Also, not all of the available SWQ methods are applicable to all types of pollutants/sediments. E.g. external pollutants are best simulated by mass-conservative methods (i.e. Methods 3 and 4).

Review Global SWQ parameters

If one of the advanced SWQ methods is to be applied, review and adjust SWQ global parameters (in the SWQ Global Data editor).

Specify SWQ parameters sets

If one of the advanced SWQ methods is to be applied in the SWQ computation for any of the pollutants or sediment fractions, SWQ parameter sets must be specified for such pollutants or sediment fractions.

Note that several parameter sets may be specified for any pollutant or sediment fraction - each for one type of catchment surface.

Specify SWQ boundary conditions

For each type of catchment surface and for each runoff component, a separate boundary condition can be specified as one of the two available SWQ boundary types. Association of the boundary condition with model sub-catchments representing a specific type of surface is achieved by specifying appropriate selection list.

Specify SWQ boundary WQ properties

For each of the specified SWQ boundary conditions, a number of WQ properties can be specified. Normally, one WQ property is defined for each pollutant (or sediment fraction) assumed to be present on the actual catchment surface type and in the actual runoff component. For details, refer to Section 5.6..

Set up a runoff simulation with SWQ

The SWQ setup is included in the runoff simulation simply by ticking the Storm Water Quality (SWQ) checkbox in the General tab of the Simulation Setup editor.

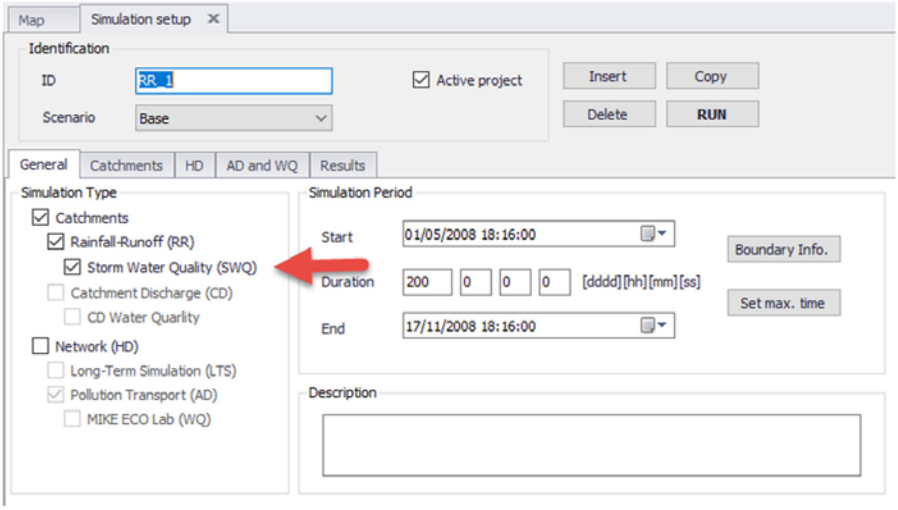


Figure 5.22 Example simulation setup for a model including Rainfall-Runoff and SWQ. Note that SWQ can only run in association with a Rainfall-Runoff simulation.

MIKE+ will automatically include relevant output files. The Default output files can be replaced or complemented by user-specified output files.

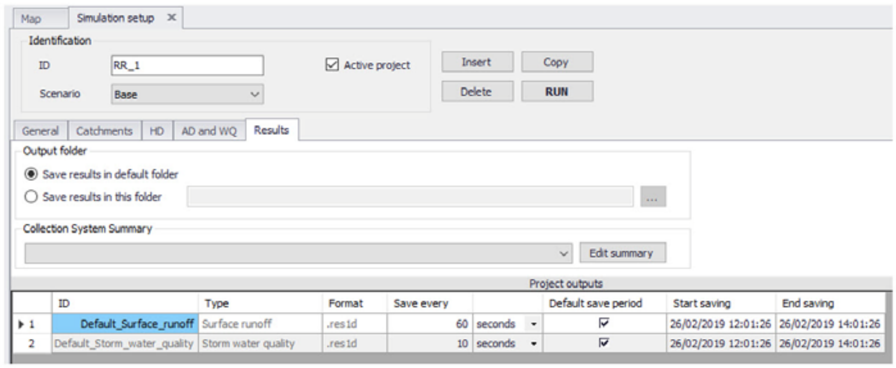


Figure 5.23 Default output is automatically provided by the system

6 Initial Conditions

Initial conditions are needed by most numerical models to define the initial state of hydrodynamic, water quality, and other module specific variables at the start of the simulation.

6.1 HD Initial Conditions

Define hydrodynamic initial conditions for network models (i.e. Rivers and/or Collection Systems) in the Initial Conditions editor in MIKE+ accessed under the Initial Conditions group on the setup tree (Figure 6.1).

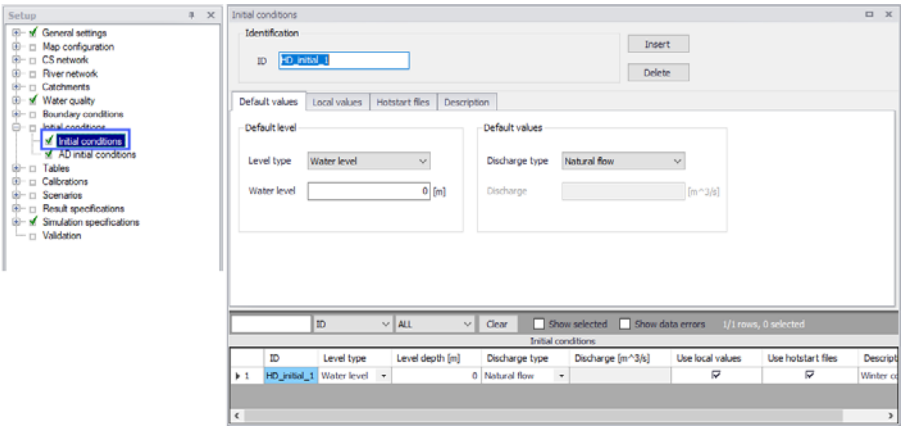


Figure 6.1 The Initial Conditions editor

Initial network HD conditions may be defined by:

- User-defined values for state variables. They may be defined as general values to be universally applied in the network models, or local values for specific parts of networks.
- Hotstart files resulting from previous simulations (i.e. previous simulation result files). Values from the hotstart file are used where available, but where computational points are not in the hotstart file, user-defined values are automatically applied.

For a set of HD initial conditions to apply in a simulation, it must be selected in this simulation definition, from the 'Simulation setup' editor.

6.1.1 Identification

The Identification group contains information on the unique ID for the initial conditions setup.

Use the 'Insert' button to add initial conditions setups in the editor. The Delete button removes the active initial conditions setup from the list.

6.1.2 Default Values

Define default initial conditions in the Default Values tab page of the Initial Conditions editor (Figure 6.2).

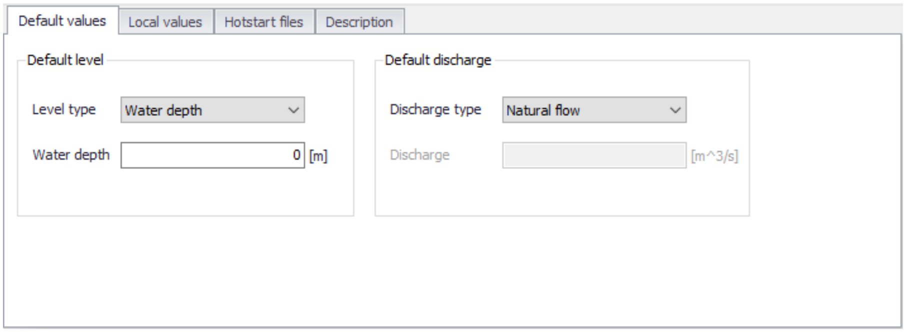


Figure 6.2 The Default Values tab page on the Initial Conditions editor

Default values for initial conditions are applied at computational points which are not included in hotstart files, and for which no local values have been defined.

Default Level

Initial water level conditions may be defined in terms of level or depth.

Level Type

How height of water at computation points is expressed. Water height may be specified as Water level (measured from a Datum) or Water depth (measured from the invert level).

Water Level/Depth

Corresponding value for the water level or depth.

Default Discharge

Discharge may be specified either as User defined flow values or calculated as Natural flow.

Discharge Type

Discharge may be specified either as 'User defined' flow values or calculated as 'Natural flow'.

Discharge

Corresponding value for 'User defined' Discharge Type.

The 'Natural flow' option calculates discharge according to the Manning formula:

$$Q = R^{2/3} \cdot A \cdot M \cdot \sqrt{I} \tag{6.1}$$

where

R = Hydraulic radius

A = Flow Area

M = Manning number*

I = Water surface slope

*Natural flow calculation can use Manning (M) or Manning (n) roughness formulations depending on the selected option under Default Roughness Formulation in the Bed Roughness editor.

6.1.3 Local Values

Initial HD conditions may be defined in specific areas of network models in MIKE+ (i.e. Rivers and/or Collection Systems). Localized initial conditions are defined in the Local Values tab page of the Initial Conditions editor (Figure 6.3).

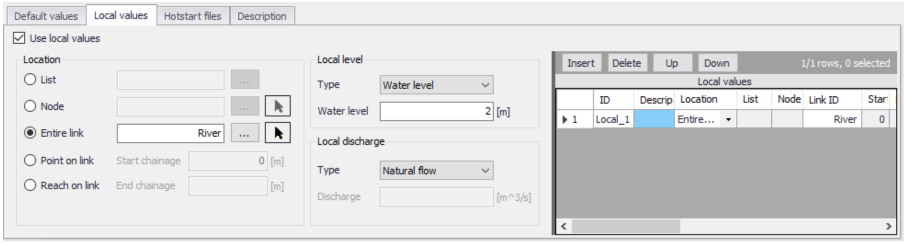


Figure 6.3 The Local Values tab page of the Initial Conditions editor

Use Local Values

Activate the 'Use local values' option to allow definition of local values for the active initial conditions setup.

Location

The location of local initial conditions may be defined by a:

- **List**: A predefined selection list of network model elements. Click on the ellipsis button to view the ID Selector dialog, which presents available selection lists. Selection lists may be defined using the Selection Manager accessed via the Map menu ribbon.
- **Node**: A particular node in the CS network. Click on the ellipsis button to view the ID selector dialog presenting a list of nodes in the project. Alternatively, use the arrow button to interactively select the node from the Main Map.
- **Entire Link**: The entire river branch or CS network link defined in the Link ID input box (see Figure 6.4). Click on the ellipsis button to view the ID selector dialog presenting a list of River branches or CS network pipes and canals that exist in the project. Alternatively, use the arrow button to interactively select the river branch or CS network link from the Main Map.
- **Point on link**: A point in the river branch or CS network link defined in the Link ID input box (Figure 6.4). Define the point along the link or river branch in the 'Start chainage' input box in the editor.
- **Reach on link**: A segment in the river branch or CS network link defined in the Link ID input box (Figure 6.4). The segment is defined by the 'Start chainage' and 'End chainage' values in the editor.

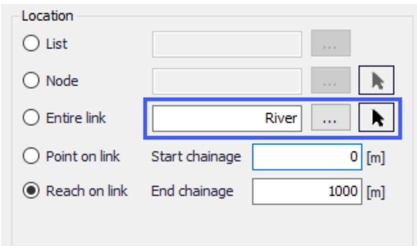


Figure 6.4 The Link ID input box for defining local initial conditions along links

Local Level

Local initial water level conditions may be defined in terms of level or depth:

- **Type**: Water height may be specified as Water level or Water depth.
- **Water Level/Depth**: Corresponding value for the water level or depth.

Local Discharge

Local discharge conditions may be specified as User defined flow values or calculated as Natural flow:

- **Type**: Discharge may be specified either as ‘User defined’ flow values or calculated as ‘Natural flow’. The ‘Natural flow’ option calculates discharge according to the Manning formula shown in Equation (6.1).
- **Discharge**: Corresponding value if Type is ‘User defined.’

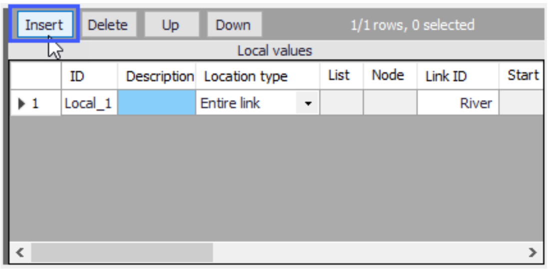
Note: If two or more local values are defined within the same river reach or link, the intermediate values will be calculated by linear interpolation.

If only one local value location is defined in a river or link, then this value will be applied at the specific location and all other values in the link will be assigned Default values.

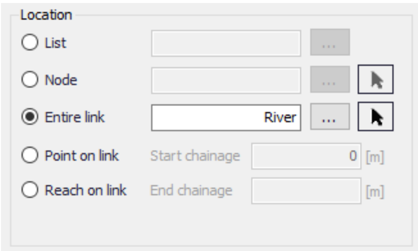
Defining Local Initial HD Conditions

To define local initial conditions:

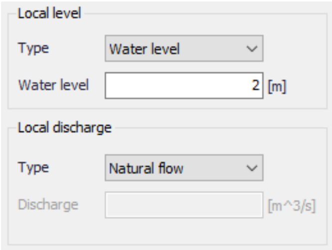
1. Activate the ‘Use local values’ option on the Local Values tab page.
2. Insert a row in the secondary table on the right side of the Local Values tab page using the ‘Insert’ button.



3. Define the location for the local initial condition under the Location groupbox.



4. Define initial water level/depth and discharge conditions under the Local Level and Local Discharge group boxes.



6.1.4 Hotstart Files

Hotstart files maybe used to define HD initial conditions for networks in MIKE+. Define hotstart files to use for an initial conditions setup via the Hotstart Files tab page of the Initial Conditions editor (Figure 6.5).

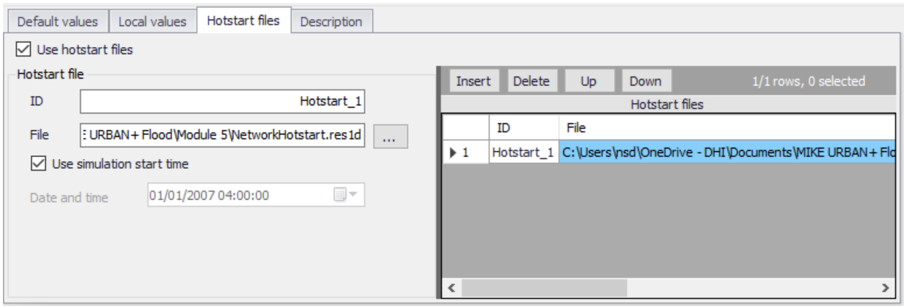


Figure 6.5 The Hotstart Files tab page in the Initial Conditions editor

The following file types may be used:

- ***.RES1D**: MIKE 1D engine computation results.
- ***.PRF**: MOUSE engine pipe flow computation results.
- ***.RES11**: MIKE 11 engine computation results.

Use Hotstart Files

Activate the ‘Use hotstart files’ option on the tab page to allow definition and use of hotstart files for the active initial conditions setup.

ID

Hotstart file ID

File

Hotstart file path and name.

Use Simulation Start Time

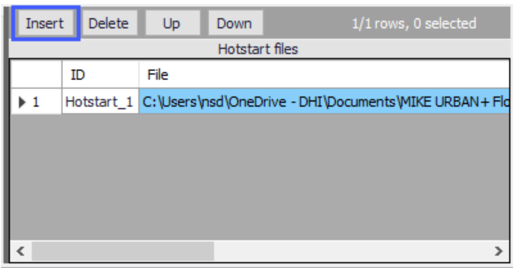
If 'Use simulation start time' option is enabled, the initial condition is automatically extracted from the hotstart file using the simulation start date and time as specified in the 'Simulation Setup' editor.

Date and Time

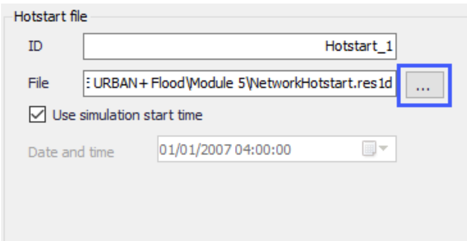
The date and time at which the initial conditions are loaded from the hotstart file. Only active if 'Use simulation start time' option is inactive.

Defining Hotstart Files for Initial HD Conditions

1. Activate the 'Use hotstart files' option on the tab page.
2. Add a row in the secondary table on the right side of the Hotstart Files tab page using the 'Insert' button. Note that several hotstart files may be added. If a part of the network is covered by more than one hotstart file the first hotstart file will be used.



3. Define the file name and path for the hotstart file under the Hotstart File groupbox in the 'File' input box using the ellipsis button.



4. Define the Date and Time in the hotstart file for which initial condition values will be taken.

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 file will be used. If a part of the network is not covered by the specified hotstart files, Default or local values are used.

If the network in a hotstart file differs from the current network e.g. the number of grid points in a link have changed, then interpolation is performed.

6.1.5 Description

Optional free text description for an HD Initial Conditions setup (Figure 6.6).

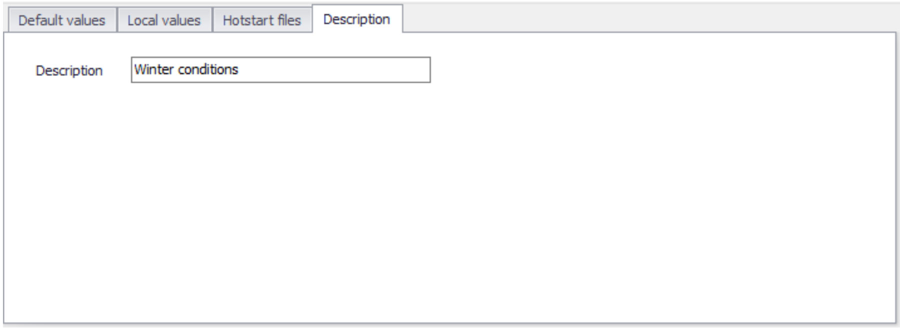


Figure 6.6 The Description tab page on the Initial Conditions editor

6.2 AD Initial Conditions

Initial water quality component concentrations in river and collection system networks are defined in the AD Initial Conditions editor in MIKE+ (Figure 6.7).

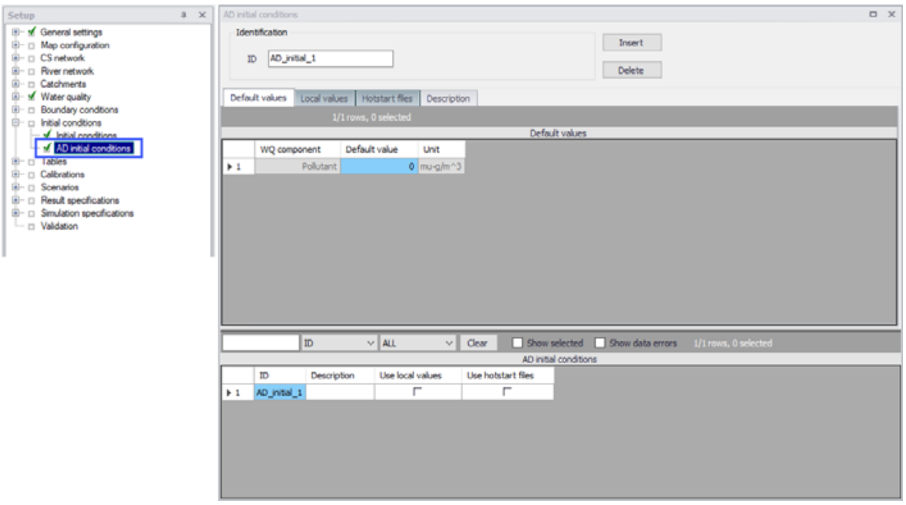


Figure 6.7 The AD Initial Conditions editor for both CS and River networks

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.

For a set of AD initial conditions to apply in a simulation, it must be selected in this simulation definition, from the 'Simulation setup' editor.

6.2.1 Identification

The Identification group holds information on the unique ID for an AD initial conditions setup.

Use the 'Insert' button to add AD initial conditions setups in the editor. The Delete button removes the active AD initial conditions setup.

6.2.2 Default Values

Define default initial AD conditions in the Default Values tab page of the AD Initial Conditions editor (Figure 7.2).

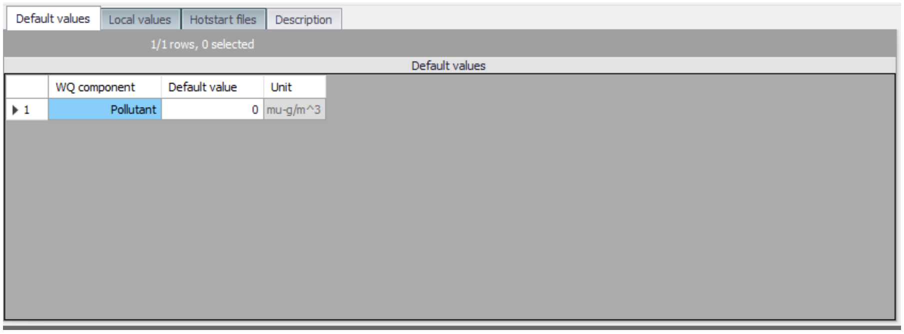


Figure 6.8 The Default Values page on the AD Initial Conditions editor

Default values for initial AD conditions are applied at computational points which are not included in hotstart files, and for which no local values have been defined.

For Water blend components, concentrations must always be given as a number between 0 and 100 %, and the sum of the two blend components must add up to 100 %.

For initial conditions specified at individual manholes, the initial conditions in connected links are calculated by linear interpolation of the concentrations specified in the upstream and downstream grid points.

6.2.3 Local Values

Initial AD conditions may be defined in specific areas of river and/or collection system network models. MIKE+ will use the initial values specified under the Default Values tab page except for those locations were local values have been specified.

Local initial AD conditions are defined in the Local Values tab page of the AD Initial Conditions editor (Figure 6.9).

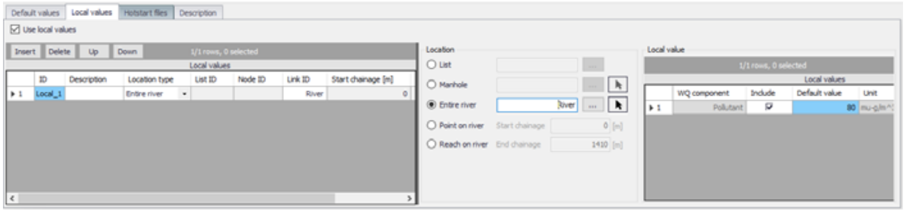


Figure 6.9 The Local Values tab page of the AD Initial Conditions editor

Use Local Values

Activate the ‘Use local values’ option to allow definition of local values for the active AD initial conditions setup.

Location

The location of local AD initial conditions may be defined by a:

- **List**: A predefined selection list of network model elements. Click on the ellipsis button to view the ID Selector dialog, which presents available selection lists. Selection lists may be defined using the Selection Manager accessed via the Map menu ribbon.
- **Node**: A particular node in the CS network. Click on the ellipsis button to view the ID selector dialog presenting a list of nodes in the project. Alternatively, use the arrow button to interactively select the node from the Main Map.
- **Entire Link**: The entire river branch or CS network link defined in the Link ID input box. Click on the ellipsis button to view the ID selector dialog presenting a list of River branches or CS network pipes and canals that exist in the project. Alternatively, use the arrow button to interactively select the river branch or CS network link from the Main Map.
- **Point on link**: A point in the river branch or CS network link defined in the Link ID input box. Define the point along the link or river branch in the ‘Start chainage’ input box in the editor.
- **Reach on link**: A segment in the river branch or CS network link defined in the Link ID input box. The segment is defined by the ‘Start chainage’ and ‘End chainage’ values in the editor.

Local Value

Here the WQ component for which local initial conditions shall be set is selected. It is possible to choose between the components defined in the WQ Components editor. Local values are defined in a table containing information on:

- **WQ Component**: WQ components defined in the project in the WQ Components editor.

- **Include**: Tick this option to set local values for the corresponding WQ component for the initial AD conditions setup.
- **Default Value**: The value for the WQ component.
- **Unit**: Units for the corresponding WQ component varying according to type.

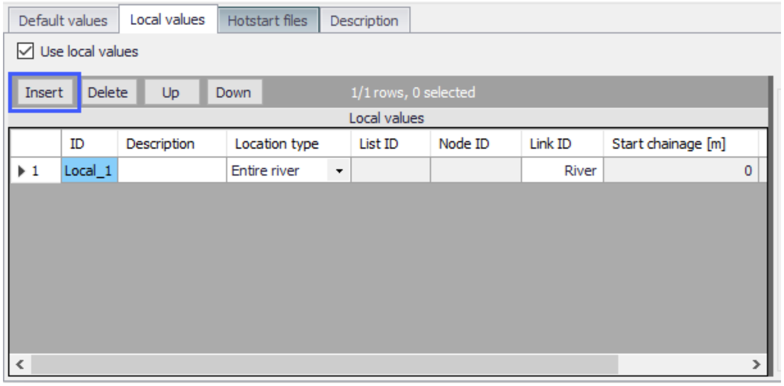
Note: If two or more local values are defined within the same river reach or link, the intermediate values will be calculated by linear interpolation.

If only one local value location is defined in a river or link, then this value will be applied at the specific location and all other values in the link will be assigned Default values.

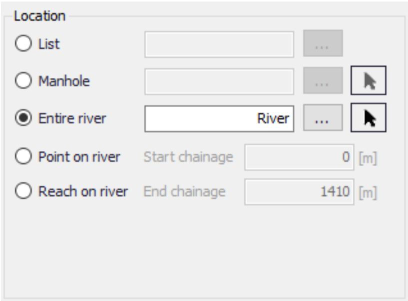
Defining Local Initial AD Conditions

To define local initial AD conditions:

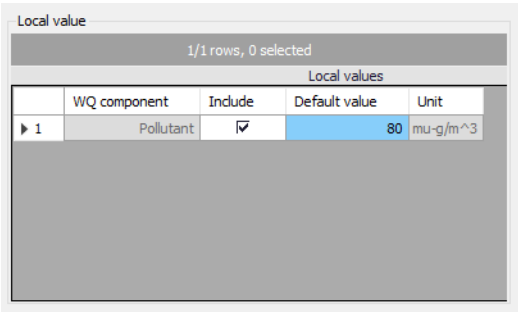
1. Tick on the 'Use local values' option on the Local Values tab page.
2. Insert a row in the secondary table on the leftmost side of the Local Values tab page using the 'Insert' button.



3. Define the location for the local AD initial condition under the Location groupbox.



4. Activate and define the value(s) for available WQ component(s) in the Local Value table on the rightmost side of the tab page.



6.2.4 Hotstart Files

Hotstart files (*.RES1D, *.TRF, or *.RES11) maybe used to define AD initial conditions for networks in MIKE+. Define hotstart files to use for an initial conditions setup via the Hotstart Files tab page of the Initial Conditions editor (Figure 7.5).

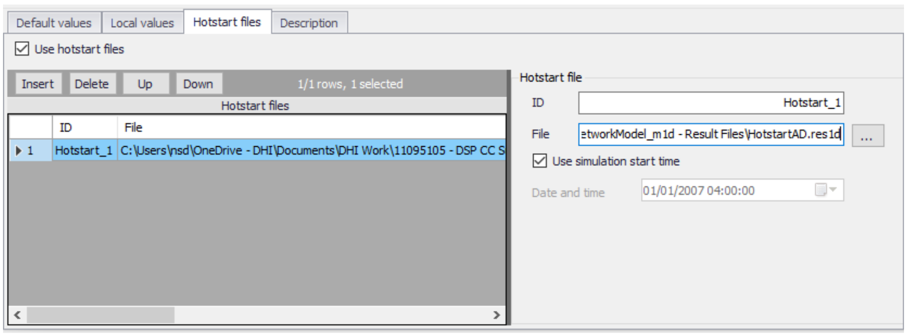


Figure 6.10 The Hotstart Files tab page in the AD Initial Conditions editor

Use Hotstart Files

Activate the ‘Use hotstart files’ option on the tab page to allow definition and use of hotstart files for the active AD initial conditions setup.

ID
Hotstart file ID

File
Hotstart file path and name. Files of type *.RES1D, *.TRF, and *.RES11 may be used as AD hotstart files. Point to the file using the ellipsis button.

Use Simulation Start Time
If ‘Use simulation start time’ option is enabled, the initial AD condition is automatically extracted from the hotstart file using the simulation start date and time as specified in the ‘Simulation Setup’ editor.

Date and Time
The date and time at which the initial AD conditions are loaded from the hotstart file. Only active if ‘Use simulation start time’ option is inactive.

Defining Hotstart Files for Initial AD Conditions

1. Activate the ‘Use hotstart files’ option on the tab page.
2. Add a row in the Hotstart files table using the ‘Insert’ button. Note that several hotstart files may be added. If a part of the network is covered by more than one hotstart file the first hotstart file will be used.
3. Define the file name and path for the hotstart file under the Hotstart File groupbox in the ‘File’ input box using the ellipsis button.
4. Define the Date and Time in the hotstart file for which initial condition values will be taken.

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 file will be used. If a part of the network is not covered by the specified hotstart files, Default or local values are used.

6.2.5 Description

Add optional free text descriptions for an AD Initial Conditions setup in the Description tab page of the AD Initial Conditions editor.

7 Boundary Conditions

In broad terms, a model boundary condition can be defined as an external interference, which forces the behaviour of the computed variables within the model domain.

A great part of the boundary conditions represent various types of water loads (rainfall, infiltration, wastewater…). The main characteristic of these ’load’ boundary conditions is that they contain a ’transport medium’ - water. Water can transport other material (dissolved pollutants, oxygen, sediments), heat (temperature) and organisms (bacteria and parasites), as well as various other properties, such as pH, conductivity, turbidity, etc. Properties of ’water load’ boundary conditions (expressed as concentrations, mass flow, temperature, etc.) are also, in strict terms, boundary conditions for e.g. pollution transport model (Advection-Dispersion). However, these are not treated as separate boundary conditions, but items inseparably associated with the water loads. E.g. a lateral inflow is a water load boundary condition for the hydrodynamic process in the network (defined as discharge item), while the temperature, concentration, etc. associated to this particular inflow are specified as pollutant items of that “load” boundary condition.

7.1 Hydraulic boundary conditions

The hydraulic boundary conditions tab allows the modeller to control the loads included and applied to the model.

In order to include a boundary in a simulation the “Apply” box must be checked.

The selected type of boundary condition controls which options are active in the different tabs. The list of available boundary condition types depends on the type of features included in the model setup (catchments, collection system network and/or river network).

7.1.1 Catchment boundary conditions

Types of variables associated with catchment boundary conditions are:

- Air temperature
- Evapotranspiration
- Rainfall
- Catchment discharge
- Stormwater loads
- Groundwater abstraction
- Solar radiation

- Degree-day coefficient
- Catchment irrigation

Any number of each boundary conditions types can be specified.

Air Temperature is a boundary condition used for snow melt calculations and describing constant or time series varying temperature. This type of boundary is used for RDI hydrological calculations.

Evapotranspiration is a boundary condition describing potential evapotranspiration. This type of boundary is used for RDI hydrological calculations.The dfs0 file needs to be of the type ‘Evapotranspiration’. Delete values will be considered equal to 0.0.

Rainfall catchment boundary condition can be defined using a constant value, a uniform time series (from .dfs0 file) or using a spatially- and time-varying file (.dfs2 file). The latter format can only be used when the spatial extent of the boundary is set to 'Grid distributed weights'. This type of boundary is used for precipitation-runoff hydrological calculations. When using a time series the dfs0- or .dfs2 file needs to use one of the following Item Types: ‘Rainfall Intensity’, ‘Rainfall’ or ‘Rainfall Depth’. Delete values will be considered equal to 0.0.

Catchment discharge is a boundary condition of the type ’catchment load’, has associated to a discharge (constant, cyclic or a time series). This type of boundary represents various kinds of hydraulic loads, such as area-based or PE-based dry weather loads, area-based infiltration, etc. The discharge can be associated with any pollutant, sediment or temperature item (constant, cyclic or a time series).

Stormwater loads are boundary conditions only used when the Transport module is also included in the simulation. For this boundary type, the stormwater flow value is obtained from the hydrological model and cannot be specified in the 'Boundary conditions' editor: the purpose of this boundary type is only to associate the stormwater flow to a pollutant or sediment concentrations from the 'WQ boundary properties' editor. Select the boundary type 'Stormwater loads (RDI)' for catchments modelled with the RDI hydrological model. Select the boundary type 'Stormwater loads (surface)' for catchments modelled with any of the other hydrological models. For the boundary type 'Stormwater loads (RDI)', a flow type must also be selected (Total runoff, Overland flow, Base flow, etc.), to specify to which flow component of the RDI model the pollutant or sediment concentration should be associated.

Groundwater abstraction is a boundary type used with the RDI hydrological model, when the 'Use abstraction' option is active in the RDI parameters and when the pumped depth is defined as varying in time. The item type defined in the selected time series must be 'Ground Water Abstraction Depth'.

Solar radiation is a boundary type used with the RDI hydrological model, when snowmelt modelling is included and when the 'Use solar radiation' option is active. The item type defined in the selected time series must be 'Sun radiation'.

Degree-day coefficient is a boundary type used with the RDI hydrological model, when snowmelt modelling is included and when the 'Degree-day coefficient type' is defined as varying in time. The item type defined in the selected time series must be 'Melting coefficient'.

Catchment irrigation is a boundary type used with the RDI hydrological model, when the 'Include irrigation' option is active in the RDI parameters. The item type defined in the selected time series must be 'Irrigation'.

The following chapters describes the work-flow to define a boundary condition for a catchment.

As the first action after inserting a new boundary conditions, a proper name (ID) must be specified. It is recommended to use a descriptive ID.

7.1.2 Network boundary conditions

In MIKE+ there is a single boundary condition editor for all types of network boundary conditions.

The following types of boundary conditions are available for collection system and/or river networks:

- Load point discharge: this boundary condition is linked to a load point, and the inflow is specified in the 'Load points' editor
- Load point discharge per PE: this boundary condition is linked to a load point, and the inflow and the number of person equivalents are specified in the 'Load points' editor
- Inflow to node: a discharge boundary condition applied to a node from the collection system network
- Inflow to link: a discharge boundary condition applied to a link from the collection system network
- Inflow from result file: a discharge boundary condition where the discharge time series are obtained from a result file from a previous simulation. This is usually a .res1d file from a Rainfall-runoff simulation. It can alternatively be a .dfs0 time series file, or a SWMM result file (regular .out file or outflow interface file in .txt format).
- Outlet water level: a water level boundary condition applied at an outlet of a collection system network. Only one water level boundary condition is allowed at each network outlet. If no boundary condition is specified for an outlet on a collection system network, a free outflow is assumed.

- Exfiltration from node: an exfiltration boundary condition applied to a node from the collection system network. . It is defined as an infiltration rate. For a manhole, the infiltration area considered during the simulation is the circular area of the manhole, and for a basin it is the surface area of the water surface, which varies in time.
- Exfiltration from link: an exfiltration boundary condition applied to a link from the collection system network, expressed as a flow per unit of length along the network.
- Open inflow to river: a discharge boundary condition applied at the end of a river
- Source inflow to river: a discharge boundary condition applied along a river. It can either be a point source applied at a point location, or a distributed source where the specified discharge is distributed evenly between a start point and an end point.
- River water level: a water level boundary condition applied at the end of a river
- River Q/h relation: the boundary condition at the end of a river is defined by a Q/h relationship, controlling the water level at the boundary as a function of the outflow.
- Free river outflow: When the free outflow type is applied, the smallest of the critical and the natural depth is applied at the boundary
- Closed river boundary: the closed boundary type is used at river end points where a zero-flux condition across the boundary is applicable.
- Inflow to storage: a discharge boundary condition applied to a storage node from the river network.
- Network rainfall: rainfall boundary conditions are specified in river reaches where the inflow of rainfall on the channel is to be represented. The rain intensity applies to the water surface, i.e. the wet part of the river area (hence excluding dry areas of the cross sections). Rainfall can be specified globally (spatial extent = All) or as a distributed source per branch. When applied globally, the network rainfall boundary condition can be defined using a constant value, a uniform time series (from .dfs0 file) or using a spatially- and time-varying file (.dfs2 file).
- Network evaporation: evaporation boundary conditions are specified in river reaches where loss of water from evaporation from the channel is to be represented. Evaporation can be specified globally (spatial extent = All) or as a distributed source per branch. When applied globally, the network evaporation boundary condition can be defined using a constant value, a uniform time series (from .dfs0 file) or using a spatially- and time-varying file (.dfs2 file).

- Wind friction: wind friction on the water surface is accounted for by including wind shear stress in the simulation. When wind friction is applied, at least one wind friction boundary condition must apply to the entire network (spatial extent = All), to define a default wind field. Extra local wind boundary conditions may be added with local values and will take priority over the global wind field. When a wind friction boundary condition is included, a related 'Wind scaling factors' editor appears in the tree view, to specify local scale factors.
- Groundwater leakage: the groundwater leakage boundary condition represents an additional loss of water from the river to the ground-water. When the groundwater leakage boundary condition is included, a related 'Leakage coefficients' editor appears in the tree view, to specify the coefficients (global and local coefficients) controlling the rate of the leakage. The groundwater leakage boundary condition can be defined using a constant value, a uniform time series (from .dfs0 file) or using a spatially- and time-varying file (.dfs2 file).

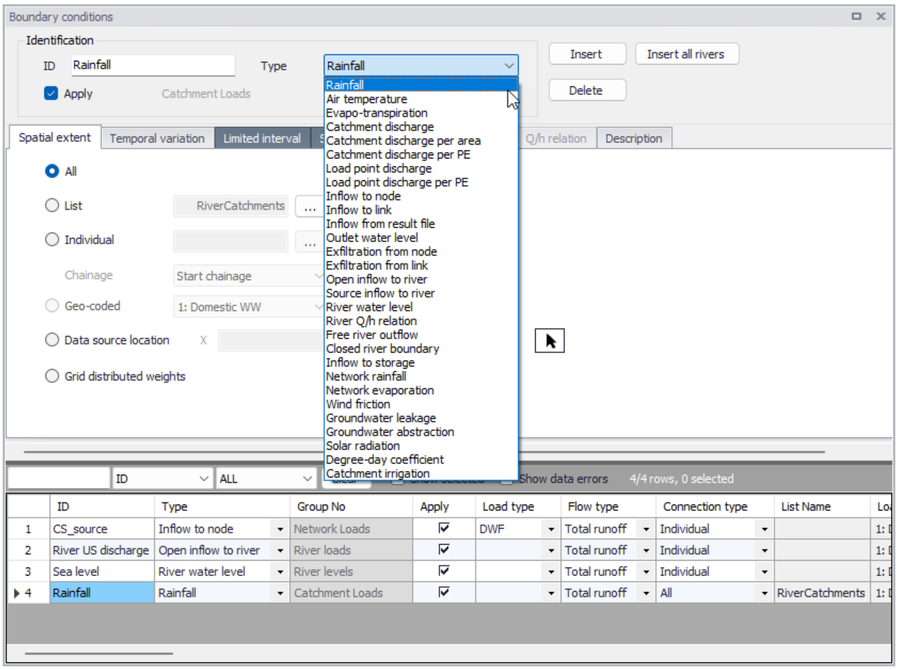


Figure 7.1 Hydraulic Boundary Conditions Dialog

Defining HD Boundary conditions

Below are steps for defining basic HD boundary conditions in MIKE+.

1. Insert a boundary condition using the 'Insert' button on the upper right part of the Boundary Conditions editor. It is recommended to specify a

descriptive ID for the new boundary condition.

2. Define the boundary condition type via the Type dropdown list. The selected type influences subsequent data requirements for the boundary condition presented in the various tab pages in the editor.
3. Define the location or spatial extent for the boundary condition in the Spatial Extent tab page.
4. Define the values for the boundary condition item in the Temporal Variation tab page of the editor.
5. Ensure the 'Apply boundary' checkbox is ticked if the boundary condition shall be used in subsequent simulations.

Special options and data requirements vary according boundary condition type defined, and so additional parameters/steps may be needed for other specific boundary types.

The subsequent sections describe the various options available in the various tab pages of the Boundary Conditions editor in greater detail.

Note: For river networks, boundary conditions are mandatory at river ends not connected to other parts of the modelled network. Rather than inserting and locating these boundary conditions manually, clicking the "Insert all rivers" button will add all the open boundaries of the river network to the boundaries table simultaneously. The created boundary conditions will be correctly located at the open ends, with a pre-defined boundary type: you then need to review the applied type, and specify the temporal variation of each boundary condition. If a boundary condition has already been defined at an open end, it will be kept unchanged when using the button.

7.1.3 Spatial extent

The **Spatial Extent** enables the modeller to define the distribution of the boundary conditions. The distribution ranges from ‘All’ (e.g. all catchments or all rivers, depending on the boundary condition type), Individual (specified locations), ‘List’ of a selection of elements (pipes, nodes, rivers or catchment depending on the boundary condition type), ‘Geo-coded’ locations, ‘Data source location’ and “Grid distributed weights”.

The application of ‘All can be applied to applied to catchment boundaries (‘Catchment discharge’, ‘Catchment discharge per area’ and ‘Catchment discharge per PE’). It can also be used for meteorological boundaries applied to catchments (‘Rainfall’, ‘Air Temperature’ and ‘Evapotranspiration’) or to the network (‘Network rainfall’, ‘Network evaporation’, and ‘Wind friction’).

Lists can be used with various types of boundary conditions to assign the condition to multiple items at once. **Lists** apply for a set of e.g. catchments, rivers, nodes or links, contained in a defined selection. The selection must first be saved in the **Selection Manager**, and then selected in the boundary condition definition.

’**Individual**’ applies for a single item (catchment, river, node or pipe), and requires a specification of the item ID.

‘Geo-coded’ option applies to ‘Load point discharge’ and ‘Load point discharge per PE’ and is used to assignr **Load points** (typically DWF) geo-coded to the network nodes. User can select which one of the geo-coded load categories applies for this boundary condition.

Further information regarding the Geo-coding of loads is discussed specifically under the ‘Load Points’ help section.

'**Data source location**' allows assigning the XY coordinates of a rainfall gauge to the measured time series. With this option, each catchment will be assigned the closest rainfall time series. If one rainfall boundary condition is defined by 'Data source location', then all other catchment rainfall boundary conditions must also be defined with the same location type.

‘**Grid distributed weights**’ can be used to assign spatially varying rainfall data. The application of 2D radar data as input for runoff computations requires that rainfall dagta are defined in DHI’s *dfs2* format files. MIKE 1D will automatically distribute the rainfall to individual catchments and apply it to the runoff computation.

Note: If a load is connected to a link, it will be uniformly distributed over all computational grid H-points along the link.

7.1.4 Temporal variation

The temporal variation describes how the boundary condition value changes in time. It can either use a constant value, a cyclic value or a time series.

Depending on the boundary condition type, the units are representative units and automatically updated.

If a cyclic temporal variation is to be applied, a pattern is also needed. The help section for ‘Repetitive profiles’ provides a more detailed description of the design and application of patterns and profiles.

The Time-series temporal variation requires a compatible time series file for input. The standard time-series file format is ‘.DFS0’. Press the '...' button to create a new time series. Press F1 from the time series editor to get more detailed information on the various settings of the time series.

For 'Inflow from result file' boundary conditions, the time series file can also be a .res1d file from a previous Rainfall-runoff simulation. It can alternatively be a .dfs0 time series file, or a SWMM result file (regular .out file or outflow interface file in .txt format). When selecting a SWMM result file, a connection type must also be selected:

- Matching nodes names: time series from the SWMM result file will be applied as boundary conditions to nodes with the same name in the MIKE 1D simulation.
- Matching catchment and node names: catchment results from the SWMM result file will be applied as boundary conditions to MIKE 1D nodes, where the MIKE 1D node has the same name as the SWMM catchment in the result file.
- rom catchment connections: the connection between SWMM catchment runoff results and MIKE 1D nodes is defined in the 'Catchment connections' editor.

For the Grid distributed weights’ spatial extent, the input file must be a time-varying 2D grid file with dfs2 format.

For 'Load point discharge' and 'Load point discharge per PE' boundary conditions, the condition can be either constant or cyclic, and the value is specified in the 'Load points' editor.

For 'Wind friction' boundary conditions, a temporal variation must be specified for both the wind velocity and the wind direction. They are specified in their respective tabs and can both be either constant or varying in time. The direction is expressed in degrees in clockwise direction from north.

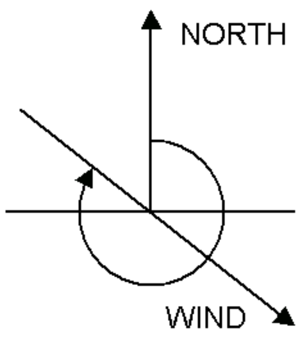


Figure 7.2 Definition of Wind field direction

7.1.5 Limited interval

The limited interval function allows the a specific time definition of when the boundary item is applied.

If a limited validity interval is to be applied to a boundary, then check ‘Use limited validity’. It is then necessary to define the start and end date & time of the validity period.

7.1.6 Scaling factor

The Scaling factor tab is used to apply a scale factor for the hydraulic load. This can be used to add a climate factor.

For 'Wind friction' boundary conditions, the scale factor for the wind velocity can be specified in the ‘Wind scaling factors’ editor.

7.1.7 Distributed weights

The 'Distributed weights' tab is used to compute and review the weights, used in combination with rainfall boundary conditions defined with the spatial extent type 'Grid distributed weights' (using input 2D radar data) or 'Data source location' (locating time series from their rain gauge coordinates). From this tab it is possible to check the distributed weights per catchment and adjust if desired.

With 'Grid distributed weights' boundaries, all catchments covered by the grid will be presented in the table. Catchments might be presented in several entries if multiple grid points cover its entirety.

There are two text fields which will display the most important data for the loaded boundary condition selected; grid file name and selected item name.

The table shows the connected grid cells (coordinates i and j) and the calculated weight percentage covering the catchment per grid cell.These percentages fractions should add up to a 100 per cent for each catchment.

The table is dynamic and shows the selected boundary condition data. It allows to insert new connecting grid cell by right clicking, or by typing a new catchment name in the last row.

The ‘Compute’ button allows to compute the weights for either all catchments or a selection of catchments, as well as for the new added grid points (not updated).

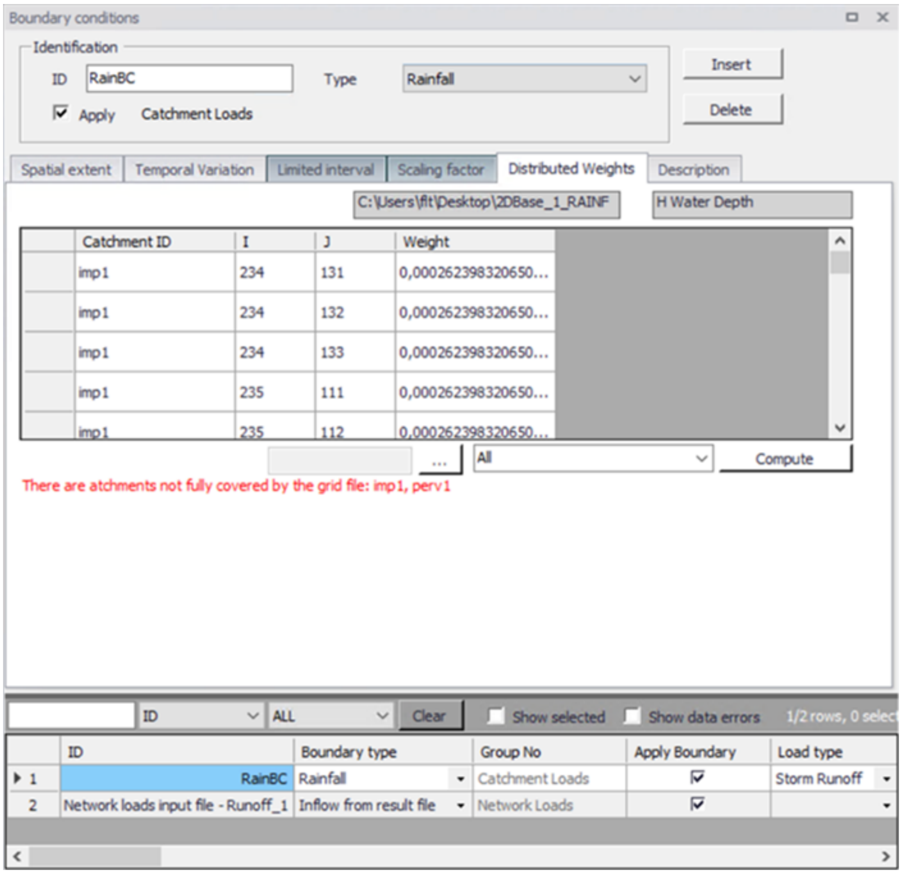


Figure 7.3 The 'Distributed weights' tab for a grid-distributed weights boundary condition

With 'Data source location' boundaries, the table is used only when the rainfall distribution type selected in the 'Simulation setup' editor (for the active simulation ID) is either 'Thiessen polygons weighting' or 'Inverse distance weighting'. The table will list all catchments having a positive weight associated to the active rainfall boundary conditions (catchments outside the Thiessen polygon surrounding the rain station, or beyond the search radius for the interpolation, are not listed in the table).

Text information above the table displays the weighting method selected in the active simulation ID, as well as the time series file name and item name used in the current boundary condition.

The 'Compute' button computes the weights for all the rainfall boundary conditions. The table can then be reviewed and updated if necessary. New rows can be added to the table by right-clicking, or by typing a new catchment name in the last row. If the tables of weights are left empty, the weights for all the rainfall boundary conditions will be computed when starting the simulation.

The 'Weights overview' button opens another table providing the complete list of weights, for each catchment and for each rainfall station.

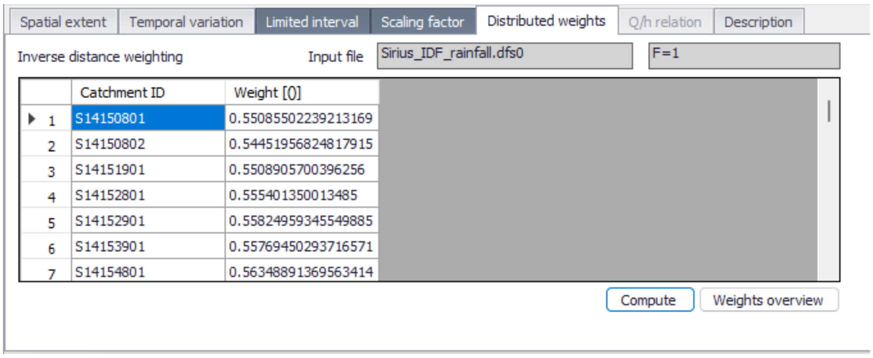


Figure 7.4 The 'Distributed weights' tab for a 'Data source location' boundary condition

During the simulation with 'Data source location' boundaries, the interpolated rain time series $\hat{R}$ related to a catchment C is a weighted time series obtained from the various boundary time series Ts_i :

$$\hat{R} = W_1 \cdot Ts_1 + W_2 \cdot Ts_2 + \ldots + W_p \cdot Ts_p \tag{7.1}$$

More details follows for the two available methods.

Inverse Distance Interpolation Method

Consider a plane with XY coordinates, the Catchment centroid coordinate (X_C, Y_C) and the sets of rain station measurements: (Mx_1, My_1) corresponding to the time series Ts_1, (Mx_2, My_2) corresponding to the time series Ts_2,…, (Mx_p, My_p) corresponding to the time series Ts_p. Then the weight of each time series Ts_j is simply weighted proportionally to one over the square distance $D(C, Ts_j)$ between the catchment center (X_C, Y_C) and the measurement location (Mx_j, My_j):

$$W'_j = \frac{1}{D(C, Ts_j)^2} \tag{7.2}$$

After computing all the weights for each measurement station, they need to be normalized so they sum up to 1:

$$W_j = \frac{W'_j}{\Sigma_j W'_j} \tag{7.3}$$

The normalized weights are used to compute the interpolated rain of the catchments.

Thiessen Interpolation Method

The Thiessen (or Voronoi) interpolation is another simple way to decide the domain of influence of the measurements towards the catchments. A Thiessen polygon Tp_i is defined for each geolocated measurement time series Ts_i, and it is roughly defined as the set of points in the plane for which the closest measurement location is the one defined by the geolocation of Ts_i. Therefore, the edges of the Thiessen polygons correspond to equidistant lines between neighbor measurement locations. The overlapping of all these equidistant lines between all neighbor time series will define a set of polygons Tp_1, Ts_2, …, Ts_p.

Consider the n Catchments C_1, C_2, …, C_n and the p measurement stations M_1, M_2, …, M_p. In the picture below, as example, 4 catchments and 6 stations:

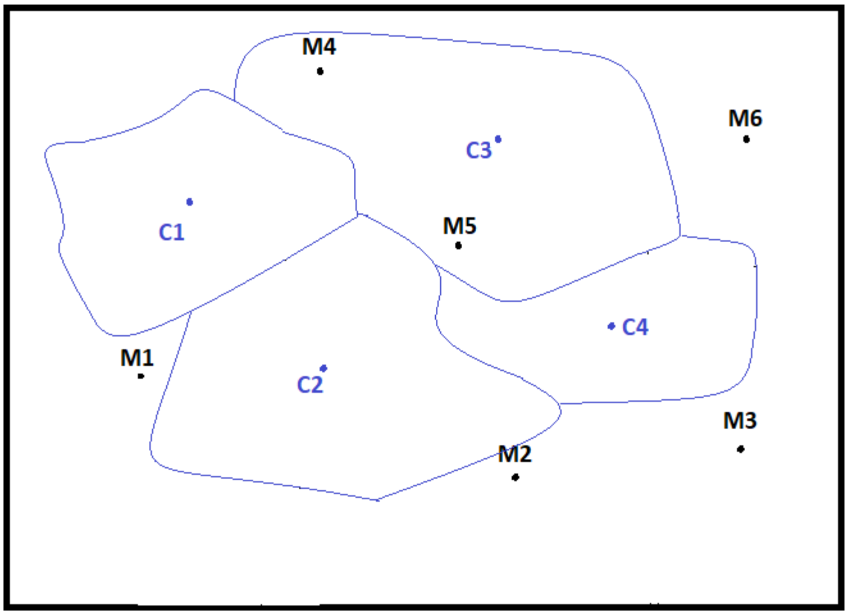


Figure 7.5 Example of catchments with geolocated rainfall stations

The stations M_1, M_2, …, M_p and the surrounding box divide the plane into Thiessen/Voronoi polygons according to the distances to stations, as shown on the next figure.

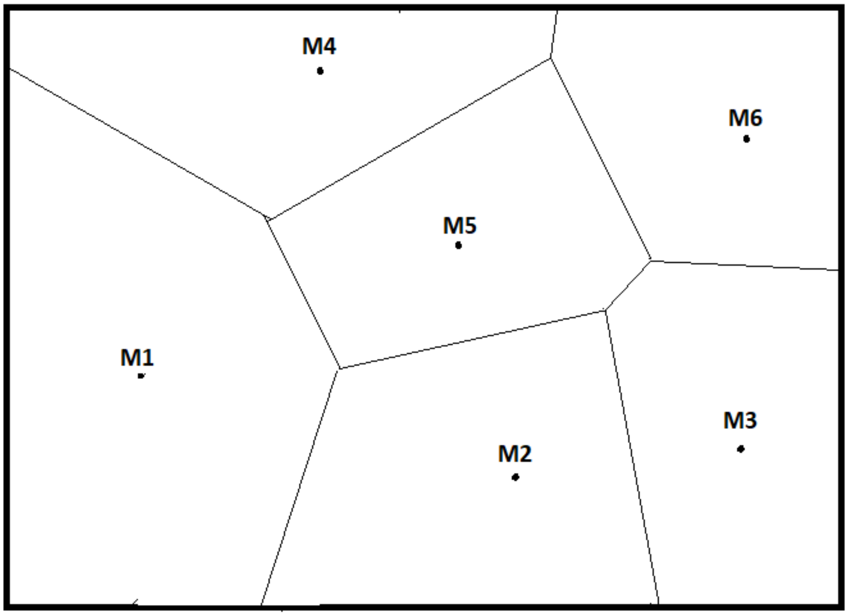


Figure 7.6 Thiessen polygons created from rainfall stations

The superposition of the Thiessen polygons divides the plane into areas where the closest station is the governing measurement which is consider for the polygon.

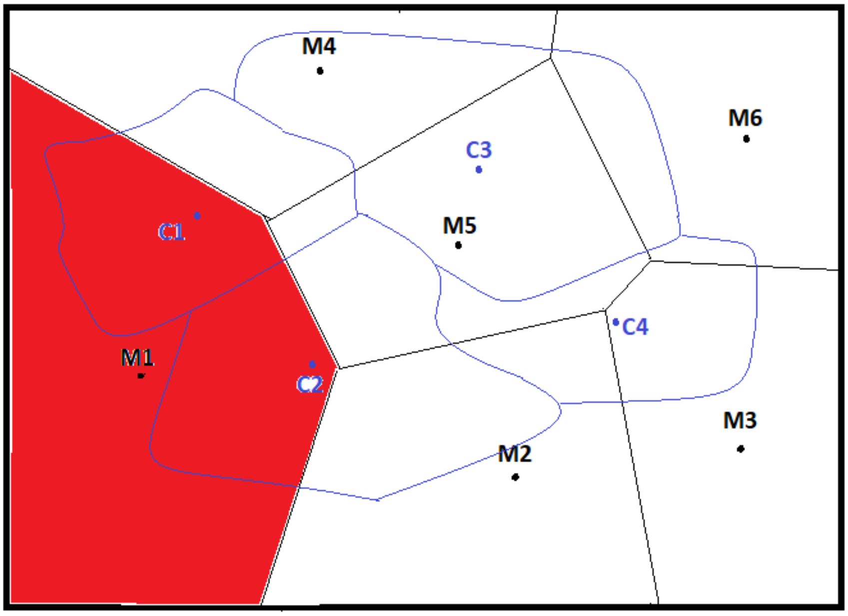


Figure 7.7 Thiessen polygon related to rainfall measurement station M1, in red

The weight related to catchment *i* and station j is the ratio between the Intersection of the catchment *i* and the Thiessen polygon j:

$$W_{ij} = \frac{\mathrm{Area}(C_i \cap P_j)}{\mathrm{Area}(C_i)} \tag{7.4}$$

For the case of station 1 and catchment 1 the weight is the ration between the green area, and the sum of the green and brown areas:

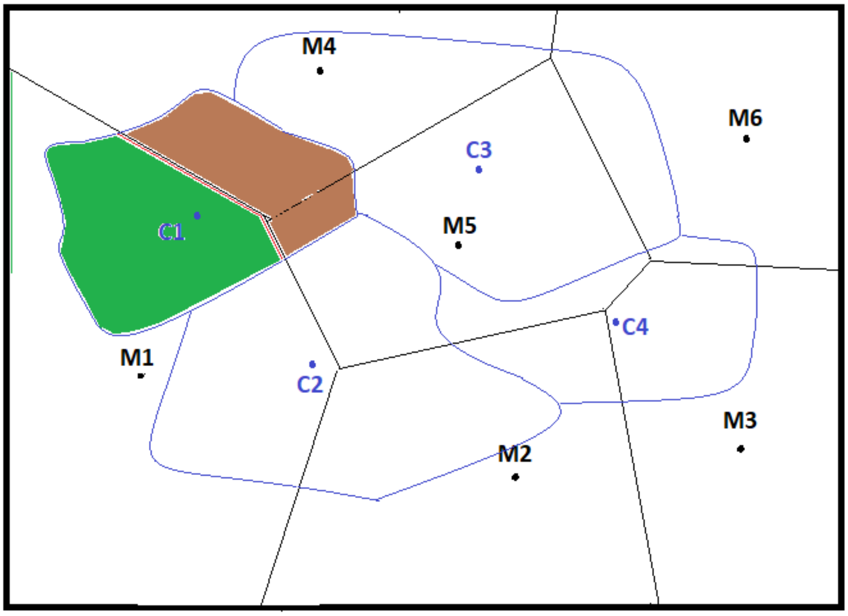


Figure 7.8 Thiessen weight calculation for catchment C1

7.1.8 Q/h Relation

The Q/h Relation tab page is relevant for 'River Q/h Relation' boundary condition types. It is where the corresponding Q/h relation table for a 'River Q/h Relation' boundary condition is defined (Figure 7.9).

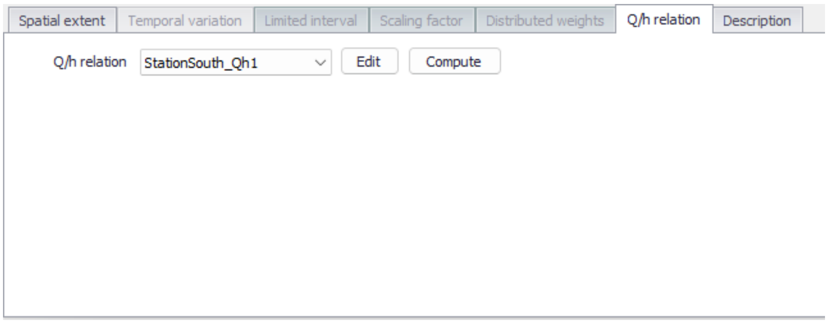


Figure 7.9 The Q/h Relation tab page relevant for River Q/h Relation boundary condition types

Select the appropriate Q/h relation table for the boundary condition from the '**Q/h Relation**' dropdown list.

Use the '**Edit**' button to access the 'Curves and Relations' editor, wherein one may edit or create various types of tabular data sets for the project.

Use the 'Compute' button to automatically compute the Q/h relation using the characteristics of the cross section located at the boundary location (a cross section must exist at this location before this tool can be used). The button opens the window below to control how to compute the Q/h relation.

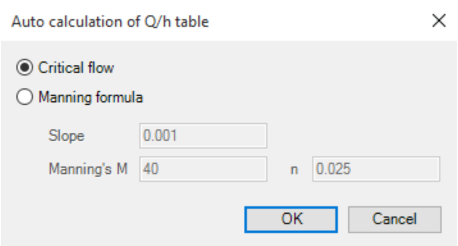


Figure 7.10 Automatic calculation of Q/h relations

H values in the Q/h relation are extracted from the processed data of the 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 value (Manning (M) or Manning (n)) must be specified.

In case the critical flow formula is used, Q is calculated from:

$$Q(h) = A(h)\sqrt{g\frac{A(h)}{W(h)}} \tag{7.5}$$

In case of uniform flow by Manning's formula, Q is calculated from:

$$Q(h) = Conv(h)\sqrt{I} \tag{7.6}$$

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

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:

$$Conv(h) = M_{QhTool} \cdot A(h) \cdot R^{2/3} \tag{7.7}$$

If the resistance type is set to Manning's M:

$$Conv(h) = M(h) \cdot A(h) \cdot R^{2/3} \tag{7.8}$$

If the resistance type is set to Manning's n:

$$Conv(h) = \frac{1}{n(h)} \cdot A(h) \cdot R^{2/3} \tag{7.9}$$

If the resistance type is set to Chezy or Darcy-Weisbach:

$$Conv(h) = C(h) \cdot A(h) \cdot \sqrt{R(h)} \tag{7.10}$$

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.

7.1.9 Description

It is possible to provide a description of the boundary condition and load type using the Description tab.

7.2 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).

The 'Wind scaling factors' editor will appear in the Setup tree view, if at least one Wind friction boundary condition exists.

Default factor

A default wind scale factor is defined at the top of the editor. It is applied throughout the channels network unless local values have been defined.

Local factors

By using local wind scaling factors, it is possible to adjust the wind shear stress to reflect local variations within the domain. Local values overwrite the default value. Local values are applied at specific locations defined by the channel ID and a chainage value.

Local values are added or deleted using the 'Insert' or 'Delete' buttons above the table.

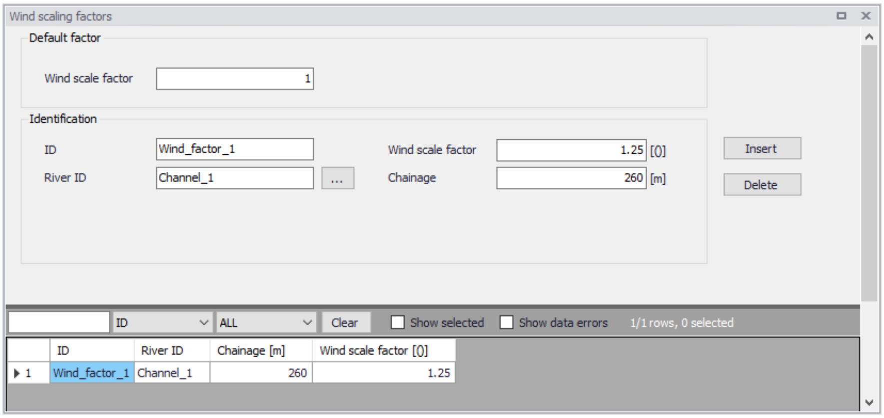


Figure 7.11 The Wind scaling factors editor

Note: If two or more local values are defined on the same channel, the values in intermediate calculation points will be calculated by linear interpolation.

7.3 Leakage coefficients

When the groundwater leakage boundary condition is included, the coefficients (global and local coefficients) controlling the rate of the leakage must be specified in the 'Leakage coefficients' editor.

Leakage coefficients may be defined by a default (global) leakage coefficient or by local coefficients that override the default value at specific locations. Hence, if leakage is to be included only at some locations, then a default value of 0 must be defined and the local reaches where groundwater leakage takes place must be defined in the Local coefficients table.

The Leakage coefficients' editor will appear in the Setup tree view, when a 'Groundwater leakage' boundary condition exists.

Default coefficient

A default leakage coefficient is defined at the top of the editor. It is applied throughout the channels network unless local values have been defined.

Local coefficients

By using local leakage coefficients, it is possible to adjust the flow towards the groundwater, to reflect local variations of soil characteristics within the domain. Local values overwrite the default value. Local values are applied at specific locations defined by the channel ID and a chainage value.

Local values are added or deleted using the 'Insert' or 'Delete' buttons above the table.

Note: If two or more local values are defined in the same river branch, the intermediate value(s) will be calculated by linear interpolation.

7.4 Water Quality Boundary Condition Properties

This editor is only available if Water Quality module is active. Water can transport other material such as pollutants, heat, organisms, as well as other properties, such as pH, conductivity, turbidity, etc. The transport of these properties may be modelled in MIKE+, requiring definition of these WQ properties associated with water load boundary conditions. WQ boundary properties for boundary conditions for CS and River network models are defined in the WQ Boundary Properties editor (Figure 7.12).

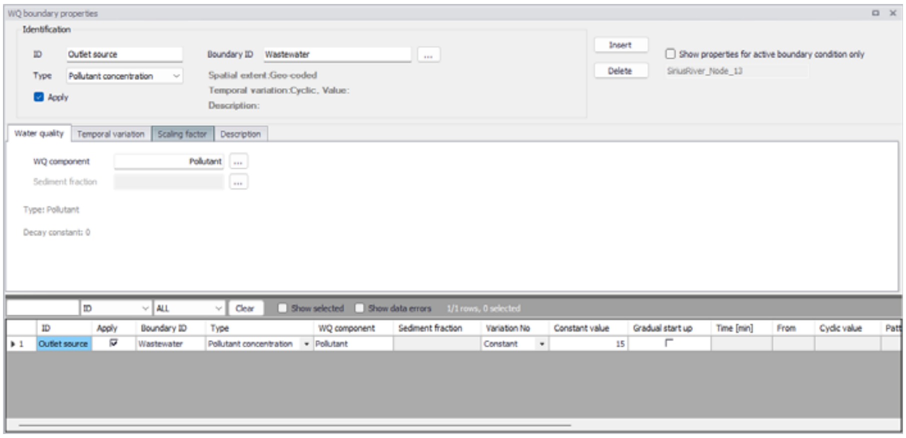


Figure 7.12 The WQ Boundary Properties editor

7.4.1 Identification

The top section of the WQ Boundary Properties editor contains the Identification group box where one defines general properties of WQ boundary setups (Figure 7.13).

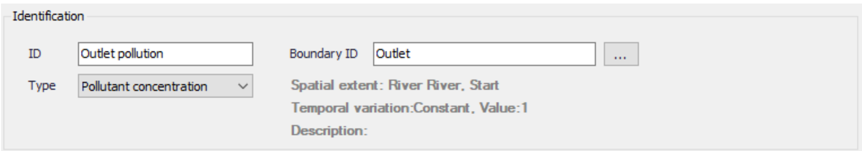
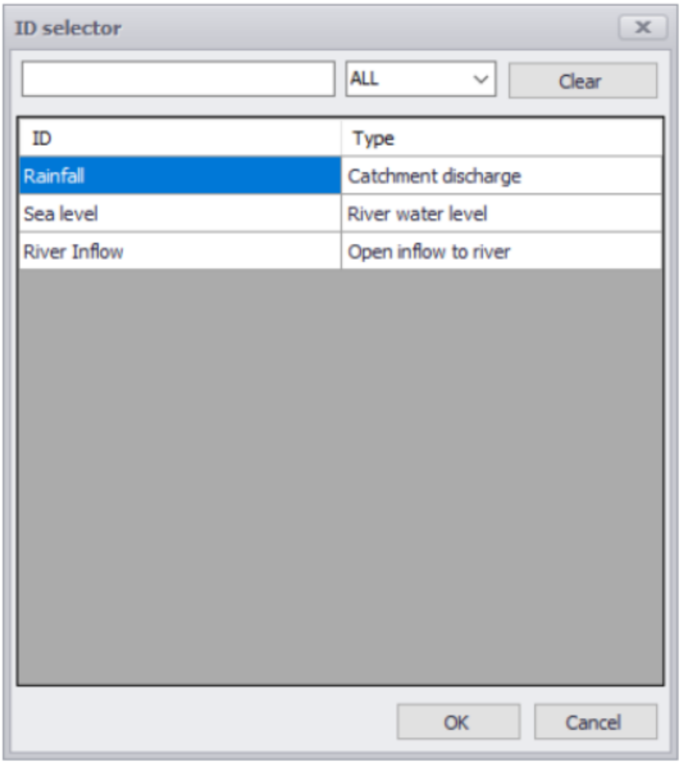


Figure 7.13 The Identification group box in the WQ Boundary Properties editor

To define a WQ boundary property in the project:

1. Use the 'Insert' button to add a new WQ boundary property to the editor.
2. One may edit the ID for the newly-added WQ boundary property to make it more descriptive.
3. Associate the WQ property to an HD boundary condition via the 'Boundary ID' input box. Click on the ellipsis button to access the Selector window, which lists existing HD boundary conditions that may be associated with WQ properties.

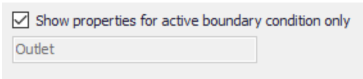


A WQ boundary property must be connected to a hydraulic boundary condition. A hydraulic boundary condition can be used with a hydraulic load set to 0 to only add a Pollutant Load or Sediment Load.

4. Define the WQ property Type from the drop-down list of possible types, which is associated with the WQ Component defined in the Water Quality tab page.
5. Define the WQ Component for the WQ boundary property in the Water Quality tab page of the editor.
6. Furnish information in the other tab pages of the editor as needed.

The top part of the editor also presents options for filtering the WQ boundary properties displayed in the editor.

Use the 'Show properties for active boundary condition only' option to display only the WQ properties associated with the active HD boundary condition in the Boundary Conditions editor. The ID for the active HD boundary condition is displayed in the text box below the option. If the option is inactive, all WQ boundary properties are shown in the editor



The available types of WQ properties are:

Pollutant concentration or load

The pollutant will be added with the concentration or load specified on the Temporal Variation tab.

Microorganism

The component is added as a micro-organism load, used for bacteria and pathogens.

Temperature

The component represents the temperature, the unit will be adjusted according to the unit system specified for the model.

pH

The component is related to the water pH, a measure of how acid or basic the water is.

Salinity

The component will reflects the measure of salts dissolved in water.,

Water Age

This option allows for water with specified age to enter the network.

Water Blend

This option allows for water with specified mixing ratio to enter the network.

Other Type of AD component

The property type 'Other' can be used in combination with WQ components defined with the type 'Other' in the 'WQ components' editor. This type is to be used for component types not available in the predefined list of types, in the 'WQ components' editor.

Sediment concentration

The sediments will be added with the concentration (mass/volume) specified on the Temporal Variation tab.

The simulation engine will multiply the water discharge at the boundary location to define the total mass entering the network at the boundary location.

Sediment load

The sediments will be added with the load (volume/time) specified on the Temporal Variation tab.

The specified load is directly applied in the simulation and the water discharge at the boundary location is hence not used to determine the mass inflow to the model.

Bed level

This property describes the elevation of the bed (surface of the sediments layers) at the boundary. For this type of boundary property, no sediment fraction needs to be specified (the boundary property applies to the boundary as a whole, no matter how sediment fractions are distributed at this location).

This type of property is only supported for open boundaries (network ends), but not for e.g. sources along the network.

Bed level change

This property describes the relative change of bed level (surface of the sediments layers) at the boundary. For this type of boundary property, no sediment fraction needs to be specified (the boundary property applies to the boundary as a whole, no matter how sediment fractions are distributed at this location).

This type of property is only supported for open boundaries (network ends), but not for e.g. sources along the network.

Mixing

This option can be applied 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 enabling the mixing property for a boundary condition, when the boundary becomes an inflow, the concentration at the boundary is not null but is estimated as a function of the concentration back in time when the boundary was an outflow. This is estimated using the halving time specified in the 'Mixing' tab,

The 'Apply' check box allows the user to toggle the Active status of the WQ boundary on and off. The simulations will omit all WQ boundary conditions that are not active.

7.4.2 Water quality

Define WQ Components associated with WQ boundary properties in the Water Quality tab page of the editor (Figure 7.14).

Possible components are listed in the ID Selector window accessed via the ellipsis button. The dialog lists components that have been defined in the WQ Components editor.

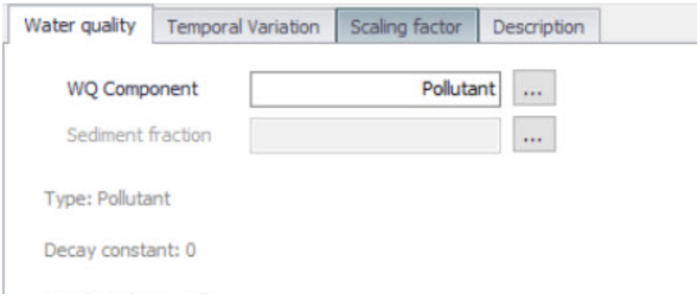


Figure 7.14 The Water quality tab in the Water Quality editor.

The Sediment Fraction setup associated with a WQ boundary property used in Sediment Transport computations is also defined in the Water Quality tab page. The ellipsis button presents a selector window listing sediment fraction definitions in the Sediment Fractions editor.

7.4.3 Temporal Variation

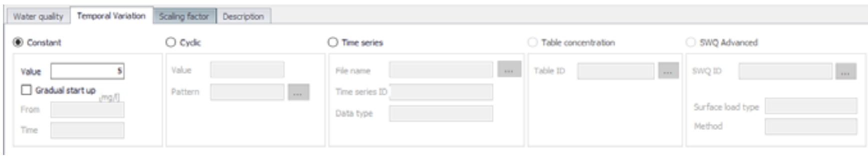


Figure 7.15 Temporal variation tab in Water Quality editor.

A water quality boundary condition can have a different temporal variation than the hydraulic boundary condition it is connected to. It can be constant, cyclic, a time series temporal variation, table concentration and SWQ.

Dependent on the water quality type, the units are representative and automatically updated.

If a cyclic temporal variation is to be applied, a pattern is also needed.

The Time-series temporal variation requires a compatible time series file for input. The standard time-series file format is '.DFS0'.

The table concentration allows to specify a variation of pollutant loads based on a runoff -concentration varying relation.

The temporal variation can also be defined through a Storm Water Quality method, build-up/ wash-off or EMC, for more details please refer to the Water Quality Module.

7.4.4 Scaling factor

The scaling factor allows to add a global factor to the load. The Default value is 1.

7.4.5 Mixing

This option can be applied 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 concentration at the boundary is adjusted according to:

$$C = C_{bf} + (C_{out} - C_{bf})e^{-t_{mix} \cdot k_{mix}} \tag{7.11}$$

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

t_{mix} is the time since the boundary became an inflow

k_{mix} is the mixing coefficient computed as a function of the halving time: k_{mix}=ln(2)/$T_{halving}$

$T_{halving}$ is the user-defined time specified in the 'Mixing' tab.

When outflows occur at the boundary, the boundary condition is defined as:

$$\frac{\partial^2 C}{\partial x^2} = 0 \tag{7.12}$$

7.4.6 Description

The Description tab allows the user to add text data to the specific WQ boundary condition.

7.5 Load Point

The allocation of geographically determined load points to the nodes and, or links of a collection system model is defined thtrough the Load Points section. The load points are geographical point features, typically representing a categorised wastewater load, such as domestic wastewater, commercial / industrial wastewater and infiltration.

It is possible to modify the additional categories to provide a tailored coding system to replicate the water loadings of the catchment.

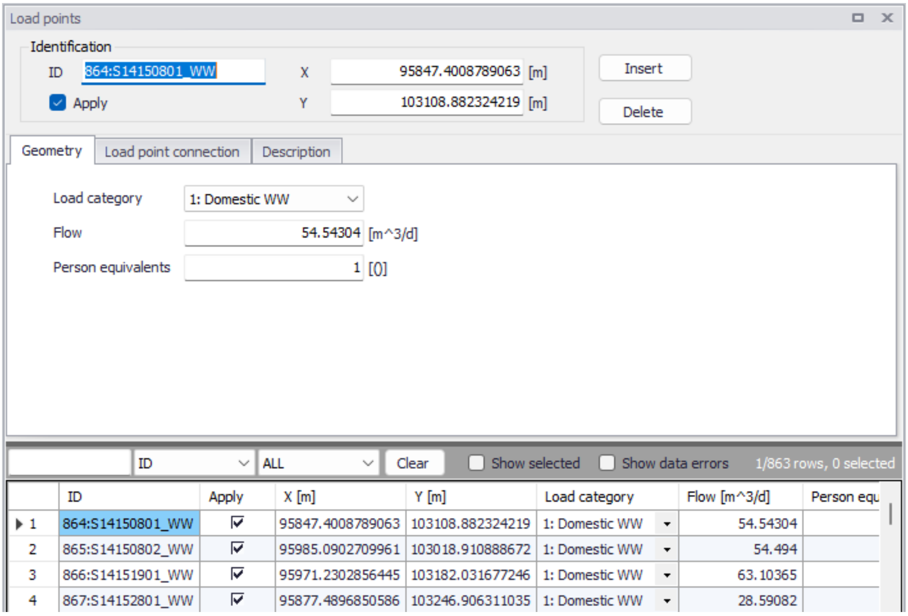


Figure 7.16 Load points grid

The 'Apply' check box allows the user to toggle the Active status of the load point on and off. The simulations will omit all load points that are not active.

There are 3 sub-tabs within the ‘Load point’ tab, ‘Geometry’, ‘Load point connection’ and Description.

7.5.1 Geometry

Within the ‘Geometry’ tab, the load can be categorised and its flow and number of person equivalents can be specified.

When the load is associated to a 'Load point discharge' boundary condition, the boundary condition value is the flow value specified for the load. When the load is associated to a 'Load point discharge per PE' boundary condition, the boundary condition value is the flow value multiplied by the number of person equivalents specified for the load.

7.5.2 Load point connection

Load points can be connected to the model network through nodes and links, each load will be coupled to a single network element.

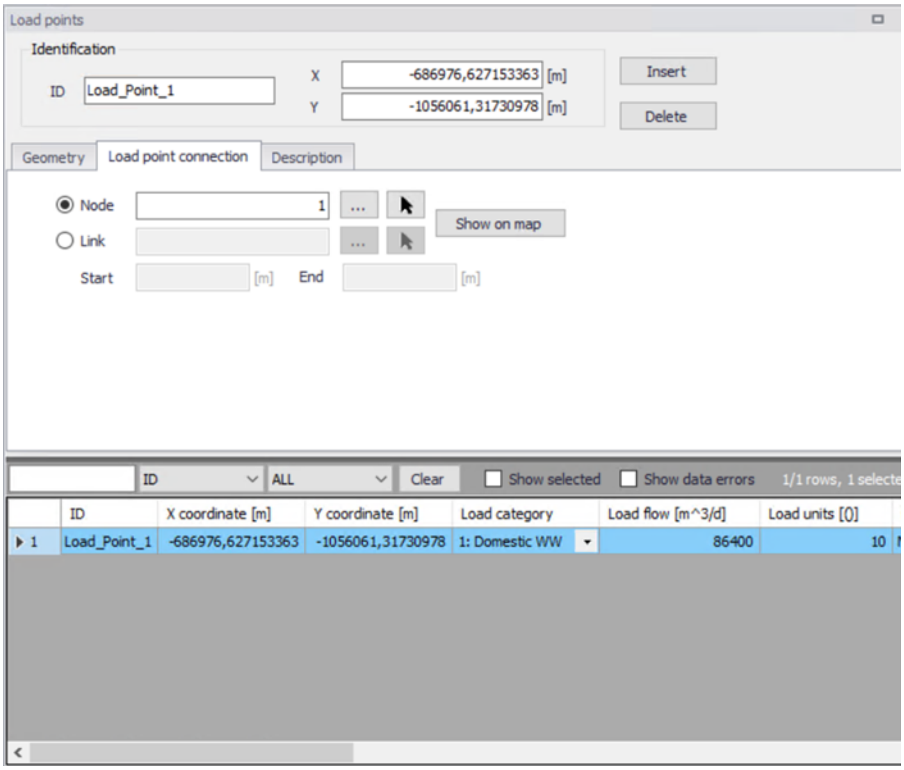


Figure 7.17 Load point connection

The connections can be made via the asset ID Selector (grid selection) or by map selection.

If using a link connection for the Load point, it is possible to defined the chainage start and end points.

7.5.3 Description

The description tab enables the modeller to record load point information.

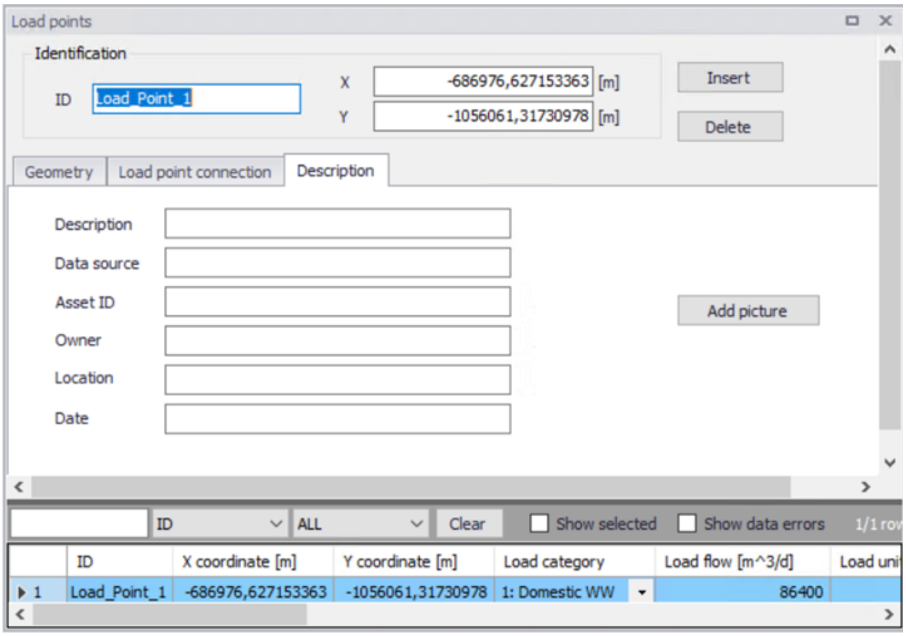


Figure 7.18 Load point description tab

7.6 Repetitive Profiles

The 'Repetitive Profile Editors' can be used for generating dimensionless, cyclic time series ('repetitive profiles') with a fixed time resolution of one hour. E.g., it can be applied for defining diurnal profiles that can describe the Dry Weather Flow (DWF) from a specific catchment.An unlimited number of repetitive profiles can be applied to different groups of catchments. For example, an industrial area will have a different dry weather flow (DWF) description to rural or residential area. DWF profiling may also vary according to the time of week and holidays.

There are four discrete parts forming the repetitive profiles:

- Diurnal patterns - used for specifying diurnal patterns
- Cyclic profiles - used for coupling of individual diurnal patterns with profile calendar definition
- Profiles calendar - used for coupling of individual diurnal patterns with profiles calendar definition
- Special days - used for specifying days that are to be considered as exceptions to the calendar (e.g. the 1st of January)

7.6.1 Diurnal patterns

A diurnal pattern consists of the pattern ID and non-dimensional coefficients (multipliers) varying with the time of the day.

A Delta T (time step) is also required, if the time information is to be automatically distributed for the entire day with a constant interval.

Figure 7.19 shows an example of a diurnal pattern. A new pattern is created by using the ‘Insert’ button. The common workflow is then to define the Delta value (minutes). The Delta value represents the interval distribution for the pattern. The ‘Distribute’ button creates the intervals determined by the Delta value assigned. For example, a Delta value of 60min returns 24 hourly intervals, a Delta value of 240 will return 6 x 4 hour intervals.The dimensionless coefficient is inserted under the ‘Multiplier’ column.

The ‘Normalize’ button adjusts the multiplier values so that their average value is equal to 1, but the relative weightings are maintained.

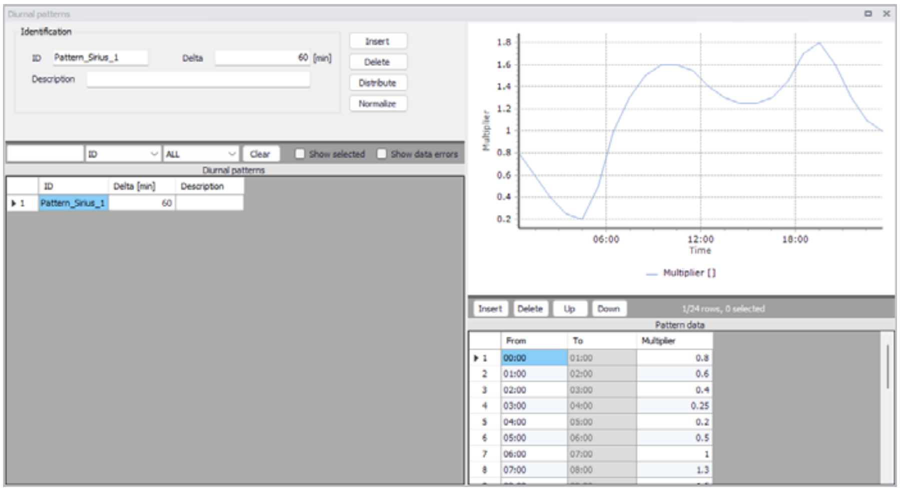


Figure 7.19 Diurnal pattern editor

Patterns times can alternatively be created or edited using the 'Insert' and 'Delete' button below the plot, instead of using the 'Distribute' button. The pattern's time steps don't necessarily have to be equidistant.

The pattern's data table shows the 'From' and 'To' time of the day relating to the corresponding multiplier. Only the 'From' time needs to be specified. The 'To' time is automatically filled with information from the next time step.

Please refer to the 'Cyclic profiles' options for more information about how the pattern is applied in the simulation.

7.6.2 Cyclic profiles

A cyclic profile links a diurnal pattern with the relevant calendar definition.

Each cyclic profile requires an ID with a defined ‘Diurnal Pattern’ and ‘Calendar’. In addition, the profile requires input regarding the interpolation of the pattern, either by ‘linear interpolation’, or ‘no interpolation’. No interpolation will apply a step function between the diurnal pattern values.

Figure 7.20 shows the Cyclic Profile editor. The visualisation shows at a glance how the profile will look (monthly or weekly) with the diurnal pattern coupled to the selected profile calendar. The Cyclic Profile example shows a ‘Residential’ diurnal profile with a ‘Weekday’ profile calendar.

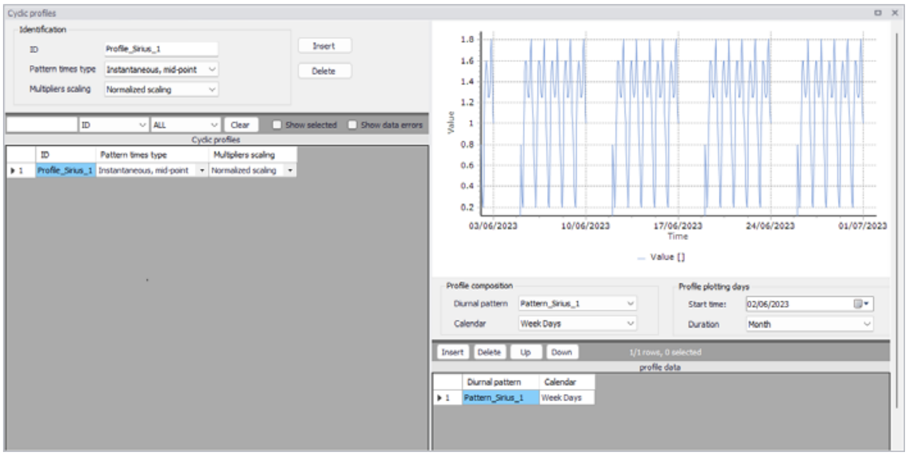


Figure 7.20 Cyclic Profile

The 'Pattern times type' controls how the pattern's multipliers apply for each of the pattern's time step:

- Step values: The multiplier in the Diurnal pattern at a given row remains constant between the 'From' and 'To' times. This leads to a stepped pattern.
- Instantaneous, mid-point: The multiplier in the Diurnal pattern at a given row is applied in the middle of the time interval for that row, and a linear interpolation applies between consecutive multipliers.

The 'Multipliers scaling' option controls whether the multipliers are scaled / adjusted during the simulation or not:

- Normalized scaling: The multipliers are rescaled, such that their average value becomes 1 over a day. Negative multipliers are not allowed in this case.

- No scaling: The multipliers are kept unchanged. Negative multipliers are allowed.

7.6.3 Profiles calendar

The purpose of the profiles calendar is to specify when the diurnal patterns are to be applied, e.g, only during summer, only February, only on weekdays, only on each first in the month, etc.

Profile calendars specify when the diurnal profile pattern is to be applied, and can be defined in three primary formats, Weekdays, Dates and Months.

There is scope to apply multiple calendar formats to create very specific calendar formats, if required.

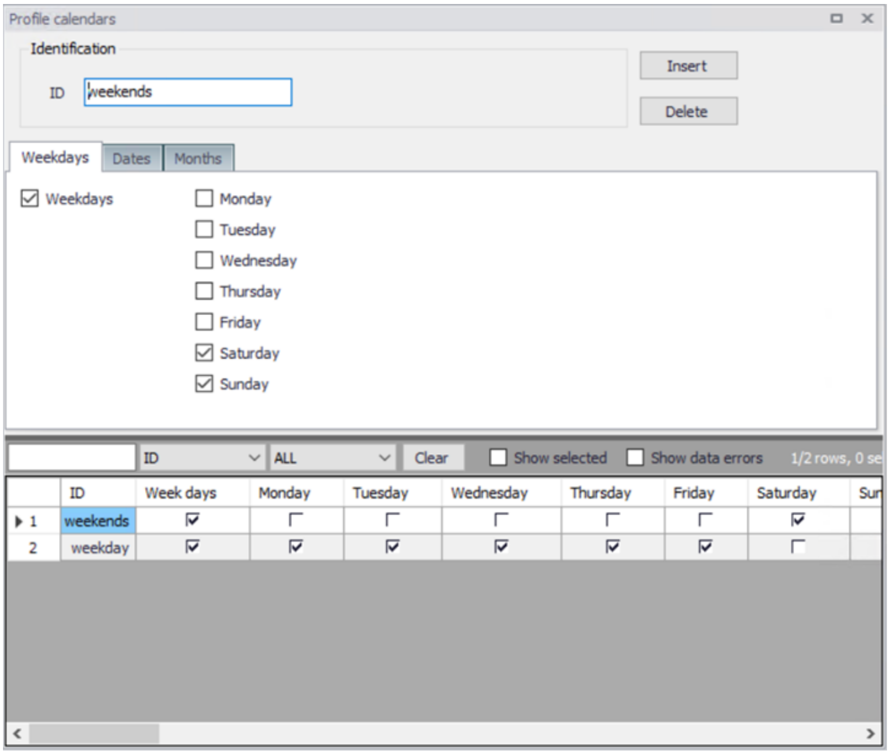


Figure 7.21 Profiles calendar weekdays

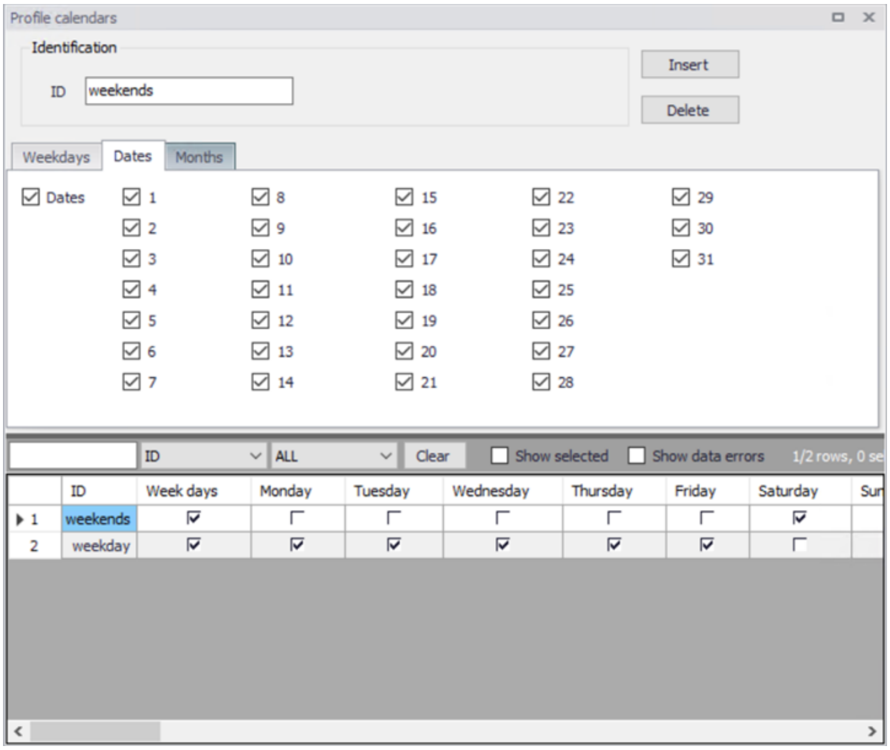


Figure 7.22 Profile calendars dates

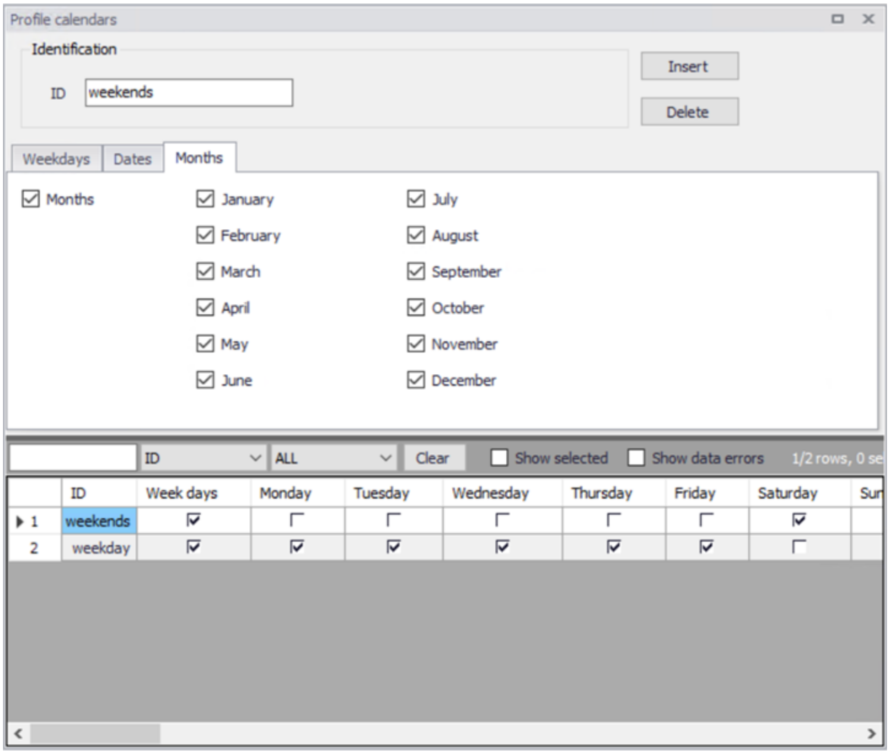


Figure 7.23 Profile calendar months

7.6.4 Special days

The ‘Special days’ is used for the specification of individual days that should be considered differently than it is given by the profiles calendar.

For example, the 1st January (the New Year day) falls on a different day each year. Typically, holiday days diurnal profiles are more representative of Week-ends. The ‘special days’ function can be used to assign the 1st January with a weekend diurnal pattern.

The special days have two categories, ‘Unique date’ and ‘Every year’. An example of this is Easter day. The date of Easter varies each year, so the ‘Unique date’ would be more applicable.

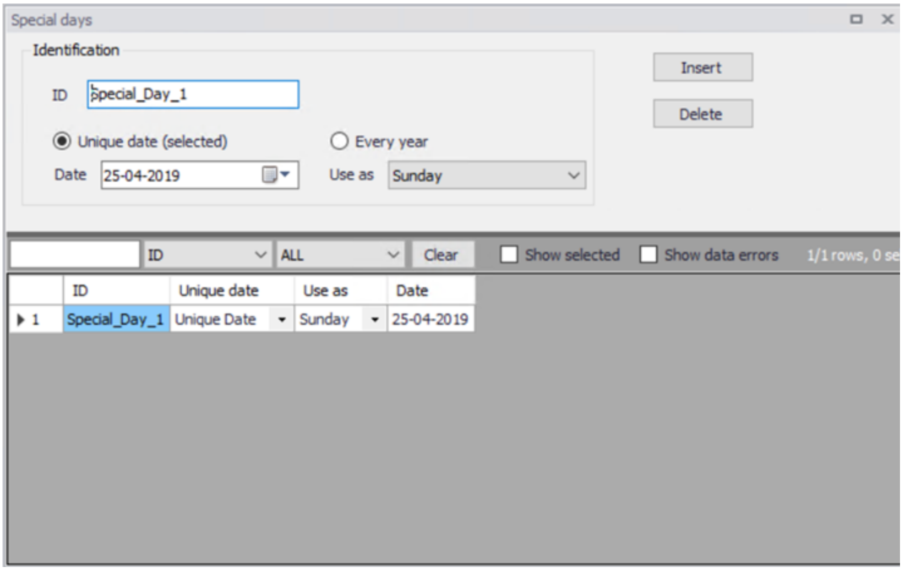


Figure 7.24 Special Days

The date calendar allows the user to select the desired date for a special day. The “Unique date (selected)” and “Every year” checkbox are used to specify the recurrence. Finally “Use as” allows the user to assign the variation pattern, weekday and weekend.

7.7 Boundary Overview

It is possible to visualize the time frame variation of all boundary conditions included in the model by means of the boundary overview dialog.

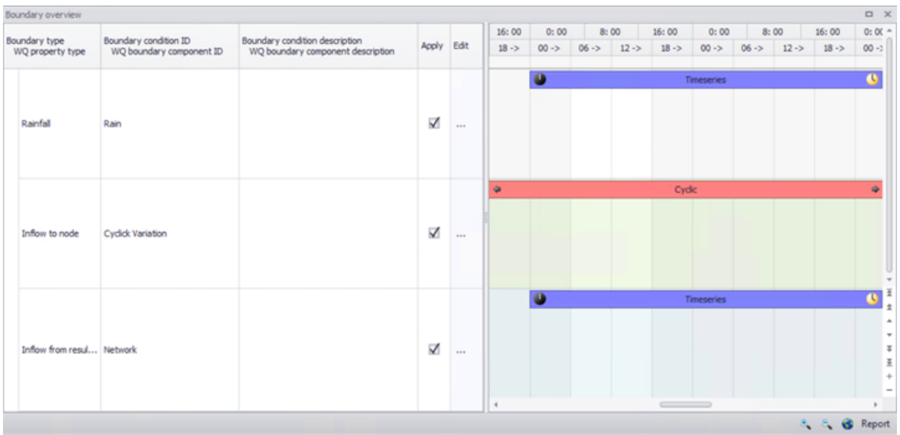


Figure 7.25 Boundary overview

The overview allow the user to apply boundary conditions and lead the user to the editor by clicking on “Edit”. The right side of the panel shows the time frame of the boundary condition.

8 Tables

The Tables Section in MIKE+ holds data for the following tabular data types:

- Curves and Relations
- Materials
- Outlet Head Loss
- On Grade Captures

8.1 Curves and Relations

In Tables|Curves and Relations (Figure 8.1), a number of tabular data used in other data dialogs are specified. These different types of tabular data are:

- Capacity Curve QH (used for Pumps)
- Capacity Curve QdH (used for Pumps)
- Pump Acceleration Curve (used for Pumps)
- Regulation Qmax(H)
- Regulation Qmax(dH)
- QH Relation (when specifying a QH relation for a node)
- Valve Rating Curve
- Time-Area Curve (used in Time-Area runoff model)
- Removal Efficiency (used for the efficiency curve for removal weirs)
- DQ Relation (used for Curb Inlets)
- QQ Relation (used for Curb Inlets and On Grade Captures)
- Capacity Curve QdH & Power
- Runoff Pollutants (used for SWQ)
- Basin Geometry (used for Basins)
- Generic control rule (used for actions in control rules)
- Control rule time series (used for actions in control rules)
- Undefined (general placeholder)

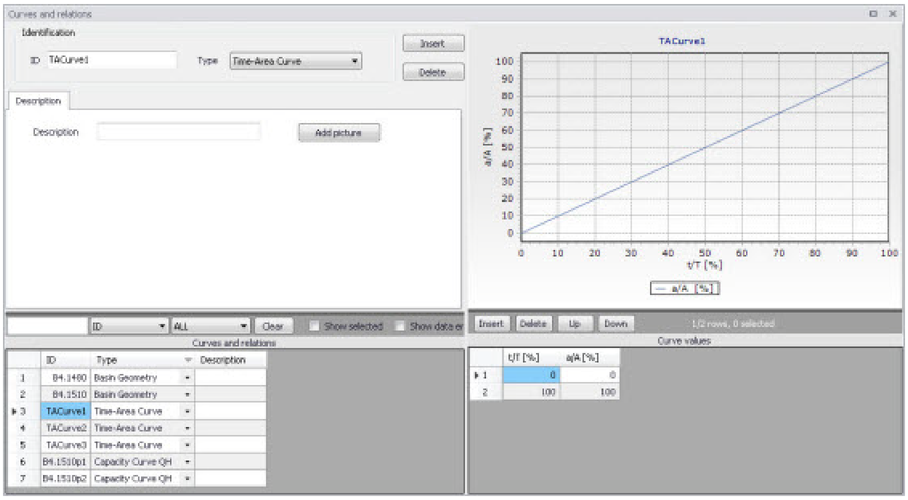


Figure 8.1 Curves and Relations Editor

There are 3 pre-defined Time-Area Curves in the database (TACurve1, TACurve2 and TACurve3), which should not be deleted.

Additional curves and relations are inserted under the Curves and Relations list (i.e. the primary table on the lower left corner of the editor) using the 'Insert' button at the top of the editor (Figure 8.2).

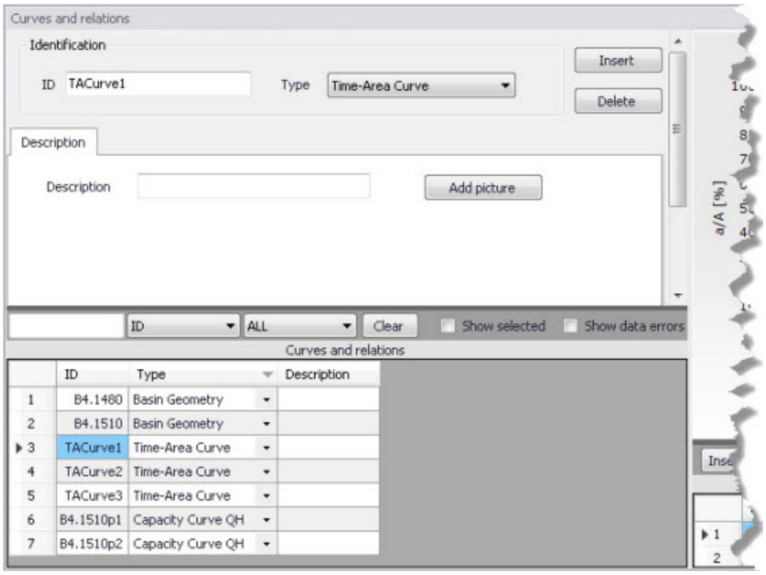
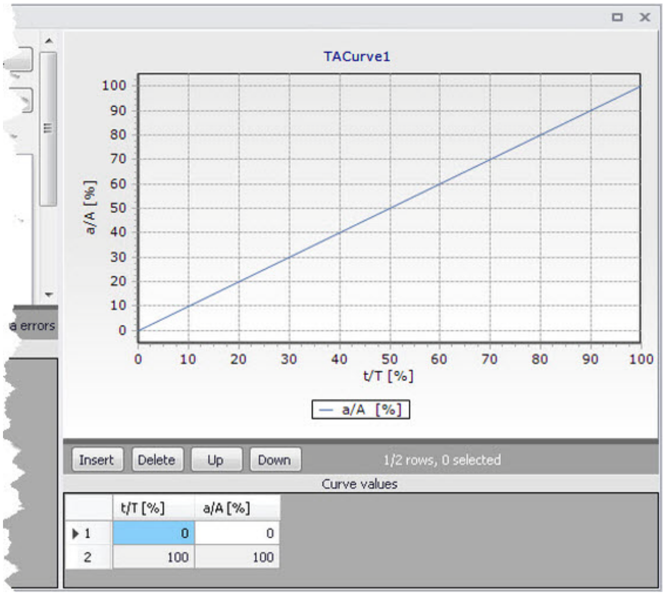


Figure 8.2 Primary table with the Curves and Relations list on the lower left side of the Curves and Relations Editor

After inserting a new tabular data item, define the corresponding data values under the Curve Values table (i.e. secondary table to the right of primary table) (Figure 8.3). Secondary table parameters/columns that should be filled vary depending on the curve and relation type.



t/T [%]	a/A [%]
0	0
100	100

Figure 8.3 Secondary table containing Curves Values on the lower right side of the Curves and Relations Editor. Also shown is the tabular data plot above the secondary Curve Values table.

A plot of the tabular data is also shown on the upper right corner of the editor (Figure 8.3).

Table 8.1 Overview of Curves and Relations Editor attributes (Table ms_Tab)

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	Tabular data identi-fier	Yes	MUID
Type	Dropdown menu for selecting tabular data type	Yes	TypeNo
Description	User's descriptive information on the tabular data	Optional	Description

8.1.1 Capacity curves

It is possible to define two types of capacity curves in MIKE+; both are used to define pump operation.

The capacity curve can be a 'Capacity Curve QH' relation (for screw pumps) or 'Capacity Curve QdH' relation (for differential head pumps).

‘H' is the absolute water level in the pump's wet well (i.e. From Node), and 'dH' is the water level difference between the (downstream) 'To Node' and (upstream) 'From Node' locations.

If an offset is specified, this will be added to the capacity curve relation.

Also note that one may specify a pump capacity curve with energy consumption (i.e. Capacity Curve QdH & Power).

8.1.2 Pump acceleration curve

Pumps may be RTC controlled. For PID-controlled RTC pumps, the acceleration of a pump can be specified as dependent on the actual flow. This pump acceleration curve is then specified as a number of ‘dQ, dQ/dt’ values.

8.1.3 Regulation curves Qmax(H) and Qmax(dH)

The regulation curves Qmax(H) and Qmax(dH) are used in the regulation of the maximum discharge in links. The regulation can either be a maximum discharge as a function of the water level in a user-specified node, or a maximum discharge as a function of the water level difference between two user-specified nodes.

8.1.4 QH relation

QH relations can be used for outlets. Using a QH relation in an outlet means that you specify the discharge out of the outlet based on the water level in the outlet.

8.1.5 Valve rating curve

A valve is a functional relation between two nodes of a sewer network. The valve rating curve specifies the relationship between the valve opening (%) and resistance (k).

8.1.6 Time-Area curve

The Time-Area curve is used in the Time-Area runoff model. A Time-Area curve represents the percentage contributing part of the catchment surface as a function of time.

MIKE+ comes with three default Time-Area curves - TACurve1, TACurve2 and TACurve3 - applicable for rectangular, divergent and convergent catchments, respectively.

One can define other Time-Area curves. Each Time-Area value table must start with a pair of values (0,0) and must end with a pair of values representing the whole catchment contribution. MIKE+ maintains T-A curves in percent (%), and the last pair of values in the table must be (100,100).

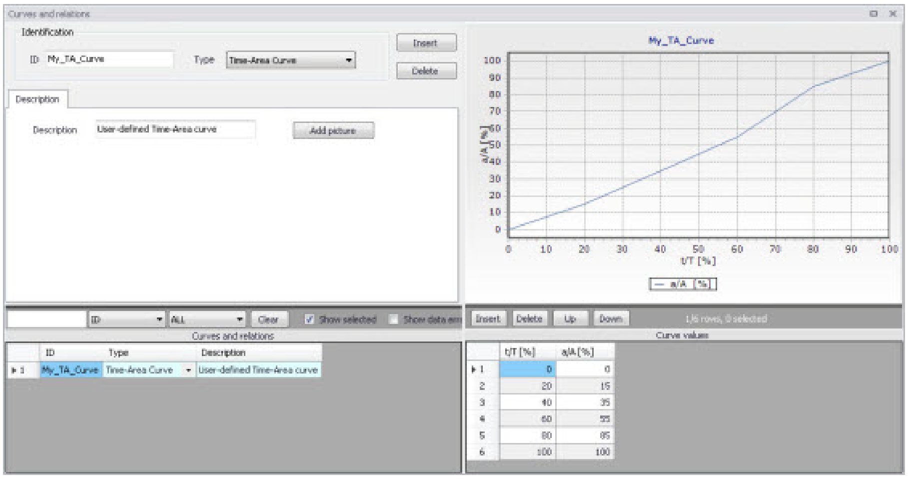


Figure 8.4 Example of user-defined Time-Area curve

8.1.7 Removal efficiency

There are three methods available for the removal of sediments in weirs. In one of these methods you specify the relation between discharge towards the weir and the removal efficiency, i.e. the efficiency curve. The removal efficiency is hence a function of Q and the efficiency (dimensionless 1/1).

8.1.8 Curb inlet DQ and QQ relations

Two curve types can be specified for two different types of Curb Inlets:

- DQ Relation (depth-discharge relation specified in the Curb Inlets dialog)
- QQ Relation (Qapproach-Qcapture relation specified in the On Grade Captures editor)

The DQ relation specifies the depth-based capacity curve for a SAG Type Curb Inlet. Values must be monotonously increasing in depth and discharge and starting at (0,0). For depths in excess of the maximum value specified in the last row of the table, the last corresponding discharge value is used.

The QQ relation specifies the relationship between approach flow in the overland flow network (Qapp) and the captured flow at the connection node for an On Grade Type Curb Inlet (Qcap). Values must be monotonously increasing and starting at (0,0). For approach discharges in excess of the maximum value specified in the last row of the table, the last corresponding capture discharge value is used.

8.1.9 Capacity curve QdH & Power

If specific power consumption in relation to pump levels is known, it is possible to include this in the model using the ‘Capacity Curve QdH & Power’ curve type.

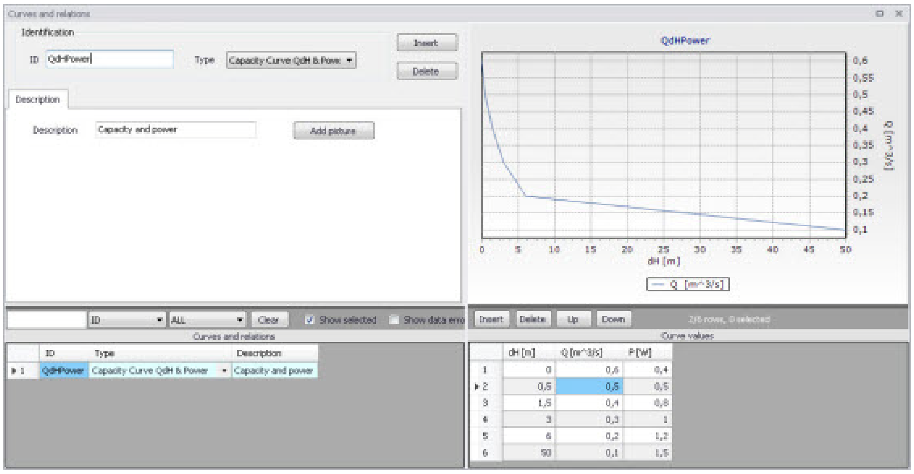


Figure 8.5 Pump capacity curve including power consumption

After the simulation with a ‘Capacity Curve QdH & Power’ the summary will contain information on the power consumption during the simulation period.

8.1.10 Runoff pollutants

This type of table is used in surface water quality (SWQ) boundary conditions as a way to define the Temporal Variation of surface stormwater loads as well as RDI stormwater loads.

The table serves as a lookup table for the boundary condition, where corresponding concentration values are determined based on runoff intensity. The tabular data set shall contain values for runoff intensity (i.e. the runoff divided by the total catchment area) and corresponding concentrations.

8.1.11 Basin geometry

Basin geometries are tabulated area-elevation functions. One specifies values for the parameters H, Ac, and As.

Ac is the cross-section area perpendicular to the main flow direction in the basin, which is used to calculate the velocity. As is the surface area of the basin (used to calculate the volume). Both parameters are specified as functions of the water level, H, in the basin.

The H-column for the basin geometry can start at any value, e.g. 0 for interpretation of H as depth in the basin. MIKE+ associates the first H-value to the bottom level of the node. This means that the same geometry can be reused in several places in the model.

The maximum level before flooding at a basin is either the highest H value of the geometry or the ground level. If the top of the basin geometry is below the ground level, the specified basin geometry is extended with additional points to allow for flooding.

The plot for 'Basin geometry' tables shows two types of points:

- Volume points: These are the raw points at levels H specified in the input table.
- Volume: These are extra points added at regular intervals between levels H specified in the input table. These extra points help better representing the actual variation of volume between input levels, which is not necessarily linear.

8.1.12 Generic control rule

Generic control rule tables are lookup tables defining the functional relation between an actual input value (e.g. sensor reading or difference between sensor readings, etc.) and the set point value (or setting). The tabulated values are linearly interpolated between defined relations.

8.1.13 Control rule time series

Control rule time series tables are lookup tables explicitly defining the setpoint value (or setting) for particular time periods (i.e. date/time). The tabulated values are linearly interpolated between defined values.

8.1.14 Undefined type

The Undefined table type is an extra generic type of table used as a place-holder for potential future functionality.

8.2 Two-dimensional Tables

The two-dimensional tables are used for tables defining an output value as a function of two input parameters.

The two-dimensional tables with type 'Time varying control rule' may be used in control rules where a relationship between an input value and a return value (output) changes over time.

Each row represents a specific date and time, for which the return values (output values) are specified in the table in the cells with white background, as a function of the input values specified in the top row with dark background.

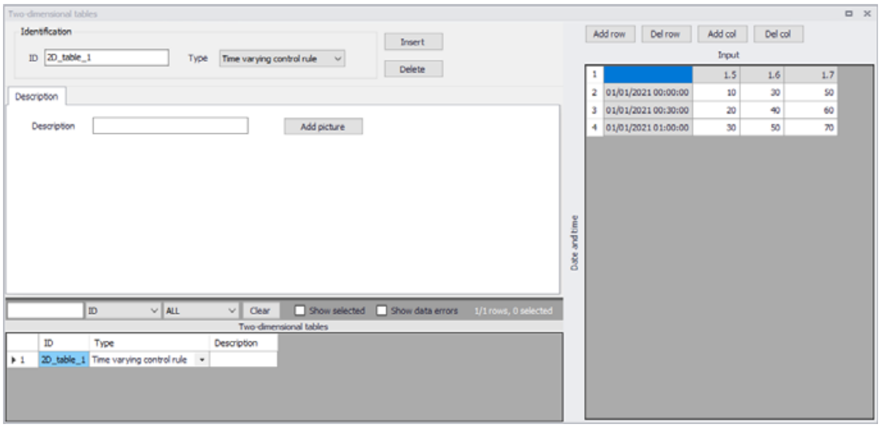


Figure 8.6 Two-dimensional tables Editor

The two-dimensional tables with type 'Generic control rule' are for use in control rules where the action value (output of the table) varies with any two input parameters. These two input parameters' values are respectively defined in the first row and the first column of the table (cells with dark background). This type of table is enabled in the 'Actions' editor when using the action type 'Set value from 2D generic table', where the two input parameters of the table are defined. Note that the unit of the output values from the table depends on the control type defined in the 'Actions' editor and is the same as what would apply with the action type 'Set value'.

The two-dimensional tables with type 'Tabulated Q' or 'Tabulated h' are used by tabulated structures on river networks, to define the relationships between the discharge in the structure and the upstream and downstream water levels.

The multiple two-dimensional tables with type 'Bridge' are used by bridge structures on river networks, to define the various coefficients that may be required for these structures.

The type of a table can only be selected when it is created, but cannot be changed afterwards.

8.3 Materials

In MIKE+, a link is characterised by material, which determines the Manning friction coefficient (Manning M or Manning n), the Colebrook-White coefficient (EQ Roughness), or Hazen-Williams coefficient (H-W Coefficient) for the conduit. It is optional to use either the default roughness values for specific materials or local values.

Specification of the different kind of materials and roughness coefficients is done through the Materials editor (Tables | Materials).

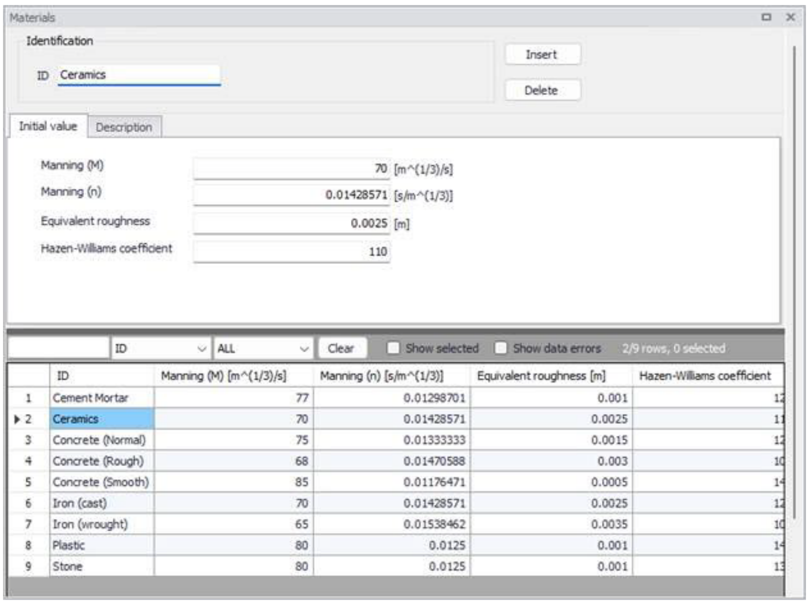


Figure 8.7 Materials Editor

MIKE+ has the following pre-defined Material types with friction loss properties:

- Cement Mortar
- Ceramics

- Concrete (Normal)
- Concrete (Rough)
- Concrete (Smooth)
- Iron (cast)
- Iron (wrought)
- Plastic
- Stone

Table 8.2 Overview of the Materials Editor attributes (Table ms_Material)

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	Material type ID	Yes	MUID
Manning (M)	Manning (M) roughness value	Yes If 'Manning (M)' is used	Manning
Manning (n)	Manning (n) roughness value	Yes If 'Manning (n)' is used	ManningN
Equivalent roughness	Equivalent roughness	Yes If 'Colebrook-White' formulation is used	EQRough
Hazen-Williams coefficient	Hazen-Williams roughness coefficient	Yes If 'Hazen-Williams' is used	HWCoef
Description	User's descriptive information on the material	Optional	Description

8.4 Outlet Head Loss

MIKE+ models outlet head losses at nodes. The general flow equations are valid only for continuous conduits, where the only resistance to flow is bottom and sidewall friction. Hydraulic conditions in nodes, i.e. at manholes and structures, serve as boundary conditions in the computation of flows in conduits. In turn, hydraulic conditions in a node depend on the flows in the inlet and outlet conduits.

These hydraulic conditions, expressed in terms of the energy conservation principle, are calculated as water levels and velocity heads. The calculation is based on the mass continuity and formulation of more or less advanced energy relation between the node and the neighbouring links, including energy losses caused by local flow disturbances at different locations in the node.

The following parameters constitute a definition of head loss calculation option in MIKE+:

Computation Method

Three different methods are available:

- **Classic**. This is a simplified computational model for energy losses in junctions based on F.A. Engelund's energy loss formulae. The total head loss comprises of node inlet loss and node outlet loss, including losses due to change in flow direction, due to change in elevation, and due to contraction, if relevant.
- **Mean Energy Approach**. This is an alternative solution which fully ignores the energy loss at the inlet. For a flow-through manhole, this means that the energy level in the manhole is set equal as at the downstream end of the inlet pipe. For manholes with multiple inlets, the energy level is calculated as the weighted average of the inlet flows (i.e. large flows contribute most to the energy level). In this formulation, the total loss at the manhole is concentrated computationally at the outlet, and can be fully controlled by the user.
- **No Head Losses**. This option ignores all local losses. Regardless of the shape of the outlets, geometrical set-up of the junction and distribution of flows among inlet and outlet conduits, water levels in the junction and the outlet conduit are set equal as if there is no change of geometry and flow conditions between the junction and outlet conduit. This means that this option should be applied only where there is no change in cross section. Inconsistent results may be generated if inappropriately applied. This option is recommended for an artificial node/junction along a straight section of conduit, where no losses actually occur.

Further details on the various head loss computation methods mentioned above are found in the MIKE 1D Reference Manual.

Effective Node Area

This parameter is only relevant for the Classic head loss computational method. In all other cases, the default total wetted node area is applied. The following choices are available:

- **Full Node Area**. Calculated as product of diameter and water depth for manholes and read from the basin geometry table (Ac) for basins. Typically results in overestimate of local loss in a node.

- **Calculated Effective Area**. The effective area in a manhole is calculated based on an empirical formula (see Section 4.6.2 ‘Headloss calculation for inflowing water’ section in the MIKE 1D Reference Manual). This results in a significantly smaller area than full wetted area and, consequently, with a more realistic flow calculation.
- **Reduced Calculated Effective Area**. The effective area in a manhole is further reduced to 50% of the calculated effective area.

Loss Coefficient

The available loss coefficient types distinguish three different interpretations of the specified head loss coefficient.

- **Km**. Interprets the specified value as the outlet 'shape' coefficient Km (see Equation (4.17) in MIKE 1D Reference Manual).
- **Contraction HLC**. Interprets the specified value as the outlet 'contraction' coefficient ζcontr(j) (see Equation (4.17) in MIKE 1D Reference Manual). This means that the model ignores the geometrical relations between the node and the outlet links (outlet shape), and applies the specified value directly as the ζcontr. The contraction losses in the outlet links are then computed by multiplying the velocity head in the respective link by the ζcontr. The total head loss for an outlet link is computed as a sum of the contraction, direction and elevation loss.
- **Total HLC**. Interprets the specified value as the total outlet head loss. This means that the model completely ignores the geometry of the node/links, and applies the specified value (Total HLC) directly as the ζout, the same for all outlet links at the node. The total head losses in the outlet links are then computed by multiplying the velocity head in the respective link by the specified ζout.

Changes to outlet head loss parameter sets used for Nodes are made via the Outlet Head Loss editor (Tables|Outlet Head Loss). Figure 8.8 also lists five pre-defined head loss parameter sets in MIKE+.

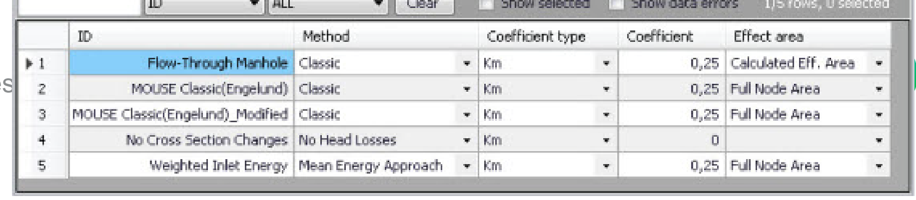


Figure 8.8 Outlet Head Loss Editor

Table 8.3 Outlet head loss parameter set Editor attributes (Table msm_LossPar)

Edit field	Description	Used or required by simulations	Field name in datastructure
Head Loss ID	Unique head loss parameter set identifier	Yes	MUID
Method	Dropdown menu to select head loss calculation method for parameter set: - Classic - No Head Losses - Mean Energy Approach	Yes	OutletShapeNo
Effective Node Area	Choice of method for the calculation of wetted area: - Full Node Area - Calculated Eff. Area - Reduced Calculated Eff. Area	Yes If Method = Classic	EffAreaNo
Loss Coefficient [dropdown menu]	Definition of the interpretation of the head loss coefficient: -Km - Contraction HLC - Total HLC	Yes If Method = Classic or Mean Energy Approach	CoeffNo
[Field next to Loss Coefficient dropdown menu]	Value for the loss coefficient	Yes If Method = Classic or Mean Energy Approach	Coeff

8.5 On Grade Captures

On Grade Captures tabular data are used as hydraulic properties for On Grade Curb Inlets describing the transfer capacity of connections. The On Grade Captures editor allows the user to group together QQ relations (tabular Curves and Relations data) that comprise a single On Grade Curb Inlet geometry (collective of QQ relations defining the capture rate as a proportion of approach flow). The transfer capacity for an On Grade Curb Inlet is dependent on the approach slope in the overland flow network, and a number of QQ relations can apply for a single curb inlet.

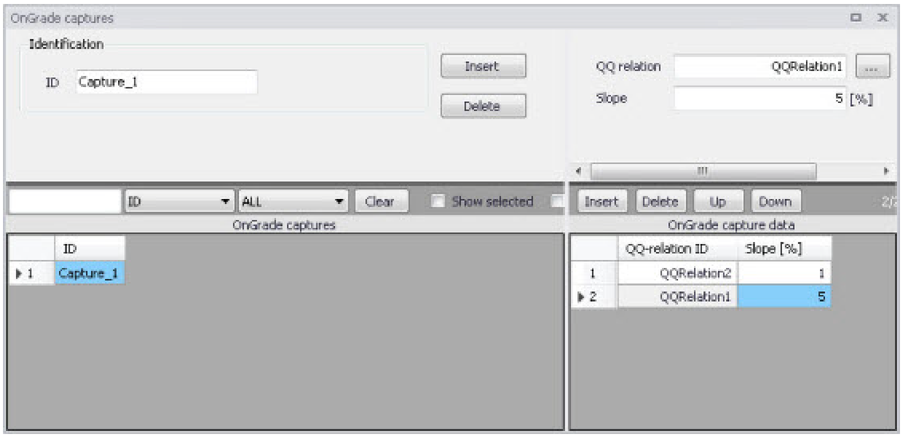


Figure 8.9 On Grade Captures Editor

For calculated or user-defined slopes in the Curb Inlet dialog that are outside the range of slopes specified in the On Grade Captures table, the closest slope curve will be used. For intermediate calculated or user-defined slopes (lying between slope curves in the On Grade Captures table), linear interpolation is applied.

In the case of an On Grade Curb Inlet capacity that is not dependent on slope of the overland flow network, the user needs to define the On Grade Capture with a single QQ relation. *Note: In this case, the calculated or user-defined slope in the Curb Inlet dialog for On Grade Type will be ignored*.

9 Control Rules

The MIKE+ Control module features advanced control capabilities for urban drainage, sewer systems and river networks. It can describe various controllable devices and makes the definition of complex operational logic for interdependent regulators fully transparent and time efficient. The following controllable devices can be specified:

- Pump
- Weir
- Gate in rectangular orifice (Blade moves from top to bottom)
- Weir in rectangular orifice (Blade moves the bottom to top)
- Valve
- Culvert
- Gate
- Direct discharge

The devices may be specified as directly controlled or PID-controlled, with control function evaluation based on a global system analysis. Each structure operates under the control logic encapsulated into a set of simple logical rules and control functions.

The MIKE 1D Control module employs an algorithm that reads arbitrary input, not necessarily limited to states of the network itself, and sets the state of the simulation. Network state conditions include measurable and derived hydraulic and water quality variables (e.g. water level, flow, pollutant concentration, level difference), device status (e.g. gate position, pump ON/OFF) and the current control function.

The control functions range from the simplest constants for the operational variables (e.g. constant weir crest setting or constant flow setpoint) to dynamic controlled variables set in a continuous functional relation with any of the measurable variables in the system (e.g. CSO discharge setpoint as a function of flow concentration or a pump START/STOP levels as functions of water level at a strategic location in the system).

9.1 Controls in Urban and River Networks

Control rules are active controls and operations of flow regulators based on measured information or target on the network.

Control of structure is feasible where it proves that flexible redistribution of water in space and time contributes to the fulfilment of the specified operational objectives based on economically- and technically-sound solutions.

Accordingly, application of structure regulation to river, drainage and sewer systems may be relevant:

- Where the system has substantial transport, storage or treatment capacity not effectively used under passive system operation;
- Where typical rainfall patterns over the catchment area exhibit high degrees of spatial variability resulting in some parts of the system becoming overloaded whilst others are underutilised;
- Where the urban wastewater system includes treatment processes whose performance is amenable to active, short term control;
- Where the assimilative capacity of the receiving waters is variable over time.

Usually, such structure regulations are implemented as an integral part of a rehabilitation/upgrade scheme also involving significant civil upgrading works to increase the transport, treatment or assimilative capacity of the river / urban network. In such circumstances, the role of structure controls is to optimise the operation of both the new and the existing facilities, thereby maximising the benefit in performance terms. Where the overall objective is to achieve compliance with specified performance targets, structure controls serve to minimise the scale and extent of the necessary works.

9.2 Architecture of Real-Time Control Systems

An RTC system includes **sensors/monitors**, which generate measurement values characterising states of the system. To be useful for RTC, the measurements must be available with relatively insignificant time lag (delay). The sensors must be accurate and reliable.

The active control is performed by **regulators** - controllable movable devices (weirs, gates, valves etc.) and pumps. Regulators may take various forms and sizes, and the regulation may be continuous within the functional range, step-wise, or discontinuous (e.g. ON/OFF, OPEN/CLOSED). The regulators may be powered mechanically, hydraulically or pneumatically.

Controllers on the basis of a pre-programmed operational strategy determine the regulator movements (the **control actions**). The operational strategy may consist of two parts: the control action(s) and, if more control actions are specified, the control logic (**conditions**) responsible for the selection of an appropriate control action. A control action establishes a relation between a **control variable** and a **controlled variable**. A controlled variable can be a regulator setting (e.g. gate position, pump START/STOP level) or some of the flow variables (e.g. water level, flow).

In the latter case, the control decisions are derived by evaluating (comparing) the current value of the controlled flow variable and the pre-defined **setpoint** value. The control algorithm is based on the numerical solution of the “continuous control problem” equation and is usually termed as **PID** (Proportional-Integral-Differential) control. The actuation signal for the regulator is generated by a PID controller, which usually appears as part of the operational strategy programmed in a Programmable Logical Controller (**PLC**).

Selection of a controlled variable is, however, subject to limits set by the variable’s “controllability”. Therefore, a controlled variable is usually selected among the flow variables (flow, water level), preferably in the vicinity of the regulator. As a controlled variable becomes more distant from the regulator, it becomes more difficult to control due to time lags, diffusion and uncontrollable interference. Control of relatively distant controlled variables is difficult and often cannot give satisfactory results.

When a regulator setting is used as a controlled variable, the control algorithm is reduced to an explicit functional relation between the control variable and the regulator setting, which controls the system response indirectly. This is much simpler than PID control, but in turn, the control results are in many cases inherently inexact and only a rough flow control can be achieved. This type of control is most suitable for regulators of the ON/OFF (or OPEN/CLOSE) type, while the application to continuously controllable regulators should be carefully considered.

If the operational strategy is based on conditions local to the regulated device (for example the ON/OFF-control of a pump based on the water level in a wet well) it is called **local control**. A PLC receives signals (measurements) from local sensors and sends the control decisions (actuation signals) to the regulators. The usual situation for a sewer system is to have a number of local controllers associated with pumps.

If the operational logic is based on global conditions, it is then called **global control**. In such a situation, a **global controller** is required. A global controller is a computer program that makes the overall system state analysis in real time and provides additional input to the local controllers, which overrides or supplements the local logic with e.g. actuator signals, or by modified setpoint values.

An additional component needed is then a data transmission system to transfer data between sensors, controllers and the global controller. In connection with the global controller function, an RTC system is usually equipped with the data management and storage facilities (databases) and the user interface. This is usually termed as SCADA (Supervisory Control And Data Acquisition) system.

The global control can also be extended to include forecast data in addition to real-time data, which is then called predictive RTC. The most comprehensive way to obtain forecast data is to include a model in the control system. Predictive control brings additional benefits in relatively inert systems, i.e. where the response time of an operational variable is long compared to the change of relevant disturbance (external input or control action).

9.3 MIKE+ Control vs. Real Life

Control rules in MIKE+ simulate reactive local and global RTC systems in river, drainage and sewer networks. The software implementation is inherently a conceptualisation of real life, of which the user must be fully aware. Some conceptualisations applied in MIKE+ are listed below.

- The program does not distinguish explicitly between local and global RTC. Per default, all elements of a modelled RTC system are assumed available for global control.
- Sensors are specified as operational devices with definition of sensor type and position in the network. Sensors with multiple functionality must be specified individually.
- When devices (weirs, gates, etc.) are specified as controllable in the MIKE+ interface, a number of Regulation parameters about the behaviour of the structure is required to describe e.g. the allowed change rates for the state of the structure.
- The actual controllers are not specified explicitly as physical devices, but their function (i.e. operational logic as a combination of operational conditions and control actions) is associated with the respective devices.
- MIKE+ controls use sampling and actuation (control loop) frequency identical to the simulation time step.
- Sensor readings are simulated as perfectly accurate and with 100% availability.
- The PID control algorithm is built into the program and is controlled by the PID constants and by factors for weighting the terms of the numerical solution of the control equation.

9.4 Sensors

A sensor is a device positioned somewhere in the system providing information on the actual value of a monitored variable.

A sensor can only monitor one variable. If more variables are measured at the same location, a corresponding number of sensors has to be described.

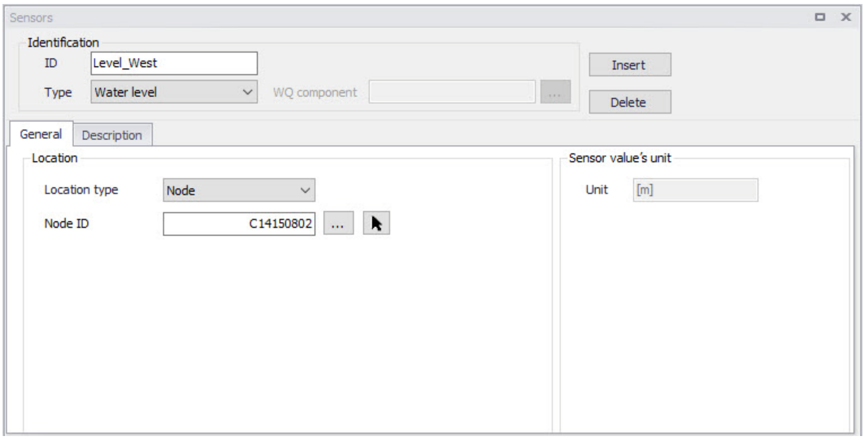


Figure 9.1 The Sensors Editor showing an example setup for a water level sensor

Table 9.1 Overview of the Sensors editor attributes (Table msm_RTCSensor)

Edit field	Description	Used or required by simulations	Field name in data structure
ID	Sensor unique identifier	Yes	MUID
Type	Type of parameter measured by the sensor	Yes	TypeNo
WQ Component	Measured water quality component	Yes If Type = Concentration or Mass flux	ComponentID
Location/ Type	Location type	Yes	LocationNo
Location/ ID	Location ID	Yes	_LocationID (prefix varies with location type)
Time Series File	Path to external time series file defining sensor values	Yes If Type = External	TSFileName
Time Series Item	Item name for the selected data series from the time series file	Yes If Type = External	TSItemName

Table 9.1 Overview of the Sensors editor attributes (Table msm_RTCSensor)

Edit field	Description	Used or required by simulations	Field name in data structure
Expression	Expression defining a user-defined variable	Yes If Type = Variable expression	Expression
Description	Free text description of the sensor	Optional	Description

Sensor ID

Each sensor needs a unique ID, which can be used to access the sensor information from other dialogs.

Type

This parameter defines the type of variable measured by the sensor. The options are:

- Level
- Discharge
- Surface Runoff
- Concentration
- Mass Flux
- Weir/Gate Position
- Pump ON/OFF
- Action Active (returns a value TRUE when the selected action is active, and FALSE when it's inactive)
- Action Active Time (returns a TimeSpan, which is the time since it was last activated. It returns zero when the action is not active. It can be turned into hours or seconds using a formula)
- Velocity
- Water depth
- Area (cross sectional / flow area)
- Volume
- Valve Opening
- External (external time series file)
- Variable expression (defines a user-defined variable, e.g. defined as a function of other sensors and mathematical functions).

WQ Component

The measured water quality component for the ‘Concentration’ and ‘Mass Flux’ sensor types.

Location Type and Location

Depending on the sensor type, various location types are available:

- Node: for this location type, the location is defined with a node ID

- Link: for this location type, the location is defined with a link ID and a 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
- Reach on link: this type is only available for the Volume type of sensor, and indicates that the volume is to be considered between two chainage values along a link

Define the location type and then specify the corresponding location ID. Note that a 'location' may be only indirectly related to a physical location, such as for 'Action Active' and 'External' sensors.

Time Series File and Item

For an 'External' sensor type, an external time series file, which will be used for sensor values, must be defined. Select the appropriate time series Item to use when loading the time series file.

Expression

The expression is used to define an 'Expression variable' sensor. This type of variable is usually a function of other sensors. The primary purpose is to define complex control rules using such 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 also be saved in the result file during the simulations, and it is therefore easier to check that the correct behaviour of the regulated structure is modelled, by comparing the applied control strategy with the variable.

The expression defining the mathematical expression of the variable can be typed manually or using the 'Edit' button. This button opens the Expression Editor, which offers a list of available sensors, functions and operators.

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 expression 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. If Variable1 is used in the expression defining Variable2, then Variable1 must be defined above Variable2 in the table. The order of the variables is changed using the 'Move up' and 'Move down' buttons.

Move up / Move down buttons

The order of the sensors in the table is important only for 'Expression variable' sensors, and only in case one variable is defined as a function of another variable.

Unit

The unit is read-only, and is shown for information. It controls in which unit the sensor values are considered in the expressions, for example in the 'Condition' expression of a control rule. The unit is set according to the sensor type. The unit can be changed for each sensor type from the 'Model type' page. Refer to page 121 for more information about units customization.

9.5 PID Parameters

This editor is used to define the functions for PID (Proportional Integral Differential) control. PID controls can apply to various controllable parameters, either from collection system structures or from river structures, e.g. weir crest level, pump flow, etc. Independently of the choice of the controlled variable, the PID algorithm adjusts the settings of the regulator according to the current error between the specified setpoint and the actual value of the controlled variable.

A single PID set can be used in multiple actions.

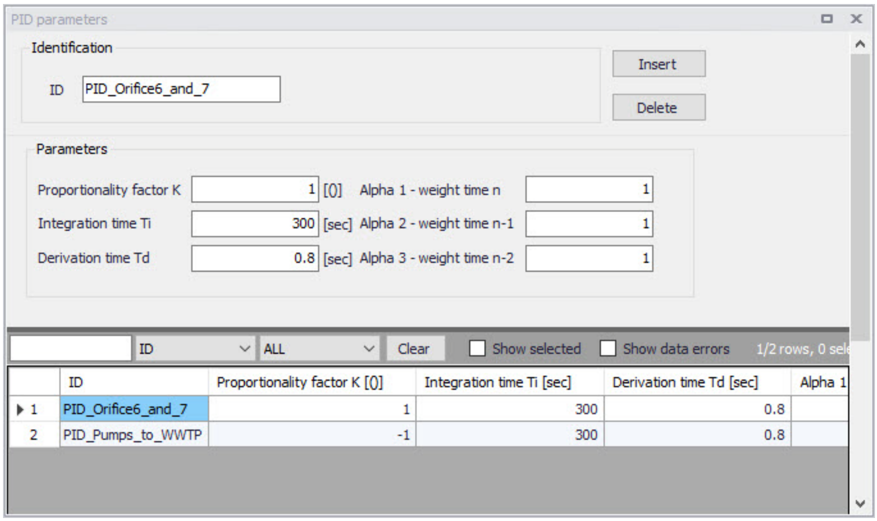


Figure 9.2 The PID Parameters Editor

ID

Each set of PID settings is identified with a unique ID. This is how the PID parameter set is accessed from other dialogs.

Proportionality Factor, Integration Time, and Derivation Time

The 3 main parameters for the PID control. These parameters are further discussed in Section 9.5.1 below.

Alpha-1, Alpha-2 and Alpha-3

Weighting factors for time level n, n-1 and n-2. These parameters are further discussed in Section 9.5.1 below.

9.5.1 Calibration of the PID Constants

Tuning of the PID constants (Ti, Td and K) is not a straightforward task. Understanding the theoretical background and the numerical solution of the control equation would be beneficial in this process.

The following values may be used as a guide:

Typical values of the PID constants and weighting factors

Table 9.2 below shows suggestions for initial values for PID constants.

Table 9.2 Summary of typical values for PID constants and weighing factors

Parameters	Pumps	Gates	Weirs
Ti	300 sec.		
Td	0.8 sec.		
K (Setpoint downstream of device)	1	1	-1
K (Setpoint upstream of device)	-1	-1	1
Alpha-1	1	1	1
Alpha-2	1	0.7	0.7
Alpha-3	1	1	1

NOTE: The sign on the K-factor is very important. If it is wrong it will cause the control function not to work at all since the device will typically move to one of the extreme positions and stay there until the end of the simulation.

Figure 9.3 to Figure 9.5 show examples of how the actual variable (flow or water level) can fluctuate around the setpoint as a consequence of PID constants values. Each figure has three different graphs depending on whether the constant is too high, too low, or adequate.

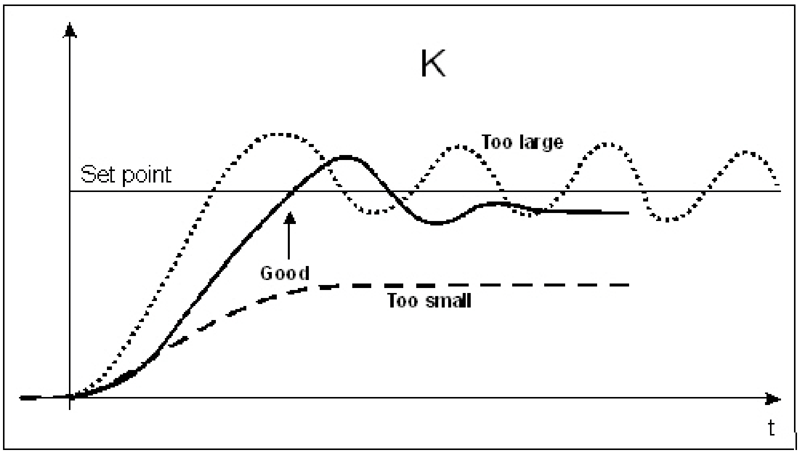


Figure 9.3 Fluctuations around the setpoint depending on the size of the proportionality factor, K

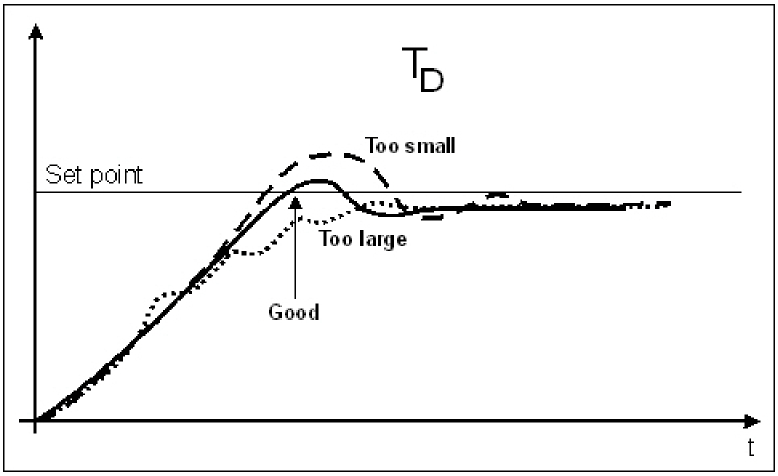


Figure 9.4 Fluctuations around the setpoint depending on the size of the derivation time, Td

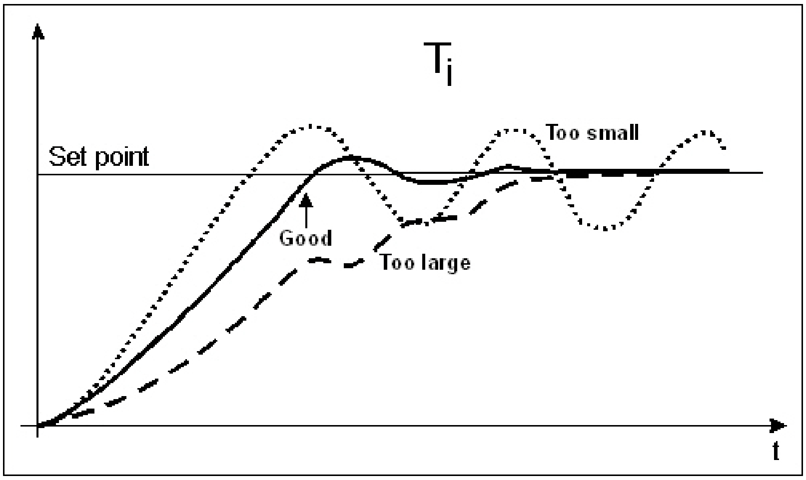


Figure 9.5 Fluctuations around the setpoint depending on the size of the integration time, Ti

9.6 Actions

The action to be performed, when the condition associated to a rule is fulfilled, is defined in the Actions editor.

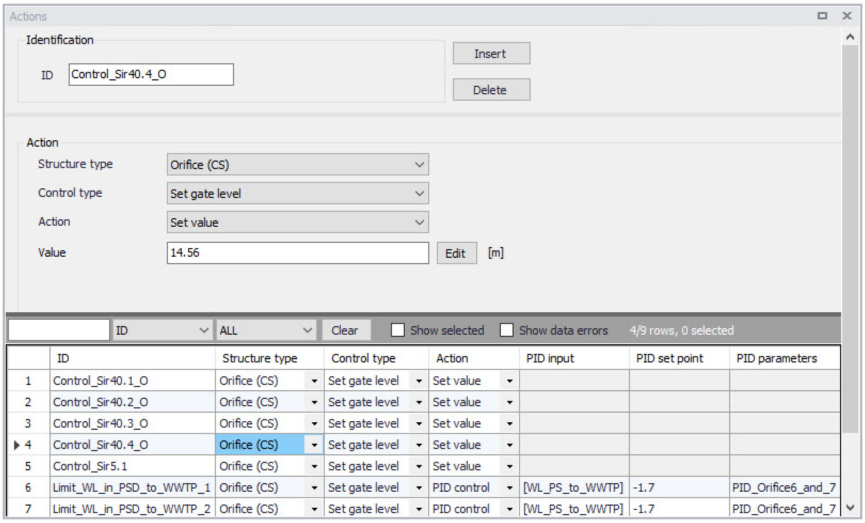


Figure 9.6 The Actions Editor

Structure Type

This list controls the type of structure to which the action can apply. On a collection system network, actions can apply to:

- Pumps
- Weirs
- Valves
- Orifices.

On a river network, actions can apply to:

- Culverts
- Weirs
- Gates
- Direct discharges
- Dambreaks
- Pumps
- Bridges
- Tabulated structures

Control Type

This field controls the variables to be regulated on the structure.

The available Control types are:

- For a pump (CS network):

- Set start level
- Set stop level
- Set flow
- Stop pump (taking the pump's deceleration time into account)
- Shutdown pump (halts the pump immediately)
- Resume pump (to be used after stopping or shutting down the pump)

- For a weir (CS network):
  - Set weir crest level
- For a valve (CS network):
  - Fully open
  - Close
  - Set valve opening
- For an orifice (CS network):
  - Set weir crest level
  - Set gate level
  - Fully open
  - Close
  - Set gate minimum level
  - Set gate maximum level
- For a culvert (river network):
  - Set flow factor
  - Set flow type
- For a weir (river network):
  - Set weir crest level
  - Set weir coefficient
  - Set flow factor
  - Set flow type
- For a gate (river network):
  - Set gate level
  - Fully open
  - Close
  - Set gate minimum level
  - Set gate maximum level
  - Apply natural flow
  - Set flow factor
- For a direct discharge (river network):
  - Fully open
  - Close
  - Set flow

  - Set minimum flow
  - Set maximum flow
  - Apply natural flow

- For a dambreak (river network):
  - Set flow factor
  - Start breach (must be combined with a ‘Wait’ action)
  - Wait (no operation: to be applied until the breach is started)
  - Set flow type

- For a pump (river network):
  - Set flow
  - Apply natural flow
  - Stop pump
  - Shutdown pump
  - Resume pump
  - Set start level
  - Set stop level

- For a bridge (river network):
  - Set flow factor
  - Set flow type

- For a tabulated structure (river network):
  - Set flow factor
  - Set flow type

When the control type applies to a variable specified in the structure editor, the value from the action will overwrite the value from the structure editor during the simulation. For example, when controlling the start level of a pump, the 'Start level' value specified in the Pumps editor is replaced by the value from the actions, when the pump is controlled.

When the control type is 'Apply natural flow', the structure is ignored and the flow is computed as in a natural channel.

Note: For a pump, it is not allowed to control simultaneously its flow and its Start / Stop levels.

Action

This controls the type of action which is performed. The following options are available:

- Keep unchanged: the controlled parameter (e.g. the gate position or the flow) remains unchanged, i.e. equal to its previous value.
- Set value: the specified value in the 'Value' field is an absolute value (e.g. level for a gate position or discharge for a flow).

- Set relative change: the specified value is a relative change from the previous state.
- Set value from time series: the time-varying absolute value is specified in a time series file.
- Set value from table: the absolute value is specified in a table, containing the relationship between an input variable and the desired value (output). The table is defined in the 'Curves and relations' editor (with curve type 'Generic control rule'), or in the 'Two-dimensional tables' editor (with type 'Time varying control rule') if this relationship varies in time.
- Set relative change from table: the relative change is specified in a table, containing the relationship between an input variable and the desired value (output). The table is defined in the 'Curves and relations' editor.
- Set value from 2D generic table: the absolute value is specified in a table, containing the relationship between two input variables and the desired value (output). The table is defined in the 'Two-dimensional tables' editor (with type 'Generic control rule'). The corresponding input variables are defined as expressions in the 'Table's X input' and 'Table's Y input' fields. Note that the unit of the output values from the table depends on the control type defined in the 'Actions' editor and is the same as what would apply with the action type 'Set value'.
- PID control: the value of the controlled parameter is assessed indirectly, by minimizing the deviation from the expected value.
- Apply natural flow: the structure is ignored and the flow is computed as in a natural channel.
- Apply structure flow: the action triggers the use of the regular energy loss calculation in a structure. It is meant to enable the calculation of the flow through the structure, when combined with the 'Apply natural flow' action.

Value

The absolute value or 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.

File name

The path to the time series containing the values to be applied.

Table ID

The table ID, as specified in the 'Curves and relations' editor, to be used to define the value. The tabular data shall represent the functional relation between the input (e.g. input sensor reading or a combination of two sensors) and the return value. The tabulated values are linearly interpolated between defined values. The selected table may also be selected from the tables listed in the 'Two-dimensional tables' editor, if the table should vary over time. With this type of table, the relationship between the input variable and the return value (output) 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. The table may also be selected from the 'Two-dimensional tables' editor if the table should vary as a function of two input variables (table type 'Generic control rule').

Table's input

Indicates the variable used as input in the table. The 'Edit' button may be used to select e.g. a sensor.

Values specified as yearly variation

This may be used to apply the same yearly pattern to multiple years. When selected, values will be interpolated between the first and the last values specified within the year, and between the last and the first values. When unselected, 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. Only available when using a two-dimensional table.

PID set point source

This controls the type of set point for the PID control. The following options are available:

- Expression: the specified value in the 'PID set point' field is an absolute value defined by an expression
- Time series: the time-varying absolute value of the set point is specified in a time series.
- Table: 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

PID input

For a PID action, the Input parameter is the parameter being evaluated against the setpoint (i.e. target) value at the setpoint for the controlled device.

The Expression editor may be used to define the Input. Note that a Double data type is expected from the evaluation of the input expression.

The input may involve two sensors e.g. if the flow is regulated as a function of the difference between two level sensor values.

PID setpoint

The setpoint value (i.e. target) for a PID control action.

When the 'PID set point source' is set to use an expression, the Expression editor may be used to define the Setpoint (constant value or functional relation). Note that a Double data type is expected from the evaluation of the setpoint expression. It is also expected that the expression of the setpoint returns a result in the same unit as the 'Input'.

When a time series is used as 'PID set point source', it holds the path to the time series file.

When the 'PID set point source' is set to use a table, the field holds the table ID, as specified in the 'Curves and relations' editor. The tabular data shall represent the functional relation between the input (e.g. input sensor reading or a combination of two sensors) and the return value (setpoint or setting). The tabulated values are linearly interpolated between defined values.

PID parameters

The set of PID parameters used in the PID control. These sets are defined in the PID parameters editor.

Table's input

Indicates the variable or expression used as input in the table for PID set point source. The 'Edit' button may be used to select e.g. a sensor or specify an expression.

Set point scale / Scale factor

The set point or time series will be multiplied by this factor. The scale can either be a constant value or an expression. The 'Edit' button may be used to specify an expression.

Figure 9.7 shows an example Action setup that is used for a PID-controlled orifice.

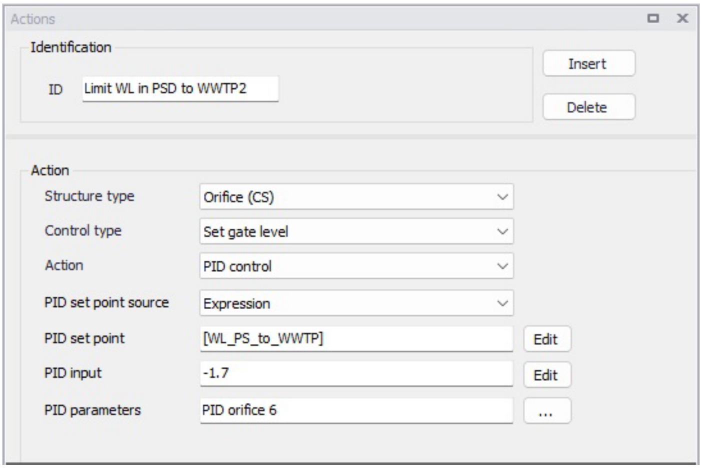


Figure 9.7 Example of PID Action

In this example, the orifice gate level shall be modified for controlling the water level at setpoint location [WL_PS_to_WWTP], where the expected water level is -1.7 m.

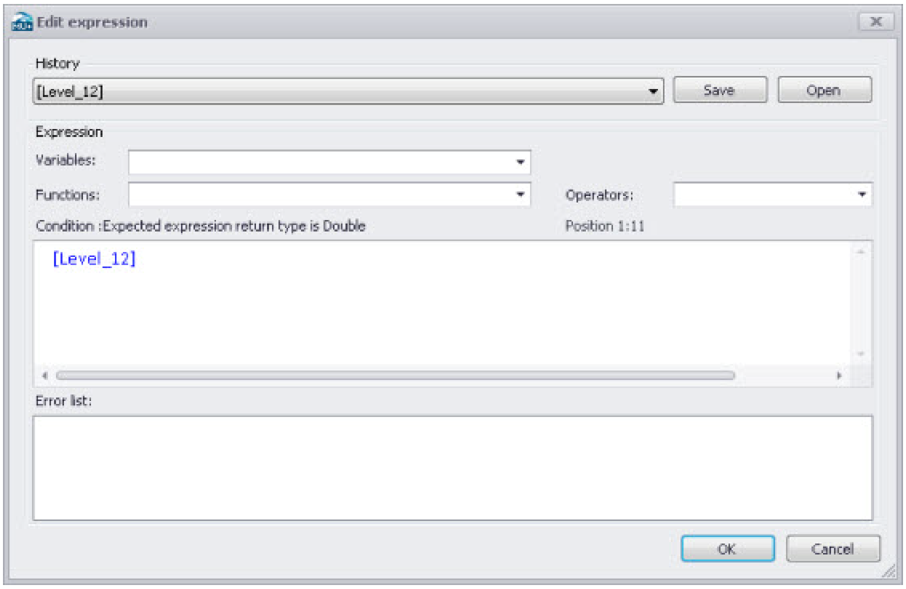


Figure 9.8 Example Input expression in Expression Editor

Initially, one may start with the Variables dropdown menu in the Expression editor when building expressions for Input. The Variables dropdown offers predefined variables, such as Sensors.

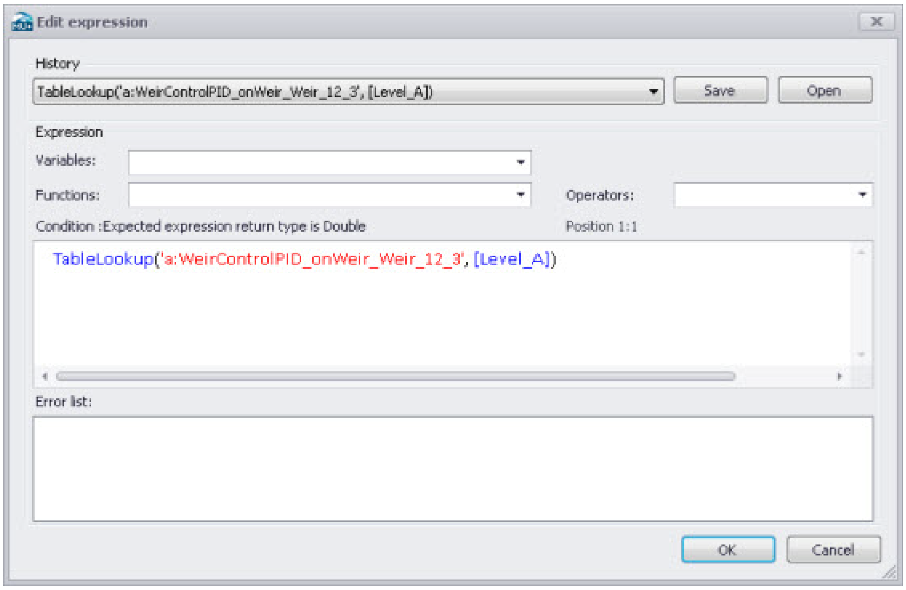


Figure 9.9 Example Setpoint expression in Expression Editor

Functions and Operators may also be used to define more complex expressions for both Input (e.g. difference between 2 sensors) and Setpoint (e.g. TableLookup).

Note: The expressions 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 'Operators' 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.

9.7 Control Rules

Define controllable devices and the respective control settings on the Control rules editor (Figure 9.10).

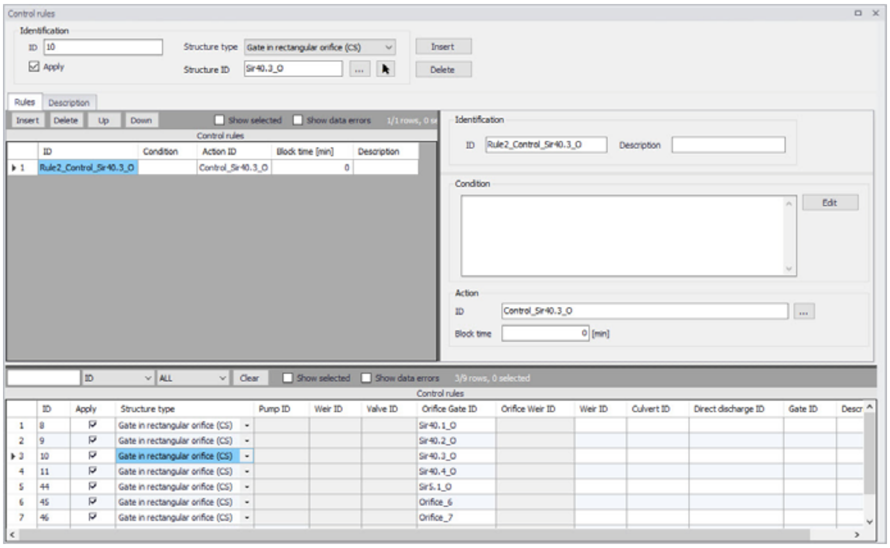


Figure 9.10 The Control rules Editor

The operational control rules for controllable devices are specified in this editor. The control is specified as a set of rules linking logical conditions and control actions. The rules are evaluated sequentially following the rules list sequence.

9.7.1 Identification

Specify the ID of the control rule in the Identification group box.

Use the ‘Insert’ button to add a new control rule configuration in the project.

Define the type of controlled device for which to apply the control rule configuration in 'Structure type' list. The types 'Gate in rectangular orifice' and 'Weir in rectangular orifice' apply to orifices on the collection system network. Therefore, a 'Weir in rectangular orifice' type will control an orifice, and not an ordinary weir.

The difference between a ‘Gate in rectangular orifice’ and ‘Weir in rectangular orifice’ is that the gate blade edge moves downwards from the top of the orifice (to the bottom), while the weir blade edge moves upwards from the bottom of the orifice.

When the device 'Type' has been selected, the controlled structure ID must be specified

The 'Apply' option controls whether the active control rules configuration is active during the simulation or not.

9.7.2 Rules

The various rules applying to the current structure are specified in the Rules tab.

Add and remove rules using the Insert and Delete buttons above the secondary table on the left.

For each rule, an ID and a description may be supplied in the Identification group, and an action must be selected.

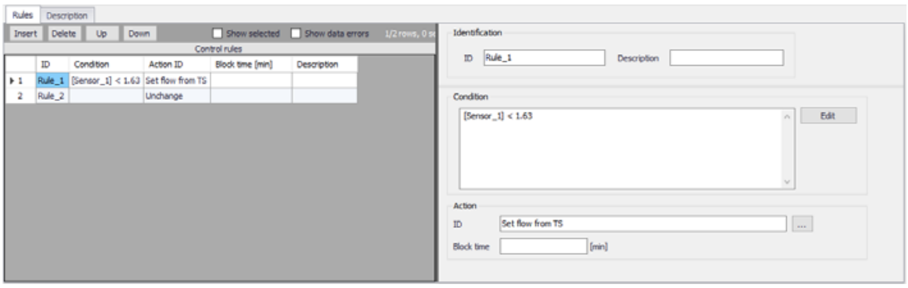


Figure 9.11 Selection of conditions and actions defining the rules

Any number of rules can be specified to control the device. When multiple rules apply, the rules are sequentially evaluated starting from the top, at every time step of the simulation. This means that appropriate sequence of rules is essential for the achievement of the desired control logic.

Use the 'Up' and 'Down' buttons above the secondary grid to modify the order of the Rules. When multiple rules are included, they must be associated to a 'Condition', controlling when a rule applies.

Evaluation of a logical condition belonging to a rule as 'TRUE', leads to the selection of the associated action. If a logical condition is 'FALSE', the evaluation proceeds to the next rule on the list.

If no logical condition is specified, the rule is unconditionally evaluated as 'TRUE'. This implies that the last rule in the sequence must not include any logical conditions in order to ensure a selection of a 'Default' control action if all previous specified conditions are found 'FALSE'.

Condition

A logical condition demarcates the boundaries of a certain operational situation in the controlled system. This frame consists of an arbitrary number of independent logical tests on the various operational variables, where the relation of the actual value (or state) of the variable (provided by a sensor) is tested against the specified threshold (limit) value (or state). The individual tests are evaluated as TRUE or FALSE, with the outcome depending on the actual variable value (or state), the threshold and the specified operator.

A condition is specified as a mathematical expression in the Condition input box is the editor (Figure 9.12).

Figure 9.12 The Condition input box in the Rules Tab

The implementation of mathematical expressions in MIKE+ is very general, supporting a comprehensive set of mathematical functions and operators. Click on the Edit button to the right of the Condition input box to launch the Expression editor.

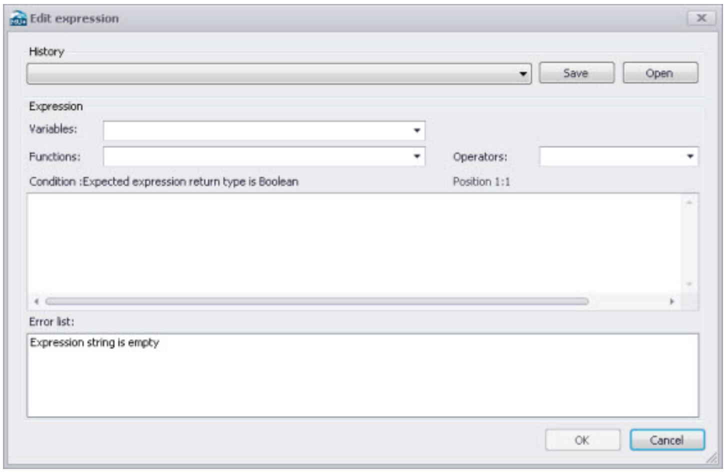


Figure 9.13 The Expression Editor

Mathematical expressions are formulas or expressions trees. The formulas support the standard mathematical functions like sine, cosine, abs, power etc. On top of these standard mathematical functions, a number of specific MIKE 1D engine control functions are available.

Note that a Boolean data type (i.e. TRUE, FALSE) is expected from the evaluation of the expression for Conditions.

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 'Operators' 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 ID

The corresponding Action for a given Condition. Options are taken from the Actions editor. The choice of the controllable device determines relevant control actions.

Block Time

The Block Time ensures that a certain rule is applied for a minimum period. This means that the system is locked to a certain rule in a period equal to the block time after it is activated, even if the condition is no longer TRUE.

Some notes when specifying rules are:

- All control actions used to control a specific device must be of the same function type - corresponding to the specified device Type and Control Type.
- For PID control, all control actions must refer to the same setpoint sensor (i.e. Input). Changing the setpoint sensor during simulation is not allowed.

PID Settings

When starting a simulation, the system checks if these conditions are fulfilled and in the case of any violation, the simulation will not start.

9.7.3 Description

One may include a description of the control rules configuration in the Description tab of the editor. An image file (e.g. of the controlled device) may also be added using the 'Add picture' button.

Figure 9.14 The Description Tab in the Control rules Editor

9.7.4 Difference between weir and weir in orifice

It is possible to define weirs in two different ways. One way is through the 'Weir' dialog, either on a collection system or on a river network, which we will refer to as an 'ordinary weir'. The second way is to use the combination of an orifice and the controllable device type 'Weir in rectangular orifice'.

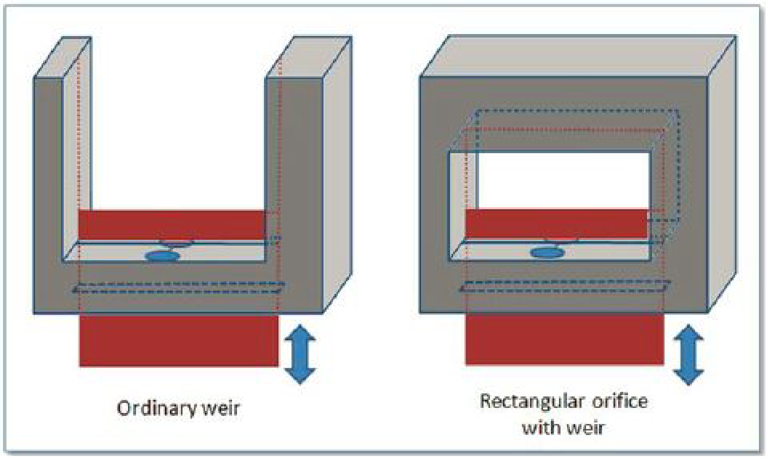


Figure 9.15 Difference between ordinary weir and a weir in rectangular orifice

The main difference is that the 'Weir in rectangular orifice' can close the orifice completely while the ordinary weir is always open upwards. It has no ceiling and in principle the flow can always pass over the weir if the water level is higher than the weir crest level.

For the 'Weir in rectangular orifice', once the weir fully closes the orifice then no flow will pass the weir even if the water level is above the crest level.

9.8 Control Rules Computations

A simulation with control rules is started from the usual Simulation Setup editor (Simulation Specifications | Simulation Setup).

Control rules computations are performed if the 'Control rules' module is active on the 'Model type' editor (General Settings | Model type) (Figure 9.16.).

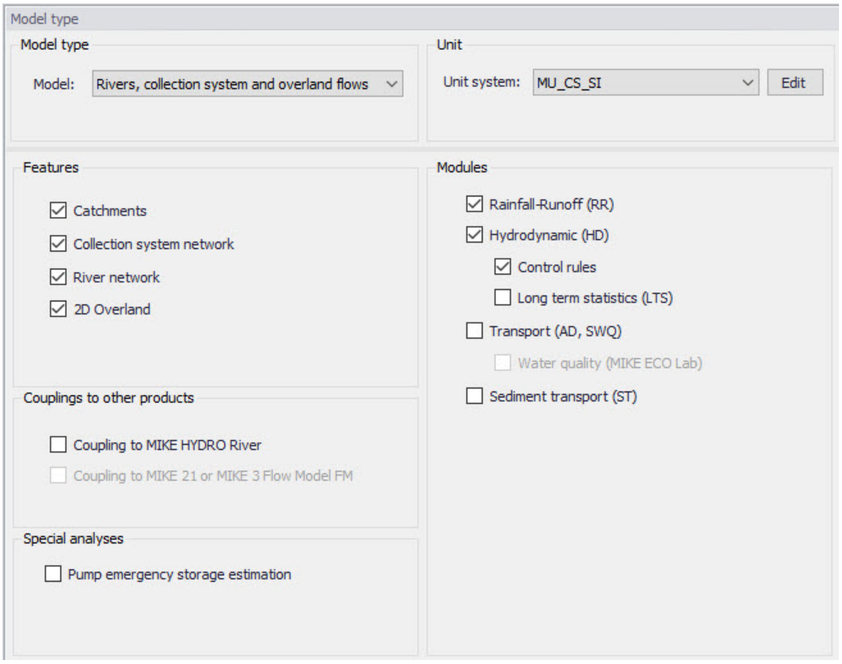


Figure 9.16 Activate the Control rules module on the Model type Editor

10 Long-Term Statistics (LTS)

The Long-Term Statistics (LTS) module performs a time-efficient simulation of wastewater collection and urban drainage systems exposed to intermittent hydrological inputs over long, continuous historical periods and the computation of relevant statistics for the operational variables (instantaneous and accumulated) of interest.

LTS features an easy way to convert a usual simulation covering a long historical period (typically 10 - 30 years) into a discontinuous series of relevant hydrological events and to define various event-based and annual statistics for the selected variables. By these means, the long-term simulation becomes feasible even for large models, as both the simulation time and the amount of the generated output data gets significantly reduced, compared to a continuous simulation over the same historical period.

Based on the specified criteria, the system automatically selects the periods for dynamic pipe flow simulations during wet weather. This reduces the simulation time significantly, while preserving the accuracy in the simulation of relevant dynamic effects in the system - CSOs, surcharges, pollution emissions, etc.

Running LTS simulations with the existing drainage system configuration and with the planned upgrades, impacts of the planned investments (e.g. new sewers, retention tanks, control schemes) on system performance can be reliably tested and evaluated based on the computed operational statistics.

This allows the user to diagnose the current operation of the system and to develop the optimal rehabilitation/upgrade strategy, e.g. for satisfying the requirements of environmental regulations.

Statistics on maximum water levels, discharges from weirs, pollution emissions, etc. generated by simulating long continuous periods subject to impact of recorded historical rainfalls is particularly useful for the evaluation of the simulated system functionality. Statistics are used to compare the effects of various mitigation measures in the planned rehabilitation process, the performance of various control strategies, the cost-benefit efficiency of the planned investments, as well as to determine whether the functionality of the sewer system is compliant with legislative requirements.

The functionality of urban drainage systems is often subject to requirements like:

- Flooding on terrain must not happen more frequently than once in every T-years.
- Flooding of basements must not happen more frequently than once in every T-years.

- Discharge from CSO weirs to receiving waters must not happen more frequently than n times per year.
- Total volume from CSOs must not exceed a certain threshold amount per year.

LTS answers these and other similar questions and is thereby an essential simulation tool for reliable assessment of drainage system performance.

Statistics on some operational variables are only reliable if the simulated number of independent events is sufficiently large, i.e. if the simulated historical period is sufficiently long. This, in turn, may imply very long simulation times (if 'normal' computing hardware is applied) and thereby spoil the feasibility of the analysis.

However, use of dynamic simulation for the hydraulic and pollution transport analysis of drainage networks is only beneficial in periods of high-flow dynamics. Otherwise, simpler and much faster simulation tools can be used with a similar level of accuracy. Since high-flow dynamics in urban drainage networks are normally associated with rainfall events, the dynamically simulated periods can be reduced to include wet weather periods only, i.e. only a few percent of the historical period included in the analysis.

The fundamental principle behind the LTS concept is therefore the ability to reduce the simulation time without losing significant information. LTS can, based on the user-specified criteria, eliminate the irrelevant periods from the simulated series and to limit the dynamic simulations to a series of disconnected events instead of the entire simulation period.

10.1 Data Input

10.1.1 LTS Job List

LTS Job list specifies intervals within the LTS simulation period that are relevant for dynamic network simulation. Duration of each job (i.e. simulation event) is defined by its START time and END time.

A job list is generated by the LTS computational engine on the basis of model setup and boundary files, the simulation period (specified in the Simulation Setup Editor, General Tab), optionally the LTS initiation hot start file(s) and user-specified job list criteria.

The job list is written into an ASCII file, recognizable by the extension *.JLF.

The job list file includes the following information:

- General information on the actual simulation setup
- List of relevant boundary conditions

- Overview over job list criteria
- Job list events, defined by
  - Start time
  - End time
  - Hot start file name and hot start time
  - Triggering job list criteria
- Job List Summary

10.1.2 Job List Criteria

Job list criteria are defined as hydraulic loads thresholds on the network, including dry weather flow (i.e. wastewater), rainfall and other loads, with the purpose of identifying and delimiting wet weather periods relevant for dynamic (network) simulation. Each job list criterion consist of a job START criterion and a job STOP criterion.

Job START criterion selects rainfall events to be included in the LTS job list, by comparing the inflow generated by all hydraulic loads to the specified part of the model (location) with the specified start-threshold value: if the threshold is exceeded continuously for at least the specified duration, the event is included in the job list. The simulation event start time is set at exactly the beginning of the identified rainfall event.

The location can be specified as an individual node, part of the system defined by a selection list or the system as the whole (General).

The START threshold value should be specified as an estimate of the hydraulic load peak just below the load that is likely to cause the operational effects of interest (e.g. overflows, inundations, activation of retention basins, etc.). By these means, many small, potentially irrelevant rainfall events are eliminated from the LTS simulation, making it feasible and time-efficient. Anyway, the specified threshold should be set sufficiently low to ensure that none of the relevant events is omitted from the simulation.

"Duration" is included in the evaluation to avoid inclusion of very short events that, despite a relatively high peak, are not likely to cause significant hydraulic effects in the system.

If multiple job list criteria are specified and activated, at least one active START criterion must be fulfilled for the event to be included in the job list.

Job STOP criterion defines the anticipated end of dynamic simulation (i.e. end-time of the simulation event). The end-time is set by comparing the inflow generated by all hydraulic loads to the specified part of the model with the specified end-threshold value: simulation end-time is set at the time when the load falls below the specified threshold and remains lower than the threshold continuously for at least the specified duration.

The simulation event end time may be extended by (optional) runtime stop criteria.

"Duration" ensures that very short drops in inflow are ignored, i.e. that the inflow load has definitely dropped below the threshold. Also, appropriately set "duration" may secure that the time offset between the actual hydraulic load and its effects in the system is included in the dynamic simulation. In the large systems including significant volumes, this offset may be quite long (e.g. emptying od a large retention basin may take many hours or even days). In such cases, instead of attempting to capture this by setting a long duration for the job list stop criterium, runtime criteria should be applied.

If multiple job list criteria are specified and activated, all active STOP criteria must be fulfilled at the simulation event end-time.

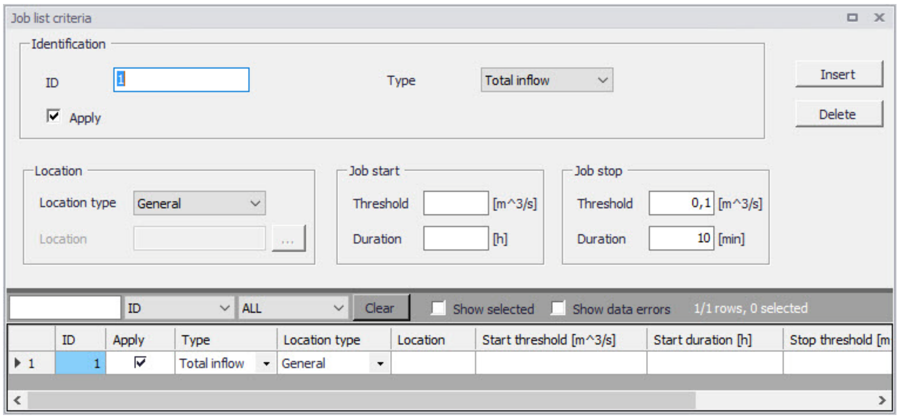


Figure 10.1 The Job List Criteria Editor.
LTS simulation requires definition of at least one set of job list criteria

Table 10.1 Job List Criteria editor attributes (Table msm_LTSJobListCriteria)

Edit field	Description	Used or required by simulation	Field name in data structure
ID	Job list criteria identifier. Up to 40 characters and case sensitive.	Yes	MUID
Type	Controls the type of criterion to be specified. In this version only “Total Inflow” can be selected	Yes	ConditionNo

Table 10.1 Job List Criteria editor attributes (Table msm_LTSJobListCriteria)

Edit field	Description	Used or required by simulation	Field name in data structure
Apply checkbox	Checkbox for activating or deactivating a Job list criteria	Yes	ApplyNo
Location Type	For the criteria of type ‘Total Inflow’, it must be specified for which part of the system the inflow must be evaluated. ‘Location type’ can be: ‘General’ (the whole system), ‘List’ (a list of elements), or ‘Individual’ (a single node)	Yes	LocationNo
Location	For the types ‘List’ and ‘Individual’, additional information must be specified in the ‘Location’ field. For the ‘List’, a Selection List must be specified. For ‘Individual’, a single node name must be types or selected from the node list.	Yes, except LocationNo=1 (General)	LocationID
Job Start Threshold	Defines the threshold that must be exceeded in order to evaluate the job start criterion as true.	Yes	StartValue

Table 10.1 Job List Criteria editor attributes (Table msm_LTSJobListCriteria)

Edit field	Description	Used or required by simulation	Field name in data structure
Job Start Duration	Criteria of type 'Total Inflow' can optionally be extended by specifying the duration of a continuous period in which the threshold must be exceeded in order to evaluate the criterion as true. The Default duration is zero.	Yes	StartTime
Job Stop Threshold	Defines the lower threshold that must be achieved in order to evaluate the job stop criterion as true.	Yes	StopValue
Job Stop Duration	Specifies the duration of a continuous period in which the stopping threshold must be fulfilled in order to evaluate the stop criterion as true. The Default duration is zero.	Yes	StopTime

10.1.3 Initial Conditions for Simulated Events

Appropriate initial conditions for the individual LTS simulation events are essential for achieving realistic statistics in hydrodynamic LTS simulations.

LTS provides two different methods for initializing the network model: The network model can either be initialized in an empty state before each job (Default), or a set of hot start files can be provided. If the latter option is used, then the specific hot start file used for a simulation event is selected based on the actual hydrological inflow to the system at the start of the event.

When applying the 'Empty system' method, the model will for all simulated events be initiated as empty, similarly as in a normal network simulation without hot start. This option is valid only for storm drainage systems where there is no water in the system during dry periods. It is not recommended to use this option in any other situation.

The method with hot start accounts for the correct initial conditions related both to the amount of infiltration and DWF load at the simulation event start time.

If a set of hot start files is provided, each hot start file must be associated with an inflow interval for total hydrological inflows. This interval is the "validity interval", meaning that the specified hot start file is valid (i.e. the file will be used) for all simulation events in the Job List where the total hydrological inflow (i.e. inflow generated by any of the hydrological models) to the network at the beginning of the actual simulation event is within that interval.

Note that for the surface runoff models, the hydrological inflows will be zero at the beginning of the simulation event in most cases. This implies that correct initial conditions may be achieved with only one hot start file, covering any possible inflow, defined as the inflow interval including zero inflow (e.g. from 0 m^3/s to 10 m^3/s). This hot start file is created by simulating the system loaded by any recurring (typically wastewater) loads, over at least one day.

In model setups including continuous hydrological models (RDI), multiple hot start files are needed in order to cover different infiltration levels that may occur at the start of a simulation event. E.g. if it is known that infiltration may vary between zero and 2 m^3/s, a set of 4 hot start files may be prepared, covering the following infiltration intervals: 0 -> 0.5 m^3/s, 0.5 -> 1.0 m^3/s, 1.0 -> 1.5 m^3/s, 1.5 -> 2.0 m^3/s. These files are created by simulating the system loaded by any recurring (typically wastewater) loads and constant infiltration loads of 0.25 m^3/s, 0.75 m^3/s, 1.25 m^3/s and 1.75 m^3/s, over at least one day. The infiltration loads should be distributed proportionally to the catchment area.

If the total inflow to the system at the beginning of job is outside of all specified validity intervals provided for the hot start files, the system will be initialized with the hot start file that is the closest to the actual inflow.

In order to account for diurnal inflow variations, the time-of-day at which the values from the hot start file should be used is set equal to the simulation event start time. However, the hot start date must be provided by the user. This is because the hot start file may cover two or more days, including the system filling phase. The date specified for the hot start must relate to the day following the completion of the filling phase.

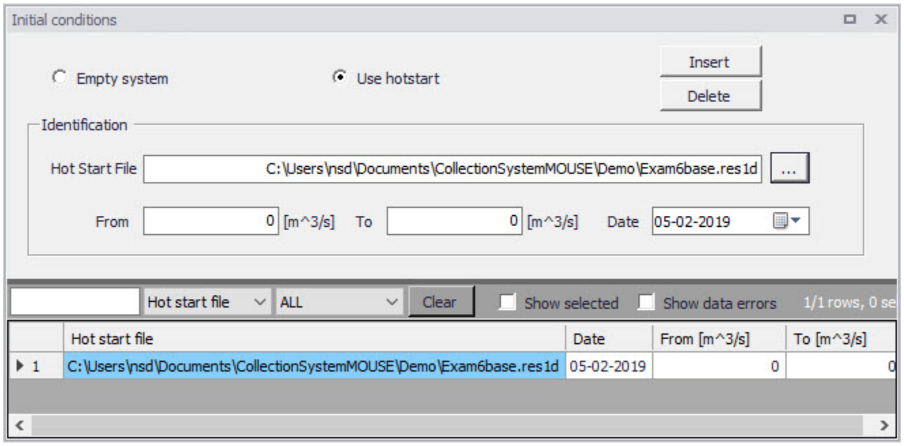


Figure 10.2   Initial Conditions Editor

Table 10.2   Initial Conditions editor attributes (Table msm_LTSInit)

Edit field	Description	Used or required by simulation	Field name in data structure
Empty System/Use Hotstart radio button	Radio buttons for selecting the initial conditions mode	Yes	-
Hotstart File	Specifies a hot start file to be used for the system initialization in association with the specified range of total inflows (excluding DWF) detected at the beginning of certain job.	Yes, if Hotstart	HotStartFilename
From	Defines a lower threshold for total inflow intervals (excluding DWF) used at the start of each job in the Job List to evaluate what hot start parameters to use.	Yes, if Hotstart	InitFrom

Table 10.2 Initial Conditions editor attributes (Table msm_LTSInit)

Edit field	Description	Used or required by simulation	Field name in data structure
To	Defines an upper threshold for total inflow intervals (excluding DWF) used at the start of each job in the Job List to evaluate what hot start parameters to use.	Yes, if Hotstart	InitTo
Date	Specifies the date in the hot start file to be used in the search for the hot start conditions associated with the specified range of total inflows (excluding DWF) detected at the beginning of a job.	Yes, if Hotstart	InitDate

10.1.4 Generating and editing Job Lists

A Job List is created based on the Job List Criteria, simulation input files, simulation period (specified in the Simulation Setup Editor, General Tab) and, optionally, the LTS initiation hot start file.

A Job List is created using the Create Job List function in the Simulation Setup Editor (Simulation Setup | LTS Tab).

The simulation periods for individual events in the resulting Job List represent the minimum simulation time (i.e. preliminary), which may be extended during run time when evaluating the Run Time Criteria.

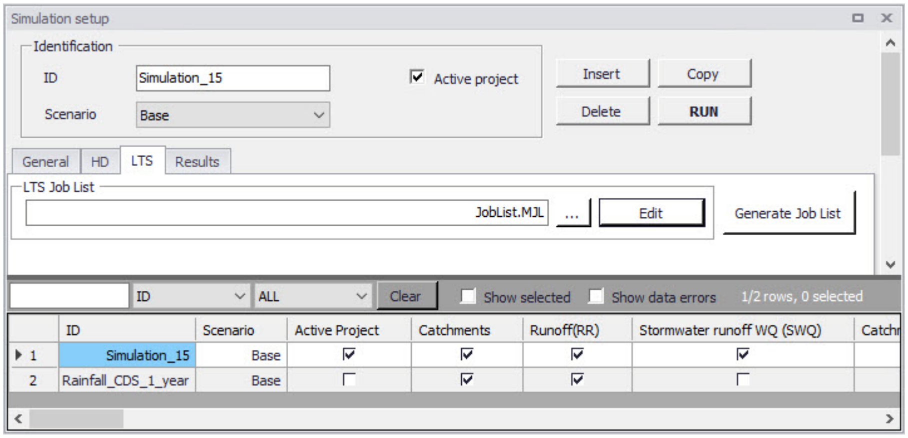


Figure 10.3 Generate Job List option in the Simulation Setup Editor LTS Tab

There is no dedicated dialog available for reviewing and editing job list files. Instead, Windows Notepad is used. Alternatively, any ASCII editor can be used.

```
Sirius_LTSfeature_1_CD0001_area2Base.MJL - Notepad
File  Edit  Format  View  Help
// Created      : 2019-09-5 10:9:54
// DLL          : C:\Program Files (x86)\DHI\MIKE URBAN\2019\bin\x64\pfs2004.DLL
// Version      : 18.0.0.13246

[MIKE1D_JOB_LIST]
   [SIMULATION_SETUP]
      Simulation_ID = 'Sirius_LTSfeature_1_CD0001_area2'
      Scenario_ID = 'Base'
      Simulation_start = 1933, 7, 22, 16, 31, 0
      Simulation_end = 1934, 5, 19, 14, 50, 0
      Live_catchments = true
   EndSect  // SIMULATION_SETUP

   [BOUNDARY_CONDITIONS]
      Catchment_source_boundaries = 'Catch_discharge'
      Global_source_boundaries = 'Rainfall'
   EndSect  // BOUNDARY_CONDITIONS

   [LTS_SETUP]
      [INITIAL_CONDITIONS]
         Hotstart1 = 'Hot_startLTSBase.res1d', 0.005, 0.01, '2019-01-01 00:00:00'
      EndSect  // INITIAL_CONDITIONS

      [JOB_START_CRITERIA]
         JL_Criterium_2 = 0.1, '00:05:00', 'SingleNode'
         JL_Criterium_1 = 0.15, '00:03:00', 'SingleNode'
      EndSect  // JOB_START_CRITERIA

      [JOB_STOP_CRITERIA]
         JL_Criterium_2 = 0.02, '00:10:00', 'SingleNode'
         JL_Criterium_1 = 0.02, '00:10:00', 'SingleNode'
      EndSect  // JOB_STOP_CRITERIA

   EndSect  // LTS_SETUP

   [SIMULATION_EVENT]
      Simulation_start = '1933-07-22 16:31:00'
      Simulation_end = '1933-07-22 18:49:00'
      Simulation_end_no_duration = '1933-07-22 18:39:00'
      Hotstart_file = 'Hot_startLTSBase.res1d'
      Hotstart_time = '2019-01-01 16:31:00'
      Hotstart_hydrological_inflow = 0.0
      Duration = '02:18:00'
      Job_number = 1
      Job_start_criterion = 'JL_Criterium_1'
      Job_stop_criterion = 'JL_Criterium_2'
   EndSect  // SIMULATION_EVENT

   [SIMULATION_EVENT]
```

Figure 10.4 Example Job List file

Optionally, the Job List file can be edited before starting the LTS calculation.

A job list file includes several info sections:

- SIMULATION_SETUP
- BOUNDARY_CONDITIONS
- LTS_SETUP

- SUMMARY

Actual simulation events are defined in sections "SIMULATION_EVENT" (one section for each event). See example:

```
[SIMULATION_EVENT]
      Simulation_sTart = '1933-07-22 16:31:00'
      Simulation_end = '1933-07-22 18:49:00'
      Simulation_end_no_duration = '1933-07-22 18:39:00'
      Hotstart_file = 'Hot_startLTSBase.res1d'
      Hotstart_time = '2019-01-01 16:31:00'
      Hotstart_hydrological_inflow = 0.0
      Duration = '02:18:00'
      Job_number = 1
      Job_start_criterion = 'JL_Criterium_1'
      Job_stop_criterion = 'JL_Criterium_2'
      DtMin = 10.0
      DtMax = 10.0
EndSect  // SIMULATION_EVENT
```

Some of the parameters in the simulation event definition may be edited:

Simulation_start: defines start time of the simulation event,

Simulation_end: defines end time of the simulation event,

Simulation_end_no_duration: defines end time of the simulation event, without "duration". This is of relevance when run-time criteria are included, i.e. defines the time when evaluation of run-time criteria commences.

Hotstart_file: name of the hot start file for the actual simulation event

Hotstart_time: hot start time for the actual simulation event

DtMin: minimum time step [s] for network simulation in the actual simulation event

DtMax: maximum time step [s] for network simulation in the actual simulation event

The remaining parameters are for information only, and editing would not affect the simulation.

10.1.5 Run Time Stop Criteria

The simulation event end time may optionally be subject to further evaluation and possible extension by a set of Run Time Criteria during the simulation.

Run time Criteria are founded on the evaluation of the operational variables within the network itself, which can potentially extend the simulation beyond the end time defined in the Job List. Since the Run Time Criteria are evaluated during the simulation, it is not possible to determine the exact duration of the dynamic simulation in advance.

LTS Run Time Stop Criteria define threshold values to be evaluated during computations (i.e. run time) in order to determine the earliest time at which the simulation can be stopped without missing important information in the closing phase of the simulated event. This may include e.g. emptying of retention basins and, generally, return of the system to dry weather situation.

The following types of Run Time Stop criteria are available:

- Outflow - Inflow

  The threshold value represents an absolute value of a difference between the total inflow into the system and the total outflow from the system. During rain events, this difference is relatively big due to the dynamic effects in the network: outflows are normally attenuated and delayed in the network. In dry weather, this difference is typically very small and relates to the attenuation and transport time of diurnally varying wastewater loads.

  This criteria is defined for the entire system only (Location = General). It is used to identify return of the system to dry weather situation.

- Outflow

  The threshold value represents outflow from the system at a specified location. The location can be specified as general, list and individual, separately for various types of outlets (outlets, pumps, weirs, orifices, valves).

  This criterium can be used to identify return of the system to dry weather situation, end of overflow, end of pumping from a retention basin, etc.

- Total volume

  The threshold value represents volume of water contained in the model elements specified as location. The location can be specified as general, list and individual.

  Knowing the water volume in the specified location in dry weather situation, this criterium can be used to identify return of the system to dry weather situation, completed emptying of a basin, etc.

- Filling degree

  The threshold value represents the filling degree (fraction of full system volume) of entire system.

This criterium can be used to identify return of the system to dry weather situation, end of overflow, etc.

- Depth

  This threshold represents water depth in a node specified as location. If location is specified as List of nodes, then the highest depth in all included nodes is evaluated against the specified threshold.

  This criterium can be used to identify completed emptying of a retention basin.

- Local flow

  This threshold represents flow at a specified individual conduit (link) or structure (weir, orifice, pump, valve) anywhere in the system, i.e. not limited to outlets.

The "Duration", associated with each run time criteria, ensures that the criterium has been fulfilled continuously over the specified duration, and that it is not just a short instance in oscillating or varying variable.

The Run Time Stop Criteria for the event being simulated are evaluated throughout the simulation. As soon as some of run time criterium is fulfilled, the time counter for the specified duration is activated. The simulation will stop only after all active run time criteria are fulfilled, but never before the "Simulation_end" time specified in the job list.

For better understanding, figures below illustrate possible situations.

In the depicted example, three active job list criteria are specified (JL1, JL2 and JL3), each with different "duration" parameter. During job list generation, the LTS engine has determined the simulation start time (equal for all three criteria, if the rainfall event exceeds the thresholds specified for all three criteria). Also, the simulation end times are determined for each criterium. The latest of the three "simulation end" times (JL2) is selected and written in the job list.

During the simulation, the three run time criteria (RT, RT2 and RT3) are continuously evaluated. Whenever one of the run time thresholds has been achieved, the duration counter for that criterium is activated. The simulation continues at least until the "simulation_end". If at that time there are any run time criteria still not fulfilled (including the specified duration), the simulation will continue until all run time criteria are satisfied.

In example 1 (Figure 10.5), when the simulation reaches the "job list simulation end", the run time criterium RT2 is still not fulfilled for the specified "duration". Therefore, the simulation continues beyond the "job list simulation end" and stop only after RT2 criterium has been satisfied.

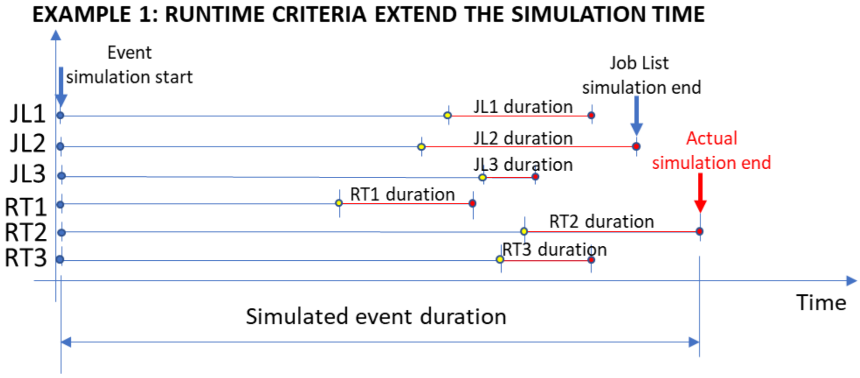


Figure 10.5 Run Time criterium RT2 extends the simulation end time determined by job list.

In example 2 (Figure 10.6), all the run time criteria (including duration) have been satisfied before the simulation reaches the "job list simulation end". Therefore, the simulation ends at the "job list simulation end".

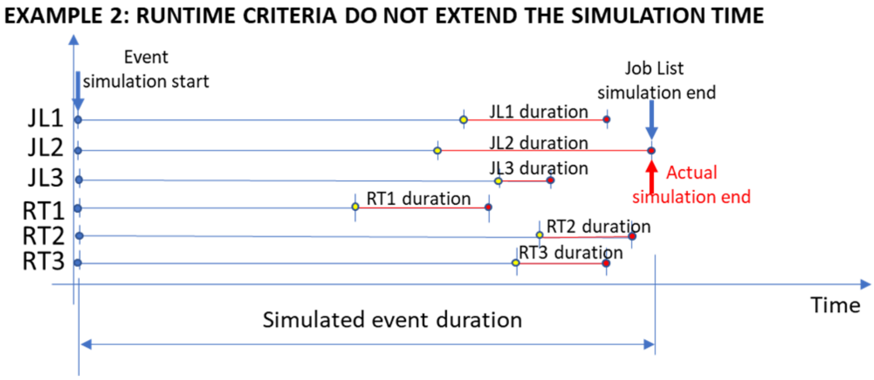


Figure 10.6 Run time criteria do not affect the "job list simulation end"

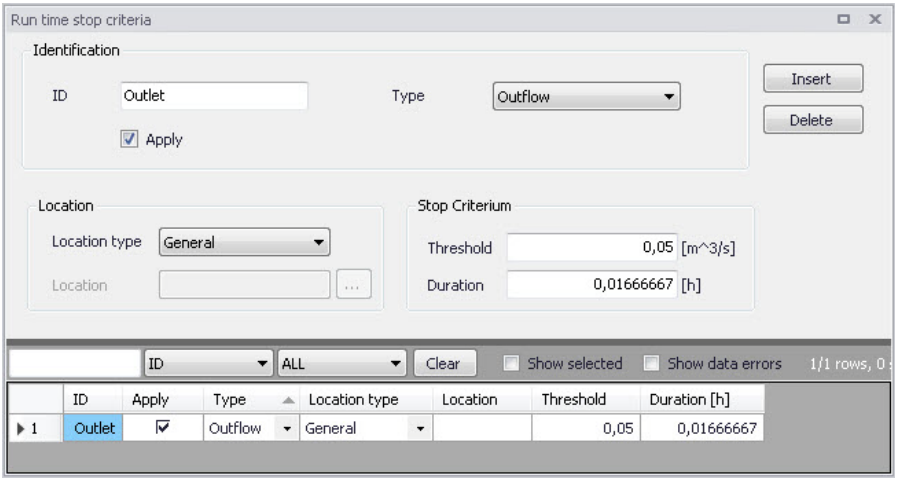


Figure 10.7 The Run Time Stop Criteria Editor

Table 10.3 Run Time Stop Criteria database attributes (Table msm_LTSRunS)

Edit field	Description	Used or required by simulation	Field name in data structure
ID	Unique name for a Run Time Stop criterion Each criterion is identified with a unique indentifier, which is used as a reference in the evaluation matrix.	Yes	MUID
Type	Controls the type of criterion to be specified: - Inflow-Outflow (difference between inflow and outflow), - Outflow (total outflow), - Total Volume (total volume stored in the system), - Filling degree (filling degree in the system), - Depth, - Local flow	Yes	ConditionNo
Apply checkbox	Checkbox for activating or deactivating a Run Time Stop Criteria set	Yes	ApplyNo

Table 10.3 Run Time Stop Criteria database attributes (Table msm_LTSRunS)

Edit field	Description	Used or required by simulation	Field name in data structure
Location Type	Defines for which part of the system the run time stop criteria must be evaluated. Location type can be: - General (the whole system), - List (a list of elements), - Node, - Outlet, - Weir, - Orifice, - Pump, - Valve, - Link	Yes	LocationNo
Location	For the ‘List’ Location Type, a Selection List must be specified. For individual element location types, a single element ID must be specified or selected from the element list.	Yes, Except for Location Type = General	LocationID
Threshold	The residual value of the variable defined by the ‘Type’ that must be achieved in order to evaluate the criterion as true	Yes	StopValue
Duration	A period in which the parameter value must be below the threshold in order to evaluate the criterion as true.	Yes	StopTime

10.1.6 LTS Global Parameters - Event Definitions

In terms of events statistics, MIKE+ LTS distinguishes events associated with extremes (maxima) of instantaneous variables (e.g. water level, discharge, mass transport), and intermittent (i.e. discontinuous) events where accumulated values (e.g. volume, pollution mass) and durations are calculated.

In order to perform statistical analysis on extreme events (maxima), MIKE+ LTS analyses the respective time series and identifies independent extreme events. For instantaneous variables (e.g. water level, discharge, concentration, etc.), the event identification must consider time between consecutive extreme events and the "depth" of the local minima in order to eliminate dependent maxima and "noise" caused by numerical instabilities, a nearby pump action, etc. In other words, MIKE+ LTS applies two interevent criteria to identify independent extreme events:

1. Interevent time criterion dT_c: Two successive events (i.e. peak occurrences) are considered independent if the time between the two events is larger than dT_c. This parameter is considered for all statistics of instantaneous values, i.e. level, flow, velocity, and concentration.
2. Interevent level criterion p_c ($0 < p_c < 1$): Two successive events are independent if the level between the events is smaller than pc times the lower of the two events. This is a threshold for the ratio between the lowest minimum value between two peaks and the lower peak.

MIKE+ LTS considers two successive events as independent only if both criteria (1) and (2) are fulfilled. This means that two peaks are independent if the time between them is longer than dT_c AND if the lowest minimum between the two peaks is smaller than the value of the smaller peak multiplied with p_c.

Both for peak statistics and for accumulated flow statistics, two events are always considered as independent if the computation has been stopped in between by the specified stop criteria. Thus, specification of a very long *dT* - longer than any individual event to be simulated would result in the number of peak values and the number of accumulated discharges and duration corresponding exactly to the number of simulated events.

Each MIKE+ LTS simulation job contains at least one statistical event. In some cases, several events may be identified within one MIKE+ LTS Job.

The interevent time criterion dT_c is user-specified through the LTS Global Parameters editor, separately for instantaneous (Q, H, v, C) and accumulated (V, T, M) values. The interevent level criterion pc is also specified through the LTS Global Parameters editor.

Note that the default value 0.75 only removes "noise" of relatively small amplitude as well as prevents false extremes in case of a flat time series. Application of smaller values would reduce the number of identified independent events.

Note that for some variables (e.g. discharge) the specified pc applies for the actual value of the variable. In other cases, e.g. water levels, pc is applied on local water depth, i.e. water level subtracted local invert level. In principle, all variables are offset so that the minimum possible local value is zero.

Definition of independent extreme events for instantaneous variables is illustrated in several examples below.

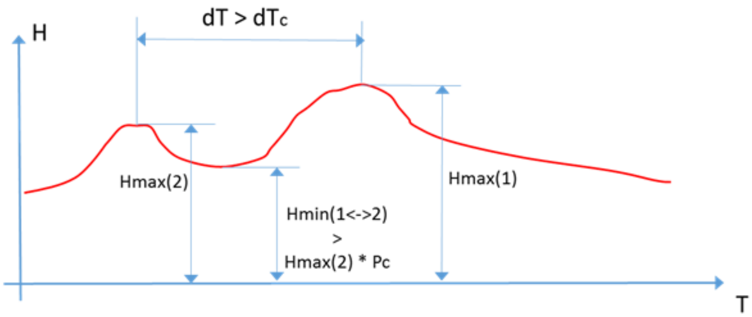


Figure 10.8 The two water level (i.e. depth) peaks in the time series are grouped under one event. The higher of the two depth peaks is clearly one extreme event in the current simulation. The other peak (Hmax(2)) is also a candidate for extreme depth event, because it occurs before the highest peak longer that the specified dTc criterion. But evaluation of the level criterion is negative - the "bottom" between the two peaks is higher that the specified criterion pc. Therefore, Hmax(2) is not considered a separate, independent event and is considered as belonging to the same event leading to the true peak Hmax(1).

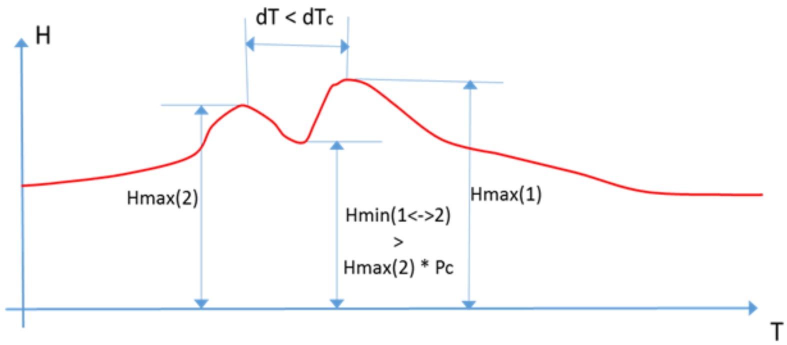


Figure 10.9 The two water level (i.e. depth) peaks in the time series are grouped under one event. The higher of the two peaks is clearly one extreme event in the current simulation. The other peak (Hmax(2)) is also a candidate for extreme depth event, but evaluation of both criteria is negative - this peak occurs too close to the highest peak and the "bottom" between the two peaks is higher that the specified criterion pc. Therefore, Hmax(2) is excluded as a separate, independent event. Rather, it is considered as belonging to the same event leading to the true peak Hmax(1).

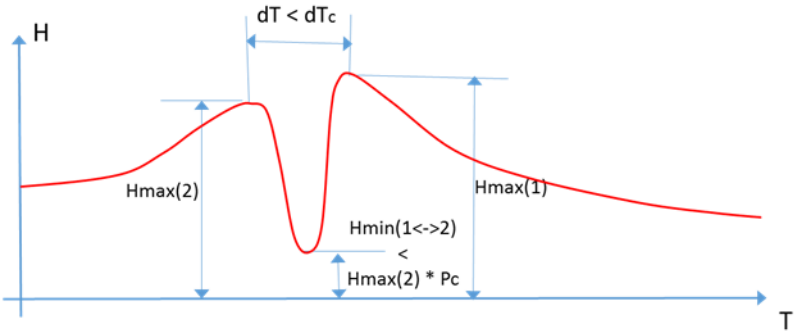


Figure 10.10 The two water level (i.e. depth) peaks in the time series are grouped under one event. The higher of the two depths (Hmax(1)) is clearly an extreme event in the current simulation. The other peak (Hmax(2)) is also a candidate for extreme depth event, but evaluation of dTc criterion is negative - this peak occurs too close to the highest peak. Therefore, Hmax(2) is excluded as a separate, independent event and is considered as belonging to the same event leading to the true peak Hmax(1).

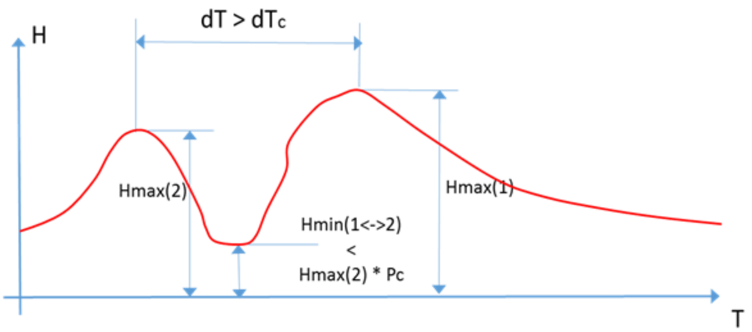


Figure 10.11 The two water level (i.e. depth) peaks in the time series are identified as two separate events. The higher of the two peaks (Hmax(1)) is clearly an extreme event in the current simulation. The other peak (Hmax(2)) is also a candidate for extreme depth event. Both criteria are evaluated positively - this peak occurs long time before the highest peak (dT > dTc) and the "bottom" between the two peaks is lower that the specified criterion pc. Therefore, Hmax(2) is also identified as a separate, independent event.

To identify intermittent events and statistics on accumulated variables (e.g. volume, duration, and mass), only the interevent time criterion dT_c applies. Consecutive independent events are separated by time intervals longer than dT_c, where the underlying instantaneous variable (e.g. discharge for volume, mass transport for mass) is below the specified threshold (default value for the threshold is zero). E.g. two overflow events (i.e. overflow spills) are considered as independent overflow events if there is zero flow between them for a time longer than dT_c.

A threshold value other than zero is relevant for inherently continuous variables, e.g. discharge in pipes: even if the pipe is empty, due to the requirements of the algorithm, a small "numeric" discharge in the pipe will be reported. In order to consider such periods as "dry", the user may want to specify a small discharge threshold other than zero.

This feature is offered via the 'LTS_DISCHARGE_THRESHOLD' parameter in the MIKE 1D engine configuration dialog accessed through the Simulation menu ribbon.

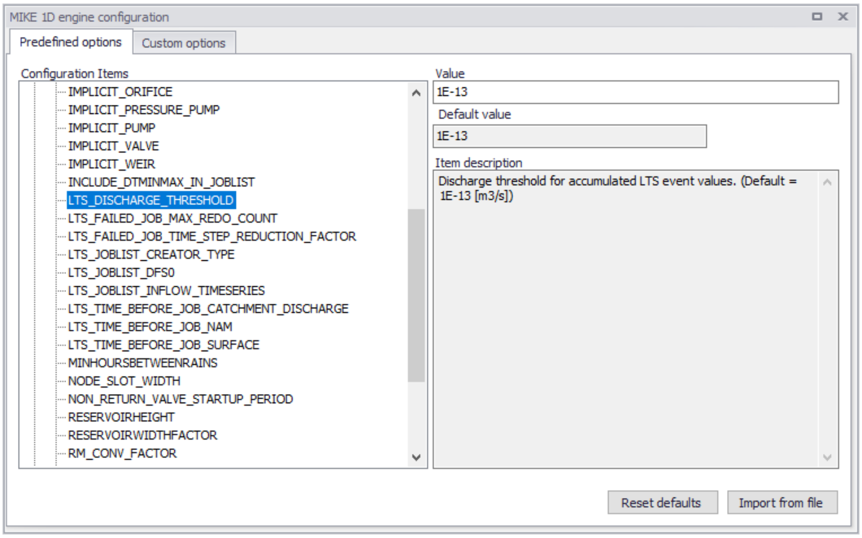


Figure 10.12 The MIKE 1D Engine Configuration dialog

Override the Default value of 1E-13 for the parameter by specifying a value in the Value input box.

Examples of identifying independent events are shown in Figure 10.13.

For some statistics parameters, the number of events during a long LTS computation can be very high and the statistics result file size can grow rapidly.

Therefore, the statistical computation can be limited to a specified number of highest events, i.e. only the limited number of most significant events is included into the calculation. This is possible if the purpose of statistical calculation is to focus only on extreme events. In such a case, all insignificant events may be eliminated from the statistics as "irrelevant".

The maximum number of events must be specified within the range 1-5000.

The main parameters which control the event definition for various types of statistics are specified in the LTS Global Parameters editor (Figure 10.14).

This editor also includes parameters for controlling the LTS outputs:

- Number of events to save: limits the saved results to the specified number of the most significant (i.e. the largest) events
- Discharge and emissions statistics frequency: allows for switching between Monthly and Yearly statistic.
- Continuous DWF TS save frequency: specifies saving frequency for continuous LTS TS outputs for intervals between dynamic simulations.

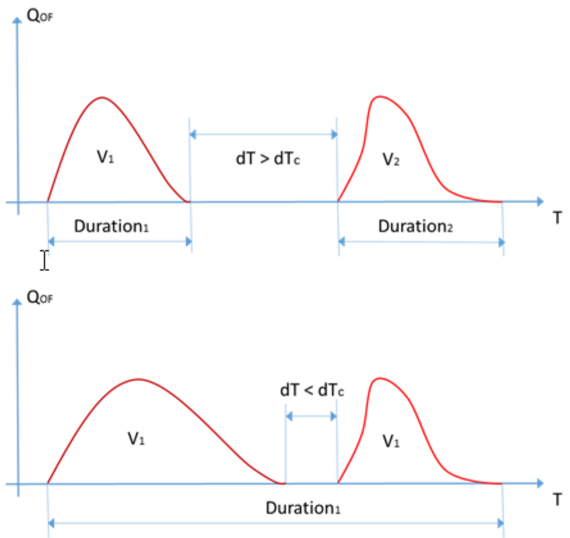


Figure 10.13 This example illustrates simulated sewer overflows occurring as two spills separated by a time period with zero flow. In the upper graph, the two spills are separated by dT which is longer than the specified dTc. In this case, MIKE+ LTS considers the two spills as two independent overflow events and calculates their volumes and durations separately. In the lower graph, the second spill starts shortly after the first one has stopped, i.e. within time interval shorter than dTc. In this case, MIKE+ LTS considers both spills as one overflow event (probably caused by the same meteorological event) and calculates the total volume and duration.

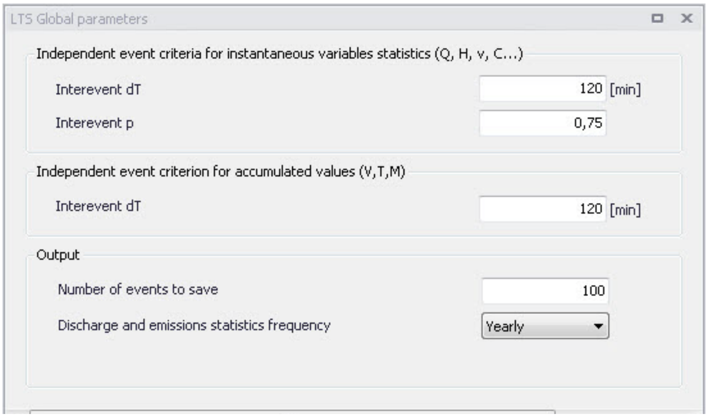


Figure 10.14 The LTS Global Parameters Editor

Table 10.4 LTS Global Parameters table attributes (Table msm_LTSResult)

Edit field	Description	Used or required by simulation	Field name in data structure
Interevent dT (For instantaneous variables statistics)	Interevent time criterion or minimum time interval between two peaks to consider them as independent	Yes	Dth
Interevent p (For instantaneous variables statistics)	Interevent level criterion or threshold for ration between the lowest minimum value between two peaks and the lower peak	Yes	Pc
Interevent dT (For accumulated variables statistics)	Interevent time criterion or minimum time interval between two events to consider them as independent	Yes	DtQV
Number of events to save	Number of largest events to save in the statistics results (1-5000)	Yes	EventLimit
Discharge and emissions statistics frequency	Choice between monthly and yearly chronological statistics	Yes	StatFrequencyNo

10.2 LTS outputs

A MIKE+ LTS simulation generates result files containing timeseries and various event-based and annual statistics.

LTS outputs are defined as any other model results, i.e. by specifying the wanted result files and their contents under result specification. The specified results are then included in the actual simulation setup, as required.

LTS outputs may take the following forms:

- "Standard" TS result outputs
- Extreme event statistics (instantaneous and accumulated)
- Chronological statistics (monthly and yearly)
- Continuous TS outputs

10.2.1 Standard TS result files

Standard TS result files with default or user-specified contents may be specified as part of an LTS simulations. These are actually "normal" result files, generated by the catchment and network models.

Common for all these files is that due to long simulation periods, they tend to be quite large and care should be taken to keep the size of such LTS outputs within reasonable limits.

The default MIKE 1D result files (*.RES1D), where the computed time series for default result items are saved for entire model, may become extremely large in long-term simulations if the saving frequency is high. On the other hand, too coarse time resolution in a result file might make the saved results useless. Consequently, for simulations covering long periods, a user-specified result file with carefully defined contents will usually be of major interest.

The user-specified result file will only contain selected result items at locations of interest. This makes it especially practical for keeping LTS results sufficiently resolved and the default result file potentially unnecessary. Detailed information on the specification of the contents of user-specified result files is provided in the Result Specifications Chapter.

- Runoff result file

  Strictly speaking, runoff result files are not specific for LTS. A fundamental difference compared to runoff results in other types of analysis is the length of the simulation periods and, consequently, size of the file. In the context of LTS simulations, runoff result files may be used as input for job list generation and/or as boundary condition for LTS simulation. Due to highly dynamic character of surface runoff, the runoff result saving frequency must remain high during rainfall. During dry weather periods, the simulation time step and saving frequency is automatically changed to much longer "DWF" time step.

RDI model is per definition continuous, so the results will be saved continuously for the entire simulation period.

Inevitably, models many sub-catchments and long simulation periods will generate extremely large runoff result files (and, optionally, SWQ result files). If accompanied with surface pollution result files with multiple pollutants, this may become a serious problem. In order to avoid this, alternative workflow for the job list creation and LTS simulation are available (see the description of simulation workflows further ahead).

- Catchment discharge result file

  Similarly, as runoff result file, in context of LTS simulations, catchment discharge result files may be used as input for job list generation and/or as boundary condition for LTS simulation. Catchment discharge normally represent slowly chaining flows (diurnal wastewater and constant infiltration, where the simulation time step and result saving frequency can be low (e.g. 1 hour) without any significant loss of accuracy. However, catchment discharge is present continuously during the simulation period. This means that models with many sub-catchments and long simulation periods will generate extremely large catchment discharge result files (and, optionally, CD WQ result files). In order to avoid this, alternative workflow for the job list creation and LTS simulation are available (see the description of simulation workflows further ahead).

- Network result file

  When generated by LTS simulation, network result files contain time series the specified result items for the entire model domain or for the specified locations of interest. The time series are discontinuous, i.e. results are only available during the dynamic simulation. Due to normally high result saving frequency, these results may become very large unless the specified contents is user-specified, focused to really needed results.

10.2.2 Statistics Result File

When running any simulation over a longer period, information on operational statistics is of major interest rather than on the raw time series. MIKE 1D LTS can save additional result files (*.RES1D) containing various statistics (event-based and annual) over the individual computed time series and the system performance as a whole:

- LTS Event-based Extremes Statistics (instantaneous and accumulated)

  These result files contain ranked extreme values for the included result items and for the specified locations. The result items represent either extremes (peaks) of instantaneous variables (e.g. water level, discharge, concentration) and/or accumulated values of cumulative variables (e.g. volume, mass, duration) for the statistical events identified as described elsewhere in this document. Such results are typically desired for overflows, water levels, pumps operations, etc.

- LTS Chronological Statistics (monthly and yearly)

  These result files contain accumulated (monthly or yearly) cumulative variables for the specified locations in the system, sorted chronologically. Such results are typically desired for overflows and pollution emissions, pumps operations, etc.

The statistics are computed according to user specifications.

The technical background for statistical computations and the process of specifying which statistical results to save are further described in the following sections below.

10.2.3 Specification of Statistics and Result Files

LTS statistical results are stored in statistical result files (*.RES1D). As a bonus, an *.ERF file is also generated, which can be loaded in MIKE View for presentation and statistics report generation.

It is possible to combine LTS hydrodynamic simulations with pollution transport (AD) calculations. In this way, one can estimate statistics on annual or event-based pollutant loads.

The contents of the statistical result file (result items and locations) must be specified prior to an LTS simulation. Content statistics is defined through the Result Files dialog (Result Specifications| Result Files).

There are two types of LTS statistics results:

1. LTS Extreme Statistics. These are divided in two sub-types:

- Instantaneous values statistics. Represent instantaneous extreme (peak) values. These include:
  - Max water level. Saves ranked peak water levels.
  - Max flow. Saves ranked peak flows.
  - Max velocity. Saves ranked peak flow velocities.
  - Max concentration. Saves ranked peak concentrations for all simulated pollutants.
- Accumulated values statistics. Represent accumulated values during events.
  - Event volume. Saves ranked event-accumulated discharge.
  - Event mass. Saves ranked event-accumulated mass for all simulated pollutants
  - Event duration. Saves ranked event duration.

2. LTS Chronological Statistics. These have the following two sub-types:

- Individual chronological statistics. These may include accumulated discharge and pollutant mass transport in individual pipes, weirs, pumps, orifices, outlets, valves, and spilling nodes. Individual statistics imply multiple item values per file.
  - Accumulated volume. Saves individual accumulated yearly/monthly discharge in chronological order.
  - Accumulated mass transport. Save individual accumulated yearly/monthly mass flux for all simulated pollutants in chronological order.
- Global accumulated statistics of discharges and emissions. These include accumulated discharge and pollutant mass transport for ALL outlets and weirs, pumps, orifices, valves discharging out of the system, and all spilling nodes, as well as TOTAL accumulated discharge and pollutant mass transport out of the system. Global statistics imply one (i.e. TOTAL) item value per file.
  - Total accumulated volume. Saves chronological total accumulated yearly/monthly discharge.
  - Total emission. Saves chronological total accumulated yearly/monthly mass flux for all simulated pollutants.

LTS statistics result items are specified in the LTS HD Items and LTS AD Items tabs in the Result Files editor. Available statistics results items are presented in Figure 10.15, Figure 10.16, Figure 10.17 and Figure 10.18.

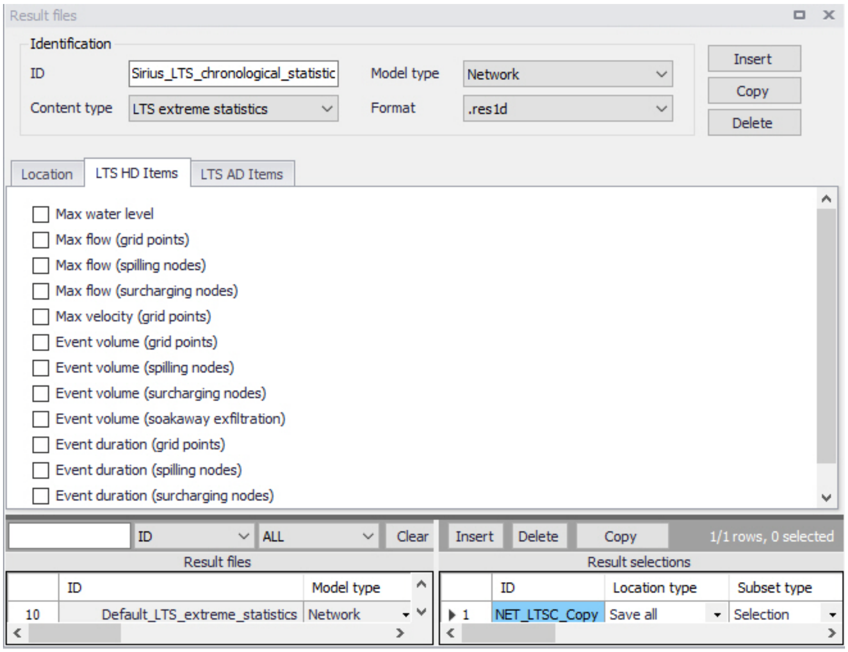


Figure 10.15 LTS HD Items for event-based extreme statistics in the Result Files Editor

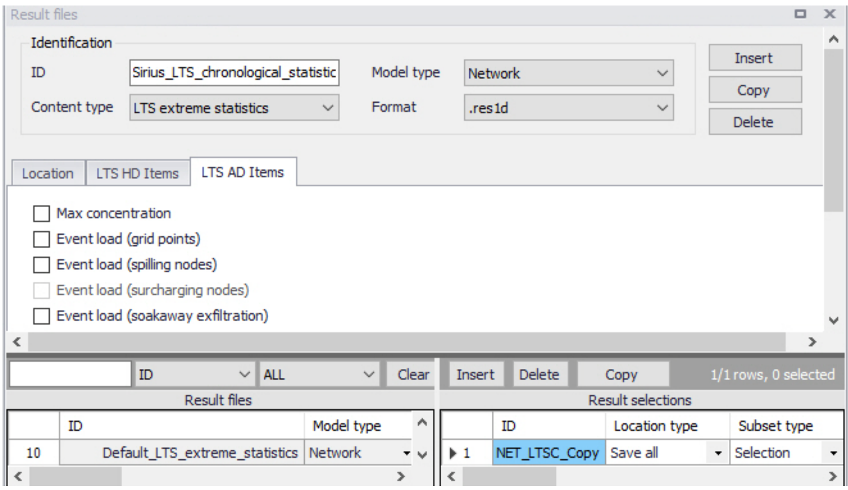


Figure 10.16 LTS AD Items for event-based extreme statistics in the Result Files Editor

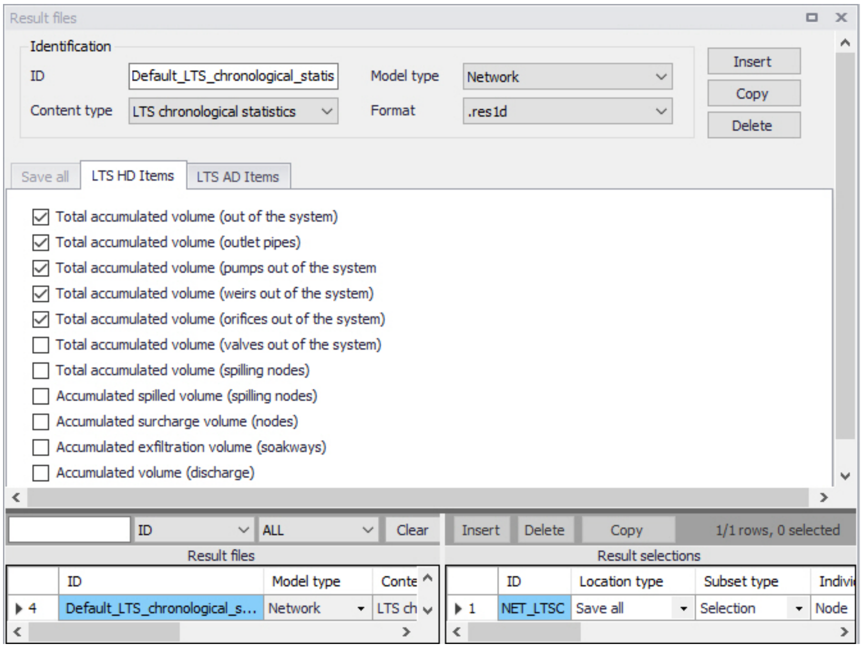


Figure 10.17 LTS HD Items for annual/monthly statistics in the Result Files Editor

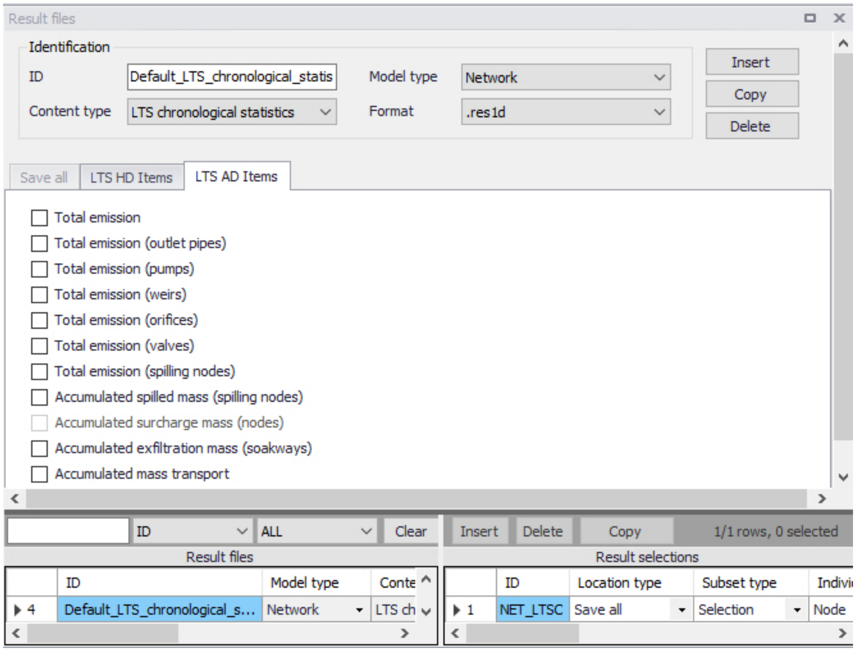


Figure 10.18 LTS AD Items for annual/monthly statistics in the Result Files Editor

10.2.4 Applicability of LTS statistics to various types of model elements

LTS statistics output is defined by the result item type and by location. Availability of certain statistics at a specific location depends on the compatibility of the location and item type, and on the actual implementation status. A complete overview of the currently available statistics is provided in Table 10.5, Table 10.6 and Table 10.7.

Table 10.5 Statistics result types in nodes (Manholes, Basins and Soakaways)

Result Type	Node (Manhole, Basin)	Spilling nodes (Cover Type = Spilling)	Surcharged nodes (WL > GL)	Soakaways
Max water level	Yes	Yes	Yes	Yes
Max flow		Yes	Yes	Yes
Event volume		Yes	Yes	Yes
Event duration		Yes	Yes	Yes
Max concentration	Yes	Yes		Yes
Event load		Yes		Yes
Total accumulated volume		Yes		
Accumulated volume		Yes		
Total emission		Yes		
Accumulated mass		Yes		

Table 10.6 Statistics result types in links (Links and Outlet pipes)

Result Type	Link (Grid point)	Outlet (Outlet pipe)
Max water level	Yes	Yes
Max flow	Yes	Yes
Max velocity	Yes	
Event volume	Yes	Yes
Event duration	Yes	Yes
Max concentration	Yes	Yes
Event load	Yes	Yes
Total accumulated volume		Yes
Accumulated volume		Yes
Total emission		Yes
Accumulated mass		Yes

Table 10.7 Statistics results in outflow structures (Pumps, Weirs, Orifices, and Valves leading out of the system)

Result Type	Pump	Weir	Orifice	Valve
Max flow	Yes	Yes	Yes	Yes
Event volume	Yes	Yes	Yes	Yes
Event duration	Yes	Yes	Yes	Yes
Max concentration	Yes	Yes	Yes	Yes
Event load	Yes	Yes	Yes	Yes
Total accumulated volume	Yes	Yes	Yes	Yes
Accumulated volume	Yes	Yes	Yes	Yes
Total emission	Yes	Yes	Yes	Yes
Accumulated mass	Yes	Yes	Yes	Yes

10.2.5 Specifying location for LTS statistics results

LTS statistics outputs are per definition user-specified. For such (non-Default) results it must be specified for which location(s) the desired results are to be saved.

As for "normal" result files, the location type for LTS statistics can be either 'Save all' (the whole system), 'Save subset' (Selection), 'Save individual' (Location ID), or 'Save within polygon'.

If 'Save subset' or 'Save individual' is selected, supplementary information on the selection list or element ID should be specified.

Additionally, for LTS statistics some pre-defined groups can be specified as subsets, such as:

- Pipes and canals
- Manholes
- Pumps
- Weirs
- Orifices
- Valves
- Outlet nodes
- Basins
- Soakaways

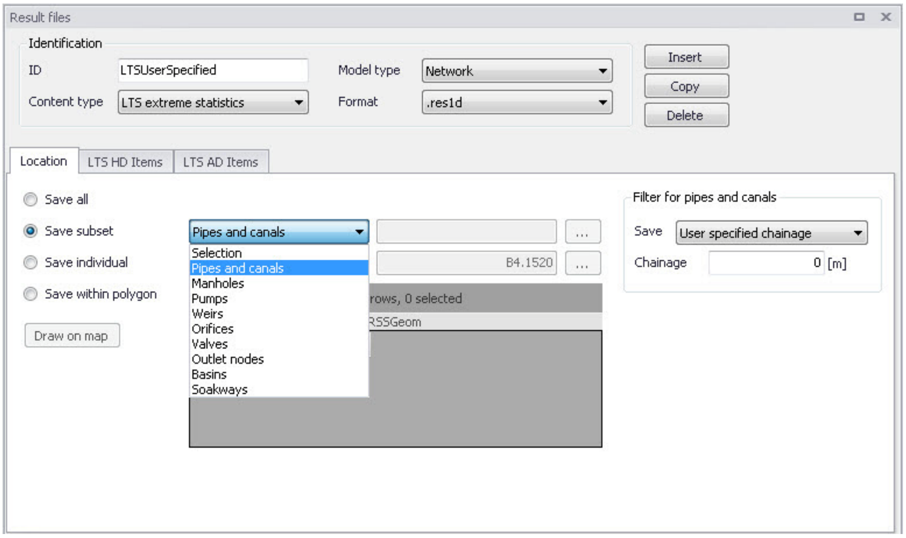


Figure 10.19 Pre-defined result location subset groups

For pipes and canals, the location for saving results can be specified as:

- All grid points
- Upstream grid point
- Downstream grid point
- Both upstream and downstream grid points
- Middle grid point
- User-specified chainage. Specify the chainage in the Chainage input box.

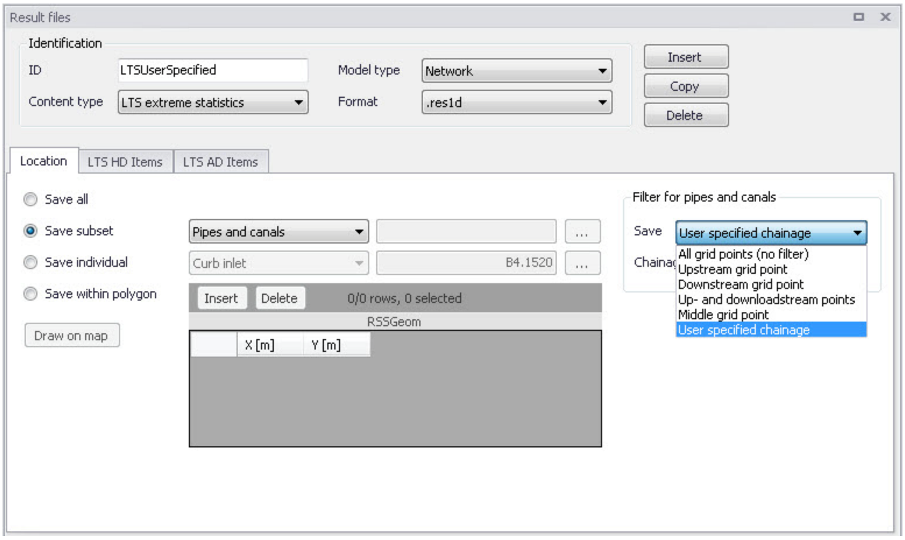


Figure 10.20 Grid point selector for result locations in pipes and canals

For chronological (annual/yearly) LTS statistics, the result items themselves define a subset of the model. E.g. result items "Total accumulated volume (weirs out of system)" contains statistics for accumulated volume of all weirs discharging out of the system. This filtering works jointly with the initially specified location, so that separate results of the same type can be achieved for various parts of the system. E.g. if definition of "Location" only includes weirs discharging into one of several recipients in the model area, the "Total accumulated volume (weirs out of system)" will be calculated and summed up for these weirs only.

If the same LTS statistics result type is used more than once (e.g. that the results from the above example are created for all local recipients separately and for the entire system), these need to be saved in separate result files.

10.2.6 Continuous LTS TS Outputs

Per definition, LTS network simulations generate discontinuous time series, with results available during dynamic simulation only. In the time intervals between events that are simulated dynamically, the variables values are set to zero.

At some locations in a modelled system it is advantageous to study a complete time series, also including the intervals between the dynamically simulated events. A typical example is the inflow to a WWTP.

MIKE+ LTS allows for the specification of any number of locations (links) in the system where such continuous discharge time series is generated by the system.

A continuous time series in the specified link is constructed by concatenation of the network simulation results during events and the sum of all hydraulic loads (boundaries) contributing to the flow in the specified link during inter-event periods. Obviously, the flows inserted in the inter-event periods are not hydraulically correct, but only an approximation. Volumes of the inserted flows are correct, but the peak amplitudes and timings inherently include an error, as the transport time and hydraulic diffusion in the network are not accounted for.

The contributing loads are user-defined by specifying the "Location" - a set of nodes and links located upstream the specified link. All loads (boundary conditions defined as runoff input, catchment discharge, load point, lateral inflows, infiltration) associated with the included model elements will be included automatically.

Accurate definition of the model elements contributing to the flows in the specified link is solely the modeler's responsibility - the model does not check if the listed elements really contribute to the specified link discharge.

The continuous time series are saved in a network result files with the contents type "LTS continuous". As for any other LTS result file, there is no default configuration available. This means that when a continuous TS output is wanted, LTS result file with continuous output TS must be created and configured "from scratch", and finally included in the actual LTS simulation setup.

The following parameters define LTS continuous result file:

File name:	user-specified
Model Type	Network
Content type:	LTS continuous
Format	res1d or dfs0
Location (contributing to the system):	Save all Save selection Save within polygon
Location (LTS continuous output in pipe of canal):	Any link
HD items	Discharge

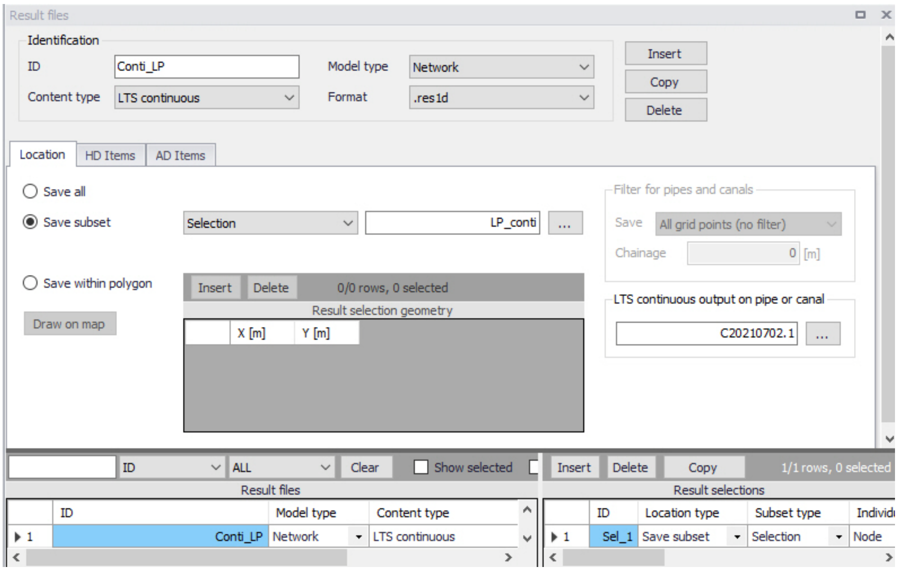


Figure 10.21 Continuous LTS TS output definition

The continuous LTS TS outputs are saved with two different frequencies:

- During dynamic simulation: user-specified save frequency in "Simulation setup | Results (Default: 60 seconds)
- Inter-event intervals: user-specified save frequency in "LTS Global parameters | Continuous DWF TS save frequency" (Default = 1 hour)

10.3 LTS Computations

10.3.1 Starting an LTS computation

LTS computations are started from the Simulation Setup editor with the Long-Term Statistics (LTS) module activated in the General Tab.

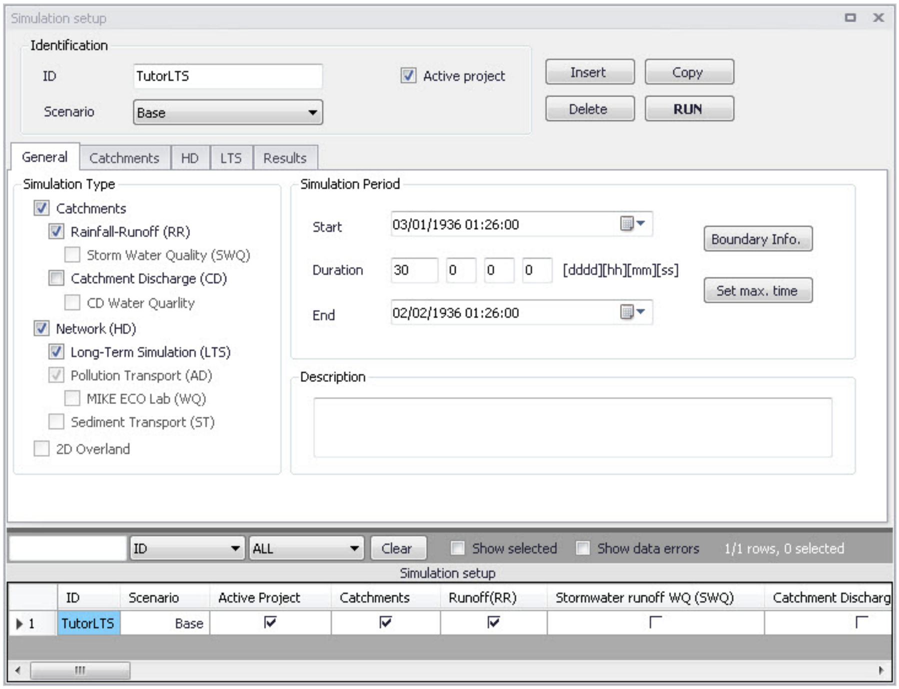


Figure 10.22 LTS computation activation in the Simulation Setup dialog General Tab

10.3.2 Generating job list files

Activating the LTS module activates the Generate Job List functionality in the LTS Tab for generating a Job List file.

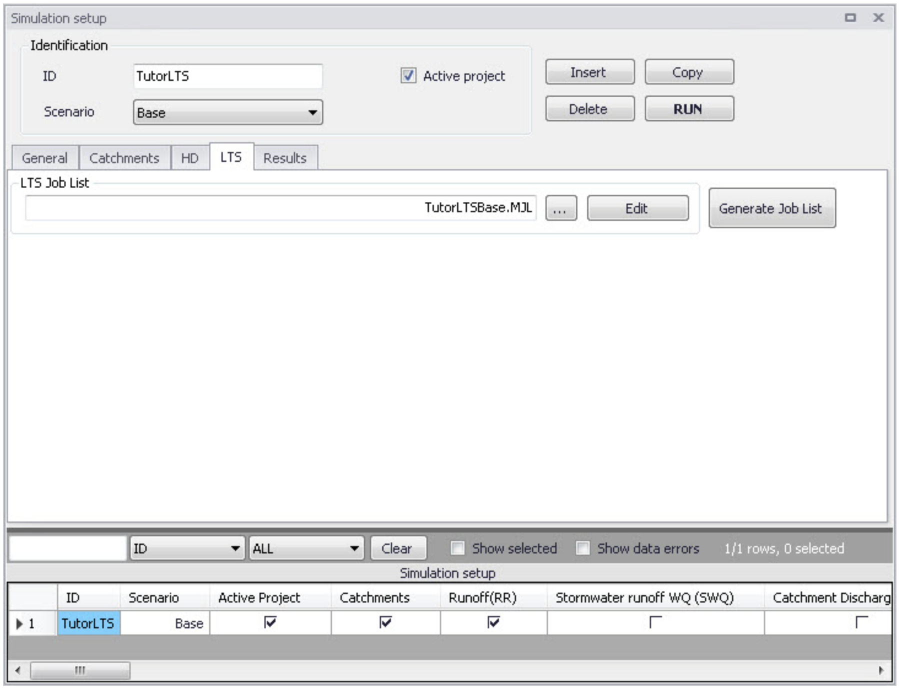


Figure 10.23 LTS Tab in the Simulation Setup dialog

10.3.3 Automatic recovery of a failed LTS simulation

An LTS simulation includes many - possibly thousands - simulation events. Depending on the size of the model, complexity and the number of the requested statistical outputs, such a simulation may represent a significant computational job. With usually available hardware, a single LTS run may take hours, days or even weeks. This sets the requirements for a stable a safe completion of such simulations very high, to avoid repetitions and delays.

Likeliness for a successful completion of a simulation improves with the application of a conservatively short time steps, at the expense of the overall longer simulation time.

Anyway, it happens occasionally that one of the simulation jobs in the LTS simulation fails. In such a case, LTS will recover automatically by repeating the failed simulation with a reduced time step.

Time step for a repeated simulation is calculated as a fraction of the time step applied in the just failed simulation. This is controlled by the parameter LTS_-FAILED_JOB_TIME_STEP_REDUCTION_FACTOR, found in the 'MIKE 1D engine configuration' dialog, from the 'Simulation' tab in the ribbon.

The system will reduce the time step and repeat the simulation as many times as specified in LTS_FAILED_JOB_MAX_REDO_COUNT, also found in the 'MIKE 1D engine configuration' dialog.

Any repetition due to a failed simulation will be reported in the simulation log.

If the repeated simulation was successful, the LTS computation will continue regularly.

If the simulation still fails after the maximum number of redo attempts was tried, this event will be excluded from any specified output and the LTS computation will continue regularly.

Any exclusion of a simulation event due to the simulation failure will be reported in the simulation log.

10.4 LTS Statistics Presentation

LTS results are saved in *.RES1D format. Due to specific contents and requirements for LTS result presentation, *.RES1D files containing LTS results are flagged to distinguish them from regular *.RES1D files containing ordinary result time series.

Essentially, there are two types of LTS result files:

1. Files containing extreme statistics (ranked extreme values) for specified variables and locations.
2. Files containing chronological yearly or monthly accumulated statistics for specified variables and locations.

10.4.1 Displaying Yearly/Monthly Statistics Bar Charts

Annual/monthly statistics may be displayed as bar charts.

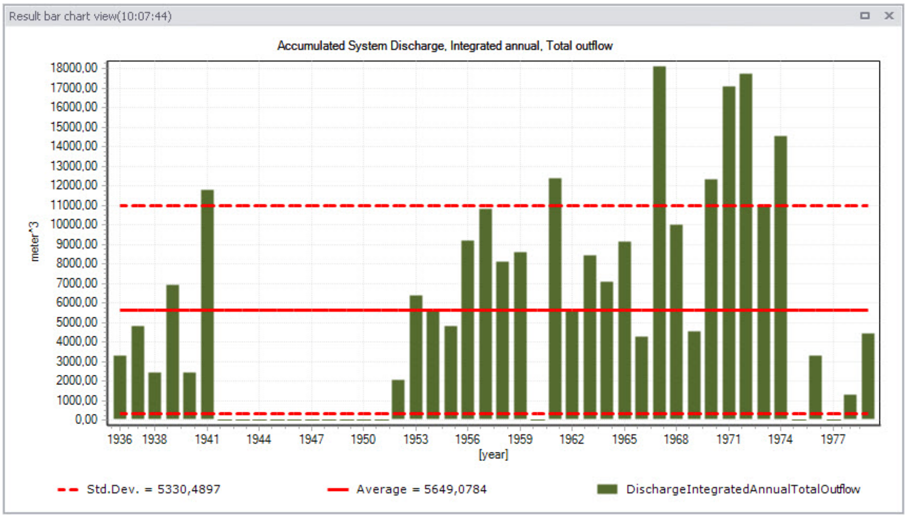


Figure 10.24 Chronological yearly statistics displayed as bar chart.

X-axis: Year axis. Yearly values are shown in case of yearly statistics. In case of monthly statistics, major tick marks are shown for years and minor tick marks for months.

Y-axis: Values axis showing with units according to type of variable for accumulated values.

Statistical information: The graph also displays the average annual/monthly accumulated (or duration) values, as well as standard deviation values plotted as line series on the graph.

The chart appearance (e.g. bar colours) may be customized.

10.4.2 Displaying Extreme Events Statistics Probability Plots

Extreme events statistics are displayed as probability plots.

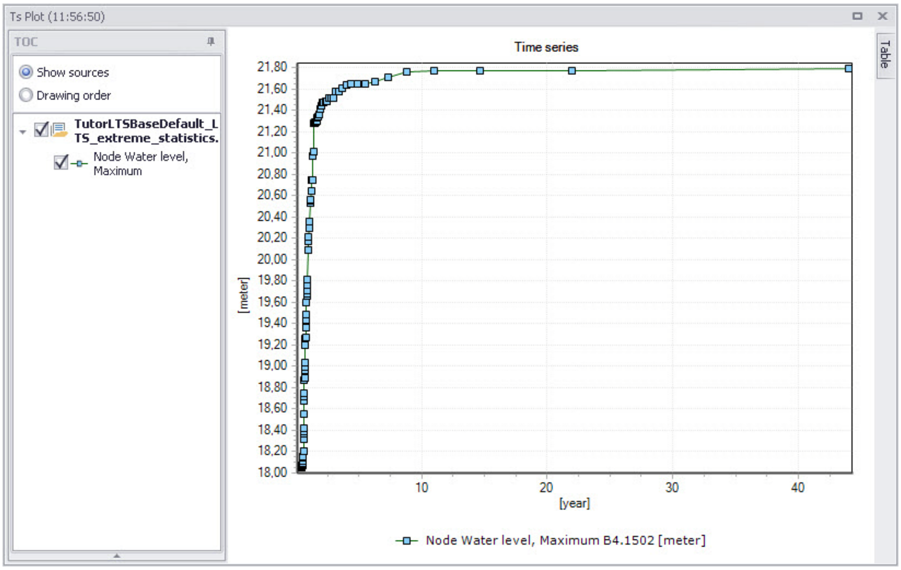


Figure 10.25 Probability plot shows ranked extreme values in reverse order (from smallest to largest, plotted against approximated recurrence interval (calculated as simulation period divided by rank).

X-axis: Recurrence interval in years.

Y-axis: Value units are displayed according to the type of variable plotted.

The user can modify plot data series colors, line thickness, marker type, etc.

10.4.3 User-specified "Observation Period"

"Observation Period" is calculated as the difference between the simulation end date/time and start date/time by Default. But sometimes, the simulated period is not continuously covered by boundary data and therefore these "empty" intervals do not contribute to the statistics.

In the bar chart example in Figure 10.24, the period between 1943 and 1952 (ten years) is not covered by rainfall data and no statistics are available for these years. In some of the other years, there are no reported results in the graph, but this is due to the actual simulations, and not due to the absence of boundary data.

To compensate for missing intervals (i.e. to calculate the exceedance frequency with the correct time base), the default "Observation Period" can be overwritten by a user-specified value:

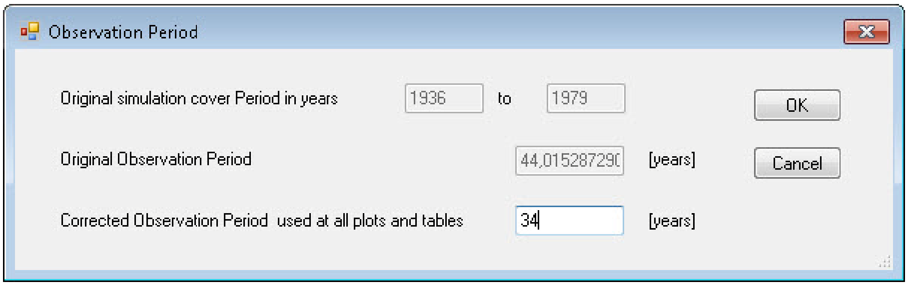


Figure 10.26 LTS Observation Period dialog

The Observation Period dialog is accessed via the “LTS user period” button in the Result Time Series window that appears when loading a result file. The option is only available for LTS extreme events statistics results.

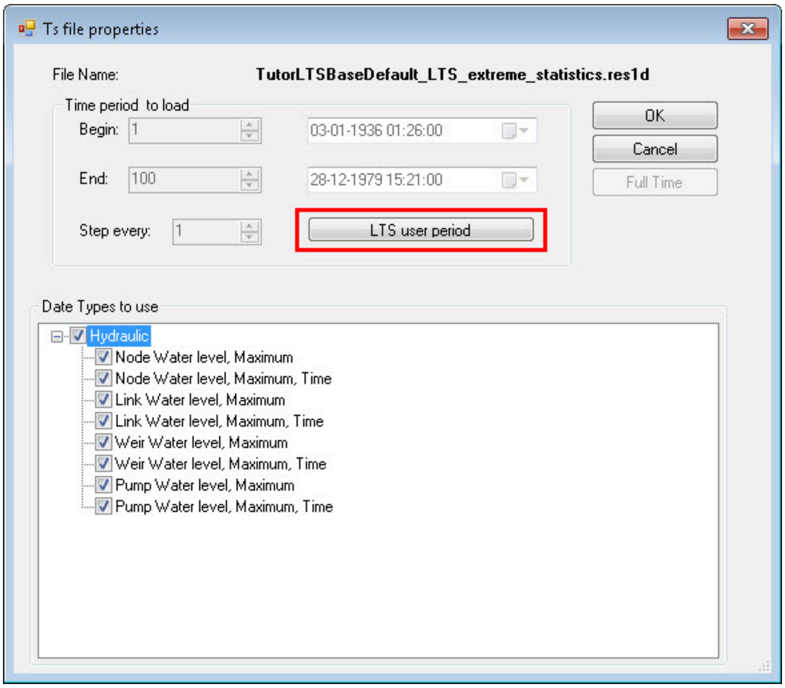


Figure 10.27 The LTS User Period button in the Timeseries File Properties window that appears when loading LTS extreme events statistics results

10.4.4 Displaying Extreme Events Statistics in Longitudinal Profiles

Recurrence intervals of exceedance for selected variables may be displayed along a longitudinal profile.

The example below shows exceedance values for link water levels along a profile for recurrence intervals 1, 3, 10 and 20 years. These are static values and cannot be animated.

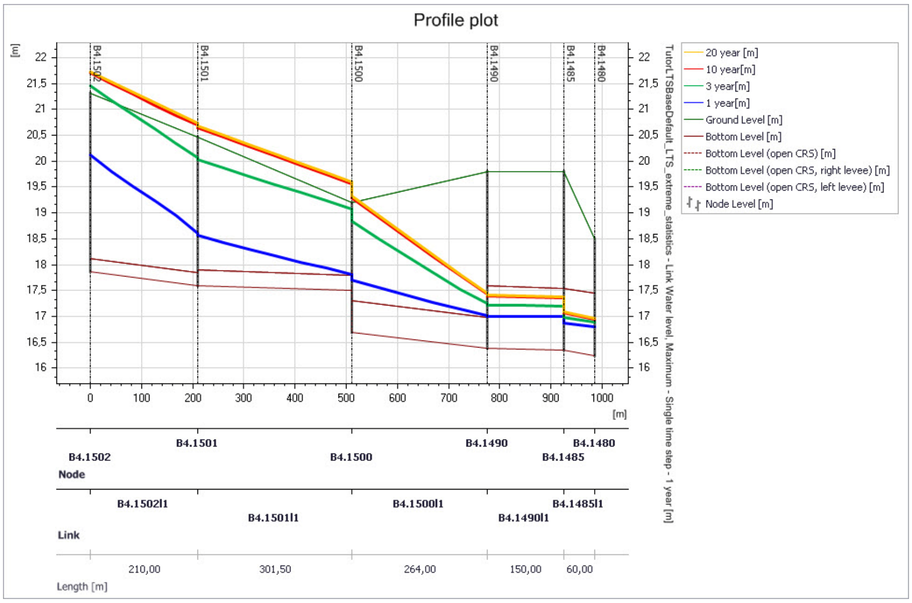


Figure 10.28 Example of link water level exceedance intervals plotted along a longitudinal profile

Exceedance values for the following variables are available for plotting along a profile:

- Maximum water level (grid points and nodes)
- Max flow (grid points, pumps, orifices, weirs, valves)
- Max velocity (grid points)
- Max concentration (grid points, pumps, orifices, weirs, valves, nodes)

After selecting the wanted variable, the exceedance values for the specified frequency are plotted for all model elements along the profile where extreme statistics are available. Intermediate values between two neighboring available values are linearly interpolated.

10.4.5 Calculating exceedance values for specified recurrence intervals

After choosing to plot exceedance values for a certain variable (e.g. water level) to a longitudinal profile plot, the user must provide the wanted recurrence interval as part of the plot specification.

The program calculates exceedance values by linear interpolation between the first available shorter recurrence interval and the first longer recurrence interval (see example below):

Number of years simulated: 15

Index (rank)	Recurrence int.	Hmax
1	15.0000	7.50
2	7.5000	7.20
3	5.0000	7.00
4	3.7500	6.70
5	3.0000	6.40
6	2.5000	6.30
7	2.1429	6.15
8	1.8750	6.10
9	1.6667	6.05
10	1.5000	6.00
11	1.3636	5.99
12	1.2500	5.97
13	1.1538	5.80
14	1.0714	5.70
15	1.0000	5.65
16	0.9375	5.50
17	0.8824	5.40
18	0.8333	5.30
19	0.7895	5.35
20	0.7500	5.34

Exceedance value for 10 years recurrence interval is calculated as (i.e. interpolation between values for the 7.5 and 15 recurrence intervals):

Hmax(10y) = 7.20 + (10-7.5)/(15-7.5)*(7.5-7.2) = 7.30

10.4.6 Displaying Extreme Events Statistics on the Map

MIKE+ can present the following results as layers on the map:

- Maximum water level exceedance in nodes and grid points for specified recurrence intervals. The values are presented in all nodes and grid points where "Max water level" statistics are available. Results in nodes are shown as point symbols, and results in grid points as line symbols.
- Maximum discharge exceedance in grid points, weirs, pumps, orifices and valves for specified recurrence intervals. The values are presented in all locations where max. discharge statistics are available. Results are shown as line symbols.

- Recurrence interval for Maximum water level exceedance of ground level in nodes and grid points. The recurrence interval is calculated similarly as in the previous section (i.e. by linear interpolation). The values are presented in all nodes and grid points where "Max water level" statistics are available.

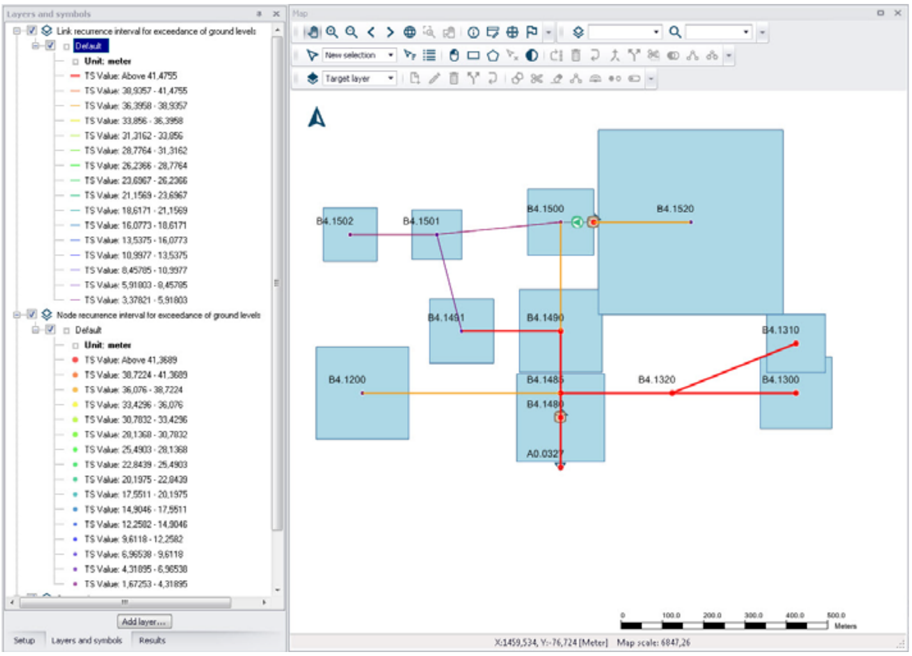


Figure 10.29 Example maximum node and link water level exceedance recurrence interval plot on the Map

Ground levels for nodes are taken from node data. Ground levels for grid points are interpolated from the neighboring nodes.

Results are shown as point symbols at nodes. The user can control the colouring and size of the symbols.

- Recurrence interval for Maximum water level exceedance of critical level in nodes. The recurrence interval is calculated similarly as described in the previous section (i.e. by linear interpolation). The values are presented in all nodes where "Max water level" statistics and "Critical level" data are available.

  Critical levels for nodes are taken from the node data in the LTS result file.

- Recurrence interval for Maximum flow exceedance of Manning discharge (i.e. full-flow discharge) in grid points. The recurrence interval is calculated similarly as described in the previous section (by linear interpolation). The values are presented in all grid points where "Max flow" statistics are available.

  Manning discharge for grid points (i.e. for links) are taken from the link data in the LTS result file.

10.4.7 Generating Reports on LTS Statistics

MIKE+ generates LTS statistics reports for the following:

1. **Summary report on extreme events statistics.** This report contains tables summarizing all calculated statistics in the file, i.e. for all individual locations, variables and statistics types. Can be generated only for result files with LTS extreme events statistics.
2. **Detailed report on extreme events statistics.** This report contains tables detailing all calculated statistics in the file, i.e. for all individual locations, variables and statistics types. Can be generated only for result files with LTS extreme events statistics.
3. **Report on chronological (yearly/monthly) statistics.** This report shall contain tables with all calculated yearly/monthly statistics for all individual locations and variables (volumes, accumulated mass, durations, number of events). Can only be generated for Chronological results (with yearly/monthly statistics).

Summary report on extreme events statistics

This report includes some or all of the tables listed below. The tables included depend on the actual contents of the LTS result file and user selection.

The following fields (columns) are included in the tables:

- **MUID**: Location ID (always)
- **Position**: Distance of the grid point from upstream node (Optional, only for location type = Link)
- **GL**: Node ground level (Optional, only for some types of statistics in nodes)
- **T_GL**: Recurrence interval for exceedance of node ground level (Optional, only for some types of statistics in nodes)
- **H_crit**: Node critical level (Optional, only for some types of statistics in nodes)
- **T_Hcrit**: Recurrence interval for exceedance of node critical level (Optional, only for some types of statistics in nodes)

- **Q_full**: Link's full-flowing Q (Manning Q) (Optional, only for discharge statistics in links)
- **T_Qfull**: Recurrence interval for exceedance of link's Q_full (Optional, only for discharge statistics in links)
- **T_spill**: Recurrence interval for occurrence of spill in a spilling node (Optional, only for some types of statistics in spilling nodes)
- **T_surcharge**: Recurrence interval for occurrence of surcharge in nodes (Optional, only for some types of statistics in spilling nodes)
- **1year**: Exceedance of variable value for recurrence interval 1 year (For LTS simulations longer than 1 year)
- **2year**: Exceedance of variable value for recurrence interval 2 years (For LTS simulations longer than 2 years)
- **5year**: Exceedance of variable value for recurrence interval 5 years (For LTS simulations longer than 5 years)
- **10year**: Exceedance of variable value for recurrence interval 10 years (For LTS simulations longer than 10 years)
- **20year**: Exceedance of variable value for recurrence interval 20 years (For LTS simulations longer than 20 years)
- **50year**: Exceedance of variable value for recurrence interval 50 years (For LTS simulations longer than 50 years)
- **100year**: Exceedance of variable value for recurrence interval 100 years (For LTS simulations longer than 100 years)

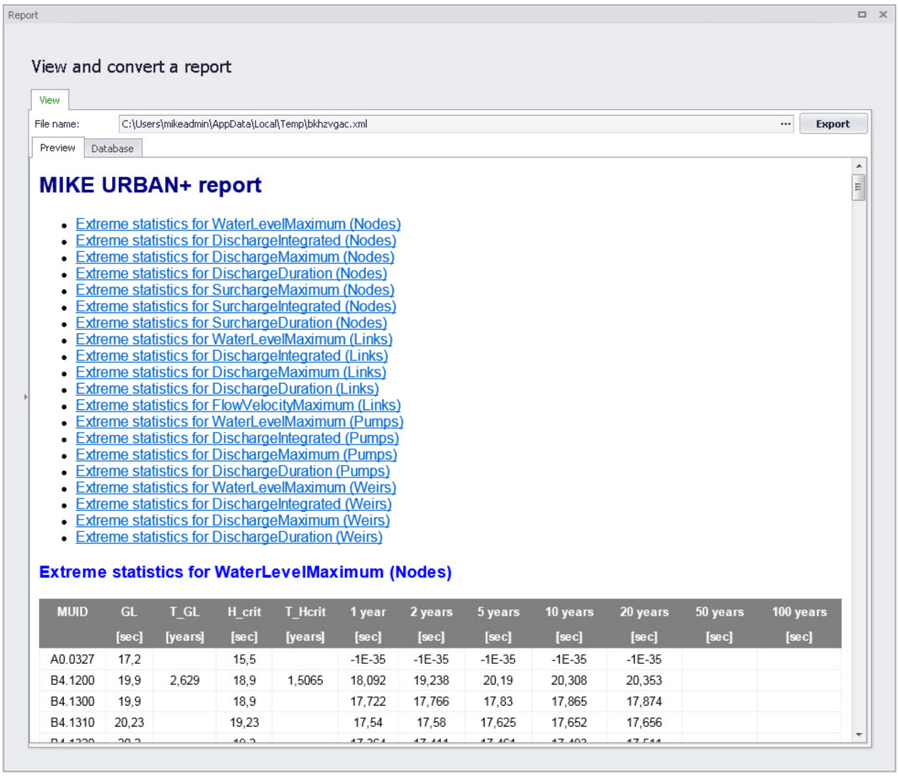


Figure 10.30 Example report on extreme statistics for water level in nodes

Detailed report on extreme events statistics

This report provides full detail of all extreme events statistics available in the LTS result file. The total number of tables corresponds to the number of available statistics types multiplied by the actual number of locations for each type of statistics.

The header of the report document has a list of hyperlinks for available statistics.

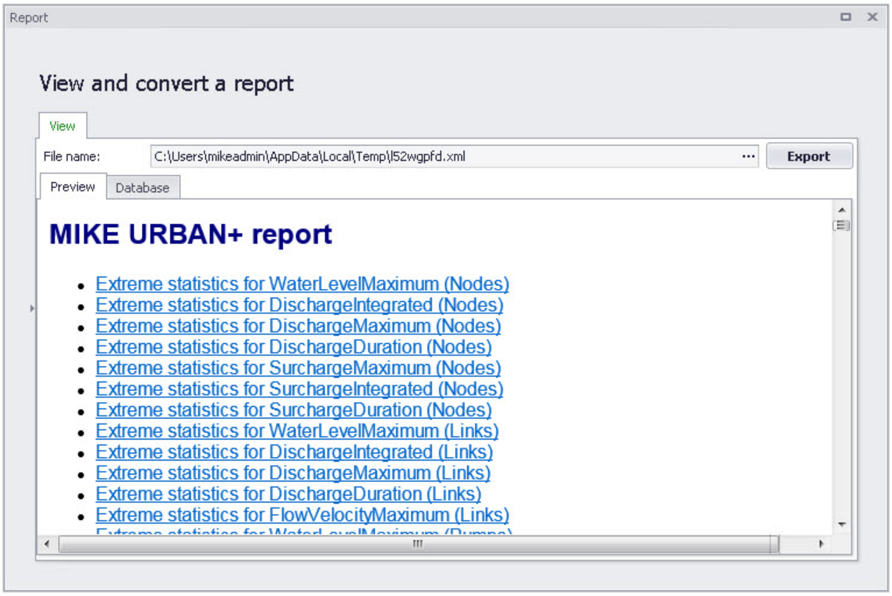


Figure 10.31 Report header hyperlinks

Report on chronological (yearly/monthly) statistics

This report includes tables according to the actual contents of the LTS chronological (yearly/monthly) statistics result file and user selections.

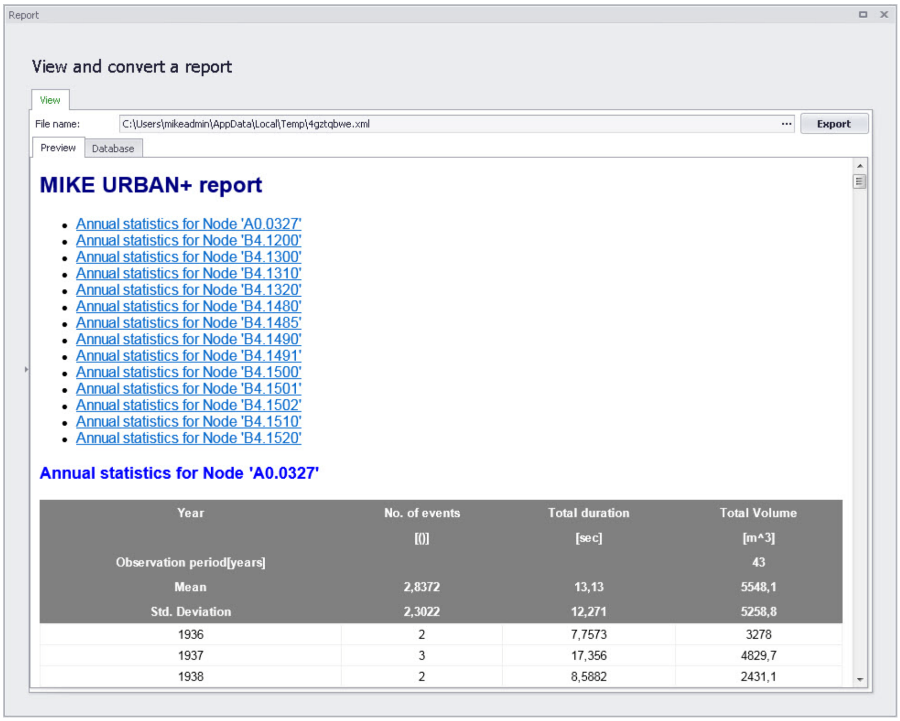


Figure 10.32 Report on chronological statistics

The number of tables in the report is a multiple of the number of included (and selected) table types and locations.

The following fields are included in the tables:

- **Year**: (Always) Actual year. The first row in the table contains the first year of simulation, and the last row contains the last year of the simulation. Intermediate rows contain the simulated years in growing order.
- **Month**: (Optional, only if monthly statistics). Actual month. The first row in the table contains the first month of simulation, and the last row contains the last month of the simulation. Intermediate rows contain the simulated months in growing order, repeating from 1 to 12 each year.
- **No. of events**: (Optional, only for individual locations). Count of events identified in the current year/month.
- **Total Duration**: (Optional, only for individual locations). Accumulated duration of all events in the current year/month.
- **Total Volume**: (Optional, if volume statistics are available and selected for this type of location). Accumulated volume for all events in the current year/month.

- **Total mass (Component xx)**: (Optional, if pollutant mass statistics are available and selected for this type of location). Accumulated pollutant mass for all events in the current year/month. NOTE: This column shall be repeated for each AD component included in the simulation.

10.5 LTS Workflows

Setting up and running LTS simulations is a modelling task that requires a thorough understanding of the LTS concepts and solid modelling skills. The following descriptions of the complete workflows and advices on individual steps complement the technical information provided in the previous chapters.

A typical LTS workflow includes the following steps:

- Preparation of an LTS model setup
  - Set-up and calibration of a collection system model (catchments + network)
  - Choice of the simulation period, preparation of the LTS catchment boundary time series and definition of the LTS catchment model boundary condition
  - Definition of job list criteria
  - Definition of runtime stop criteria (optional)
  - Preparation of hot start files and definition of initial conditions for LTS simulation events (optional)
  - Review and modification (optional) of LTS global parameters
  - Definition of LTS outputs
    - "Standard" result outputs
    - Extreme event statistics (instantaneous and accumulated)
    - Chronological statistics (monthly and yearly)
    - Continuous TS outputs
- Setting-up and execution of an LTS simulation
  - Set-up and execution of the catchment model simulation (runoff and catchment discharge)
  - Creating a job list
  - Set-up and execution of the network LTS simulation
- Review, analysis and presentation of LTS results

The steps in the LTS workflow are described in the following chapters.

10.5.1 Preparation of an LTS setup

Set-up and calibration of a collection system model

This step is identical as for any other collection system model. I.e. the model shall be set-up and calibrated, following a good modelling practice and calibration principles.

The model stability shall be tested by simulating high-intensity rainfall events (historical or synthetic) so that an appropriate simulation time step for the LTS simulation can be determined: a shorter time step contributes to the simulation stability but may compromise the simulation efficiency.

Choice of the simulation period, preparation of the LTS catchment boundary time series and definition of the LTS catchment model boundary condition

An LTS simulation normally covers long historical periods, ranging from several years to several decades (10-30 years). Duration of the simulation period may be limited by the purpose of the LTS study and, often, by the limited availability of reliable historical rainfall time series. A longer simulation period is to be preferred, as it creates a better foundation for the statistical analysis.

The applied rainfall time series should be as complete as possible. Application of a time series with frequent drop-out events may generate misleading statistical results.

If the rainfall time series includes events known to be recorded incorrectly or if such events have been removed from the time series, they should be replaced by correctly recorded rainfall at the given time at a nearby rain gauge (if available).

Any selected rainfall time series (one or several) shall be included as the catchment model rainfall boundary condition. If the hydrological simulation includes RDI model, then the potential evapotranspiration time series for the model area, covering the LTS simulation period shall also be provided as boundary condition. Similarly, where relevant, air temperature time series must be provided to support the simulation of snow accumulation/snow melt processes.

Definition of job list criteria

At least one job list criterium shall be specified. Otherwise, any number of job list criteria may be specified.

The specified START threshold shall be defined so that any event of relevance for the actual analysis is included in the LTS job list. A too high threshold value may cause important loss of information (i.e. exclusion of events that contribute to the statistics. On the other hand, a too low threshold means inclusion of many, potentially insignificant events in the LTS job list, which may compromise the efficiency of the LTS simulation.

A good guideline for the threshold value may be intensity of specific runoff (LT^{-1}), i.e. runoff per unit area. E.g. if it is known that overflows are likely to occur if the specific runoff exceeds certain intensity, the threshold may be calculated based on the total catchment area connected to the specified location.

"Duration" depends greatly on the specified location (size of the connected catchments). Small catchments may generate very short runoff peaks associated with spikes in recorded rainfall. Specifying a longer "duration" may eliminate such events from the LTS simulation.

The specified END threshold should reflect a situation when the inflow is low enough to be considered as insignificant for the processes of interest. Usually, it can be defined as a fraction of the START threshold (e.g. 10%). To ensure that the delayed hydraulic effects in the network are included in the simulation, appropriate "Duration" is specified. This depends on the size of the system, emptying time of the retention basins, etc.

Note that a better and a more precise way to control the END of the event simulation is by means of runtime criteria.

Definition of runtime stop criteria (optional)

It is recommended to apply run time stop criteria, as this is much more precise method to control the end of the event simulation than job list stop criteria. Precisely specified runtime stop criteria will result in the shortest total simulation time, without any loss of important information due to prematurely ending event simulations.

Preparation of hot start files and definition of initial conditions for LTS simulation events (optional)

Realistic initial conditions for the LTS simulation must be provided in order to achieve correct results. The following cases should be considered:

- Storm drainage system loaded by a surface runoff model

  It is assumed that the system is empty in dry weather. The runoff model generates intermittent runoff loads associated with rainfall events.

  Appropriate initial conditions for such a setup are "empty system", i.e. no hot start file is required.

- Wastewater and/or combined drainage system with some dry weather loads and runoff loads generated by a surface runoff model

  In dry weather, the system carries dry weather loads, typically wastewater. At the beginning of rainfall event, hydrological load is zero.

Appropriate initial conditions for such a setup is with a single hot start file, created with dry weather loads only. The hot start file shall include at least one day of dry weather flows, excluding the filling phase. This can be achieved either running the DWF hot start simulation over two or several days, until a fully stationary (repetitive) situation is established.

- Wastewater and/or combined drainage system with some dry weather loads and runoff loads generated by RDI runoff model (continuous infiltration)

  In dry weather, the system carries dry weather loads, typically wastewater, and some infiltration load. At the beginning of rainfall event, the hydrological load (i.e. infiltration) may have a different scale, depending on previous hydrological events, season, etc.

  Appropriate initial conditions for such a setup are established with several hot start files, created with dry weather loads and a constant infiltration component. The hot start files shall cover the expected range of possible infiltration in uniform intervals. The hot start files shall include at least one day of dry weather flows, excluding the filling phase. This can be achieved either running the DWF hot start simulation over two or several days, until a fully stationary (repetitive) situation is established.

When a hot start file (or files) is/are prepared, initial conditions for the LTS event simulations shall be specified in the dedicated "LTS initial conditions editor".

Review and modification (optional) of LTS global parameters

Default values for LTS global parameters represent a reasonable choice for the most case. It is, however, a good idea to review the global parameters and consider modifications, if it deems appropriate for the actual LTS setup.

Definition of LTS outputs

Depending on the specific requirements, LTS output files should be specified and included in the actual simulation setup.

10.5.2 Setting up and executing LTS simulations

After the LTS model has been set-up according the previous steps, the LTS simulation itself must be set-up and executed.

Several alternative workflows for running LTS simulations are available. These are described and commented in the following.

"Classic" LTS workflow

Running LTS simulation in a "Classic" LTS workflow includes three steps as described and illustrated below.

- STEP 1: Running runoff and catchment discharge (optional) simulation. This a "normal" catchment simulation, except that it runs over long historical periods.
  - Rainfall boundary condition(s) and, optionally, evapo-transpiration, temperature and SWQ boundary condition(s) must be specified for the entire simulation period and activated.
  - Catchment discharge boundary condition(s) (optional dry weather flow) must be specified for entire simulation period and activated.
  - Set-up a catchment simulation with runoff and catchment discharge (optional) included. Simulation must include at least the period to be included in the LTS simulation.
  - Define adequate simulation time step for runoff and catchment discharge simulations
  - Result files for runoff and catchment discharge shall be included in the simulation. They must include results for all catchments, saved with the frequency enough to resolve the simulated dynamics. For surface runoff, saving frequency is typically 60-300 seconds, for RDI 1- 12 hours and for catchment discharge 1 hour.
  - Start the catchment model simulation by pressing "RUN"

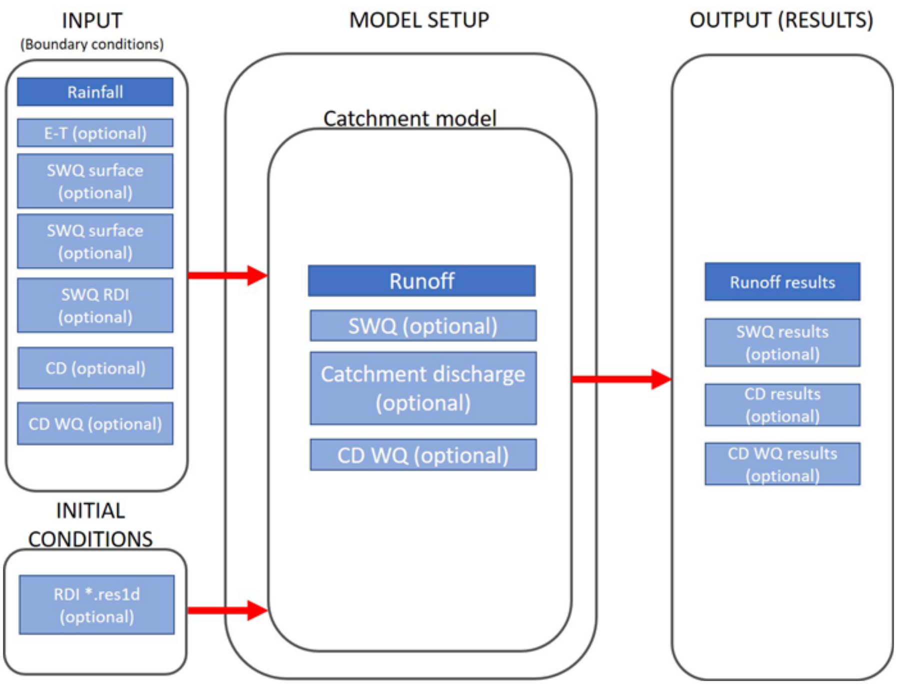


Figure 10.33 "Classic" workflow - STEP1

- STEP 2: Create job list
  - Set-up a network simulation and include LTS. The simulation period shall extend over the wanted historical period (inside the available rainfall boundary data).
  - Create and activate boundary conditions of the type "inflow from file" with simulated runoff and catchment discharge
  - In the HD TAB of the simulation setup editor define the network model simulation time step. Make sure that the specified time step is adequate for the actual model
  - In the LTS TAB of the simulation setup editor press "create job list" button. This will initiate the job list creation process. A job list file *.jlf will be created in the actual project directory. The file's name includes the actual simulation job name.
  - Upon completed job list creation, load the job list file into the simulation setup by opening the file browser (…) and picking the *.jlf file. Open the file to review and (optionally) manually modify its contents (e.g. change time step for some known extreme event). Make sure that all included boundary conditions are OK. Pay attention to the reported overall number of simulation jobs (event) and total duration of the simulation. If this deviate significantly from expectation, review and modify the job list criteria and repeat job list creation. Finally, close and save (optionally) the *.jlf file.

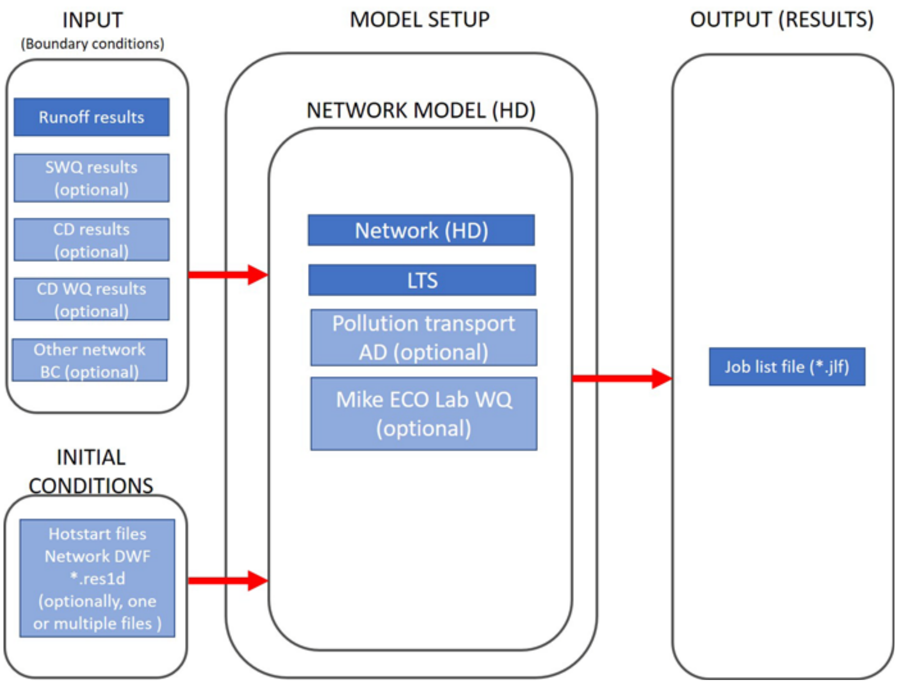


Figure 10.34 "Classic" workflow - STEP2

- STEP 3: Running LTS network simulation
  - Include the wanted output files in the simulation setup. All LTS outputs (except ASCII version for computed statistics *.ERF) are optional. The output will typically include LTS extreme statistics and LTS chronological statistics. LTS continuous output and "standard" network result files may be added if such outputs are wanted.
  - Start the network LTS simulation by pressing "RUN". Depending on the size of the model, length of the simulation period, number of specified statistics and the available hardware, this simulation may take a long time.

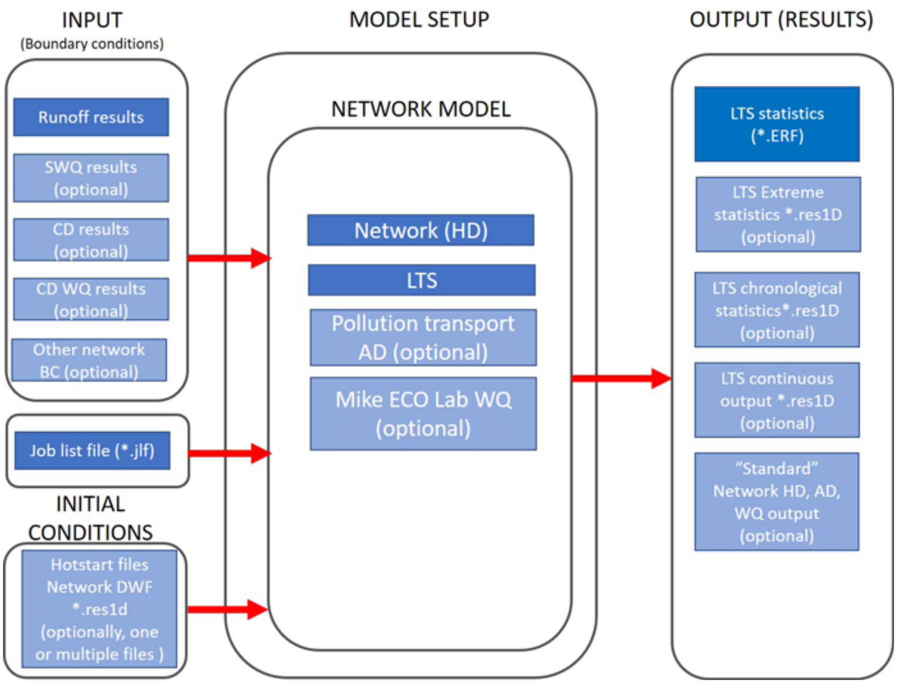


Figure 10.35 "Classic" workflow - STEP3

Disadvantage of this workflow is its inefficiency and generation of large intermediate result files (runoff and catchment discharge) with the sole purpose to provide input to the network LTS simulation.

Integrated 2-step LTS workflow

This workflow includes two steps as described and illustrated below.

- STEP 1: Create job list "on the fly" i.e. simultaneously with runoff and catchment discharge simulations

- Set-up an integrated catchment &network simulation, including run-off and catchment discharge, as well as LTS. The simulation period shall extend over the wanted historical period (inside the available rainfall boundary data).
- Define adequate simulation time step for runoff and catchment discharge simulations
- In the HD TAB of the simulation setup editor define the network model simulation time step. Make sure that the specified time step is adequate for the actual model
- Rainfall boundary condition(s) and, optionally, evapotranspiration, temperature and SWQ boundary condition(s) must be specified for the entire simulation period and activated.
- Catchment discharge boundary condition(s) (if dry weather flow included) must be specified for entire simulation period and activated.
- Any existing network boundary condition (inflow from file) representing runoff and/or catchment discharge shall be de-activated or removed.
- No result files should be specified, as we want to avoid large intermediate result files. However, if wanted, result files for runoff and catchment discharge can be included in the simulation.
- In the LTS TAB of the simulation setup editor press "create job list" button. This will initiate the job list creation process. A job list file *.jlf will be created in the actual project directory. The file's name includes the actual simulation job name.
- Upon completed job list creation, load the job lit file into the simulation setup by opening the file browser (…) and picking the *.jlf file. Open the file to review and (optionally) manually modify its contents. Make sure that all included boundary conditions are ok. Pay attention to the reported overall number of simulation jobs (event) and total duration of the simulation. If these outcome deviates significantly from expectation, review and modify the job list criteria and repeat job list creation. Finally, close and save (optionally) the *.jlf file.

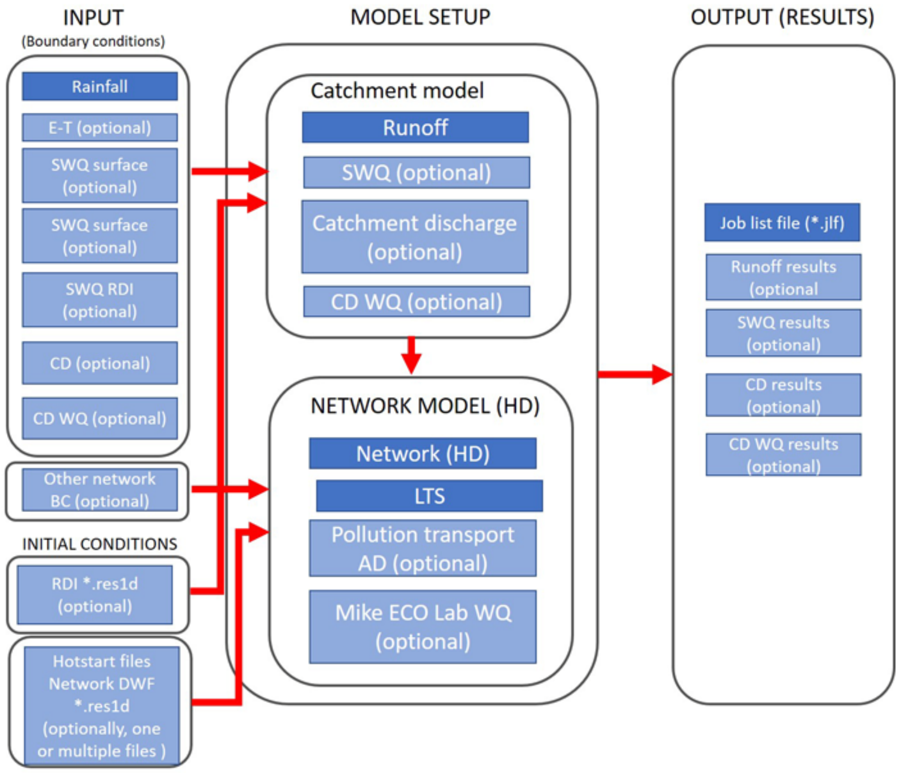


Figure 10.36 2-step integrated simulation workflow - STEP1

- STEP 2: Running integrated LTS catchment and network simulation
  – Include the wanted output files in the simulation setup. These will typically include LTS extreme statistics and LTS chronological statistics. Optionally, LTS continuous output and "standard" network result files may be added.
  – Start the integrated catchment and network LTS simulation by pressing "RUN". Depending on the size of the model, length of the simulation period, number of specified statistics and the available hardware, this simulation may take a long time.

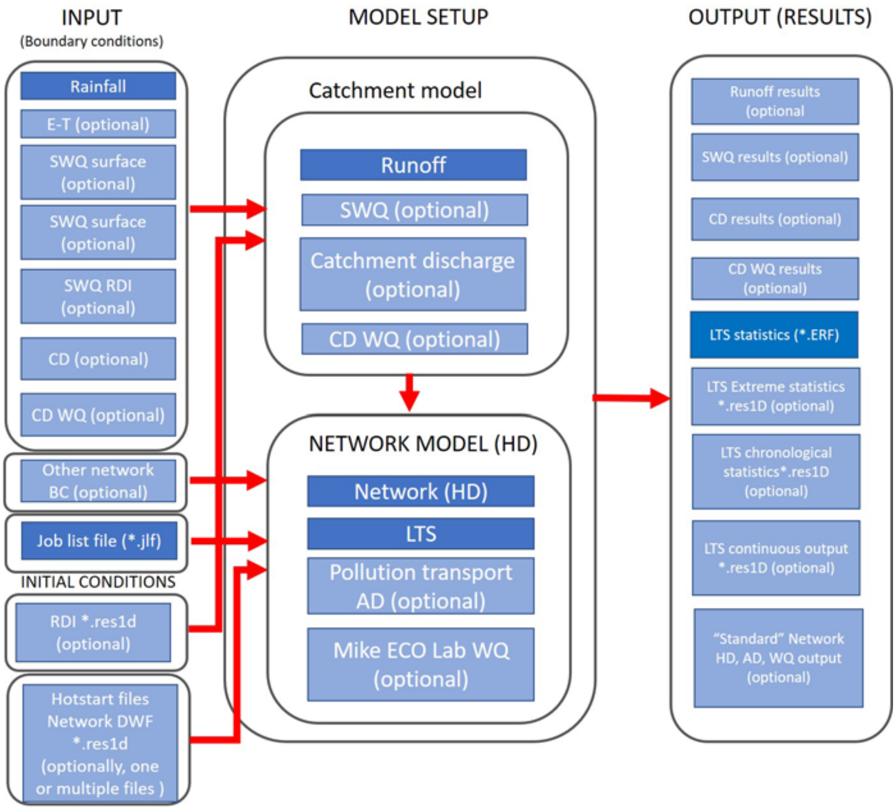


Figure 10.37 2-step integrated simulation workflow - STEP2

Advantage of this LTS workflow is its relative simplicity and avoiding creation of large runoff and catchment discharge result files.

Integrated simulation single step LTS workflow

This workflow executes the LTS simulation in a single step as follows:

- Set-up an integrated catchment &network simulation, including runoff and catchment discharge, as well as LTS. The simulation period shall extend over the wanted historical period (inside the available rainfall boundary data).
- In the "Catchments" TAB of the simulation editor, define adequate simulation time step for runoff and catchment discharge simulations
- In the "HD" TAB of the simulation setup editor define the network model simulation time step. Make sure that the specified time step is adequate for the actual model
- In the "LTS" TAB of the simulation setup editor, make sure that the job list file field is empty.

- Rainfall boundary condition(s) and, optionally, evapotranspiration and temperature boundary condition(s) must be specified for the entire simulation period and activated.
- Catchment discharge boundary condition(s) (dry weather flow) must be specified for entire simulation period and activated.
- Any network boundary condition (inflow from file) representing runoff and/or catchment discharge shall be ad-activated or removed.
- Include the wanted LTS output files in the simulation setup. These will typically include LTS extreme statistics and LTS chronological statistics. Optionally, LTS continuous output and "standard" network result files may be added
- No result files for the runoff and catchment discharge models should be specified. However, if wanted, result files for runoff and catchment discharge can be included in the simulation.
- Start the integrated catchment and network LTS simulation by pressing "RUN". Starting an LTS simulation without specified job list file, will trigger the single-step LTS simulation. Depending on the size of the model, length of the simulation period, number of specified statistics and the available hardware, this simulation may take a long time. This simulation will create a job list "on the fly" and apply it for the LTS network simulation.

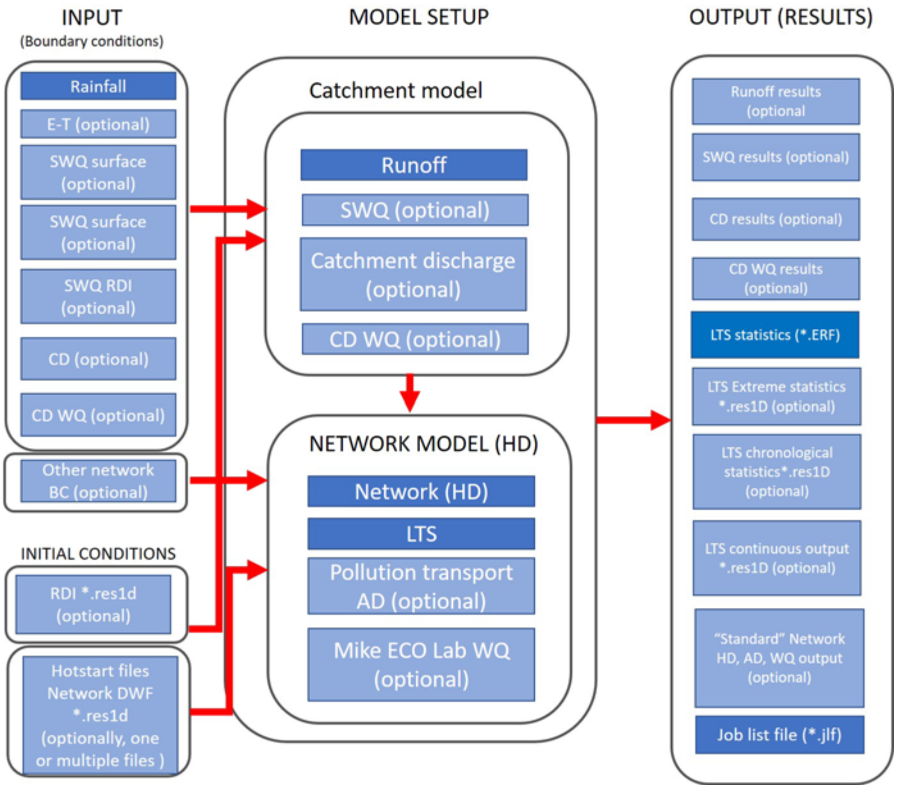


Figure 10.38 Single step integrated simulation workflow - STEP2

This workflow is the simplest and most efficient. Disadvantage is that it does not allow for review and optional editing of the job list file. Therefore, it is recommended for relatively simple LTS setups or for repeated LTS simulation with minor modifications of the LTS setup.

10.6 Controlling the LTS computations

In addition to data and parameters supplied through the general model setup and LTS editors, several engine parameters are available for controlling and adjusting the default performance of the LTS computation.

These parameters can be accessed through the 'MIKE 1D engine configuration' dialog, from the 'Simulation' tab in the ribbon.

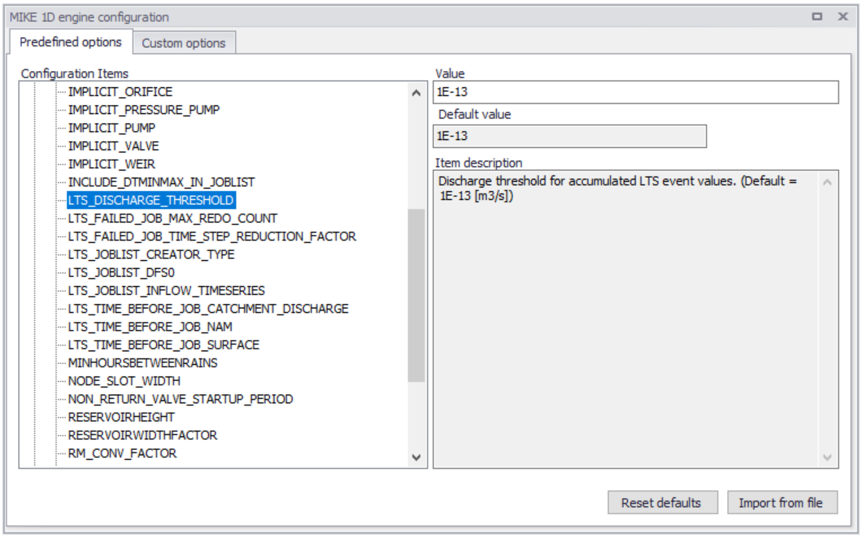


Figure 10.39 LTS engine parameters can be modified through "MIKE 1D engine configuration"

10.6.1 INCLUDE_DTMINMAX_IN_JOBLIST

When value of INCLUDE_DTMINMAX_IN_JOBLIST is "ON", the job list generator will include lines with minimum and maximum timestep in each simulation event definition (see example below):

```
[SIMULATION_EVENT]
      Simulation_start = '1933-07-22 16:31:00'
      Simulation_end = '1933-07-22 18:49:00'
      Simulation_end_no_duration = '1933-07-22 18:39:00'
      Hotstart_file = 'Hot_startLTSBase.res1d'
      Hotstart_time = '2019-01-01 16:31:00'
      Hotstart_hydrological_inflow = 0.0
      Duration = '02:18:00'
      Job_number = 1
      Job_start_criterion = 'JL_Criterium_1'
      Job_stop_criterion = 'JL_Criterium_2'
      DtMin = 10.0
      DtMax = 10.0
EndSect  // SIMULATION_EVENT
```

By "manually" editing these parameters for individual simulation events, user can ensure that e.g. a particularly difficult rain event is simulated with shorter time steps than generally specified.

Possible values:	ON (writes DTmin and DTmax in the job list) OFF (does not write DTmin and DTmax in the job list)
Default value:	OFF

10.6.2 LTS_DISCHARGE_THRESHOLD

This parameter controls the definition of statistical intermittent events for accumulated variables (discharge, mass transport) in network conduits.

Per definition, discharge and mass transport events (duration, volume and accumulated mass) are delimited by the events start time and end time.

Due to "numerical water" in empty conduits, discharge and, possibly, mass transport is never zero during a simulation. Accordingly, without a specified threshold discharge (or specified very low, i.e. smaller than expected "numerical" discharge), every simulated accumulated event in a conduit would have duration equal to the simulation period, and the accumulated volume and/or mass would include numerical water and mass. The two consecutive discharge events, even they are sufficiently separated (dT > Interevent dT) would be reported as a single event. The effect of correctly/incorrectly specified LTS_DISCHARGE_THRESHOLD is illustrated in Figure 10.40:

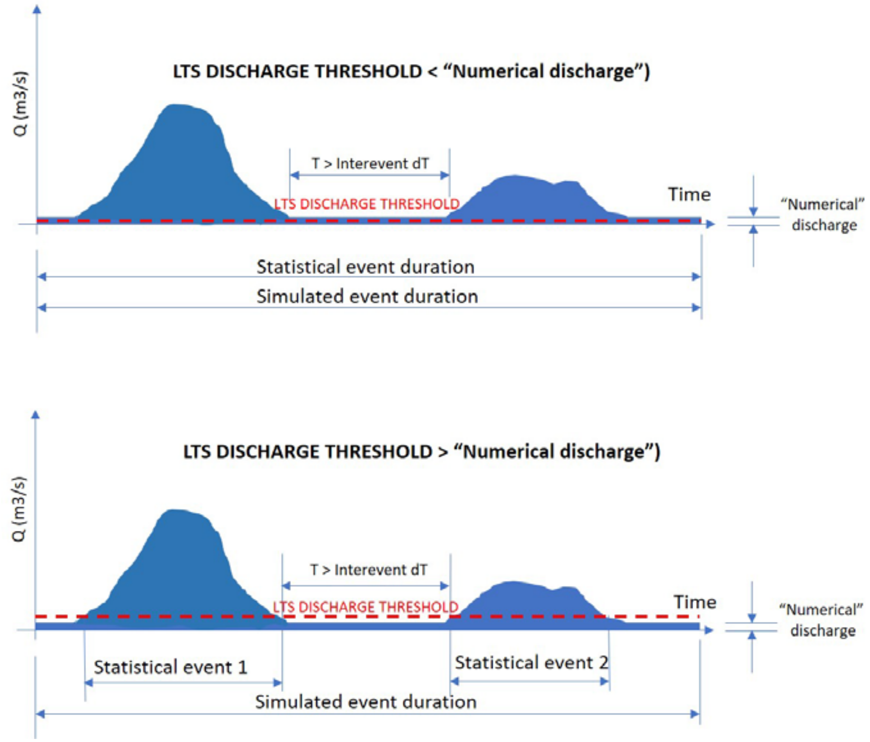


Figure 10.40 LTS discharge threshold

Possible values: any positive value

Default value: 10-13 [m3/s]

10.6.3 LTS_FAILED_JOB_MAX_REDO_COUNT

This parameter is related to the automatic recovery of failed jobs during an LTS simulation. It specifies how many times a failed job may be re-simulated with a shorter time step.

Possible values: 0, 1, 2, 3, 4, 5...

Default value: 1

Note: Time step for an LTS simulation should be specified so to ensure a stable and accurate simulation of the most intensive events included in the LTS simulation. Re-simulating accidentally failed jobs with a shorter time step should be considered as exception.

10.6.4 LTS_FAILED_JOB_TIME_STEP_REDUCTION_FACTOR

Likewise LTS_FAILED_JOB_MAX_REDO_COUNT, this parameter is related to the automatic recovery of failed jobs during an LTS simulation. It specifies how much the simulation time step shall be reduced in the following attempt to simulate a failed job..

Possible values: 0 < LTS_FAILED_JOB_TIME_STEP_REDUCTION_FACTOR < 1..

Default value: 0.5

10.6.5 LTS_JOBLIST_CREATOR_TYPE (0/1)

This parameter controls the process of job list creation, with focus on speed or accuracy.

Choice of possible values (0 or 1) has the following effects:

- LTS_JOBLIST_CREATOR_TYPE = 0
  Calculates the inflows used by job list criteria only when it is raining. This effectively skips potential jobs where boundary inflow other than rainfall satisfies a job start condition, but there is no active rain. This contributes to the job list creation efficiency, as wet periods typically represent a smaller part of the simulated period.
- LTS_JOBLIST_CREATOR_TYPE = 1
  Calculates the inflow continuously, i.e. also during periods without rainfall. This option is relevant in cases when the model includes significant intermittent inflows which are not directly related to the local rainfall. This slows down the process of job list creation.

Possible values: 0 (calculates inflow only during rain)
1 (calculate inflows continuously)

Default value: 0

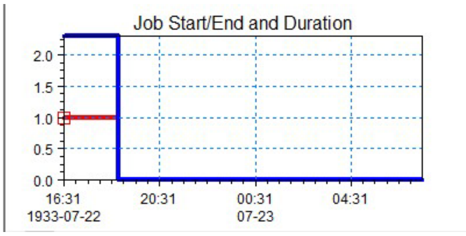
10.6.6 LTS_JOB_LIST_DFS0 (Off/On)

When this parameter is set to "On", the LTS job list creation process generates a *JobStartEnd.dfs0 file containing the following time series items:

Job ON/OFF (): 1 (job = ON); 0 (job = OFF)

Duration (hours): Simulation job duration

An example of such file shown in figure below:



	Time	1:Job ON/OFF [0]	2:Duration [hour]
0	22/07/1933 16:31:00	1	2.3
1	22/07/1933 18:49:00	0	0
2	31/07/1933 18:56:00	1	1.15
3	31/07/1933 20:05:00	0	0

Figure 10.41 Example of dfs0 file containing time series of job activity and duration

Possible values: On (create *.JobStartEnd.dfs0 file)
Off (do not create *.JobStartEnd.dfs0 file)

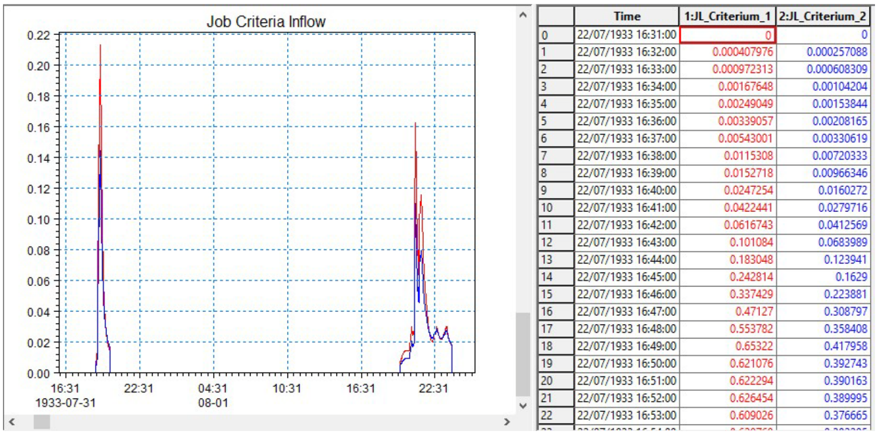
Default value: Off

10.6.7 LTS_JOB_LIST_INFLOW_TIMESERIES (Off/On)

When this parameter is set to "On", the LTS job list creation process generates a *BaseJobCriteriaInflow.dfs0 file containing the time series of inflows into the system, used in the evaluation of job list criteria. I.e. the file includes as many time series as there are job list criteria in the actual LTS setup.

The time series are created by summing up all inflows (catchment discharges, runoff, infiltration, any other lateral inflow) connected to the specific location (Individual), part of the system (List, defined by a selection) or to the whole system (General).

Figure 10.42 below illustrates example with two job list criteria. The time series are intermittent, i.e. only contain inflow discharges for the period during the simulation jobs.



	Time	1:JL_Criterium_1	2:JL_Criterium_2
0	22/07/1933 16:31:00	0	0
1	22/07/1933 16:32:00	0.000407976	0.000257088
2	22/07/1933 16:33:00	0.000972313	0.000608309
3	22/07/1933 16:34:00	0.00167648	0.00104204
4	22/07/1933 16:35:00	0.00249049	0.00153844
5	22/07/1933 16:36:00	0.00339057	0.00208165
6	22/07/1933 16:37:00	0.00543001	0.00330619
7	22/07/1933 16:38:00	0.0115308	0.00720333
8	22/07/1933 16:39:00	0.0152718	0.00966346
9	22/07/1933 16:40:00	0.0247254	0.0160272
10	22/07/1933 16:41:00	0.0422441	0.0279716
11	22/07/1933 16:42:00	0.0616743	0.0412569
12	22/07/1933 16:43:00	0.101084	0.0683989
13	22/07/1933 16:44:00	0.183048	0.123941
14	22/07/1933 16:45:00	0.242814	0.1629
15	22/07/1933 16:46:00	0.337429	0.223881
16	22/07/1933 16:47:00	0.47127	0.308797
17	22/07/1933 16:48:00	0.553782	0.358408
18	22/07/1933 16:49:00	0.65322	0.417958
19	22/07/1933 16:50:00	0.621076	0.392743
20	22/07/1933 16:51:00	0.622294	0.390163
21	22/07/1933 16:52:00	0.626454	0.389995
22	22/07/1933 16:53:00	0.609026	0.376665

Figure 10.42 Example of dfs0 file with inflow time series associated with two job list criteria

Possible values: On (create *.BaseJobCriteraInflow.dfs0 file)
Off (do not create *.BaseJobCriteraInflow.dfs0 file)

Default value: Off

10.6.8 LTS_TIME_BEFORE_JOB_CATCHMENT_DISCHARGE

This parameter controls the running of catchment discharge model in the dry periods between simulated events, during a "live" run of catchment and network model in an LTS simulation.

When running the catchment discharge model (typically generating wastewater inputs to the network) and the network model simultaneously in an LTS simulation, the catchment discharge model may be set to run only during the network simulation, or it can be specified to start running several days before the start time of the network simulation. Ultimately, the catchment discharge model can be set to run continuously.

Purpose of stopping the catchment discharge simulation between LTS simulation events is to minimize the simulation time and size of the catchment discharge result file. As catchment discharge establishes insta9bntaneously, default setting is "0", i.e. the catchment discharge simulation will start exactly at the same time as the network simulation.

Possible values:	-1	catchment discharge simulation runs continuously between events
	0	catchment discharge simulation starts at the same time as network simulation
	N (1,2,3,…)	catchment discharge simulation starts N days before the network model simulation
Default value:		0

10.6.9 LTS_TIME_BEFORE_JOB_NAM

This parameter controls the running of RDII (NAM) hydrological model in the dry periods between simulated events, during a "live" run of catchment and network model in an LTS simulation.

When running the RDII hydrological model (generating continuous hydrological inputs to the network) and the network model simultaneously in an LTS simulation, the RDII model may be set to start running a number of days before the network simulation or, ultimately, the RDII model can be set to run continuously. The latter option is preferred, because RDII model simulates slow hydrological processes and only a continuous simulation between LTS events ensures correct results..

Possible values:	-1	RDII simulation runs continuously between events
	0	RDII simulation starts at the same time as network simulation
	N (1,2,3,…)	RDII simulation starts N days before the network model simulation
Default value:		1

10.6.10 LTS_TIME_BEFORE_JOB_SURFACE

This parameter controls the running of any surface runoff model in the dry periods between the simulated events, during a "live" run of catchment and network model in an LTS simulation.

When running a surface runoff model (typically generating storm runoff inflows to the network) and the network model simultaneously in an LTS simulation, a surface runoff model may be set to run only during the network simulation, or it can be specified to start running a number of days before the start time of the network simulation. Ultimately, the surface runoff model can be set to run continuously.

Purpose of stopping a surface runoff simulation between LTS simulation events is to minimize the simulation time and size of the runoff result file. However, this must be done with a due care to avoid incorrect simulation. Models involving processes which depend on significant "hydrological memory" (e.g. infiltration) may need to be run before the actual start of the network simulation, in order to capture the effects of small rainfall events not included in the LTS job list. .

Possible values:	-1	Surface runoff simulation runs continuously between LTS events
	0	Surface runoff simulation starts at the same time as network simulation
	N (1,2,3,…)	Surface runoff simulation starts N days before the network model simulation
Default value:		1

10.6.11 MINHOURSBETWEENRAINS

This parameter allows to consider consecutive recorded rainfalls, separated by relatively short dry intervals, as one continuous rainfall event. This may affect the start time of an LTS job (it is set to the rainfall start time) in cases when a rainfall event included in the LTS job list is preceded by a small, low-intensity rainfall. This is illustrated in figures below.

If the preceding small rainfall is separated from a large rainfall event by a **dry interval longer than the specified MINHOURSBETWEENRAINS**, the two rainfalls are considered as two separate events. The small rainfall is ignored as it generates the inflow to the system smaller than the threshold specified by any of the job list START criteria. The big rainfall is included in the job list (it exceeds the threshold discharge for the time longer than the specified duration, for at least one of the job list START criteria. The start time for the simulation is set to the start of the big rainfall event.

The end of the job list event is set to the time when the inflow drops below the specified threshold for the time longer than the specified duration, for all specified job list STOP criteria.

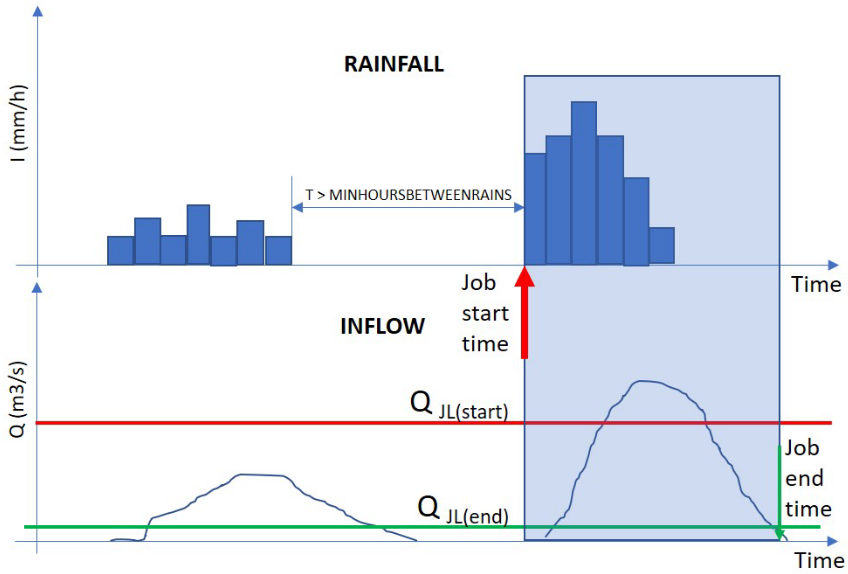


Figure 10.43 Definition of event start for rainfall events separated by a long dry interval

If the preceding small rainfall is separated from a large rainfall event by a d**ry interval shorter than the specified MINHOURSBETWEENRAINS**, the two rainfalls are considered as one continuous event. The small rainfall is included in the simulation, as it is treated as part of the big event. The joint rainfall event is included in the job list (it exceeds the threshold discharge for the time longer than the specified duration, for at least one of the job list START criteria. The start time for the simulation is set to the start of the joint rainfall event, effectively at the beginning of the small rainfall.

The end of the job list event remains unaffected, it is set to the time when the inflow drops below the specified threshold for the time longer than the specified duration, for all specified job list STOP criteria.

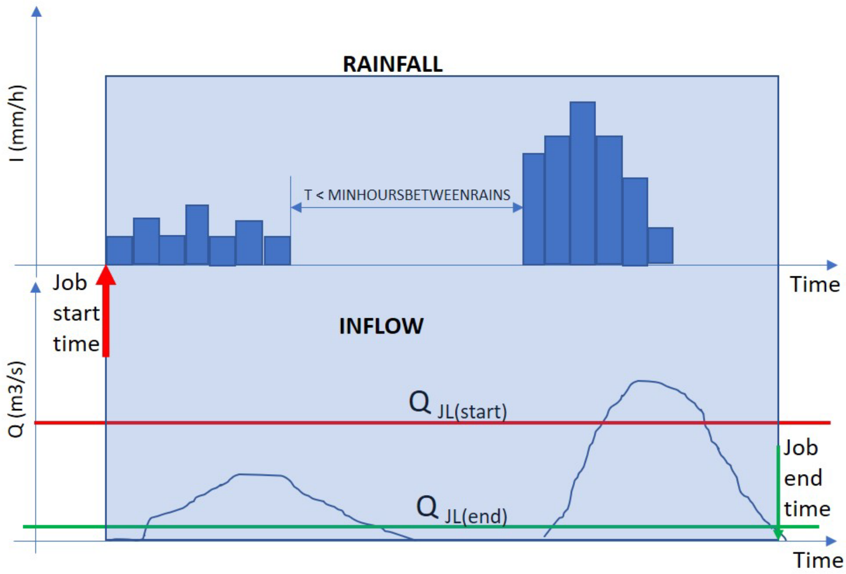


Figure 10.44 Definition of event start for rainfall events separated by a short dry interval

11 Water Quality

MIKE+ has several modules for the simulation of water quality for catchment surfaces, sewer systems, and river networks.

MIKE+ can model complex water quality and pollutant transport mechanisms using Stormwater Runoff Quality (SWQ), Advection-Dispersion (AD), and the MIKE ECO Lab engine to solve Biological Processes equations. The MIKE ECO Lab framework offers versatile and 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.

11.1 Advection-Dispersion (AD)

Advection-Dispersion (AD) simulates the transport of dissolved substances and suspended fine sediments in the network. Conservative materials as well as those that are subject to a linear decay can be simulated. The computed flow discharges, water levels, and cross-sectional flow areas are used in the AD computation. The solution of the advection-dispersion equation is obtained using an implicit, finite-difference scheme which has negligible numerical dispersion. Concentration profiles with very steep fronts can be accurately modelled. The computed results can be displayed as longitudinal concentration profiles and pollutants graphs, which could be used at the inflow to a sewage treatment plant or an overflow structure. The AD can be linked to Long Term Statistics modelling to provide long-term simulation of pollutant transport.

The option to simulate water age and blend in percentages between two sources can be done with the AD module.

The Advection-Dispersion model can be used to calculate transport of dissolved or suspended substances, age of water, blend in percent between two sources, and for modelling of water temperature variation within the network. The model is based on the one-dimensional transport equations for dissolved materials. The equations reflect two transport mechanisms: the advective transport with the mean flow velocity, and the dispersive transport due to concentration gradients in the water. The transport equations are solved by use of an implicit finite difference scheme, which is fully time and space centred, in order to minimize the numerical dispersion. The main assumptions of the model are:

- The considered substance is completely mixed over the cross-sections. This implies that a source 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 can be applied, i.e. the dispersive transport is proportional to the concentration gradient.

Special considerations have been given to the transport at manholes and other structures.

The Advection-Dispersion model requires two types of data: time series of concentrations at the model boundaries and data for full definition of the components to be modelled, e.g. initial concentrations, dispersion coefficients and decay rates.

WQ Components

Each of the water quality (WQ) components (substances) to be included in Advection-Dispersion computations must be specified in this section shown in the WQ Components dialog. Naming of component is absolutely flexible, and no 'reserved' or 'standard' component names are prescribed.

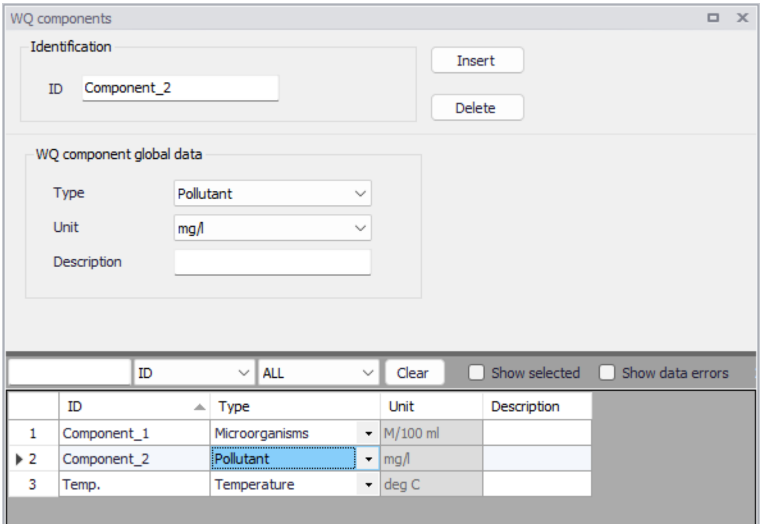


Figure 11.1 Define water quality components in the WQ Components editor

The specified WQ components can be declared as 'Pollutant', 'Microorganisms', 'Temperature', 'pH', 'Salinity', 'Water Age', 'Water Blend', 'Other' or 'Fixed matter'. This categorization is used for handling of units: each component type has its own list of possible units.

When working with a water quality model using MIKE ECO Lab, each MIKE ECO Lab state variable is associated to a WQ component from this 'WQ component' editor, and the biological processes are controlled by properties defined in the MIKE ECO Lab editors. The 'Fixed matter' type of WQ component is a special type which is relevant for this MIKE ECO Lab models, and is used to describe e.g. components deposited at the bottom and not moving with water.

An optional description of the component can also be specified.

Advection-Dispersion initial conditions for the WQ components can be specified in the AD Initial Conditions editor. Also see Chapter 6.2 AD Initial Conditions *(p. 218)*. Advection-Dispersion boundary conditions for the WQ components can be specified in the 'WQ boundary properties' editor. Also see Chapter 7.4 Water Quality Boundary Condition Properties *(p. 242)* for more description. If not specified otherwise, AD initial conditions and boundary conditions are assumed to be equal to 0.

Decay

For each WQ component a decay coefficient may be specified. The decay coefficient cannot be given for water age and water blend types. A decay set to 0 describes a conservative component, whereas a positive decay describes a non-conservative component. For a non-conservative component, the concentration is assumed to decay according to the first order expression:

$$\frac{dC}{dt} = KC \tag{11.1}$$

K = the decay coefficient (h^{-1})

C = the concentration

The decay constant (Decay) is defined as a uniform decay over the entire model for each component.

The decay cannot apply for water age and water blend types. The decay is also not relevant for WQ components associated to a MIKE ECO Lab state variable, for which the decay will be simulated as part of the biological processes modelling.

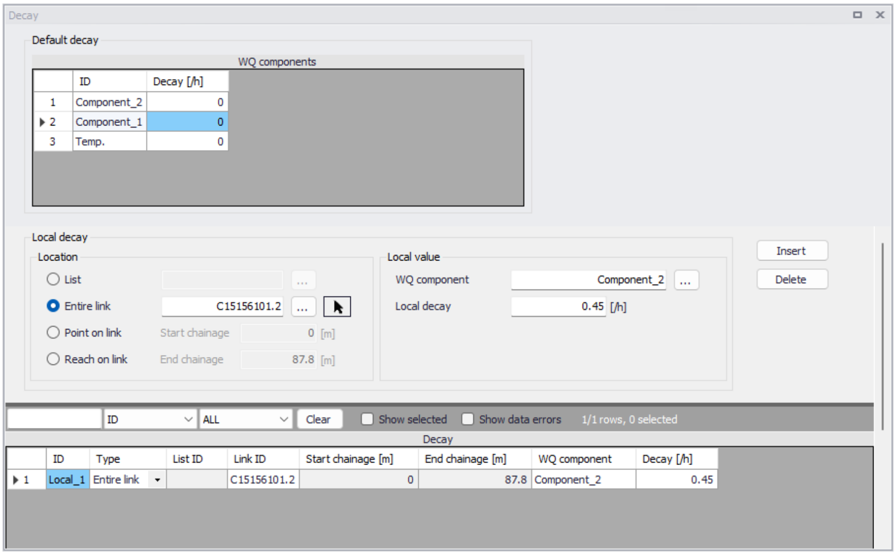


Figure 11.2 Define components' decay in the Decay editor

The decay value can be specified either globally using a default value, or locally in parts of the networks (CS and River). The default value will be used at all locations except for locations where local values have been specified. These local values 'overrule' the global specification.

When a local value is located with a list, the specified local decay value applies to all rivers and pipes included in the selected list.

AD Dispersion

The dispersion coefficient is specified as a function of the flow velocity. The function is given as:

$$D = au^b \tag{11.2}$$

where:

D = the dispersion coefficient (m2/s),

a = the dispersion factor,

u = the flow velocity (m/s),

b = a dimensionless exponent.

If the exponent is set to zero, then the dispersion coefficient is constant and independent of the flow velocity. The unit for the dispersion factor will then be m2/s. If the exponent is 1, i.e. the dispersion coefficient is a linear function of the flow velocity, then the unit of the dispersion factor will be meter, and the dispersion factor will in this case be equal to what is generally termed the dispersivity. It is possible to specify values of the minimum and the maximum dispersion coefficients in order to limit the range of the dispersion coefficient calculated during the simulation.

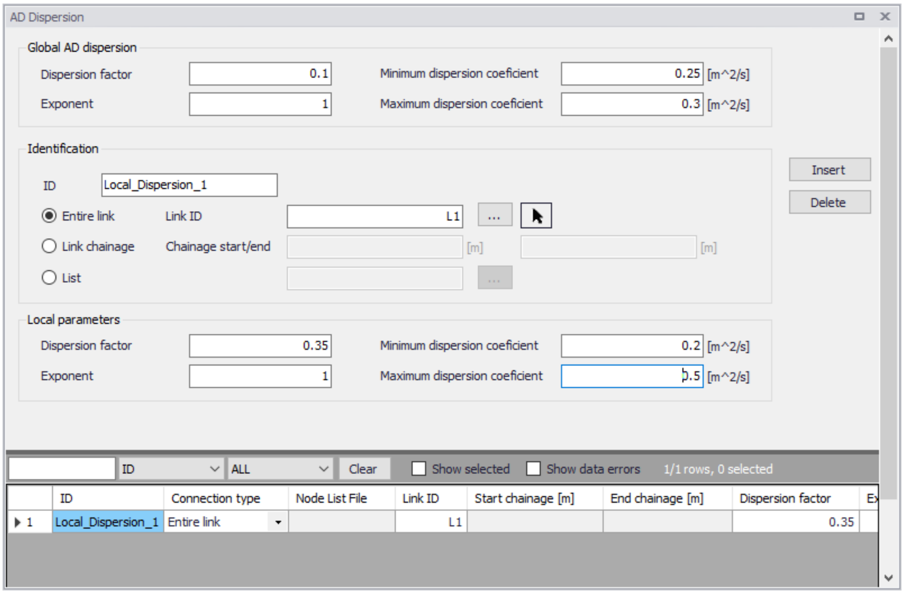


Figure 11.3 Define global and local network model dispersion parameters in the AD Dispersion editor.

The dispersion coefficient can be given either globally, or locally in parts of the networks (CS and River).

The global description will be used at all locations except for locations where local conditions have been specified.

The locally specified dispersion coefficients 'overrule' the global specification.

11.2 Water Quality with MIKE ECO Lab

Biological processes can be modelled through MIKE ECO Lab. It simulates the reaction processes in multi-compound systems, including degradation of organic matter, bacterial fate, exchange of oxygen with the atmosphere and oxygen demand from eroded sediments. This allows realistic analysis of complex phenomena related to water quality.

Biological processes can e.g. include diurnal variation of foul flow discharges and user-specified concentrations of foul flow components. The sediment types included in the interaction with biological processes are foul flow organic sediments, fine and course mineral in-pipe sediments originating from catchment runoff, potholes, and stilling basins. Biological processes can account for:

- Decay of BOD/COD in biofilm and water phase
- Hydrolysis of suspended matter
- Growth of suspended biomass
- Oxygen consumption from decay of BOD/COD, biofilm, and erosion of sediment
- Re-aeration
- Bacterial fate
- Interaction with sediments for nutrients and metals

Hydrodynamic and advection-dispersion (transport) simulations can be coupled with MIKE ECO Lab for efficient and versatile simulation of water quality processes. This option offers practically an unlimited range of processes to be simulated, as long as these can be described by a consistent model. MIKE ECO Lab in MIKE+ is supported by a set of MIKE ECO Lab editors:

- MIKE ECO Lab Templates
- MIKE ECO Lab State Variables
- MIKE ECO Lab Forcings
- MIKE ECO Lab Constants

MIKE ECO Lab and these four editors are used to model the biological processes. Each MIKE ECO Lab state variable must be associated to an Advection-Dispersion component (from the 'WQ components' editor), and therefore the initial conditions and boundary conditions for the state variables are defined by initial conditions and boundary conditions of these WQ components.

11.2.1 Notes on MIKE ECO Lab

MIKE ECO Lab is a numerical lab for Ecological Modelling. It is a highly flexible and open framework for the formulation of water quality models. It is an open and generic tool for customising Aquatic Ecosystem models to describe water quality, eutrophication, heavy metals, and ecology. 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 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, sea grasses, and mussels. Another use is in online forecasts of water quality.

Templates

MIKE ECO Lab uses template files (*.ecolab file) where water quality models are transparently defined. MIKE ECO Lab template files are customized collections of equations and parameters required for a specific type of Water Quality simulation.

The user may use a predefined MIKE ECO Lab Template installed with the software or may choose to develop their own. The MIKE+ installation includes some pre-defined water quality templates that can be applied “as is” or adjusted to conform to specific project requirements. A MIKE ECO Lab template is an ASCII file which can be accessed and edited through the MIKE ECO Lab editor.

Note: It is necessary to install MIKE Zero in order to edit a template file in the MIKE ECO Lab editor.

Please consult the MIKE Zero ECO Lab documentation on how to create or modify model templates, i.e. how to work with the MIKE ECO Lab editor.

A simulation job with the coupled HD/AD and MIKE ECO Lab model is set up and launched from within MIKE+ in the usual way. Also see Chapter 11.2.6 Running MIKE ECO Lab Simulations *(p. 385)*.

Integration Methods

Water quality calculations are based on predefined MIKE ECO Lab templates which contain models defined by a number of coupled differential equations solved through numerical integration and interactions between each equation.

Several Integration Methods are available for solving the coupled ordinary differential equations defined in the MIKE ECO Lab file. These options are offered in the Simulation Setup editor in MIKE+. See Chapter 11.2.6 Running MIKE ECO Lab Simulations *(p. 385)*.

Three 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 ad 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.

11.2.2 MIKE ECO Lab Templates

A MIKE ECO Lab template, describing the water quality processes, must be included in the project. Multiple templates can be used, if they apply to different spatial locations on the network. Templates are selected via the MIKE ECO Lab Templates editor (Figure 11.4).

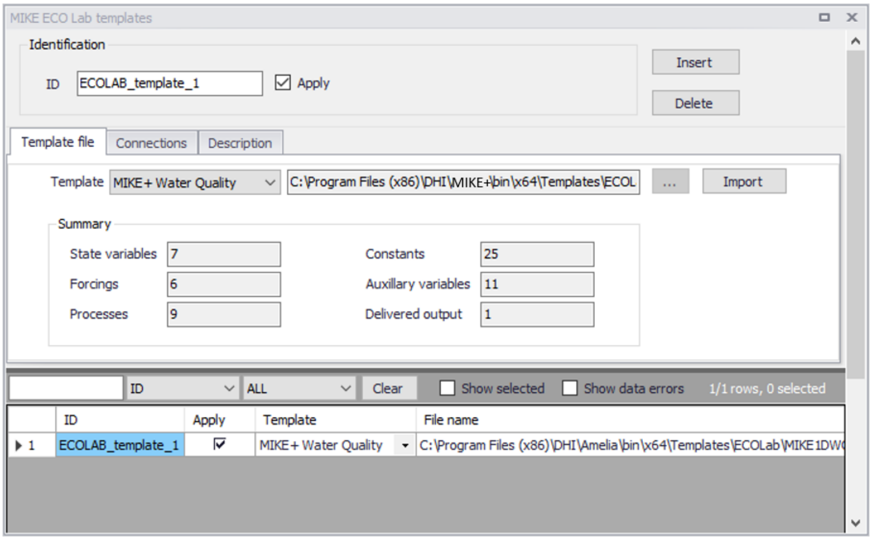


Figure 11.4 The MIKE ECO Lab Templates editor in MIKE+

For coupled simulations, coupling a 1D network to the 2D overland model, it is therefore possible to include one or more templates to model water quality in the network, and another template to describe water quality in the 2D domain.

A template is included in the simulation only if its 'Apply' check box is ticked, and is ignored otherwise.

Template File

Predefined MIKE ECO Lab templates are included in the MIKE+ installation located in the folder: 'C:\Program Files (x86)\DHI\MIKE+\<Version>\Templates\ECOLab'.

Point to the MIKE ECO Lab template file in the Template File tab page of the editor:

- **Template**: Define the MIKE ECO Lab template to which to connect using the drop-down list (or the ellipsis button for custom templates).
- **Import**: Use the Import button to read and import the contents of the template file (state variables, forcings and constants) into the MIKE+ project and editors. Pressing the Import button after the template has already been loaded (imported) will reload the template and reset all state variables, forcings and constants to default settings.
- **Summary**: Displays general information on the contents of the MIKE ECO Lab template file, such as the number of State variables, Forcings, Processes, Constants, Auxiliary variables, and Delivered output defined in the template.

Connections

Define for which model parts the loaded MIKE ECO Lab template is applied:

- **Global network**: The template is used over the whole network model (River and CS)
- **Nodes**: The template is used for CS network nodes. It may be in All nodes, a List of nodes, or Individual nodes selected from a list accessed via the ellipsis button, or interactively from the Map using the arrow button.
- **Links**: The template is used for links (i.e. River branches and/or CS pipes and canals). It may be in All links, a List of links, or Individual links selected from a list accessed via the ellipsis button, or interactively from the Map using the arrow button.
- **2D overland**: The template is used for the entire 2D domain as part of a 2D overland simulation. Only one template can apply at a time for a 2D overland simulation.

Figure 11.5 The Connections tab page in the MIKE ECO Lab Templates editor

Description

Add a free text description for a MIKE ECO Lab template on the Description tab page of the MIKE ECO Lab Templates editor.

11.2.3 MIKE ECO Lab State Variables

The MIKE ECO Lab State Variables editor is used for coupling MIKE+ WQ components to MIKE ECO Lab state variables and for possible reconciliation of differences in units used in MIKE+ and MIKE ECO Lab, respectively.

- **ID**: State variable identifier in MIKE+.
- **MIKE ECO Lab template**: MIKE+ ID for the template to which the state variable belongs.
- **State Variable**: State variable identifier in MIKE ECO Lab template file.
- **WQ Component**: MIKE+ WQ component coupled to the MIKE ECO Lab state variable.

- **Conversion Factor**: Conversion factor for possible units conversion between MIKE+ and MIKE ECO Lab
- **Description**: Add a free text description for a state variable via the Description tab page of the editor.

In the MIKE+ WQ Components editor, there must be one component defined for every MIKE ECO Lab state variable defined in the MIKE ECO Lab template. This means that all MIKE ECO Lab state variables are subject to both AD transport mechanisms and MIKE ECO Lab transformations.

For a **State Variable**, the associated **WQ Component** must be selected by the user. These WQ components must represent the same constituents as those in the MIKE ECO Lab template, but their names can be freely defined.

For coupled models, coupling a 1D network to the 2D overland model, different templates have to be used for the 1D network and for the 2D domain. When state variables from the different templates describe the same component, they must be associated to the same WQ component in order to be coupled during the simulation

The specified initial conditions and boundary conditions for the WQ components will therefore control the initial and boundary conditions of the state variables.

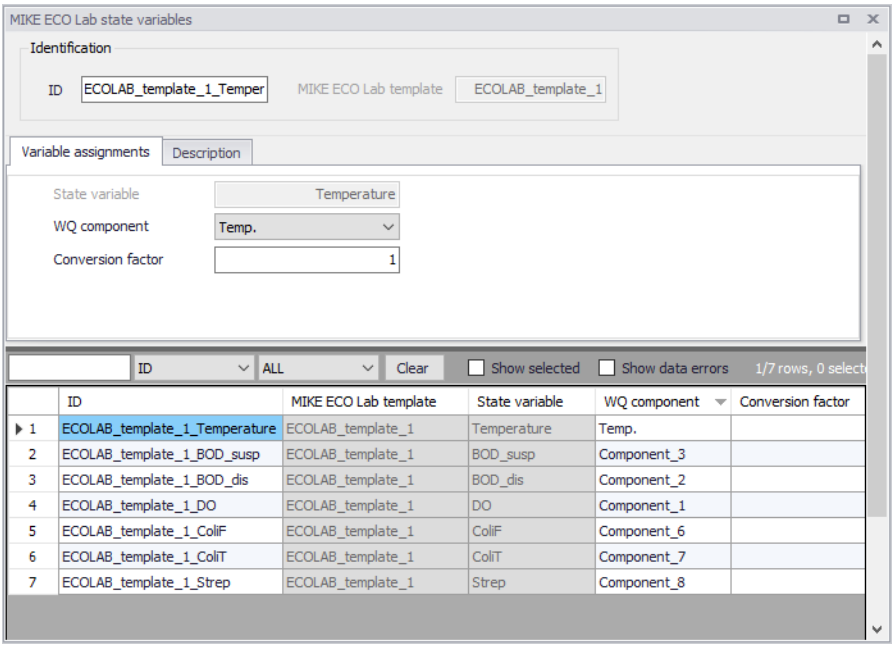


Figure 11.6 MIKE ECO Lab State Variables

The import operation described in the MIKE ECO Lab Templates editor will automatically create the required entries in the MIKE ECO Lab State Variables editor. The user must then couple each of these MIKE ECO Lab state variables to the relevant WQ component. Select the appropriate component from the WQ Component dropdown list, which is populated by components defined in the WQ Components editor.

The **Conversion Factor** (default value 1, i.e. no conversion) may be set different from 1.0 if the units for the AD components and MIKE ECO Lab state variables are different.

Note: The active fields in the editor are only used for coupling the MIKE+ WQ components with MIKE ECO Lab state variables, and for specifying a conversion factor. Edits to the state variables properties, if needed, shall instead be done in the MIKE ECO Lab template file.

Note: If multiple templates contain state variables with the same name, then these variables will be shared between the templates during the simulation, i.e. they will have the same state in all templates.

11.2.4 MIKE ECO Lab Forcings

Forcings are external variables which affect some of the processes on the state variables. They are automatically listed in this editor when the MIKE ECO Lab template is imported, from the 'MIKE ECO Lab templates' editor.

Forcings are “Built-In” or “User-Defined”, depending on the forcing definition in the MIKE ECO Lab template. “Built-in” means that forcings are provided by the hydraulic model through coupling with MIKE ECO Lab. “User Defined” forcings must be provided by the user, and can be either constant values (constant in time and domain), varying in time or varying in domain and time (the latter option is only enabled when the forcing is defined as varying horizontally in the MIKE ECO Lab template.

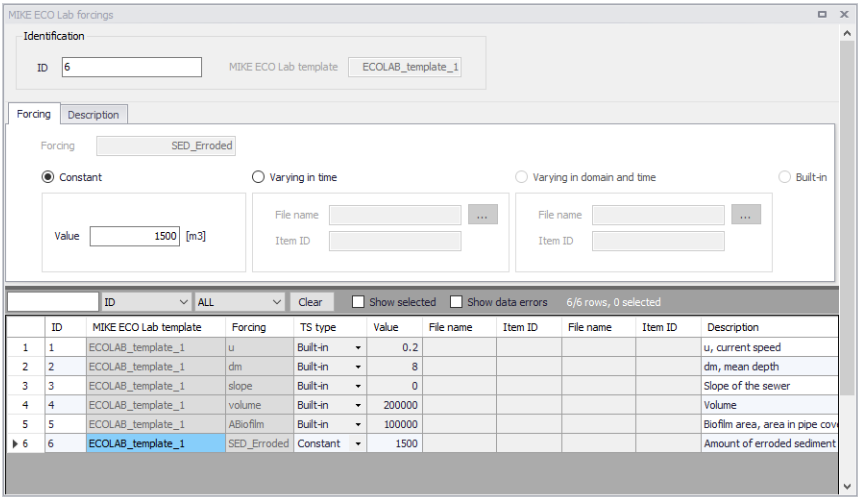


Figure 11.7 MIKE ECO Lab Forcings editor

The editor is used for setting the user-defined forcings associated with the loaded MIKE ECO Lab templates:

- **ID**: Forcing identifier in MIKE+.
- **MIKE ECO Lab template**: MIKE+ ID for the template to which the forcing belongs.
- **Forcing**: Forcing identifier in MIKE ECO Lab template.
- **Value**: Constant value for the forcing.
- **File Name**: Path/file name for the *.DFS0 time series file containing forcing time series data, or for the 2D data file when the forcing is varying in domain and time.
- **Item**: Data item selected in the file.
- **Description**: Free text description for the forcing defined via the Description tab page.

Built-In forcings are also imported into MIKE+, but don't need any configuration.

Note that the active fields in the editor are only those for specifying forcing constant values or input files. The remaining fields are read-only.

Note that for templates connected to the 1D network, some forcings may be defined as built-in in the template but appear as user-defined in MIKE+, in case the corresponding variable does not exist in the model setup. That is especially the case for Temperature and Salinity forcings. In order for them to actually work as a built-in forcing, it is necessary to include a WQ component with the corresponding type (temperature or salinity).

11.2.5 MIKE ECO Lab Constants

Constants are values which are fixed in time and affect some of the processes on the state variables. They are automatically listed in this editor when the MIKE ECO Lab template is imported, from the 'MIKE ECO Lab templates' editor.

This editor is used for setting the values of the constants used in the MIKE ECO Lab model, typically during model calibration:

- **ID**: Unique MIKE ECO Lab constant identifier in MIKE+.
- **MIKE ECO Lab template**: MIKE+ ID for the template to which the constant belongs.
- **Constant ID**: Constant identifier in MIKE ECO Lab template.
- **Default / uniform value**: Value for the constant, when it’s defined as uniform
- **File name**: Path and file name for the 2D data file containing the constant values, when the constant is defined as varying in domain (only available when the MIKE ECO Lab template is connected to the 2D overland model).
- **Item**: Data item selected in the file.
- **Local values**: From the 'Local values' tab, it is possible to apply local values on the 1D network, which will override the default value. For MIKE ECO Lab templates connected to the 2D overland model, use the option 'Varying in domain' instead. Multiple local values can be defined for the same constant, with the following settings:
  - Interpolate this constant between local values: If this option is selected, local values for the same constant will be interpolated in space when they are defined on the same river or pipe. If not selected, values will be interpreted as local values valid only for the specified chainage.
  - Location: each local value is specified either for a specific node or storage (respectively from the CS network and the river network), an entire link (river or pipe), a point on a link (at a single chainage), a reach on a link (defined from a start chainage to an end chainage), or finally using a list. When using a list, the local value is applied to all nodes, storages, rivers and pipes in this list.
  - Local value: local value for the constant.
- **Description**: Free text description for the constant defined in the Description tab page of the editor.

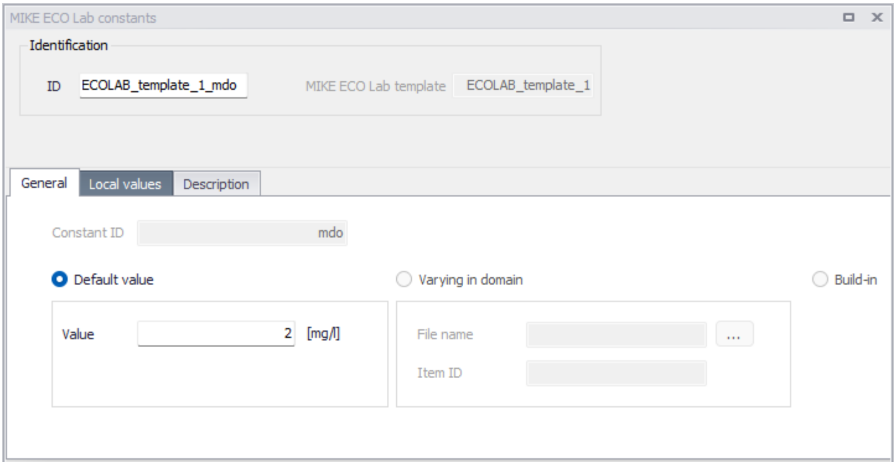


Figure 11.8 MIKE ECO Lab Constants editor in MIKE+

11.2.6 Running MIKE ECO Lab Simulations

To run a MIKE ECO Lab simulation, the wanted MIKE ECO Lab template(s) should first be imported to MIKE+. Then, WQ components must be created in the 'WQ components' editor as appropriate and each MIKE ECO Lab state variable must be associated to a WQ component. Make sure that the WQ component is set to use the unit which is expected for the state variable in the template file. Possible unit inconsistencies must be considered and resolved by providing proper conversion factors. Note that no other unit conversion / correction will be performed during the simulation.

Values of constants should be reviewed and modified as appropriate, as well as user-defined forcings.

Initial conditions and boundary conditions of the WQ components in the 1D network must also be specified when they should differ from 0. They must be defined for all components for the 2D overland model, even when they equal 0.

Result items from the MIKE ECO Lab model to be saved to result files during the simulation, must be configured in the 'Result files' editor.

The simulation with MIKE ECO Lab is activated by activating the ‘Transport (AD, SWQ)’ and ‘Water quality (MIKE ECO Lab)’ modules for the simulation setup in the Simulation Setup editor (Figure 11.9).

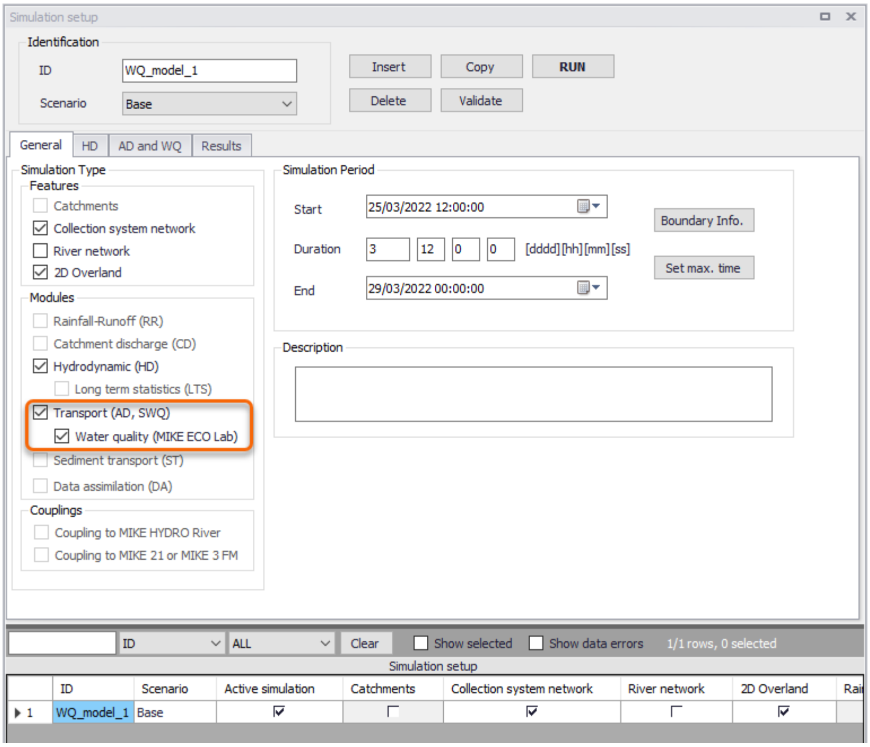


Figure 11.9 Setting up a MIKE ECO Lab simulation in the Simulation Setup editor

In the ‘AD and WQ’ tab page, define the MIKE ECO Lab update frequency and the MIKE ECO Lab Integration method to use for the simulation. The update frequency is a parameter that defines how often the water quality processes will be calculated during the simulation. It is defined as a multiple 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 a positive integer. Definition of the frequency should be based on careful considerations. 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 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. Also see Chapter 11.2.1 Notes on MIKE ECO Lab *(p. 376)* for more information on the Integration Methods.

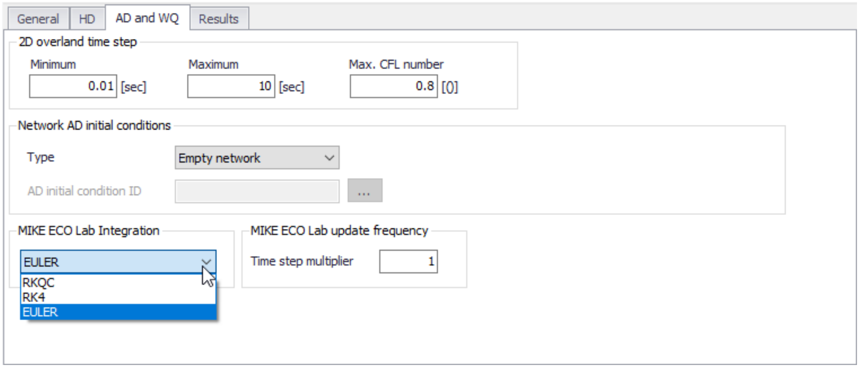


Figure 11.10 Define the MIKE ECO Lab Integration Method in the AD and WQ tab page of the Simulation Setup editor.

In the 'Results' tab, select the result files to be saved during the simulation.

12 Sediment Transport (ST)

Urban drainage networks are exposed to sediment loads carried by wastewater and/or by stormwater runoff. Such sediment loads may vary strongly both in space and time, as well as they may include a variety of particle types, ranging from large pieces of solid waste, street litter, sand and gravel from construction sites and unpaved areas, wind-borne dust, traffic debris, as well as small organic particles originating from wastewater. These sediments, if present in forms and quantities larger than the self-cleansing capacity of the sewer can handle, may create significant operational problems, such as reduced hydraulic capacity, increased overflows and full blockages of sewer pipes. These problems are typically associated with higher operational costs.

Modelling sediments in urban drainage networks does not belong to mainstream modelling work. However, the sediment modelling may be motivated by the wish to understand and eliminate existing or anticipated sediments-related problems, or by the need to document compliance with design criteria in terms of self-maintaining the sediment-free network.

In river contexts, modelling sediment transport can be used to assess long term morphology changes and alluvial resistance changes, considering both cohesive and non-cohesive sediments.

MIKE+ ST provides a modelling platform for such analyses. The ST module can be used in two main modes: a basic one, primarily designed to represent transport in pipe networks, and of a full-scale sediment transport model developed for river modelling applications. 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. Scientific background for the implemented solutions can be found in the document “MIKE1D Reference Manual, Sediment Transport (ST)”.

Inevitably, keeping the modelling apparatus reasonably simple and practically applicable requires some conceptualization of the complex reality. In MIKE+ this is achieved by limiting the number of model parameters and computational options to those most important ones. The following are examples of simplifications and conceptualization in MIKE+:

- Various sediment fractions are distinguished by the median grain size (D50), relative density and the applied transport mechanism. Typically, the sewer sediments are represented by one coarse fraction (D50>=0.1 mm) and one fine fraction (D50< 0.1 mm).
- Transport of the coarse fraction is preferably computed as non-cohesive transport, using one of the available sediment transport formulae.
- Transport of the fine fraction is preferably computed as cohesive transport, with advection-dispersion transport mechanism.

A detailed description of the ST editors and work flows is provided in the following paragraphs.

12.1 General Parameters

Three types of sediment transport analyses are supported by MIKE+ ST:

- Basic morphological analysis: in this type of analysis, the bed levels / sediment deposits are updated dynamically during the simulation (erosion/deposition) due to supply of sediments through the model's boundary condition and transport of sediments in the network. The basic mode offers simple and limited simulation options, primarily tailored for modelling the sediment transport in pipe networks.
- Advanced morphological analysis: this type simulates the same processes than the basic mode, but gives access to all sediment transport modelling options. This type is primarily designed for applications to river networks.
- Hydraulic effects only: this is an explicit method with disabled sediment transport processes. For this simulation type, the bed (sediments) levels are fixed during the simulation and controlled by the user-defined initial sediment depths, and any sediment boundary conditions is ignored.

With ‘Hydraulic effects only’ analyses, the bed level is fixed throughout the simulation and the only feed-backs from the sediment transport computations to the hydrodynamics are established via the reduced cross-section area and flow resistance (Manning number).

The flow resistance in a conduit with sediment deposits is calculated as a weighted average of the Manning number for the horizontal sediment bed and the wetted conduit walls. The Manning number of sediment deposits is controlled in the 'Pipes roughness' editor, and can be provided by the user or computed automatically based on the specified sediment fractions grain sizes.

No sediment is moved around in the system and any active sediment boundary condition is ignored. The only result items available for this type of analysis are: "Bed level", "Manning number" and "Bed shear stress".

This type of analysis is used to:

- investigate the hydraulic capacity in pipes with sediment deposits, i.e. document changes of hydraulic capacity and overflow volume and frequency due to sediment deposits and their removal;
- calculate and map bottom shear stress under representative hydraulic conditions. Result of this analysis may e.g. be used for planning of preventing maintenance (sediment removal) and identification of sedimentation-prone locations.

In morphological analyses (both basic and advanced), the sediment transport continuity equation is solved, based on the corresponding values of the hydrodynamic parameters (i.e. discharge, water levels, etc.). The feedback to the hydrodynamic module is established through dynamically changed flow area and flow resistance (Manning number).

The morphological ST model in MIKE+ allows for calculation of sediment transport for any number of specified sediment fractions, i.e. to perform the analysis for non-uniform sediments.

12.1.1 Basic morphological analysis

With 'Basic morphological analysis' type of simulations, the following additional settings must be defined in this editor.

Transport model

The transport of the coarse, non-cohesive fractions is modelled by one of four non-cohesive sediment transport formulae:

- Engelund - Hansen: The formula determines the total sediment transport directly. It has been derived from consideration of the work done by the flow on the sediment being transported. Originally, the formula was derived for a dune covered bed, but it was found applicable to the upper regimes (plane bed and anti-dunes) as well.
- Ackers - White: This formulae determines directly the total sediment transport. The formula is semi-empirical, based partly on dimensional analysis and partly on physical arguments.
- Engelund - Fredsøe - Deigaard: The formula calculates the total transport as the sum of the bed load transport and the suspended transport. The sediment transport is calculated from the skin friction, i.e. the shear stress acting on the surface of the bed. In this formula, it is possible to describe the development of sand dunes in pipes and hence include the resulting friction into the computations. The total bed resistance is then calculated as the sum of a contribution from the skin friction acting on the dune and an expansion loss behind the dune.

- van Rijn: In the van Rijn sediment transport formula, the sediment transport is divided into bed load and suspended load. The bed load is calculated from the saltation height, the particle velocity and the bed load concentration. The bed load computations follow the approach of Bagnold (1973), which assumes that the motion of the bed load particles is dominated by the gravity forces. When the bed shear velocity exceeds the fall velocity, the sediment is transported in suspension. The suspended load is calculated as the depth integration of the local concentration and flow velocity. The method uses the reference concentration computed from the bed load transport. The formula has been verified for particles in the range 200 - 2000 my-m. The verification based on 600 data sets, showed that 77% of the predicted bed load rates were within 0.5 and 2 times the observed values, van Rijn (1984a). The verification for the suspended load, using 800 data sets showed that 76% of the predicted values were within 0.5 and 2 times the observed values, van Rijn (1984b).

All non-cohesive fractions will be described using the selected transport model, in this type of analysis.

No general guidelines can be given for the preference of any one formulation over another, as this may be guided by the modeller's preference.

The implemented formulae demonstrate that the sediment transport is a highly non-linear function of the flow velocity: depending on the formulation, the sediment transport is proportional to the velocity raised to the power from 3 to 5. Obviously, a correct description of the hydrodynamics in the model setup is an essential prerequisite for a meaningful and accurate simulation of sediment transport.

Relative density

The density of all sediments in the model, relative to water. This density is assumed uniform throughout the entire network.

Porosity of deposits

The fraction of sediment deposits' volume filled with pores. This fraction is assumed uniform throughout the entire network.

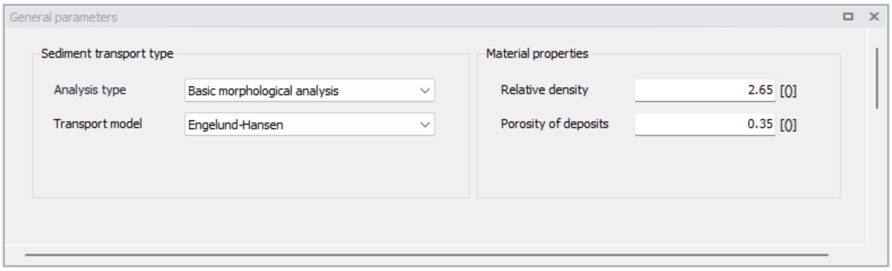


Figure 12.1 The 'General parameters' editor for basic morphological analyses

12.1.2 Advanced morphological analysis

With 'Advanced morphological analysis' type of simulations, the following additional settings must be defined in this editor.

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 records in the 'Sediment fractions' editor.

Include 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 parameter's 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.

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)
- aw of the wall (skin friction depth)

Use Egiazaroff hiding function

When multiple sediment fractions are simulated, the Egiazaroff hiding function can be applied.

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.

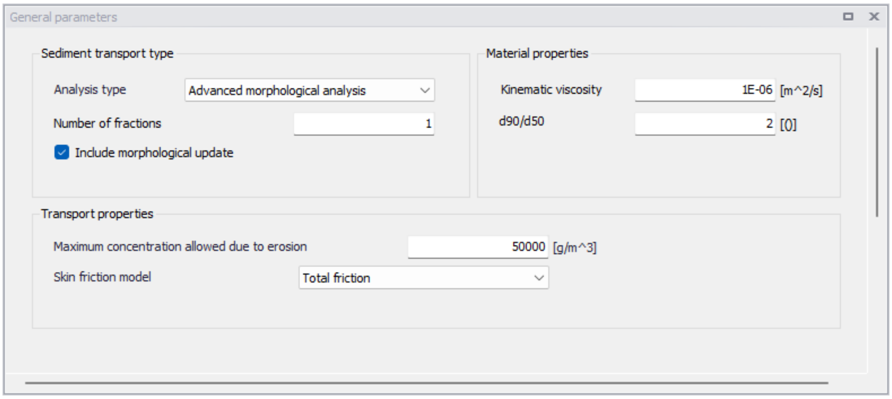


Figure 12.2 The 'General parameters' editor for advanced morphological analyses

12.2 Sediment Fractions

Any number of individual sediment fractions can be specified (at least one). Sediment fractions are inserted and edited in the "Sediment fractions" editor.

The content of this editor changes depending on whether the basic or advanced type of simulation is selected.

12.2.1 Sediment fractions in basic analysis

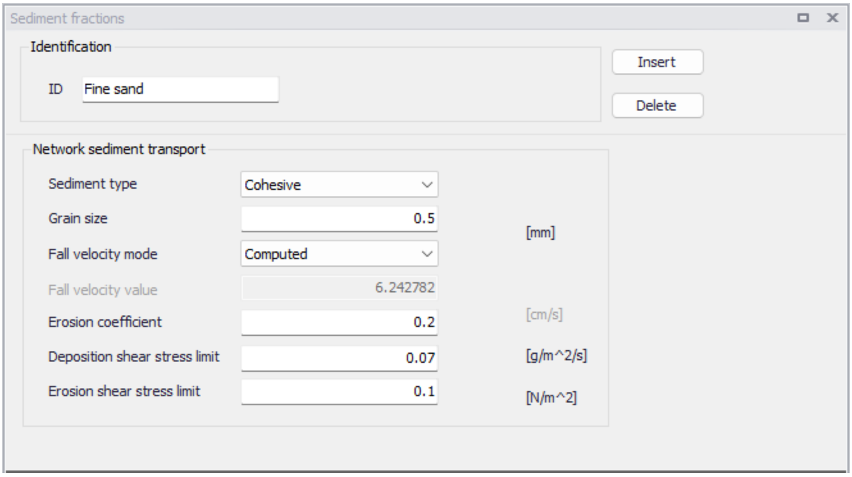


Figure 12.3 The "Sediment fractions" editor for basic morphological anayses

Sediment fractions are defined by the following properties.

ID

Each fraction is identified by a unique name.

Sediment type

The choice between cohesive or non-cohesive fraction type is selected from this list.

Non-cohesive fractions will be described using the transport model selected in the 'General parameters' editor.

Cohesive fractions will be described by an Advection-Dispersion mechanism including a description of the erosion and deposition processes. The transport of the fine fraction (D50 less than 0.1 mm) should preferably be modelled as cohesive fractions, as the transport of particles transported in suspension is better described by used of the advection-dispersion equation. The fractions transported by the advection-dispersion module are still included in the morphological calculation, but the description of erosion/deposition is changed.

Note: The dispersion of cohesive fractions is controlled by the dispersion factors defined in the 'AD Dispersion' editor, which is accessed from the 'Water quality' group in the Setup tree.

A special type 'Fixed material' can also be selected for very coarse fractions, which are not expected to be transported during the simulation.

Grain size

The median grain size (d50) value, which is applied throughout all calculation points in the network.

Fall velocity mode

The fall velocity can either be user-defined or can automatically be computed using the Rubey formula by MIKE+. The Rubey formula computes the velocity based on sediment fraction grain size, sediment relative density and water viscosity (at 20 deg.C).

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.

Erosion coefficient

Determines the rate of erosion when bottom shear stress exceeds the erosion shear stress limit (only for cohesive fractions).

Deposition shear stress limit

The maximum value of bottom shear stress at which sedimentation of the actual fraction will occur (only for cohesive fractions).

Erosion shear stress limit

The minimum value of bottom shear stress at which erosion of the actual fraction will occur (only for cohesive fractions).

Note: Erosion and deposition only take place in links, but not in nodes.

12.2.2 Sediment fractions in advanced analysis

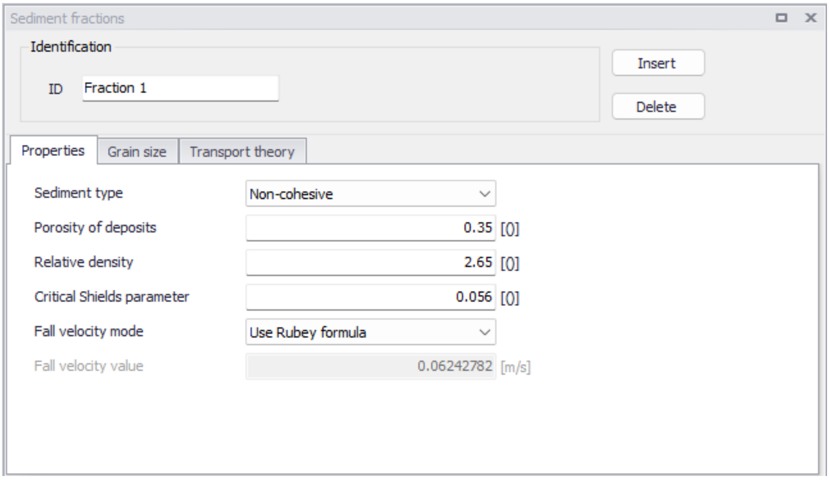


Figure 12.4 The "Sediment fractions" editor for advanced morphological analyses

ID

Each fraction is identified by a unique name.

Properties

Sediment type

The choice between cohesive or non-cohesive fraction type is selected from this list.

Non-cohesive fractions will be described using the transport model selected in the 'General parameters' editor.

Cohesive fractions will be described by an Advection-Dispersion mechanism including a description of the erosion and deposition processes. The transport of the fine fraction (D50 less than 0.1 mm) should preferably be modelled as cohesive fractions, as the transport of particles transported in suspension is better described by use of the advection-dispersion equation. The fractions transported by the advection-dispersion module are still included in the morphological calculation, but the description of erosion/deposition is changed.

A special type 'Fixed material' can also be selected for very coarse fractions, which are not expected to be transported during the simulation.

Porosity of deposits

The porosity of the sediment fraction, i.e. the fraction of sediment deposits' volume filled with pores. Porosity of the bottom sediment is usually in the range of 0.3 to 0.7.

Relative density

The density of the sediment fraction, relative to water (solid/ water). 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 mode

The fall velocity can either be user-defined or can automatically be computed using the Rubey formula by MIKE+. The Rubey formula computes the velocity based on sediment fraction grain size, sediment relative density and water viscosity (at 20 deg.C).

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

Default grain size

A default global grain size value, which is applied throughout all calculation points in the network, unless local values have been defined.

Local grain size

Local grain size values can only be specified for single-fraction models, and when the fraction is non-cohesive. Local grain size values are applied at specific locations defined either:

- List: the local grain size is applied to all links (pipes or rivers) in the selected list.
- Entire link: the local grain size is applied to the selected link.
- Point on link: the local grain size is applied to a specific point (identified by its chainage) on the selected link. The local grain size will be interpolated in space between the different points, when multiple points are defined on the same link.

Transport theory for non-cohesive fractions

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 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
- 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 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 faction.

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.

Transport theory for cohesive fractions

The following parameters are required to describe the suspended load of cohesive fractions. Default values are applied globally throughout all calculation points in the network, unless local values have been defined. Local values are added to the secondary table, and are located by a point identified with a link ID (pipe or river ID) and a chainage along this link.

Exponent

The exponent used in the erosion equation, from MIKE 1D reference manual.

Erosion shear stress limit

The minimum value of bottom shear stress at which erosion of the actual fraction will occur.

Erosion coefficient

The erosion coefficient used in the erosion equation, from MIKE 1D reference manual. It determines the rate of erosion when bottom shear stress exceeds the erosion shear stress limit.

Deposition shear stress limit

The maximum value of bottom shear stress at which sedimentation of the actual fraction will occur.

12.3 Pipes roughness

The hydraulic resistance in a conduit with sediment deposits originates from the pipe wall and from the sediment deposits on the bottom of the sewer. In general, the flow resistance from the sediment deposits consists of two contributions: one part originates from the grain friction and the other part originates from the expansion loss behind the bed forms (ripples, dunes, etc.). In MIKE+ ST, only the flow resistance from the sediment grains is calculated.

The total roughness in the pipe is calculated as a weighted average of the Manning number for the horizontal sediment surface and the wetted conduit walls. The roughness value of the horizontal sediment surface can be provided by the user or it can be calculated based on the sediment grain size and bed shear stress. Both methods can be applied globally (i.e. for entire model) and for individual conduits. Any local definition overrides the global specification.

The average shear stress in a pipe with sediment deposits is calculated from the Einstein side-wall elimination procedure. The calculation is based on the pipe roughness and the bed shear stress calculated from the sediment deposits.

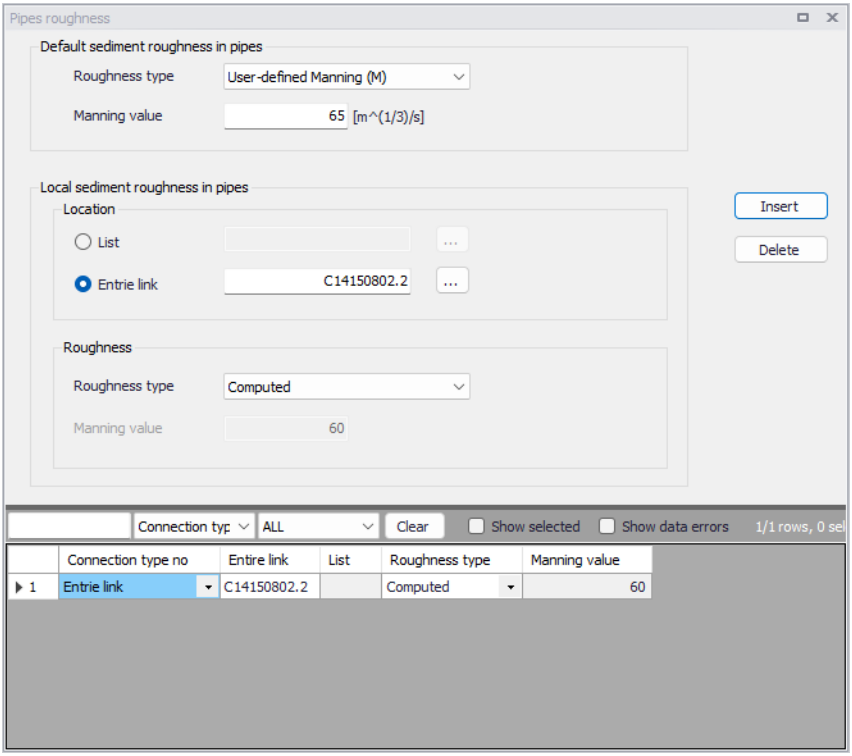


Figure 12.5 The “Pipes roughness” editor

When the local value is applied to a 'List', then all pipes included in the selected list will be applied the selected method.

12.4 Bed parameters

For advanced morphological analyses, the 'Bed parameters' editor controls how the bed of the channels is described in the model.

Layers layout

The bed material may be represented by one or two layers. The active layer is the layer in which sediment transport occurs. This active layer overlays a passive layer. Each layer is divided into the number of fractions specified in the 'Sediment fractions' editor.

The 'Layers layout' selection defines 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 when that fraction is non-cohesive.

Target thickness of active layer

The following parameters are required for the target thickness of the active layer. Default 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 Link ID (river or pipe ID) 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 selection of the morphological update method applies to river networks, but only when the morphological update is included, in the 'General parameters' editor.

A default 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 (in the 'Cross sections' editor). Primarily for use in cross sections with floodplain to exclude the floodplain from sediment transport calculation.

The default method is applied throughout all calculation points in the river network unless local methods have been defined. Local methods may be specified in the secondary table, with locations defined by River ID and Chainage.

Note: For pipe networks, the morphological update method is always 'Horizontal distribution'.

12.5 Passive links

For advanced morphological analyses, it is possible to exclude sediment transport calculations in selected links (rivers or pipes) when this option is active. Links in which sediment transport should not be calculated must be listed in this table.

No sediment will enter or leave a passive link, and therefore no morphological changes will take place in such a link.

12.6 Non-scouring bed level

For advanced morphological analyses, 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.

Default settings are applied throughout all calculation points in the network unless local values have been defined. Local values may be specified in the secondary table, with location defined by Link ID (river or pipe) and Chainage.

Apply unlimited scouring

When this option is selected, the scouring is unlimited. The combinations of default 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 default or local non-scouring bed level.

12.7 Sediment removal in basins

Basins act as "sediment traps" because the water flow kinetic energy and turbulence (hence, dynamic forces acting on sediments) are much lower than in the conduits. Sedimentation in basins may in some cases be an unwanted side-effect and an operational problem (need for frequent sediment removal), while in other cases it may be deliberately designed to induce sedimentation.

MIKE+ ST does not simulate sedimentation in basins: both non-cohesive and cohesive fractions continues unaffected by default through basins (actually, through any node in the model).

However, effect of basins on sediment transport of non-cohesive fractions may be emulated by defining a sediment removal efficiency in individual basins. This function simply removes the specified fraction of the transport rate (named 'Settling factor') from the model.

Cohesive sediments are not affected by this function.

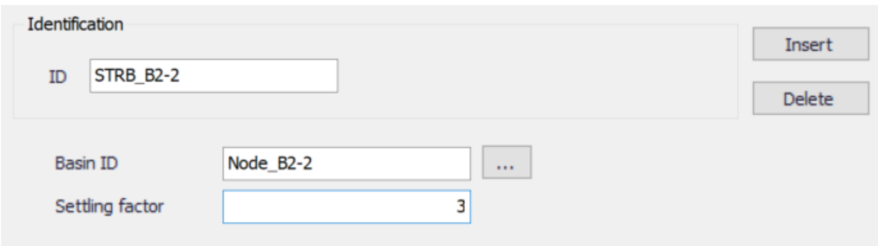


Figure 12.6 The "Sediment removal in basins" editor

"Sediment removal in basins" editor fields are described in Table 12.1..

Table 12.1 "Sediment removal in basins" editor data fields

Edit field	Description	Unit SI	Unit US
ID	Unique identifier of sediment removal definition	-	-

Table 12.1 “Sediment removal in basins” editor data fields

Edit field	Description	Unit SI	Unit US
Basin ID	Identifier of actual basin	-	-
Settling factor	Turbulence constant indicating the settling performance of the basin 1: poor performance 3: good performance 5: very good performance ∞ : ideal performance	-	-

The sediment removed in basins can be reported in result files for each basin individually as sediment and mass transport (instantaneous and accumulated) for each ST fractions separately and for total sediment transport.

Note: This functionality is available only for basin nodes on collection system networks. It is not available in storages from river networks.

12.8 Sediment removal in weirs

Weirs act as efficient barriers for non-cohesive sediment transport, as the weir crest levels are normally significantly higher that the conduit's invert. This means that significant part of non-cohesive sediment cannot be transported over weirs.

MIKE+ ST simulates effect of weirs on transport of non-cohesive sediments by defining a constant sediment removal efficiency or removal efficiency as function of weir discharge. This function simply removes the specified fraction of the non-cohesive fractions’ transport rate from the weir flow. This removal fraction can either be constant in time, or be a function of the discharge (as defined in a 'Removal efficiency' table).

Cohesive sediments are not affected by this function.

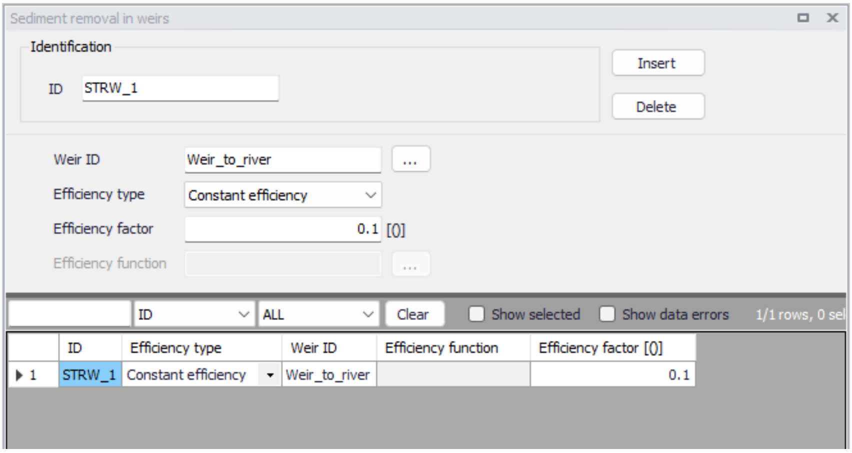


Figure 12.7 The "Sediment removal in weirs" editor

"Sediment removal in weirs" editor fields are described in Table 12.2..

Table 12.2 “Sediment removal in weirs” editor data fields

Edit field	Description	Unit SI	Unit US
ID	Unique identifier of sediment removal in weirs definition	-	-
Weir ID	Identifier of actual weir	-	-
Method	Constant efficiency: sediments removal by efficiency factor Efficiency (flow): sediment removal efficiency specified as function of weir flow	-	-
Efficiency factor	Fraction of non-cohesive sediment to be removed from the actual weir flow (only with "Constant efficiency")		
Efficiency function	Reference to a tabular function of type "Removal efficiency" (only with "Efficiency(flow)")		

The sediment removed in weirs can be reported in result files for each weir individually as sediment and mass transport (instantaneous and accumulated) for each ST fractions separately and for total sediment transport.

Note: This functionality is available only for weirs on collection system networks. It is not available in weirs from river networks.

12.9 Sediment transport boundary conditions

Sediment transport boundary conditions are only relevant for morphological analysis. For "Hydraulic effects only" type of analysis, no sediment boundary conditions are needed. Any active sediment boundary conditions is ignored if the sediment transport analysis is specified as "Hydraulic effects only".

Sediment transport boundary conditions may be specified at any inflow boundary in the model (upstream boundary) and at outlet water level boundaries (downstream boundary).

Sediment inflows to a drainage or river model can be specified either in association with catchment model or with network model, defined as "WQ boundary prop-erties" for the any hydraulic or SWQ boundary conditions.

For most of boundary condition types, the sediment boundary properties are specified directly as sediment concentration in the inflowing water. For SWQ boundaries, sediments inflows are defined either as a tabular function (specific runoff vs. sediment concentration) or are calculated by SWQ advanced methods (BuildUp/WashOff and EMC).

A schematic overview over the possible types of sediment boundary conditions for collection system networks is presented in Table 12.3.

Note that when running an integrated simulation (Catchment + Network simul-taneously), sediment outputs from the catchment model are automatically transferred to the network model at the locations of catchments connections to the network model. If the simulations of catchment and network models are run separately in a sequence, the catchment model's sediment outputs must be specified as WQ boundary properties of the corresponding hydraulic boundary conditions to the network model.

Table 12.3 Overview over possible sediment boundary conditions for collection system networks.

	Boundary condition type	WQ boundary property (for each sediment fraction)				
		Constant	Cyclic	Time series /result file	Table concentration	SWQ advanced
Catchment model	Catchment discharge	YES, sediment concentration	YES, sediment concentration	YES, sediment concentration	NO	NO
	Catchment discharge per PE	YES, sediment concentration	YES, sediment concentration	YES, sediment concentration	NO	NO
	Catchment discharge per area	YES, sediment concentration	YES, sediment concentration	YES, sediment concentration	NO	NO
	Stormwater loads (surface)	YES, sediment concentration	NO	YES, sediment concentration	YES, sediment concentration as function of runoff intensity	YES, SWQ calculation
	Stormwater loads (RDII)	YES, sediment concentration	NO	YES, sediment concentration	YES, sediment concentration as function of runoff intensity	YES, SWQ calculation
Network model	Inflow to node	YES, sediment concentration	YES, sediment concentration	YES, sediment concentration	NO	NO
	Inflow to link	YES, sediment concentration	YES, sediment concentration	YES, sediment concentration	NO	NO
	Inflow from result file	YES, sediment concentration	YES, sediment concentration	YES, sediment concentration	NO	NO
	Exfiltration from node	NO	NO	NO	NO	NO
	Exfiltration from link	NO	NO	NO	NO	NO
	Outlet water level	YES, sediment concentration, AD transport only	YES, sediment concentration, AD transport only	YES, sediment concentration, AD transport only	NO	NO

Refer to the 'Water Quality Boundary Condition Properties' editor for additional details about definition of boundary conditions (page 242).

12.10 ST initial depths

Initial sediment depths are defined in the 'ST initial depths' editor, available from the 'Initial conditions' group in the Setup tree.

Initial sediment depths are given default settings which apply globally (for all links in a model) unless local values are specified for selected links. The local definitions override the default settings.

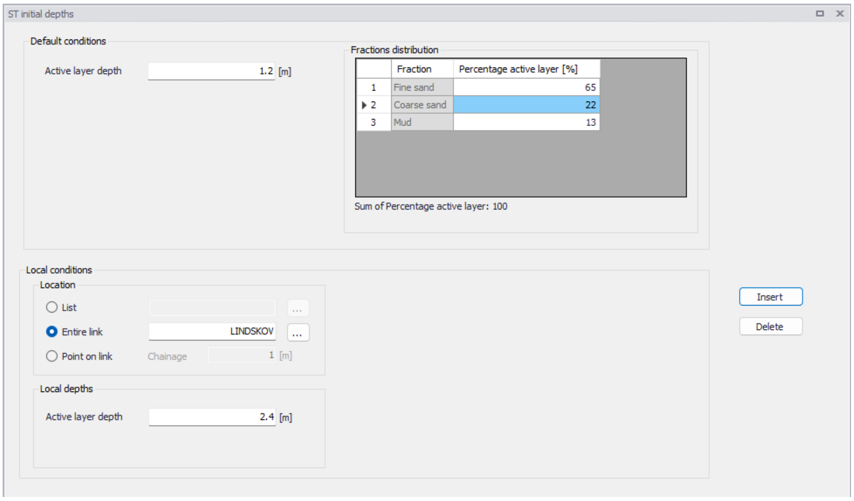


Figure 12.8 The 'ST initial depths' editor

A description of all available settings follows.

Active layer depth
The initial depth of the active layer.

Limited passive layer depth
The option to specify the initial depth of the passive layer is available only for advanced morphological analyses when the layers layout is set to 'Active and passive layers'.

Passive layer depth
The initial depth 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.

Fractions distribution table
This table contains the initial percentage of each fraction within the sediment layer(s). Depending on the type of analysis being conducted, the table contains the distribution for the active layer only, or for both the active and the passive layers. The sum of the initial percentages should equal 100%, otherwise the percentage values will be scaled so that the sum equal 100% during the simulation.

Local conditions locations
Local settings for the initial sediment depths can be specified. For each local condition, the location may be defined by:

- List: the local depth is applied to all links (pipes or rivers) in the selected list.
- Entire link: the local depth is applied to the selected link.
- Point on link: the local depth is applied to a specific point (identified by its chainage) on the selected link. The local depth will be interpolated in space between the different points, when multiple points are defined on the same link.

Local depths
For each local condition, the local values of initial depth need to be specified here. The local depth of the active layer is mandatory. The local passive layer depth must also be specified for advanced morphological analyses when the layers layout is set to 'Active and passive layers' and when 'Limited passive layer depth' is selected for the default initial depth.

Local fractions distribution table
This table contains the initial percentage of each fraction within the sediment layer(s), at the location of the local condition.

Note: initial sediment depths are not used for advanced morphological analyses when the layers layout is set to 'Fully alluvial model (no layer)'.

12.11 ST initial concentrations

Initial sediment concentrations can be specified for cohesive fractions, in advanced morphological analyses. They are defined in the 'ST initial concentrations' editor, available from the 'Initial conditions' group in the Setup tree.

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+.

Initial sediment concentrations are given default settings which apply globally (for all links in a model) unless local values are specified for selected links. The local definitions override the default settings.

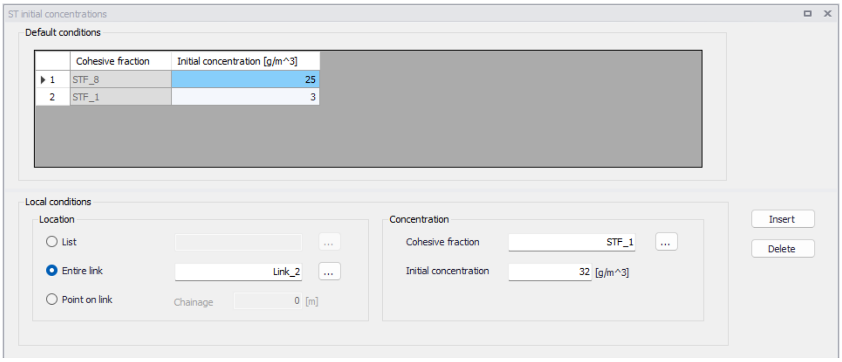


Figure 12.9 The 'ST initial concentrations' editor

A description of all available settings follows.

Default conditions table

This table contains the default initial concentration of each cohesive fraction.

Local conditions locations

Local settings for the initial sediment concentrations can be specified. For each local condition, the location may be defined by:

- List: the local concentration is applied to all links (pipes or rivers) in the selected list.
- Entire link: the local concentration is applied to the selected link.
- Point on link: the local concentration is applied to a specific point (identified by its chainage) on the selected link. The local concentration will be interpolated in space between the different points, when multiple points are defined on the same link.

Local concentration

For each local condition, the local value of initial concentration needs to be specified here. It is defined by the local concentration value and by the corresponding sediment fraction to which it applies.

12.12 Sediment Transport Modelling Workflows

Several typical work flows in modelling sediment transport are presented below.

12.12.1 Analysis of hydraulic effects of sediment deposits

This analysis focusses on the hydraulic effects (changes in flow capacity, overflows, surcharge, etc.) as consequence of fixed bed sediment deposits in a pipe network.

Step 1: Prepare a stable and well-calibrated drainage network model, loaded with representative hydraulic inflows

Step 2: Set the "Sediment transport | General parameters" analysis type to "Hydraulic effects only".

Step 3: From the ‘ST initial depths’ editor, specify initial sediment depths (fixed-bed sediment depths), globally and, optionally, locally. The local sediment depth specifications override global specification.

Step 4: From the ‘Pipes roughness’ editor, specify flow resistance for sediment deposits, globally and, optionally, locally. The local Manning number specifications override global specification.

Step 5: Review the contents of the default result file for sediment transport (Default_Network_ST) and include the wanted output items. Note that this type of analysis generates only result items "Bottom level", Bottom shear stress", and "Manning number". Alternatively, make a copy of default network result file, change its contents type to "Mixed contents" and add the wanted ST result items.

Step 6: Set up ST simulation. Create a simulation job, including "Hydrodynamic (HD)" and "Sediment transport (ST)" modules. Set appropriate simulation period, simulation time step and hot-start file (optionally).

Step 7: Review result files to be generated and set appropriate result saving frequency

Step 8: Run a simulation

Step 9: Review results

12.12.2 Analysis of wastewater sediments transport in a drainage network

This is a morphological analysis of sedimentation in a wastewater collection network. The analysis includes sediment loads from wastewater inflows generated on urban catchments, and their transport, sedimentation and erosion in the sewer network.

Step1: Prepare a stable and well-calibrated drainage network model, loaded with representative hydraulic inflows, defined as "Catchment discharge", representing wastewater generated by population in the catchments. Wastewater load is defined by quantity and diurnal variation.

Step 2: Set the "Sediment transport | General parameters" analysis type to “Basic morphological analysis". Choose the ST formula and review basic sediments properties (relative density and porosity of deposits).

Step 3: Specify at least one sediment fraction (Sediment transport | Sediment fractions) and review/modify its properties.

Step 4: Specify initial sediment depths), globally and, optionally, locally. The local sediment depth specifications override global specification.

Step 5: Specify flow resistance for sediment deposits, globally and, optionally, locally. The local Manning number specifications override global specification.

Step 6: Specify WQ Boundary properties for the wastewater boundary condition, as sediment concentration in wastewater, separately for each sediment fraction. The concentration may be given as a constant or as diurnal variation.

Step 7: Review the contents of the default result file for sediment transport (Default_Network_ST) and include the wanted output items. Alternatively, make a copy of default network result file, change its contents type to "Mixed contents" and add the wanted ST result items.

Step 8: Set up ST simulation. Create a simulation job, including "Catchment discharge”, “Hydrodynamic (HD)”, “Transport (AD,SWQ)” and “Sediment transport (ST)" modules.

Set appropriate simulation period, simulation time step for catchment discharge and for network simulation, and hot-start file for the network model (optionally).

Step 9: Review result files to be generated and set appropriate result saving frequency.

Step 10: Run a simulation

Step 11: Review results

12.12.3 Analysis of stormwater sediments buildup/washoff ("first flush") and sediment transport in a drainage network

This is a morphological analysis of sedimentation in a wastewater collection network. The analysis includes sediment loads from storm runoff, generated on urban catchments, and their transport, sedimentation and erosion in the sewer network

Step 1: Prepare a stable and well-calibrated drainage network model, loaded with representative rainfall, defined as "Rainfall" boundary condition.

Step 2: Set the "Sediment transport | Generel parameters" analysis type to "Basic morphological analysis". Choose the ST formula and review basic sediments properties (relative density and porosity of deposits).

Step 3: Specify at least one sediment fraction (Sediment transport | Sediment fractions) and review/modify its properties.

Step 4: Specify initial sediment depths), globally and, optionally, locally. The local sediment depth specifications override global specification.

Step 5: Specify flow resistance for sediment deposits, globally and, optionally, locally. The local Manning number specifications override global specification.

Step 6: Define buildup/washoff parameters for the specified sediment fractions or the model's catchments

Step 7: Define SWQ boundary condition (Stormwater loads (surface) and its WQ properties, based on SWQ advanced method "BuildUp Washoff".

Step 8: Review the contents of the default result file for sediment transport (Default_Network_ST) and include the wanted output items. Alternatively, make a copy of default network result file, change its contents type to "Mixed contents" and add the wanted ST result items.

Step 9: Set up ST simulation. Create a simulation job, including "Rainfall-Runoff (RR)", Transport (AD, SWQ)", "Hydrodynamic (HD)" and "Sediment transport (ST)" modules.

Set appropriate simulation period, simulation time step for runoff and for network simulation, and hot-start file for the network model (optionally).

Step 10: Review result files to be generated and set appropriate result saving frequency

Step 11: Run a simulation

Step 12: Review results

13 Result Specifications

MIKE+ allows flexible generation of model simulation result files and summaries. Various result file setups may be specified in the Result Specifications section, which may then be used in a simulation as needed by the user.

Result files obtained from simulations may be customised with respect to the number of result files generated, file types, spatial extent of saved results, and types of result items included in the files.

In MIKE+, **one result file** may be set up to contain **multiple result sets** comprised of **various location-item combinations** (see Sections 13.1.6 and 'User-specified results').

Result file and custom network summary specifications are defined under the following editors:

- **Result Files.** Result files obtained from simulations may be customised with respect to the number of result files generated, file types, spatial extent of saved results, and types of parameters included in the files.
- **Network Summary**. Basic HTML summaries for network simulations may be extended by adding supplementary tables to the summary.

13.1 Result Files

The Result Files editor (Figure 13.1) provides a facility for viewing and specifying result file setups in a project. The types or results available depend on the active features and modules for the project (General Settings | Model type).

Result setups are a mix of Default results and User-specified results.

Default results

The editor is initially filled with Default result setups. A Default set is shown for each active Module in the project. These records are distinguished by the "Default_" prefix in their IDs.

The following table shows an overview of the various Default result files that are possible in MIKE+.

Table 13.1 Overview of possible Default result files in MIKE+

Default ID	Model Type	Format	Content Type
Default_Surface_runoff	Catchments	.RES1D	Surface runoff
Default_RDII	Catchments	.RES1D	RDII

Table 13.1 Overview of possible Default result files in MIKE+

Default ID	Model Type	Format	Content Type
Default_Storm_water_quality	Catchments	.RES1D	Storm water quality
Default_Storm_water_sediments	Catchments	.RES1D	Storm water sediments
Default_LIDs	Catchments	.DFS0 (currently hard-coded)	LIDs
Default_Catchment_discharge	Catchments	.RES1D	Catchment discharge
Default_Catchment_discharge_quality	Catchments	.RES1D	Catchment discharge quality
Default_Network_HD	Network	.RES1D	Hydrodynamic
Default_Network_RTC	Network	.RES1D	Real time control
Default_Network_AD	Network	.RES1D	Pollution transport
Default_Network_MIKE_ECOLab	Network	.RES1D	MIKE ECO Lab
Default_LTS_extreme_statistics	Network	.RES1D	LTS extreme statistics
Default_LTS_chronological_statistics	Network	.RES1D	LTS chronological statistics
Default_2D_overland	2D Overland	DFSU / DFS2	2D area
Default_2D_overland_AD	2D Overland	DFSU / DFS2	2D area, AD
Default_2D_Flood_statistics	2D Overland	DFSU / DFS2	2D flood statistics
Default_2D_Volume_balance	2D Overland	DFS0	Volume balance

Default result sets include results in all model elements, saving values for the basic calculation parameters.

Default result sets may be modified with respect to selection of results items to be included in the file. Only basic result items are initially included in Default result setups.

Note that the saving locations may not be modified for Default results, and results will always be saved in all elements.

Also note that for some content types, only one file format is allowed and cannot be changed.

User-specified results

The user may also define custom result file setups according to their modelling needs.

The following properties may be customised for user-specified results:

- File format
- Location: Spatial extent or network elements for which results are saved in the file.
- Result items: Calculated parameters to be included in the file.

Note that **multiple result sets comprised of various location-item combinations may be specified for one user-specified result file** (see Section 13.1.6 for more details).

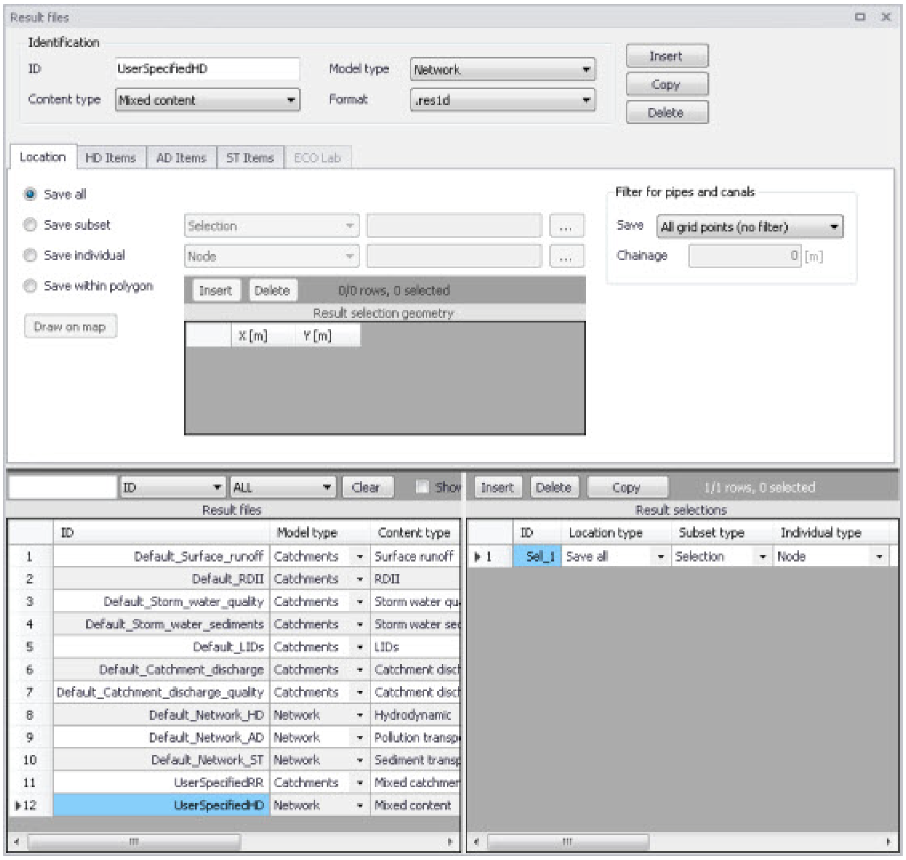


Figure 13.1 The Result Files Editor

The various data tabs and components of the Result Files editor are described in succeeding sections.

13.1.1 Identification

The Identification group box on the Result Files editor contains general information on a result file item.

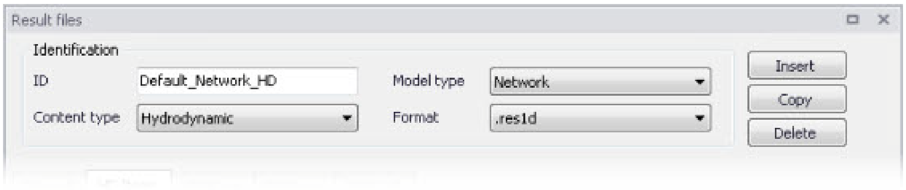


Figure 13.2 The Identification group box on the Result Files Editor

Model Type

Each result file item is categorised based on the type of model from which it is generated. The model may either be a Catchment model, a Network model (comprising CS network and/or River network) or a 2D Overland model.

Content Type

This parameter characterises result files according to the calculation modules, and filters the available result items that may be included in the result file setup. The available categories depend on the selected Model Type for a result file setup.

Note that there are also “mixed content” types, which allow flexibility in terms of mixing result items across various active computational modules in one result file setup (See Section 13.1.6).

Content Type can be:

- For ‘Catchments’ Model Type:
  - Mixed catchment contents. Content type for which result items across various active catchment model-related computational modules may be included.
  - Surface runoff
  - RDII
  - Storm water quality
  - Storm water sediments
  - LIDs
  - Catchment discharge
  - Catchment discharge quality

- Statistics. For this content type, the result file will save maximum, time of maximum, minimum, time of minimum, and average values for the selected result items. For the relevant result items, it will also save the accumulated values over time (e.g. volume accumulated over time, for a discharge results). This result file contains a single time step.

- For ‘Network’ Model Type:
  - Mixed content. Content type for which result items across various active network model-related computational modules may be included (e.g. HD, AD, ST, and MIKE ECO Lab).
  - Hydrodynamic
  - Pollution transport. If the ‘Water Quality (AD)’ module is active.
  - MIKE ECO Lab. If the ‘Water Quality (MIKE ECO Lab)’ module is active.
  - LTS extreme statistics. If the ‘Long Term Statistics (LTS)’ module is active.
  - LTS chronological statistics. If the ‘Long Term Statistics (LTS)’ module is active.
  - Statistics. For this content type, the result file will save maximum, time of maximum, minimum, time of minimum, and average values for the selected result items. For the relevant result items, it will also save the accumulated values over time (e.g. volume accumulated over time, for a discharge results). This result file contains a single time step.
  - 2D map. This content type provides a 2D result file where results from the 1D simulation are distributed across cross sections and interpolated along the river line. The maps are saved in dfs2 file format (rectangular grids) and constructed through interpolation in space of the 1D 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.
  - State files (initial conditions). State files store results from all active modules in the simulation (Hydrodynamic, Rainfall-runoff, Transport, etc.) and for a given time step. When 'State files' are included in a simulation, there are actually several files saved during the simulation (one per time step, with the saving period selected in the 'Simulation setup' editor). State files are designed to be used as initial condition afterwards, for another simulation. They contain more detailed information than .res1d files and therefore offer more accurate initial condition definitions. Using state files as initial conditions is also activated in the 'Simulation setup' editor, in the 'HD' tab.

- Decoupling. This produces a special hydrodynamic result file, to be used as input for a decoupled Transport (Advection-Dispersion or Water Quality) simulation. The created special hydrodynamic result file stores the simulated water level and the average discharge per saved time step. Decoupling a transport simulation from the hydrodynamic simulation speeds up the simulation by getting the hydrodynamic conditions from the decoupled result file, instead of running the hydrodynamic simulation at the same time. The decoupling of the transport simulation is activated in the 'Simulation setup' editor.

- For '2D Overland' Model Type:
  - 2D area: a map result file containing instantaneous results at regular time intervals
  - 2D flood statistics: a map result file containing a single time step with statistical results (e.g. maximum values over time)
  - Time series: time series results from one or more cells from the 2D domain
  - Volume balance: a time series providing total volumes over the 2D domain
  - Section discharge: a time series providing the total discharge computed through a cross section
  - 2D area, AD: a map result file containing instantaneous water quality results at regular time intervals
  - Time series, AD: time series of water quality results from one or more cells from the 2D domain
  - 2D culverts results: time series of results in selected culverts. The time series will save for each culvert the target discharge (discharge calculated using the empirical formulas), the discharge (effective discharge, which can be less than the target discharge if the upstream water depth is too low), accumulated discharge, and water level on both sides of the culvert (for a short culvert the left and right water level is the mean water level in the real wet elements to the left and right of the section of faces; for a long culvert the start and end water levels are the mean water level in the real wet elements to the right of the two end sections).
  - 2D weirs results: time series of results in selected weirs. The time series will save for each weir the target discharge (discharge calculated using the empirical formulas), the discharge (effective discharge, which can be less than the target discharge if the upstream water depth is too low), accumulated discharge, and water level on both sides of the weir (the left and right water levels are the mean water level in the real wet elements to the left and right of the section of faces).
  - Section statistics: a *.dfs0 file with a relative item axis, showing the maximum water depth and/or maximum water level in each element face from the mesh, along a defined cross section. The axis of this result file is the index number of the element faces along the cross section.

Format

Some result files may be saved in various file formats.

An overview of the Identification group attributes in the Result Files editor is shown in Table 13.2 below.

Table 13.2 Overview of the Identification group box attributes (Table msm_RS)

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	Unique identifier for the result file setup	Yes	MUID
Model Type	Categorises the model used to generate the result file as either: - Catchment model, or - Network model - 2D overland model	Yes	ModelTypeNo
Content Type	The type of result set under which the result file item falls under.	Yes	ContentTypeNo
Format	The file format for the generated result file is either: - .RES1D -.DFS0 - DFSU - DFS2 - state1d - txt	Yes	FormatNo

The buttons to the right of the Identification group box control the data rows in the left overview table in the Result Files editor.

'Insert' button

Adds a new result item in the Result Files table.

‘Copy' button

Creates a copy of an active result file item. The ID of the copied item is set the same as the original item’s ID plus the suffix ‘_Copy’.

'Delete' button

Deletes the current selected rows from the left overview table in the editor.

13.1.2 Location

The flexibility in results-saving in MIKE+ extends to possibilities for selecting elements or specifying locations for which to save results in the file.

Result locations may be defined for user-specified result files, but not for Default results. Note that Default result files will always save results in all model elements (i.e. ‘Save all’ option).

Result saving locations are specified in the Location tab in the Result Files editor (Figure 13.3).

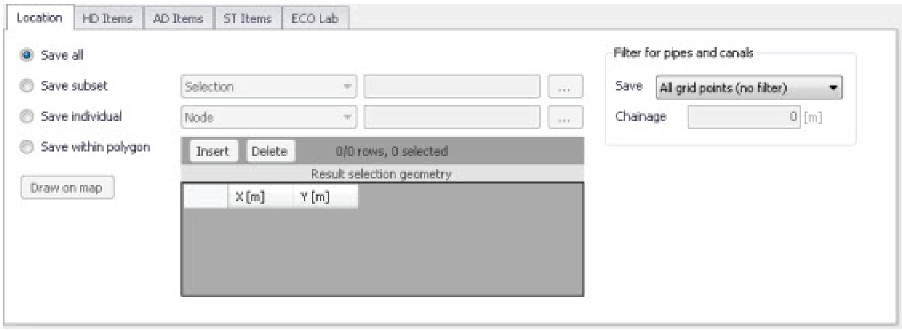


Figure 13.3 The Location Tab in the Result Files Editor

Table 13.3 Overview of the Location tab attributes in the Result Files editor (Table msm_RSS)

Edit field	Description	Used or required by simulations	Field name in datastructure
[Location radio buttons]	Radio button for selection result saving location: - Save all - Save subset - Save individual - Save within polygon - Save list / all but list	Yes	SelectionNo
[Save Subset dropdown menu]	Dropdown menu for selecting the type of subset	If Location = Save subset	SubsetNo

Table 13.3 Overview of the Location tab attributes in the Result Files editor (Table msm_RSS)

Edit field	Description	Used or required by simulations	Field name in datastructure
[Selection list input box]	Input box for a Selection List	If Location = Save subset and Subset = Selection	SelectionListID
[Save Individual dropdown menu]	Dropdown menu for selecting the type of model element for which to save results	If Location = Save individual	IndividualNo
[Element input box]	Input box for an element selection	If Location = Save individual	ElementID
Save [Filter for Pipes and Canals]	Option for selecting the calculation grid point(s) along pipes and canals for which to save results	Yes If Model Type = Network, and results are saved in Pipes and Canals	GridPointNo
Chainage [Filter for Pipes and Canals]	Input box for specifying the chainage (i.e. distance from upstream node) of grid point along the pipe or canal for which to save results	Yes If Model Type = Network, results are saved in Pipes and Canals, and Save = User-specified chainage	Chainage

Save all

This option saves results in all model elements. All Default result files (see Default results, page 415) use this option, which may not be modified. Also note that this option is not available when results are saved in .DFS0 format (see Format section).

Save subset

This option offers a dropdown list of possible subset groups for which to save results. The list varies according to Model Type associated with the results. A selection list must be defined (Figure 13.4) when the subset is a 'Selection'.

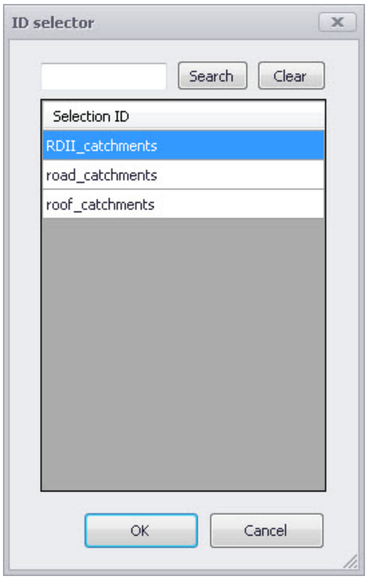


Figure 13.4 Selection lists in the ID Selector window

Save individual

This option offers a dropdown list of model elements for which to save results. The list varies according to the Model Type associated with the results. An element ID must be defined for the selected model element.

Save within polygon

Results from only the network elements or the 2D domain elements within a specified polygon are saved in the file. The polygon is characterised by vertex XY coordinates defined in the secondary grid on the Location tab (Figure 13.5).

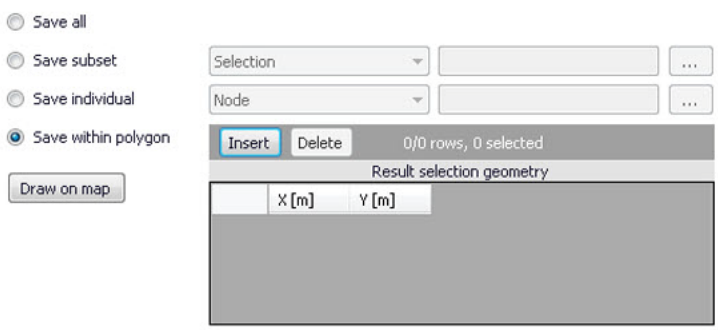


Figure 13.5 The Result Selection Geometry secondary grid in the Location Tab

The location polygon may be defined by:

- Defining values in the secondary table.
  Polygon vertex locations are added and removed from the table using the 'Insert' and 'Delete' buttons at the top of the secondary grid.
- Drawing the polygon on the map using the 'Draw on map' button to the left of the table.
  Use the 'Draw on map' button to define a new polygon on the map. The ‘Draw on map’ button activates the Map view (Figure 12.5). Draw a polygon on the map by defining vertex locations. Double-click on the map to finish the polygon editing. The polygon coordinates are then shown in the secondary table in the Location tab of the Result Files editor.

  The 'Draw on map' button is renamed 'Edit on map' as soon as a polygon has been defined and the secondary table is filled.

  Note that the location polygon is shown on the Map only while drawing. The polygon is no longer shown on the map once the polygon has been drawn.

  When the polygon already exists (i.e. when the Location tab secondary table is not empty), the 'Edit on map' button allows for editing the existing polygon. The Map is shown, where polygon vertices may be moved, deleted, or added.

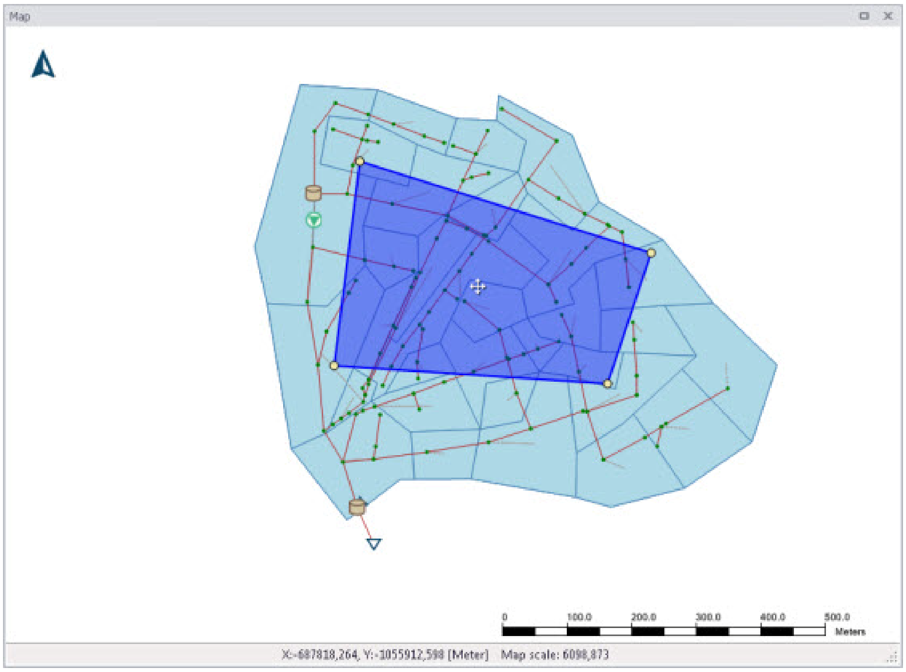


Figure 13.6 Defining a result location polygon on the Map

Coordinates

For 2D time series data, the X and Y coordinates of the time series must be specified in the table. When multiple coordinates are specified for the same result file, each location will be saved as an individual item in this file. For each time series, the raw results for the 2D domain element in which the coordinates fall will be saved.

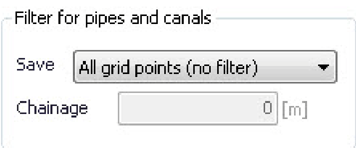
For a 2D section discharge result file, coordinates must be specified for the cross section polyline, through which the discharge will be computed.

For a 2D section statistics result file, coordinates must be specified for the cross section polyline, along which the maximum water depths and water levels will be reported.

Defining saving grid points

The Filter for Pipes and Canals group box is shown on the right side of the Location tab when the result is from a network model (i.e. Model Type = Network).

This section is used to select grid points along pipes and canals for which results are saved in the result file.



The 'Save' dropdown list offers the following options:

- All grid points (no filter)
- Upstream grid point
- Downstream grid point
- Up- and downstream points
- Middle grid point
- User specified chainage. If ‘User specified chainage’ is selected, specify the grid point location for which results are saved in the 'Chainage' field below. Results are saved for the grid point closest to specified chainage value.

Location of 2D maps

For the 2D map content type, the Location tab holds the following information to define the location and the resolution of the rectangular grid on which the 1D results are interpolated.

Cell size: The cell size of the dfs2 result file. The cell size controls the spatial resolution of the map.

Rotation: The rotation of the rectangular grid. 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 definition: This controls the way the extent of the map in the two spatial dimensions is defined. Two options are available:

- Length: with this option, you specify the width and height of the rectangle to define the dimensions along the two horizontal axes. MIKE+ will derive the extent of the file from the origo and these lengths.
- Number of cells: with this option, the user specifies the number of cells of the rectangular grid in the two spatial dimensions.

Lower left corner: The X coordinate (in the first column) and the Y coordinate (in the second column) of the lower left corner of the rectangular grid.

Lengths: The length of the grid along the X coordinate (in the first column) and along the Y coordinate (in the second column). These fields can be edited only when the extent is defined using the option 'Length'. When the extent is defined using the option 'Number of cells', the fields display the lengths which are automatically derived from the number of cells and the cell size.

Number of cells: Number of cells of the grid along the X coordinate (in the first column) and along the Y coordinate (in the second column). These fields can be edited only when the extent is defined using the option 'Number of cells'. When the extent is defined using the option 'Length', the fields display the numbers of cells which are automatically derived from the lengths and the cell size.

The grid may alternatively be defined by drawing the rectangle on the map using the 'Draw on map' button. This button activates the Map view: click and drag on the map to draw the rectangle. Use the icons on the map to move, resize or rotate the rectangle. Right-click on the map to finish editing. The 'Draw on map' button is renamed 'Edit on map' as soon as a grid has been defined.

Note that the grid is shown on the Map only while drawing. The grid is no longer shown on the map once editing is finished.

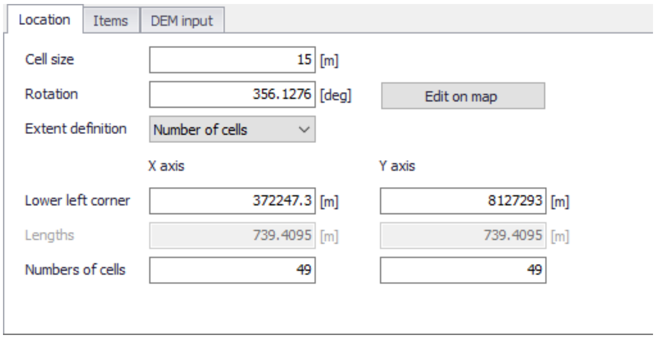


Figure 13.7 Defining the rectangular grid for a 2D map result file

Note: The results are mapped only within the extent of cross sections as illustrated in Figure 13.8.

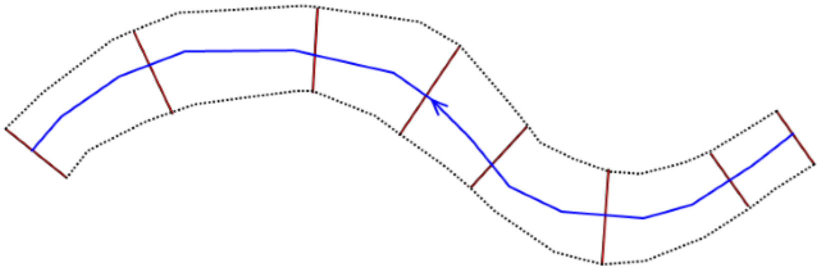


Figure 13.8 Illustration of area covered by the 2D map result, based on cross sections extent (red) and river line (blue).

No results will be mapped outside the cross sections extent, and the obtained maps will therefore represent exactly the extent of the flow during the 1D simulation.

An example of a flood map produced through the map-feature is presented in Figure 13.9.

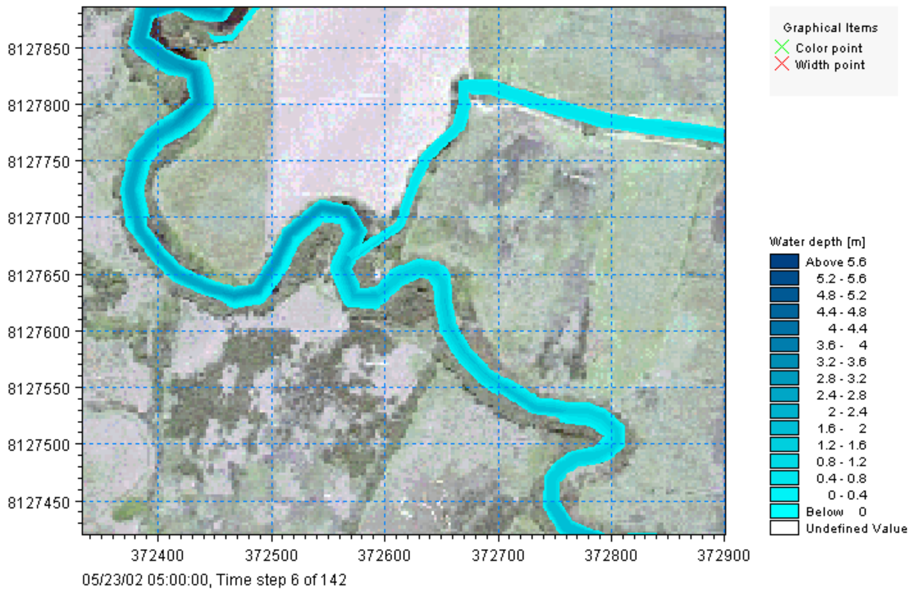


Figure 13.9 An example of a 2D flow map

Save list / all but list

The 'Save list' and 'Save all but list' options are only available when saving network hydrodynamic results in a .dfs0 or .txt format.

The result file will respectively save all calculation points covered by the table, or all calculation points from the network except those covered by the table. The table is used to define a list of links (pipes or rivers) with upstream and downstream chainages encapsulating all calculation points between these two values.

Note that it is not possible to save these results in .dfs0 or .txt format in nodes or storages.

13.1.3 Format

When network hydrodynamic results are saved to .txt format, this tab is used to control the format of the created text file.

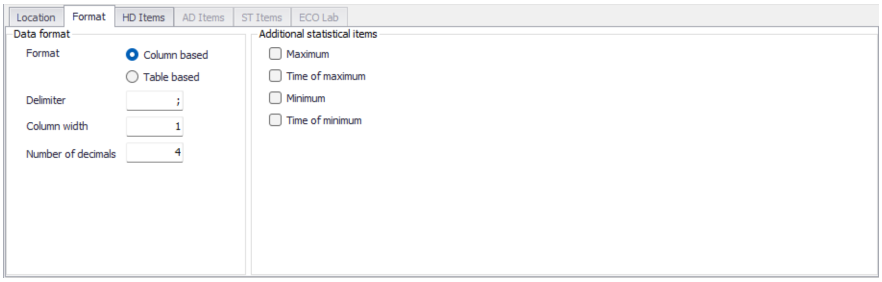


Figure 13.10 The Format tab in the Result Files editor.

Format

Two file formats are available:

- Column based: this will generate a text 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.
- Table based: this will generate a text 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.

Delimiter

This controls the special character (e.g. semicolon, comma, etc.) to be used between each column.

Colum width

This is the desired minimum width of each column. The actual width in the created file will be larger if the content of the column doesn't fit in the minimum width.

Number of decimals

The number of decimals written to the file, for result values.

Additional statistical items

For each result item saved to the file, it is also possible to save some statistics, including:

- Maximum: a separate table at the bottom of the file will show the maximum value of the saved result items.
- Time of maximum: a separate table at the bottom of the file will show the time of maximum value of the saved result items.
- Minimum: a separate table at the bottom of the file will show the minimum value of the saved result items.
- Time of minimum: a separate table at the bottom of the file will show the time of minimum value of the saved result items.

13.1.4 Items

Tabs in the Result Files editor are used to select items that will be stored in the result file. Different tabs are shown depending on the Model Type and active project Modules.

Note that customising Items related to LIDs is currently not available. Also, modifying items related to MIKE ECO Lab results is done in the MIKE ECO Lab template (.ECOLAB) and not the MIKE+ interface.

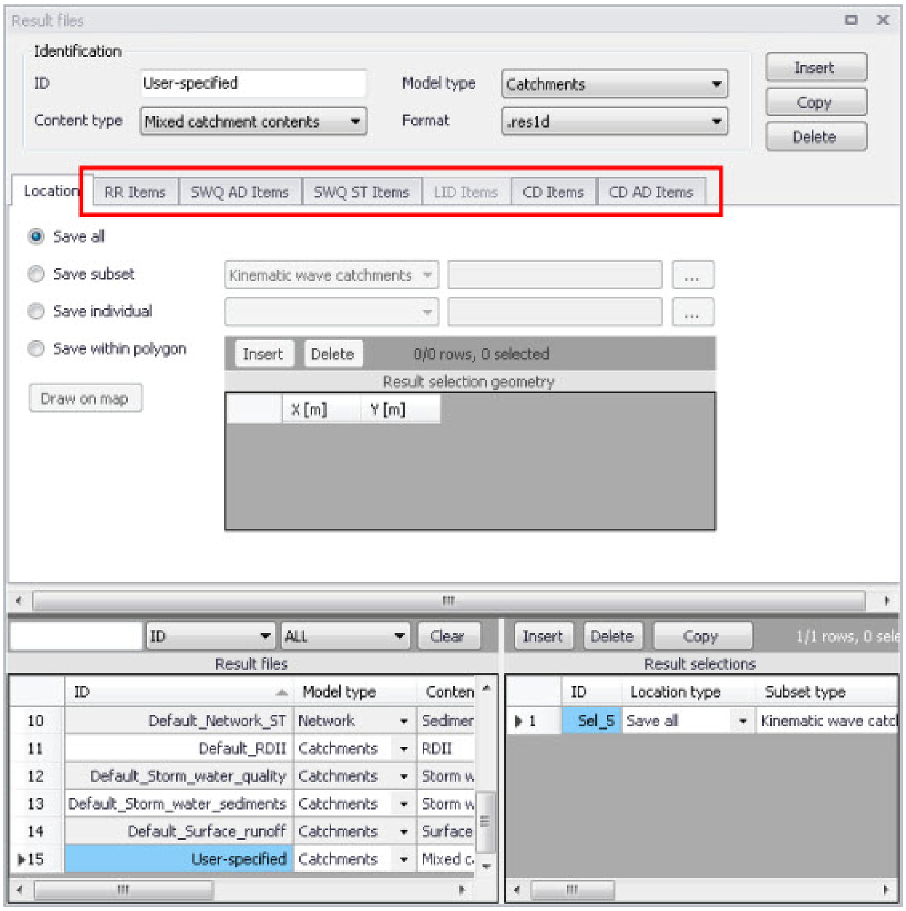


Figure 13.11 Item Tabs in the Result Files Editor. This figure shows available tabs for results from Catchment models.

Each tab shows items related to a computation Module, and the items are categorised as:

- Basic items. Primary result parameters for a simulated process.
- Additional items. Additional result items that provide greater detail on the simulated processes for the system.

The following sections describe the various result items available in MIKE+.

RR Items (Surface runoff and RDI)

These are catchment rainfall-runoff modelling result items.

Table 13.4 summarises items that may be saved in surface runoff result files.

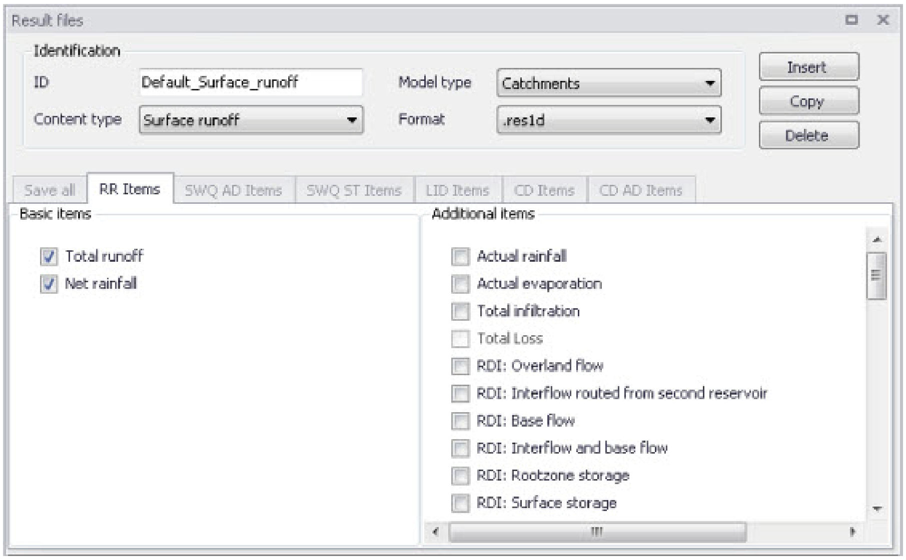


Figure 13.12 The RR Items Tab in the Result Files Editor. The 'Total runoff' and 'Net rainfall' items are initially included in Default Surface Runoff results.

Table 13.4 Overview of Surface Runoff result items in the RR Items Tab

Basic Items	Additional Items
Total runoff Net rainfall	Actual rainfall Actual evaporation Total infiltration RDI: Overland flow RDI: Interflow routed from second reservoir RDI: Base flow RDI: Interflow and base flow RDI: Rootzone storage RDI: Surface storage RDI: Groundwater depth RDI: Infiltration to groundwater RDI: Overland first reservoir flow, from first to second reservoir RDI: Interflow first reservoir flow, from first to second reservoir RDI: Capillary flux RDI: Overland first reservoir storage RDI: Overland second reservoir storage RDI: Lower base flow SnowStorage RDI: Snow ZoneTemperature RDI: Snow ZoneRainfall RDI: Snow ZoneWaterRetention RDI: Snow ZoneMeltingCoefficient RDI: Snow ZoneAreaCoverage RDI: Snow ZoneMeltingWater TimeArea: InitialLossStorage UHM: Excess Rainfall KW Runoff [ImperviousSteep, Flat] KW Runoff [PerviousSmall, Medium, Large] KW Depth [ImperviousSteep, Flat] KW Depth [PerviousSmall, Medium, Large] KW WettingLoss [PerviousSmall, Medium, Large] KW WettingLoss [ImperviousFlat] KW StorageLoss [ImperviousFlat] KW StorageLoss [PerviousSmall, Medium, Large] KW Infiltration [PerviousSmall, Medium, Large] KW InfiltrationPotential [PerviousSmall, Medium, Large]

SWQ AD Items (Stormwater quality)

These are results related to the modelling of water quality of stormwater from catchments.

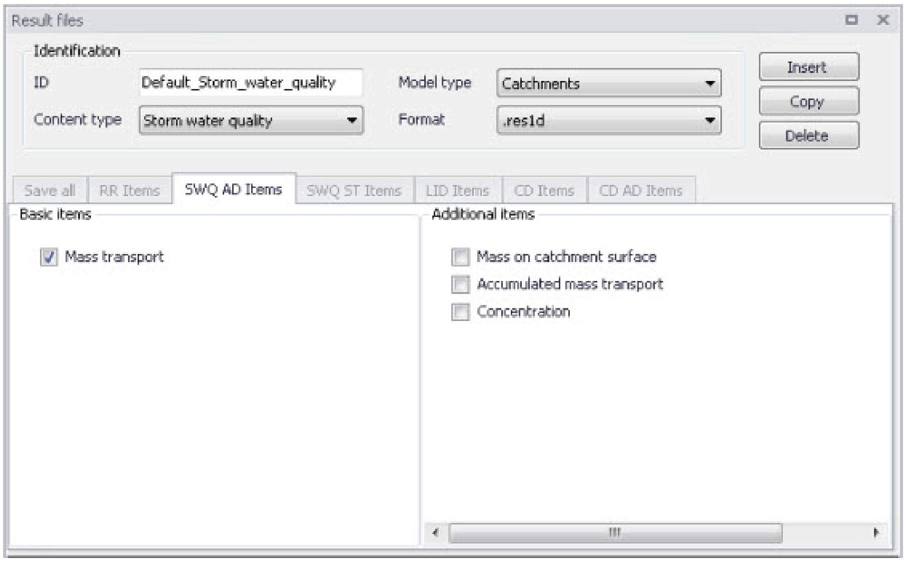


Figure 13.13 The SWQ AD Items Tab in the Result Files Editor. The 'Mass transport' item is initially included in Default Stormwater Quality results.

Table 13.5 Overview of Stormwater Quality result items in the SWQ AD Items Tab

Basic Items	Additional Items
Mass transport	Mass on catchment surface Accumulated mass transport Concentration

SWQ ST Items (Stormwater sediments)

These are results related to the modelling of sediment transport with stormwater over catchments.

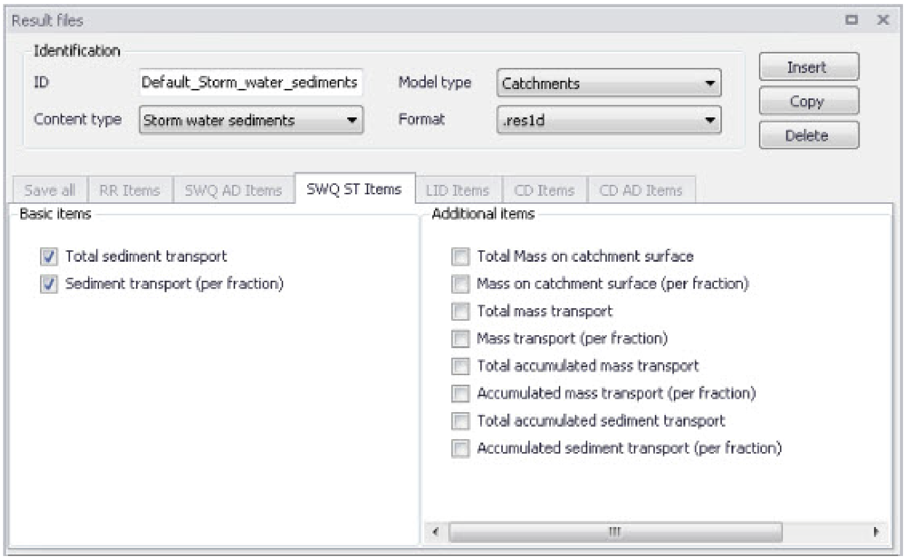


Figure 13.14 The SWQ ST Items Tab in the Result Files Editor. The 'Total sediment transport' and 'Sediment transport (per fraction)' items are initially included in Default Stormwater Sediments results.

Table 13.6 Overview of Stormwater Sediments Transport result items in the SWQ ST Items Tab

Basic Items	Additional Items
Total sediment transport Sediment transport (per fraction)	Total Mass on catchment surface Mass on catchment surface (per fraction) Total mass transport Mass transport (per fraction) Total accumulated mass transport Accumulated mass transport (per fraction) Total accumulated sediment transport Accumulated sediment transport (per fraction)

LID Items

It is currently not possible to customise result items for LID results in MIKE+. All result items are saved by Default. Table 13.7 shows the LID result items from a catchment model saved in .DFS0 format.

Table 13.7 Overview of LID result items

Basic Items	Additional Items
Rain	Evaporation
Inflow	Surface Depth
Surface Flow	Soil Moisture
Drain Flow	Storage Depth
Infiltration	Surface to Soil
	Soil to Storage
	Drain Storage Depth (Green Roof)
	Soil to Drain Storage (Green Roof)
	Pavement Moisture (Porous pavement)
	Surface to pavement (Porous pavement)
	Pavement to Storage (Porous pavement)
	Mass Checksum

Please refer to the 'LID Deployment Result File' section under the 'Rainfall-Runoff Modelling' chapter in this Collection System User Guide for more details on the LID result items listed above.

CD Items (Catchment discharge)

Catchment discharge consists of person equivalent (PE)-based or area-based inflows from catchments (e.g. wastewater inflows).

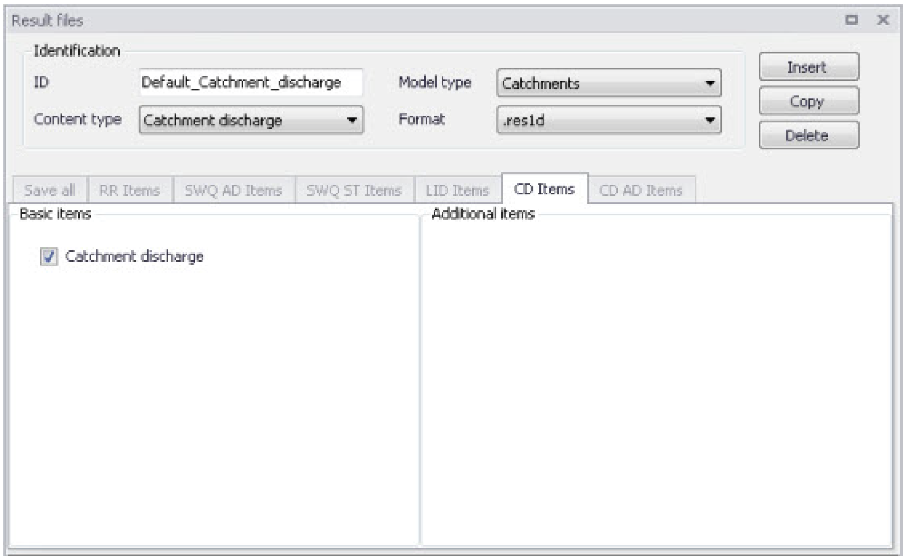


Figure 13.15 The CD Items Tab in the Result Files Editor. The 'Catchment discharge' item is initially included in Default Catchment Discharge results.

CD AD Items (Catchment discharge quality)

These are computational items related to the modelling of pollutant transport with catchment discharge.

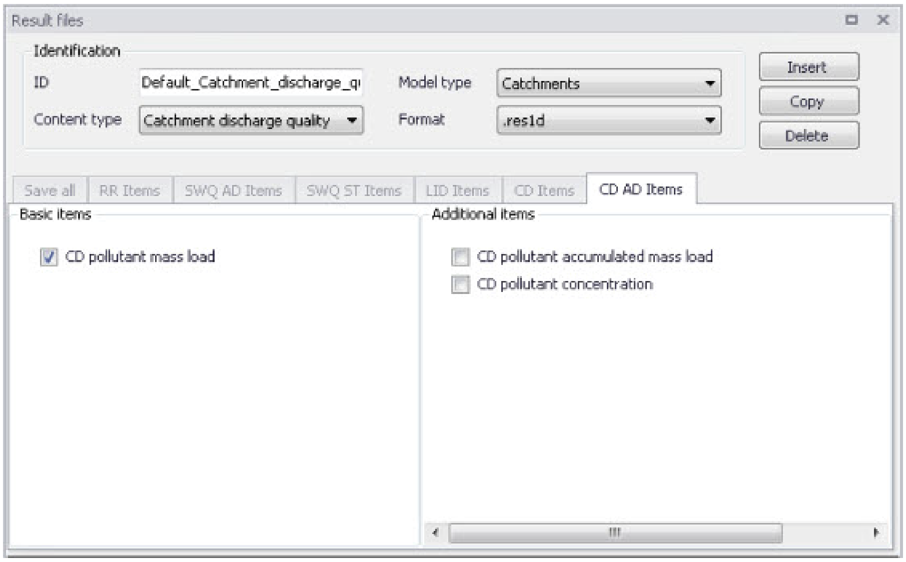


Figure 13.16 The CD AD Items Tab in the Result Files Editor. The 'CD pollutant mass load' item is initially included in Default Catchment Discharge Quality results.

Table 13.8 Overview of Catchment Discharge Quality result items in the CD AD Items Tab

Basic Items	Additional Items
CD pollutant mass load	CD pollutant accumulated mass load CD pollutant concentration

HD Items

These are result items related to hydrodynamic calculations in the network, including items related to control rules.

Some additional items related to control rules (setpoint value, pump start level, pump stop level) are always saved even though they are not listed in this tab, because they are essential in case the result file is to be used as hot-start file.

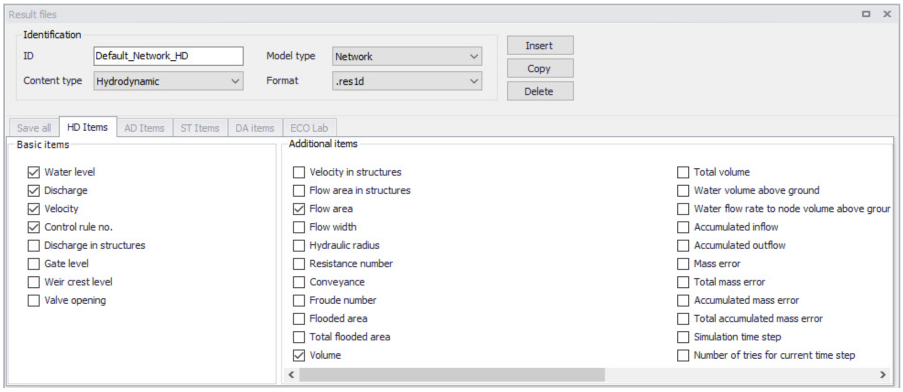


Figure 13.17 The HD Items Tab in the Result Files Editor.

Note: Some result items are not available when saving to .txt file format, and are therefore disabled.

Details on some of the additional items:

- Mass error: this gives the difference between the calculated and geometric (true) volumes. The calculated volume is evaluated using the inflow/outflow of a grid point at each time step. The geometric volume is evaluated by integrating the width of the cross-section (from its processed data table) from the bottom up to the water level and multiplying by the grid point's length. This result item is called 'Mass error, generated water' in the result file.
- Accumulated mass error: this gives the values of 'Mass error' accumulated over time. This result item is called 'Mass error, signed accumulated' in the result file.

AD Items (Pollution transport)

These are result items related to the modelling of the transport of pollutants in the sewer network.

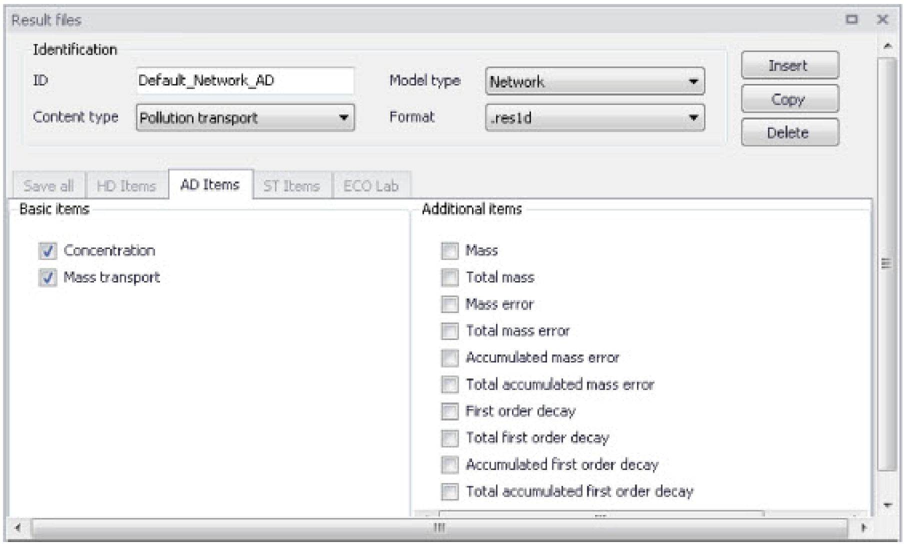


Figure 13.18 The AD Items Tab in the Result Files Editor. The ‘Concentration’ and ‘Mass transport’ items are initially included in Default Pollution Transport results.

Table 13.9 Overview of pollution transport items in the AD Items tab

Basic Items	Additional Items
Concentration Mass transport	Mass Total mass Mass error Total mass error Accumulated mass error Total accumulated mass error First order decay Total first order decay Accumulated first order decay Total accumulated first order decay

Note: if the Decay is set to 0 for a component, the decay-related result items which are not relevant for this component are never saved to the result file. That means that decay-related result items are saved only for WQ components with decay different than 0.

ECO Lab (MIKE ECO Lab water quality)

These are the result items from the MIKE ECO Lab simulation. The 'MIKE ECO Lab template' group on the left shows the list of templates included in the project. Select one of the templates to access the selection of result items for this template.

The 'Basic items' group in the center shows the list of state variables and derived outputs from the selected template.

The 'Additional items' group on the right shows the list of auxiliaries and processes from the selected template.

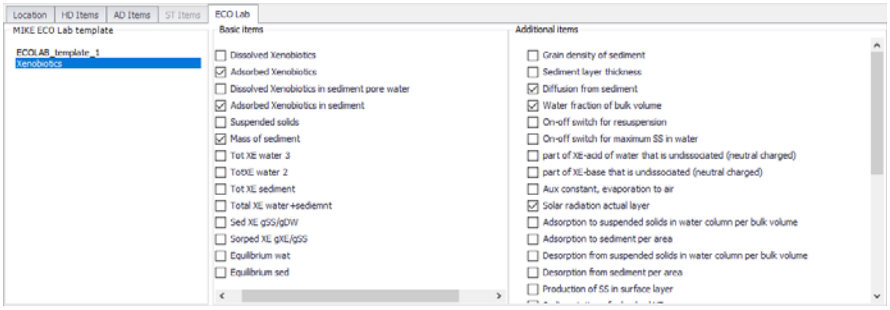


Figure 13.19 The MIKE ECO Lab items in the 'Result files' editor.

Note that these lists depend on the State Variables, Auxiliary Variables, Processes and Derived Outputs defined in the MIKE ECO Lab template used in the simulation (Figure 13.20). This template can only be edited in the MIKE ECO Lab editor, in MIKE Zero.

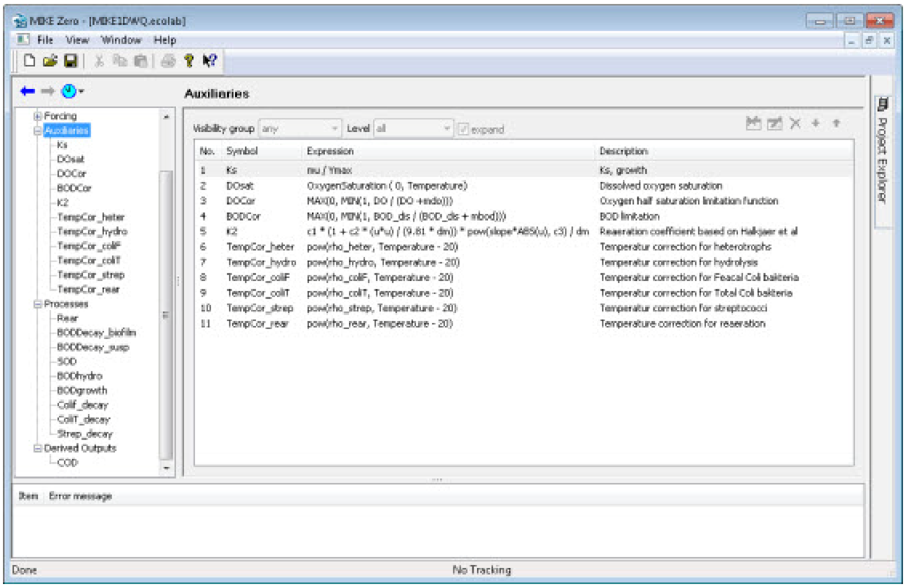


Figure 13.20 Example MIKE ECO Lab template (*.ECOLAB) showing the Auxiliaries, Processes, and Derived Outputs sections

LTS HD Items (LTS extreme and chronological statistics)

These are LTS statistics results related to LTS hydrodynamic modelling in sewer networks.

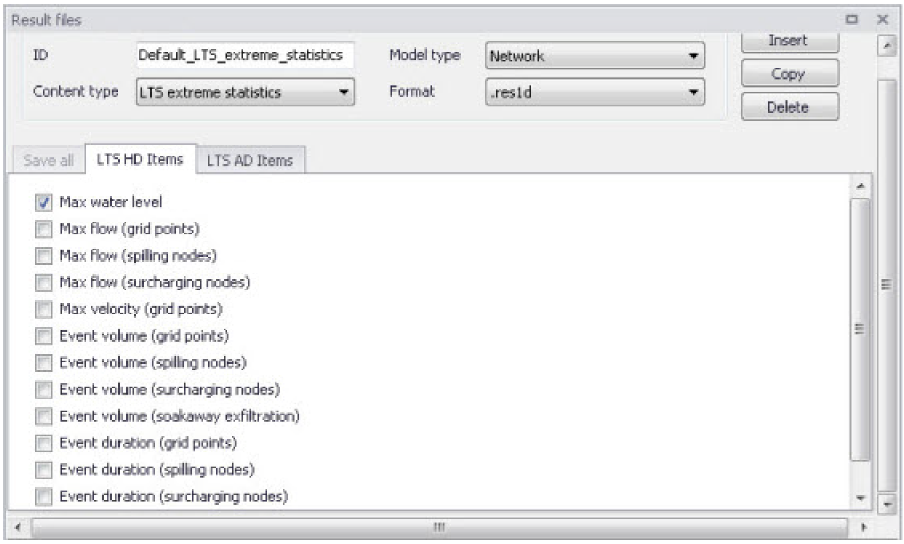


Figure 13.21 The LTS HD Items Tab in the Result Files Editor. The figure shows pre-selected items for Default LTS Extreme Statistics results.

Table 13.10 Overview of items in the LTS HD Items Tab related to LTS Extreme and Chronological Statistics Results

Extreme Statistics	Chronological Statistics
Max water level Max flow (grid points) Max flow (spilling nodes) Max flow (surcharging nodes) Max velocity (grid points) Event volume (grid points) Event volume (spilling nodes) Event volume (surcharging nodes) Event volume (soakaway exfiltration) Event duration (grid points) Event duration (spilling nodes) Event duration (surcharging nodes) Event duration (soakaway exfiltration)	Total accumulated volume (out of the system Total accumulated volume (outlet pipes) Total accumulated volume (pumps out of the system) Total accumulated volume (weirs out of the system) Total accumulated volume (orifices out of the system) Total accumulated volume (valves out of the system) Total accumulated volume (spilling nodes) Accumulated spilled volume (spilling nodes) Accumulated surcharge volume (nodes) Accumulated exfiltration volume (soaka-ways) Accumulated volume (discharge)

LTS AD Items (LTS extreme and chronological statistics)

These are LTS statistics results related to LTS modelling of pollution transport in sewer networks.

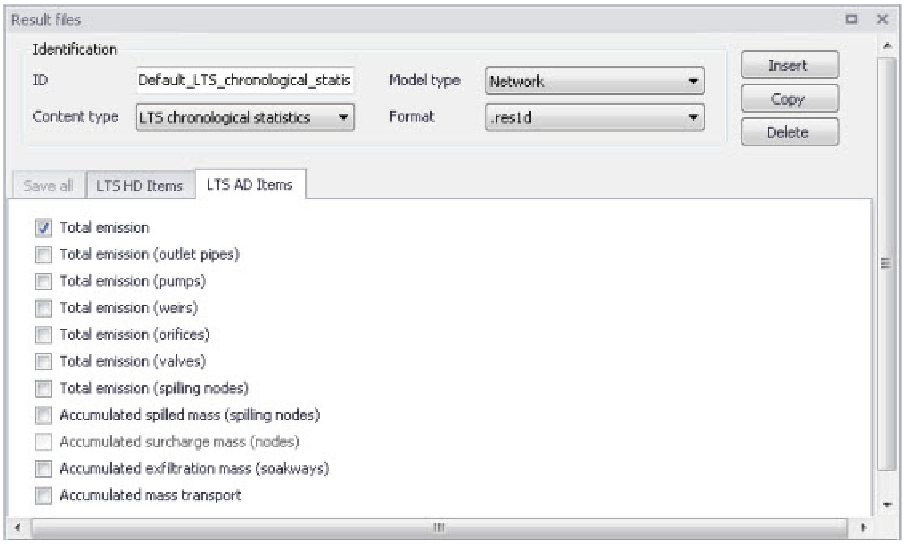


Figure 13.22 The LTS AD Items Tab in the Result Files Editor. The figure shows pre-selected items for Default LTS Chronological Statistics results.

Table 13.11 Overview of items in the LTS AD Items Tab related to LTS Extreme and Chronological Statistics Results

Extreme Statistics	Chronological Statistics
Max concentration Event load (grid points) Event load (spilling nodes) Event load (surcharging nodes) Event load (soakaway exfiltration)	Total emission Total emission (outlet pipes) Total emission (pumps) Total emission (weirs) Total emission (orifices) Total emission (valves) Total emission (spilling nodes) Accumulated spilled mass (spilling nodes) Accumulated surcharge mass (nodes) Accumulated exfiltration mass (soakaways) Accumulated mass transport

DA Items (Data assimilation)

These are the result items related to Data assimilation calculations. Some result items are only applicable with the DA mode 'Updating with Kalman filter'. Some result items are applicable with the DA modes 'Updating with Kalman filter' and 'Uncertainty prediction'. The DA mode 'Updating with weighting function' does not create specific result items, but only affects the regular result items (e.g. water level and discharge), so the 'DA items' tab is not relevant for this mode.

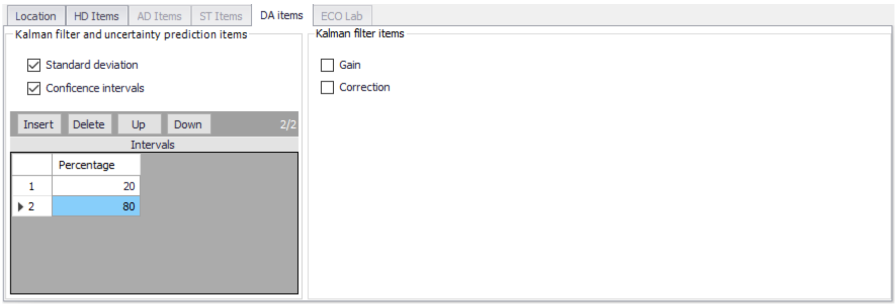


Figure 13.23 The DA Items tab in the Result Files editor.

Confidence interval result item requires specification of a percentage value for each confidence interval. Percentage values are defined in the table located below the 'Confidence intervals' check box.

Note that DA results are always saved to a separate result file, with a suffix (e.g. _HDStat.res1d).

2D map items

This controls the item being mapped in a result file with content type '2D map'. Only one item can be mapped in each file.

Various results items may be mapped:

- 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. This bed level can optionally be defined by a DEM, and is otherwise defined by the cross sections.
- 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.
- DEM: this option generates a digital elevation model (DEM) for the river bed, based on the topography from the cross sections and the river direction.

- Depth and fluxes (h, p, q): this option produces a file containing the three items Water depth, P-flux and Q-flux. This map type is primarily used to display flow vectors (using the fluxes information) on top of the water level or depth.
- WQ component: this option creates a map of the concentration of the selected component.

Please refer to the MIKE 1D reference manual for details about the mapping procedure.

Besides, three types are available:

- 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.

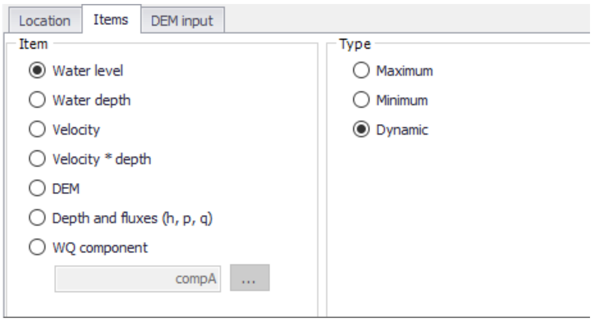


Figure 13.24 Selecting the result item for a 2D map result file

13.1.5 DEM input

An additional option while generating 2D maps of water depth is to include ground level information from external DEM files. In that case, the water levels are mapped to the DEM to compute the water depth, instead of interpolating the ground levels from the cross sections.

To include this option, activate the 'Use input DEM' box.

One or more Digital Elevation Model files must then be supplied in the table. Items are added or removed using the 'Insert' and 'Delete' buttons above the table. The use of multiple DEM files is intended to support multiple DEM tiles to cover the entire area: in that case all the tiles can be supplied in the table, with no need to merge them in a single file. If some files overlap, the topography in the overlapping area will be defined by the first DEM in the list.

Note: The list of DEMs is shared by all 2D map result files i.e., editing the list of DEMs for one 2D map result file will also update any other 2D map result file.

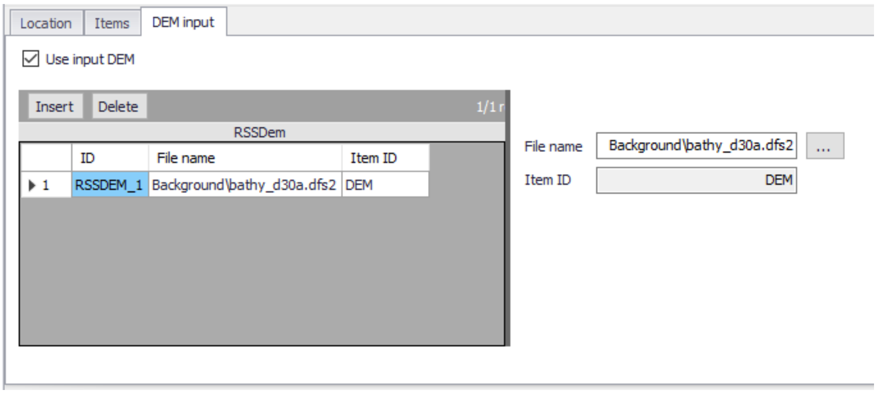


Figure 13.25 The optional list of DEMs for 2D map result files

13.1.6 Combining various result items in one file

MIKE+ offers high flexibility in configuring result files obtained from simulations. The secondary table (Figure 13.26) in the Result Files editor is used to specify different combinations of items and locations for user-specified result setups.

User-specified results (i.e. non-Default) may be configured to contain different result sets with varying combinations of:

- Location
- Items. Items may be from the same Content Type or from across different (active) Content Types.

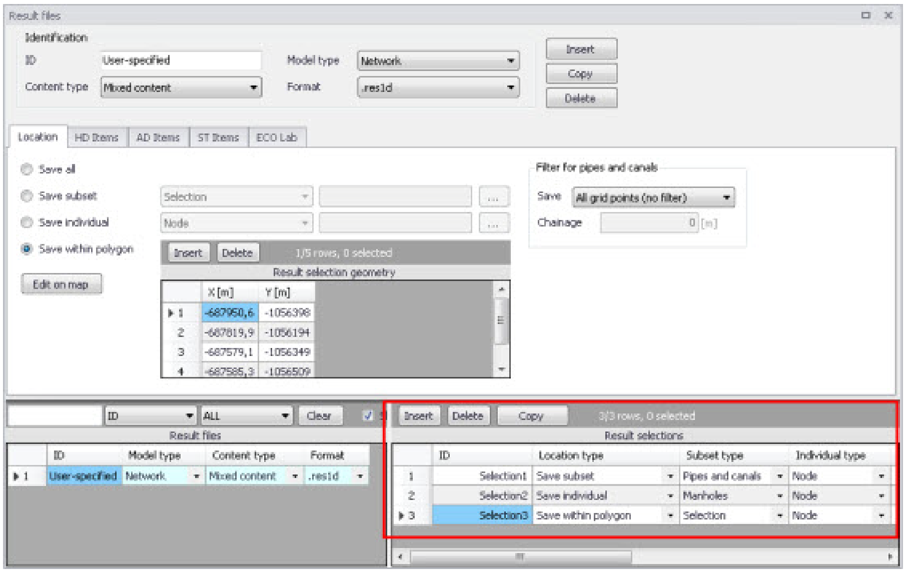


Figure 13.26 The secondary table in the Result Files Editor

The example shown in Figure 13.26 is a 'Network' result file setup that has 'Mixed content' and includes 3 Location-Items combinations in one .RES1D result file. Result items from across content types (i.e. active modules) may also be saved in one file. Use Content Type = 'Mixed content' for the result file setup to allow this option (see example in Figure 13.26).

The Location-Item combinations are is the example are summarised in the table below:

Table 13.12 Example Location-Items combinations from across content types that may be combined in one Mixed Content User-Specified result file

Result	Location	Items
Selection1	Save subset = Pipes and canals	HD Items = Water level, Velocity
Selection2	Save individual = Node 7	AD Items = Concentration
Selection3	Save within polygon	HD Items = Discharge

The content of the secondary table is controlled with the following buttons:

'Insert' button

Creates a new item in the table, with Default properties.

'Delete' button

Deletes the current selected rows from the secondary table.

‘Copy' button

Creates a copy of the active row.

13.2 Network Summary

The basic contents of HTML summaries generated at the end of simulations may be customised in MIKE+. Various Network Summary configurations may be defined through the Network Summary editor (Result Specifications | Network Summary) (see Figure 13.28). These configurations may then be invoked in the Simulations Setup editor when running Network (HD) simulations.

The basic HTML summary shows information on the simulation, overall results summary, and boundary inputs. Adding a network summary configuration will extend the basic summary with the ‘HD Results Summary’ section (Figure 13.27).

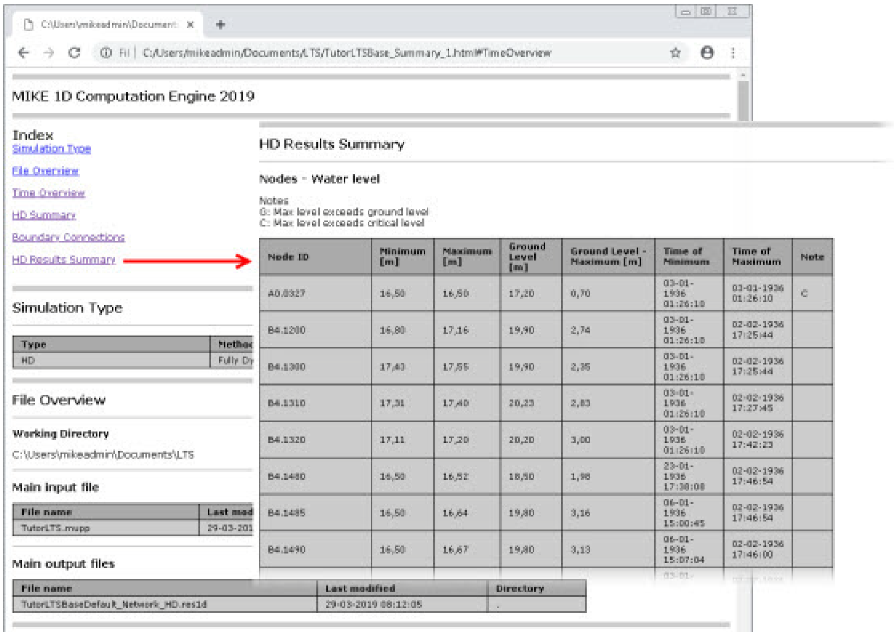


Figure 13.27 Example network simulation HTML summary augmented with an additional HD result summary table for Nodes.

Additional items may be activated for each network summary setup in the editor. The Network Summary editor is organized into the following groups:

- Identification
- Tables with min, max and accumulated values for
- Summary of input data

- Use selection to reduce summary tables

Activate the various options on the editor to customise additional contents for the network simulation HTML summary. An overview of the various options in the editor is shown in Table 13.13.

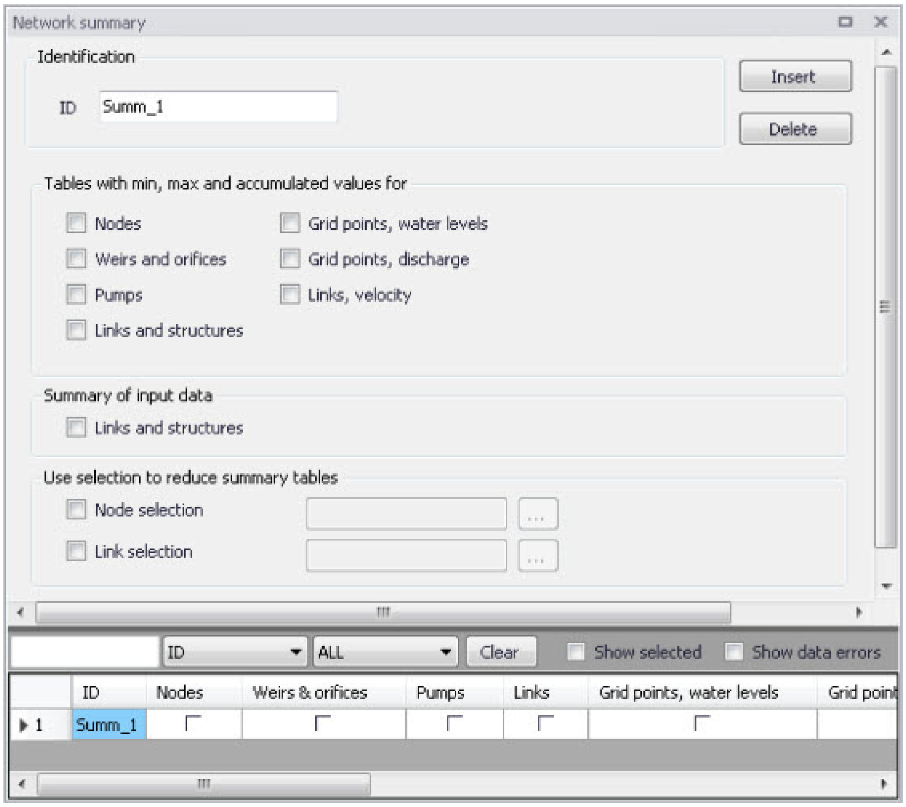


Figure 13.28 The Network Summary Editor

Table 13.13 Overview of the Network Summary editor attributes (Table msm_HtmlSummary)

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	ID of summary- user specified	Yes	MUID
Nodes checkbox	Activate to include tables with min, max and accumulated values in the summary	Optional	SummaryNodeNo

Table 13.13 Overview of the Network Summary editor attributes (Table msm_HtmlSummary)

Edit field	Description	Used or required by simulations	Field name in datastructure
Weirs and offices checkbox	Activate to include tables with min, max and accumulated values in the summary	Optional	SummaryWeirNo
Pumps checkbox	Activate to include tables with min, max and accumulated values in the summary	Optional	SummaryPumpNo
Links and structures checkbox	Activate to include tables with min, max and accumulated values in the summary	Optional	SummaryLinkNo
Grid points, water levels checkbox	Activate to include tables with min, max and accumulated values in the summary	Optional	SummaryLinkLevelNo
Grid points, discharge checkbox	Activate to include tables with min, max and accumulated values in the summary	Optional	SummaryLinkDischargeNo
Links, velocity checkbox	Activate to include tables with min, max and accumulated values in the summary	Optional	SummaryLinkVelocityNo
Summary of input data, Links and structures checkbox	Activate to include a summary of input data in the HTML summary	Optional	SummaryLinkInputNo
Node selection file checkbox	Activate to specify nodes for which to include tables with min, max and accumulated values in the summary	Optional	NodeSelectionNo
Node selection input box	Input box for specifying the Node selection list	Yes If NodeSelectionNo=1	NodeSelectionName

Table 13.13 Overview of the Network Summary editor attributes (Table msm_HtmlSummary)

Edit field	Description	Used or required by simulations	Field name in datastructure
Link selection file checkbox	Activate to specify links for which to include tables with min, max and accumulated values in the summary	Optional	LinkSelectionNo
Link selection input box	Input box for specifying the link selection list	Yes If LinkSelectionNo=1	LinkSelectionName

Use the following button functions on the editor to add or remove network summary setups for the project:

‘Insert’ button

Adds a new Network Summary setup in the editor.

‘Delete’ button

Removes the active/selected Network Summary setup from the editor.

14 Simulation Specifications

MIKE+ simulations are started from the Simulation Specifications section. This section includes the following menus:

- **Simulation Setup**. Where various combinations of different types of simulations may be setup and run.
- **Batch Simulation**. Controls batch simulations involving the automatic sequential launch of several simulation jobs.

Note: Simulations can also be executed without opening the editor, through command lines. Refer to Section 6.4.11 Predefined export from command lines *(p. 191)* for more information.

14.1 Simulation Setup

The Simulation Setup editor has several tabs, which are shown depending on the active features and modules for the project:

- **General.** Includes general parameters, such as definition of the simulation period, selection of simulation types, and free text description of the simulation setup.
- **Catchments**. Includes parameters specific for Runoff simulation.
- **HD**. Includes parameters specific for HD simulation.
- **AD and WQ**. Includes parameters specific for network AD simulation and MIKE ECO Lab.
- **LTS**. Includes parameters specific for Network LTS simulation.
- **Results**. Includes specification of results (output) to be generated by the simulation.

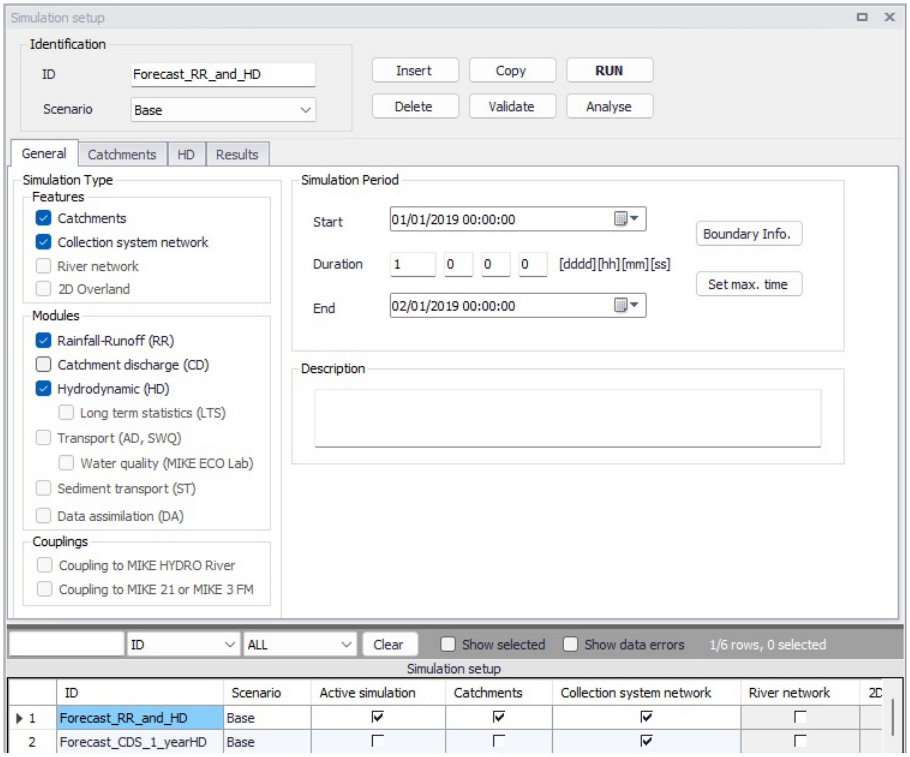


Figure 14.1 The Simulation Setup Editor

The Identification group at the top and the scrollable grid table at the bottom of the editor are common across all tabs.

Table 14.1 Overview of the Simulation Setup Identification data group (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	User-specified ID of simulation. ID will be reflected in the name of result files	Yes	MUID

Table 14.1 Overview of the Simulation Setup Identification data group (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Scenario	Dropdown menu for selecting ID of Scenario for the simulation	Yes	ScenarioName
Active simulation check box	Defines a simulation setup as "in the tray", i.e. as the one chosen among simulation setups to be used when extracting data by external application, e.g. MIKE FLOOD, and for running directly from the Simulation toolbar. Only one job at a time may be set as “Active”.	Yes	ActiveProject

The following buttons are also located at the top of the editor with the Identification group:

‘Insert’ button

Inserts a new record in the Simulation Setup editor with a default ID.

‘Copy’ button

Duplicates an existing (currently active) simulation setup record.

‘Delete’ button

Deletes a currently active simulation record.

‘RUN’ button

Triggers export of the current simulation job and execution of the simulation.

'Validate' button

MIKE+ employs automatic validation, checking for data errors as one navigates through the various editors. However, one may also collectively check the consistency of the MIKE+ project data through the validation of the simulation. Press the 'Validate' button to open the validation window:

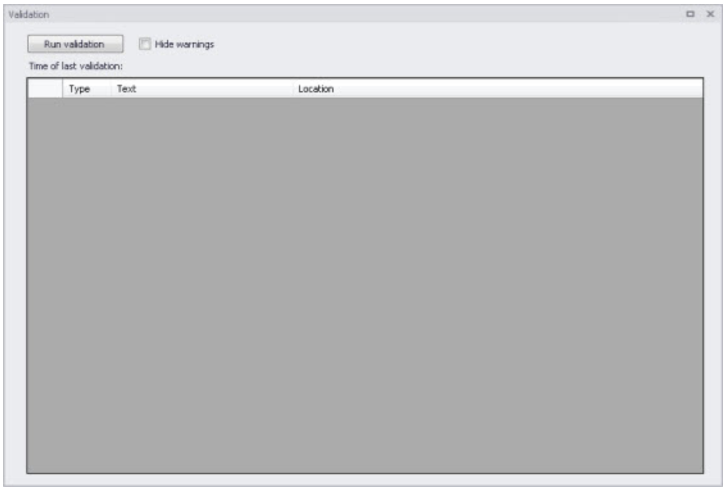


Figure 14.2 The Simulation Validation window

The validation applies to the simulation set as 'Active simulation' in the 'Simulation setup' editor. From this window, press the 'Run validation' button to start the data validation and see the list of warnings and errors in the table. The validation time depends on the database size, number of active modules and number of errors found.

A description of the warning or error is included in the 'Text' column in the list. The description usually indicates the data type or editor in which the error should be corrected.

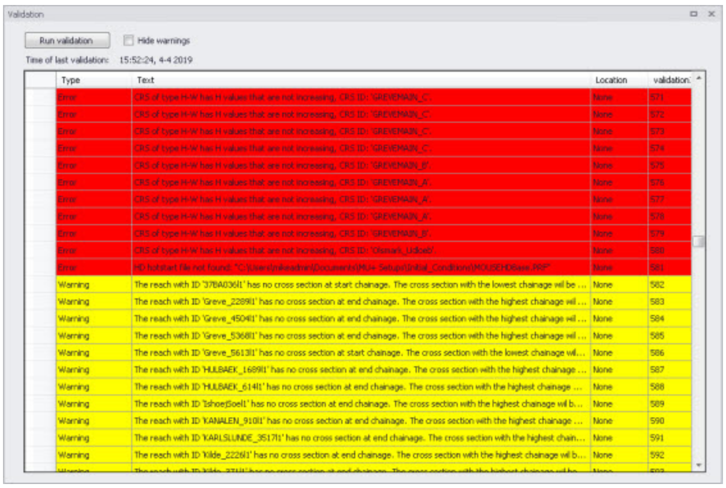


Figure 14.3 The validation list indicates whether an item is an Error (highlighted in red) or a Warning (highlighted in yellow).

It is possible to activate the 'Hide warnings' option to only show errors without warnings in the list.

'Analyse' button

This button opens a tool that can be used to find elements of a river or CS network that may be contributing to instabilities during an HD simulation. The tool produces a list of h-points ranked according to one of the three following criteria, all ranked from smallest to largest:

- Time step required to achieve a specified stability criterion
- Surface area
- Length

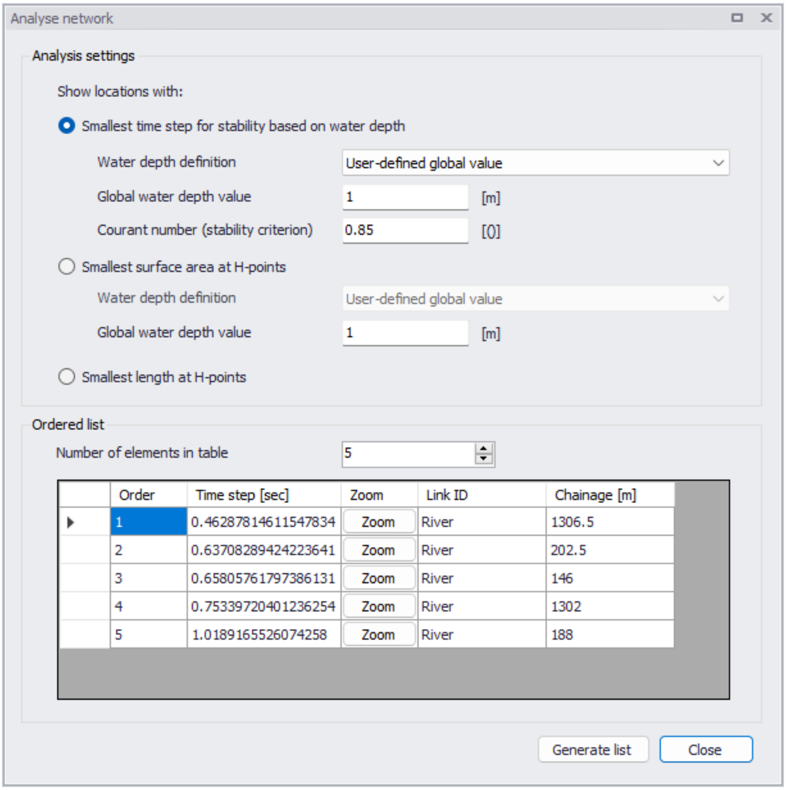


Figure 14.4 The Analyse network tool.

More information about the three options is provided below.

Time step required to achieve a specified stability criterion

When using this option, the tool finds the time step needed to achieve a specified Courant number for each h-point. This option can be used to find an appropriate time step for a stable simulation or identify parts of the model network that should be streamlined to enable a stable simulation at a longer time step.

Because the Courant number evaluation depends on the depth of water, a depth definition must be provided. Four options are available:

- User-defined global value: single value applied throughout network.
- Full-running pipe or river (min. of markers 1 and 3): for rivers, this option uses the minimum of markers 1 and 3 as water level in each cross section. A full-running pipe is used for collection system modelling.
- Full-running pipe or river (min. of markers 1 and 3): for rivers, this option uses the minimum of markers 4 and 5. If markers 4 and 5 are not defined, then the minimum of markers 1 and 3 is used. A full-running pipe is used for collection system modelling.
- Initial conditions: Water depth definitions are taken from the initial conditions specification.

Surface area

Under this option, the tool computes the surface area associated with each h-point, in other words the surface area from the upstream Q-point to the downstream Q-point (or from the calculation point to the nearest upstream or downstream Q-point, if the calculation point is located at the start or end of a river branch).

A depth is needed to compute a surface area. The same four options are available as for 'Time step required to achieve a specified stability criterion'.

For rivers, the area is calculated assuming a constant width equal to the width of the cross-section at the specified depth. For collection system modelling, the area is calculated using a very small width corresponding to a pipe that is almost full. The flow area criterion is not particularly useful for analysis of pipes.

Length

Under this option, the tool computes the length associated with each h-point, in other words the length from the upstream Q-point to the downstream Q-point (or from the calculation point to the nearest upstream or downstream Q-point, if the calculation point is located at the start or end of a river).

14.1.1 General

The General tab includes parameters relevant for the entire simulation setup. The following parameters are specified in the General tab:

- Simulation type
- Simulation period

- Description

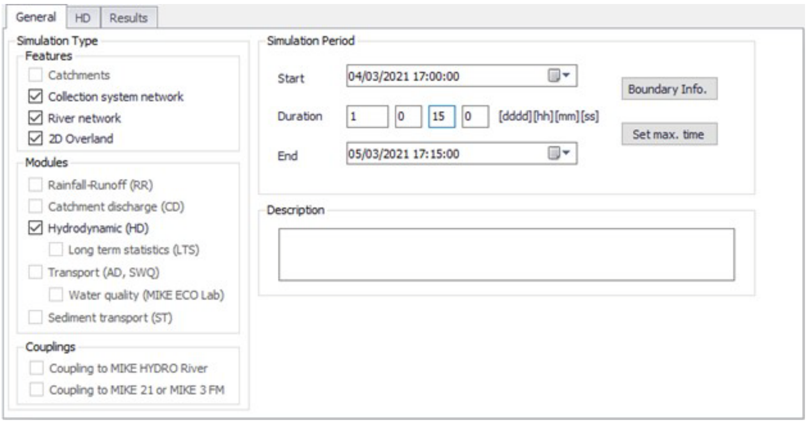


Figure 14.5 The Simulation Setup Editor General tab

An overview of the editor fields and corresponding database attributes is provided in Table 14.2 below.

Table 14.2 Overview of the Simulation Setup General tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Catchments	Activates catchment-related simulation types	Yes	Enable_Catchment
Collection system network	Activates CS related simulation type	Yes	Enable_CS
River network	Activates river related simulation type	Yes	Enable_River
2D Overland	Activates 2D related simulation types	Yes	Enable_2DOverland
Rainfall-Runoff (RR)	Activates runoff simulation	Yes	Enable_RR
Catchment Discharge (CD)	Activates catchment discharge computations	Yes	Enable_CD
Hydrodynamic (HD)	Activates HD simulations for CS, Rivers and/or 2D overland	Yes	Enable_HD

Table 14.2 Overview of the Simulation Setup General tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Long Term Statistics (LTS)	Activates LTS simulation	Yes	Enable_LTS
Transport (AD, SWQ)	Activates AD simulations for CS, Rivers and/or 2D overland and/or SWQ simulations for catchments	Yes	Enable_AD
Water quality (MIKE ECO Lab	Activates MIKE ECO Lab simulations for CS, Rivers and/or 2D overland	Yes	Enable_ECOLAB
Sediment Transport (ST)	Activates CS network sediment transport simulation	Yes	Enable_ST
Data Assimilation (DA)	Activates DA network update simulation	Yes	Enable_DA
Coupling to MIKE HYDRO River	Activates coupling to MIKE HYDRO River. HD coupling is always activated. AD coupling is activated if Transport (AD,SWQ) module is also activated (see note below).	Yes	Enable_MHRiver_Coupling
Coupling to MIKE 21 or MIKE 3 FM	Activates coupling to MIKE 21 FM or MIKE 3 FM. HD coupling is always activated. AD coupling is activated if Transport (AD,SWQ) module is also activated (see note below).	Yes	Enable_M21FM_Coupling
Start	Specifies start date and time for the simulation.	Yes	ComputationBegin

Table 14.2 Overview of the Simulation Setup General tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Duration	Displays the duration of the simulation in days, hours. minutes and seconds. Automatically adjusted based on Start and End time/date. May be edited, adjusting End date/time accordingly.	Yes	-
End	Specifies end date and time for the simulation. Adjusted automatically according to user's specification of duration.	Yes	ComputationEnd
Description	Free text description of the simulation setup	Optional	Description

Note: Coupling of AD components or MIKE ECO Lab state variables from external model setups is also supported under the following conditions:

- When coupling to a MIKE HYDRO River file, and when the 'Transport (AD, SWQ)' module is included in the simulation: all components defined in the 'WQ components' editor in MIKE+ will be coupled to AD components from MIKE HYDRO River having the same name.
- When coupling to a MIKE HYDRO River file, and when the 'Water quality (MIKE ECO Lab)' module is included in the simulation: all components defined in the 'WQ components' editor in MIKE+ will be coupled to state variables from MIKE HYDRO River having the same name.
- When coupling to a MIKE 21 or MIKE 3 FM file, and when the 'Transport (AD, SWQ)' module is included in the simulation: all components defined in the 'WQ components' editor in MIKE+ will be coupled to AD components from MIKE 21 / 3 FM having the same name.
- When coupling to a MIKE 21 or MIKE 3 FM file, and when the 'Water quality (MIKE ECO Lab)' module is included in the simulation: all components defined in the 'WQ components' editor in MIKE+ will be coupled to state variables from MIKE 21 / 3 FM having the same name.

- When coupling to a MIKE 21 or MIKE 3 FM file, and when the 'Transport (AD, SWQ)' module is included in the simulation: if the density in the MIKE 21 or MIKE 3 FM file is defined as a function of the temperature and/or salinity, these built-in temperature / salinity components will be coupled respectively to components called 'Temperature' and 'Salinity' in the 'WQ components' editor in MIKE+, if any.

‘Boundary Info.’ button

The Boundary Info. button opens the Boundary Overview window with a horizontal bar chart showing time extent of all active boundary conditions from all included modules.

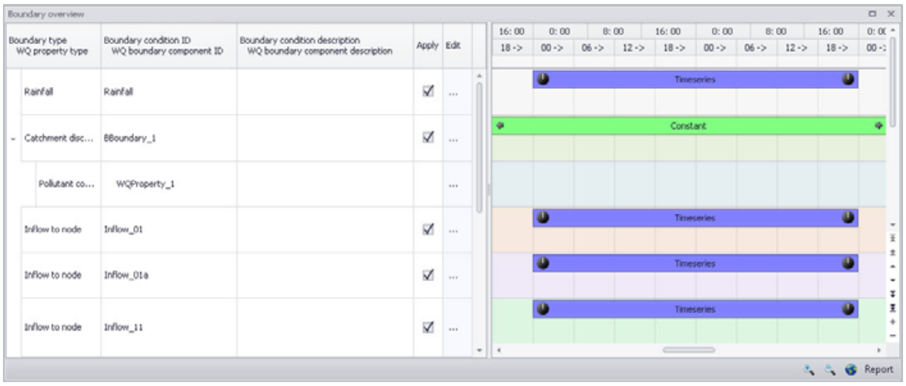


Figure 14.6 The Boundary Overview appears when pressing the ‘Boundary Info.’ button

‘Set max. time’ button

The ‘Set max. time’ button sets the maximum simulation time by filling in the start and end times of the simulation. The start time of a simulation is considered the latest start time of all boundaries. Likewise the end time for the simulation is considered the earliest end time of all boundaries.

Each boundary contains a number of items which can cover different parts of the simulation.

If a limited validity interval is specified for a boundary condition, this specifies the start and end time. If a validity is not specified, only items specified as timeseries have a start and end time. If either a constant or cyclic value is given without validity interval, the item is not included in the evaluation.

14.1.2 Catchments

The following parameters can be specified on the Catchments tab:

- Surface runoff model simulation time step
- RDI simulation time step

- Catchment discharge simulation time step
- Spatial distribution of rainfall
- RDI hotstart specification
- Additional parameters specification

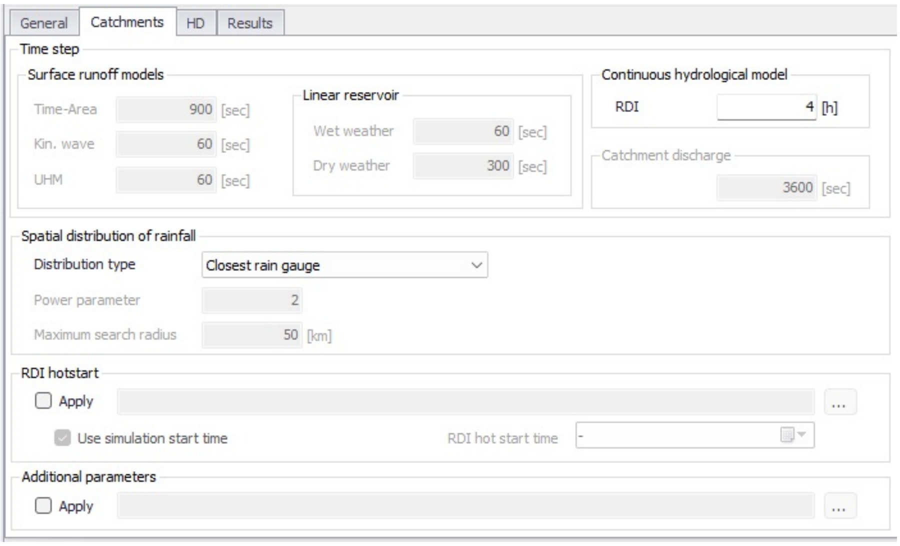


Figure 14.7 The Simulation Setup Catchments Tab

Table 14.3 The Simulation Setup Catchments tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Time-Area	Fixed time step for Time-Area runoff model	Yes if at least one catchment is set for simulation with Time-Area model	TSDt
Kin. Wave	Fixed time step for Kinematic Wave runoff model	Yes if at least one catchment is set for simulation with Kinematic Wave model	KWDt
UHM	Time step for UHM model	Yes if at least one catchment is set for simulation with UHM runoff model	UHMDt

Table 14.3 The Simulation Setup Catchments tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Wet weather	Time step during wet periods	Yes if at least one catchment is set for simulation with Linear Reservoir runoff model	DtWetPeriod
Dry weather	Time step during dry periods	Yes if at least one catchment is set for simulation with Linear Reservoir runoff model	DtDryPeriod
RDI	RDI slow runoff component time step. The fast runoff component time step is the corresponding surface runoff model time step.	Yes if at least one catchment is set for simulation with RDI runoff model	SRCDt
Catchment Discharge	Time step for catchment discharge	Yes if Catchment Discharge simulation active	CDDt

Table 14.3 The Simulation Setup Catchments tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Distribution type	When rainfall boundary conditions are defined with the geographical location of the rain station (boundary type 'Data source location'), it is possible to control how the rainfall intensity from the various rain gauges (boundary conditions) are applied to the different catchments. Three methods are available: assign the closest rainfall time series to each catchment, apply weighting using Thiessen polygons, or apply weighting using an inverse distance interpolation. Weights can be reviewed in the 'Boundary conditions' editor.	Yes, if rainfall boundary conditions are of spatial extent type 'Data source location'	DistribTypeNo
Power parameter	The power coefficient, in the inverse distance interpolation equation.	Yes, when the distribution type is set to 'Inverse distance weighting'.	Power
Maximum search radius	The maximum distance for searching rainfall stations, in the inverse distance interpolation equation.	Yes, when the distribution type is set to 'Inverse distance weighting'.	SearchRadius
RDI Hotstart Apply checkbox	If this checkbox is ticked, a hotstart file for RDIlmust be specified.	Optional	RDIHotStartNo

Table 14.3 The Simulation Setup Catchments tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Use simulation start time	If 'Use simulation start time' option is enabled, the initial condition is automatically extracted from the hotstart file using the simulation start date and time as specified in the 'Simulation Setup' editor.	Optional	RDIHotStart-TimeUseNo
RDI hotstart time	The date and time at which the initial conditions are loaded from the hotstart file. Only active if 'Use simulation start time' option is inactive.	Optional	RDIHotStartTime
Additional Parameters Apply checkbox	If this checkbox is ticked, an *.ADP file for the runoff simulation can be specified	Optional	ADPRunoffFileNo

The additional parameter file (*.ADP file) is a separate file with additional settings for the simulation. Please refer to the seperate documentation on this file for further information.

14.1.3 HD

For a network (CS network and/or river network) simulation, the tab holds parameters specific to the hydrodynamic simulation setup:

- Fixed simulation time step, or
- Adaptive simulation time step settings
- Network initial condition type
- Additional parameters

For a simulation including 2D overland, the time step parameters are changed to:

- A fixed simulation time step, used by the network simulation. This time step is also used to determine the saving frequency of 2D overland result files, and to synchronize the HD and AD modules for the 2D overland simulation.
- Adaptive time step settings, applying only to the 2D overland simulation.

The tab is active if a hydrodynamic (HD) module is activated and if relevant data exist in the project (e.g. if at least 1 conduit is specified).

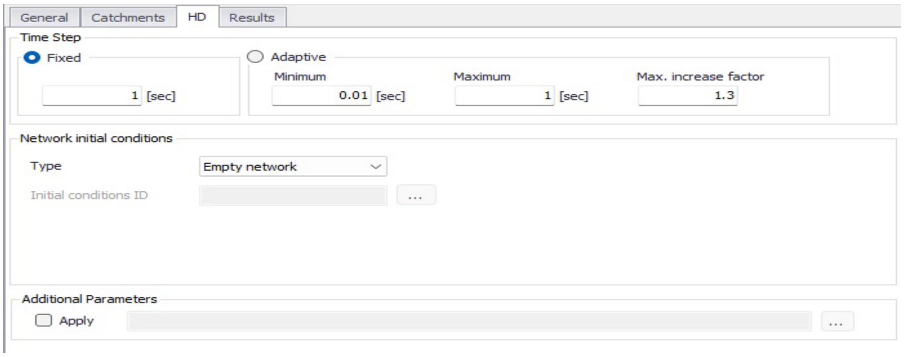


Figure 14.8 HD Tab of the Simulation Setup Editor

For network simulations, the initial condition definition may be of the following types:

- Empty network: with this option, the network will be empty at the first time step of the simulation, except at outlet nodes where water levels from possible boundary conditions will propagate into the network.
- User-specified: with this option, a set of initial conditions must be selected. Sets of initial conditions contain definition of default values, local values, and hotstart files. They are defined in the 'Initial conditions' editor.

- State file: with this option, initial conditions for all modules included in the simulation (Hydrodynamic, Rainfall-runoff, Transport, etc.) are obtained from a state file, which is a detailed result file from a previous simulation. It is therefore required that state files have been saved during a previous simulation (see 13.1.1 Identification *(p. 418)*). For this type of initial condition, it is required to provide a path to the folder containing the state files created from the previous simulation, along with a date and time controlling which time step (i.e. which state file) will be used as initial condition. 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. If 'Select state file using simulation start time' is ticked, the state file with the date and time the closest to the simulation start date will automatically be selected, otherwise the state file with the date and time the closest to the user-defined date and time will be used. A set of 'Initial conditions ID' must also be selected: initial conditions from the state file have the highest priority, but if the network has e.g. been extended and some calculation points do not exist in the state file, the initial conditions in these points will be controlled by the user-defined initial conditions.
- Steady state: with this option, initial conditions will be based on a steady state calculation for all rivers in the model, except for locations where local user-defined values are defined in the optional user-defined 'Initial conditions ID'. Default values and hotstart files defined in the selected 'Initial conditions ID' are ignored. This option is not supported for collection system networks.

Table 14.4 The Simulation Setup HD tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Fixed/Adaptive radio buttons	Toggles between alternative time step type	Yes	HDTimeStepType
Fixed	Specifies a fixed time step for the network simulation	Yes if Fixed time step type or if including 2D overland	HDDtFixed
Minimum	Specifies a minimum time step for network or 2D overland model	Yes if Adaptive time step type or if including 2D overland	HDDtMin
Maximum	Specifies a maximum time step for network or 2D overland model	Yes if Adaptive time step type or if including 2D overland	HDDtMax

Table 14.4 The Simulation Setup HD tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Max. Increase Factor	Specifies maximum increase factor for adaptive time step for network model	Yes if Adaptive time step type	HDDtIncreaseFactor
Max. CFL number	Specifies the expected maximum CFL number in the simulation, to control the adaptive 2D overland time step	Yes if including 2D overland	M2DHDMaxCFL
Network initial conditions type	Specifies the type of initial condition for the network (CS and/or river)	Yes if including CS network or River network simulation	HDInitCondTypeNo
Initial conditions ID	Specifies the ID of the set of initial conditions, defined in the 'Initial conditions' page	Yes except if defined as 'Empty network'	HDInitCondID
State files folder	The path to the folder containing the state files saved during a previous simulation	Yes if 'State file' is chosen	StateFilesFolder
Select state file using simulation start time	If selected, the simulation will use the state file with date and time matching the start time of the simulation.	Yes if 'State file' is chosen	UseSimulationTypeNo
Date and time	The user-defined date and time to select the state file from all state files saved during a previous simulation	Yes if 'Select state file using simulation start time' is not chosen	StateStartTime
Additional Parameters Apply checkbox	Activates *.ADP file with network-relevant input. Define *.ADP file name and path if activated.	Optional	ADPNetworkFileNo

The additional parameter file (*.ADP file) is a separate file with additional settings for the simulation. Please refer to the seperate documentation on this file for further information.

14.1.4 AD and WQ

The "AD and WQ" tab includes parameters specific for the AD and MIKE ECO Lab (WQ) simulation setup.The tab is available if the Transport (AD, SWQ) module is activated in the 'General' tab, otherwise it is hidden.

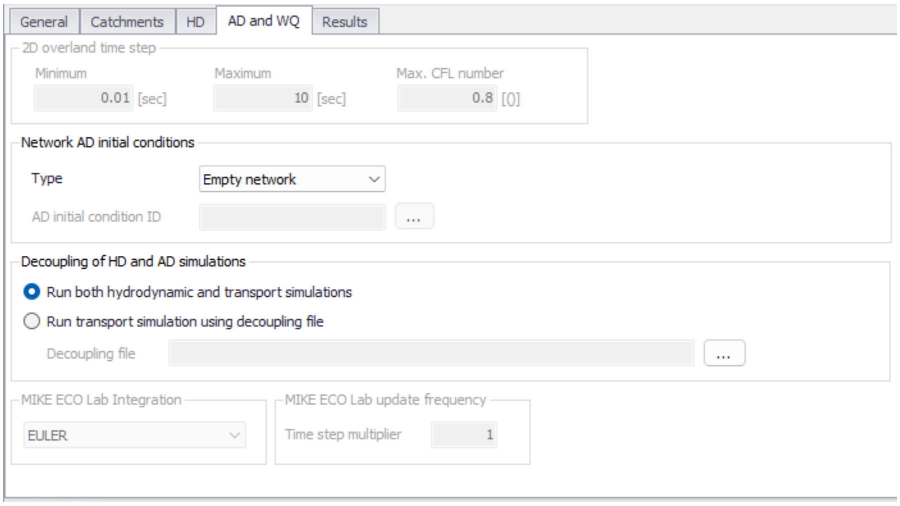


Figure 14.9 The Simulation Setup AD and WQ Tab

Table 14.5 The Simulation Setup AD and WQ tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Minimum	Specifies a minimum time step for 2D overland model	Yes if including 2D overland	M2DADDtMin
Maximum	Specifies a maximum time step for 2D overland model	Yes if including 2D overland	M2DADDtMax
Max. CFL number	Specifies the expected maximum CFL number in the AD simulation, to control the adaptive 2D overland AD time step	Yes if including 2D overland	M2DADMaxCFL

Table 14.5 The Simulation Setup AD and WQ tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Type	Controls whether the simulation start with an empty network (no component) or with predefined initial conditions	Yes	ADInitContTypeNo
AD initial condition ID	The ID of the selected set of Network AD initial conditions	Yes if Type = User-specified	ADInitCondID
Decoupling of HD and AD simulations	Decoupling a transport simulation (AD or WQ) from the HD simulation speeds up the simulation by getting the hydrodynamic conditions from a decoupled result file, instead of running the HD simulation at the same time. This is especially relevant when running multiple transport simulations with identical HD conditions.	Yes	DecouplingADHD-TypeNo
Decoupling file	The decoupled result file to be used as input for the decoupled transport simulation. It is a result file with content type 'Decoupling' which must be saved during a previous simulation.	Yes, if 'Run transport simulation using decoupling file' is chosen	DecouplingPath
MIKE ECO Lab Integration	Specified ECOLab integration method	Yes If Simulation Type = MIKE ECO Lab (WQ)	ELIntegrationNo
Time step multiplier	Multiplier used to scale the MIKE ECO Lab time step as a function of the HD time step	Yes	ELUpdateFre-quency

14.1.5 LTS

The LTS tab includes parameters specific for Network LTS simulations. The tab is shown if the Long Term Statistics (LTS) module is activated in the project and if at least 1 Job List Criterion is specified.

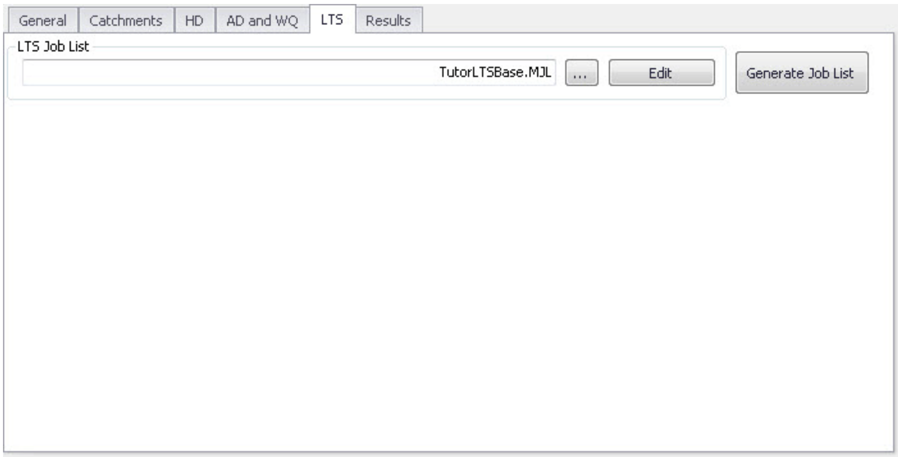


Figure 14.10 The Simulation Setup Editor LTS Tab

Table 14.6 Simulation Setup LTS tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
LTS Job List	Defines the Job List *.MJL file name and path to be used in the LTS simulation	Yes If Simulation Type = Long Term Simulation (LTS)	MJLFileName

The LTS tab also has buttons for the following functions:

'Edit' button

Opens the specified Job List file (*.MJL) in text editor. If the Job List File dialog is empty, or the specified file does not exist or is empty, the 'Edit' button opens an empty ASCII file.

'Generate Job List' button

Starts a Job List generation process. This is a special form of simulation where the output is a Job List file (*.MJL).

More details on Lob List generation and editing are found in the Long Term Statistics (LTS) User Manual.

14.1.6 Results

The Results tab includes parameters for defining output from a simulation setup.

Multiple result files may be specified for each simulation setup.

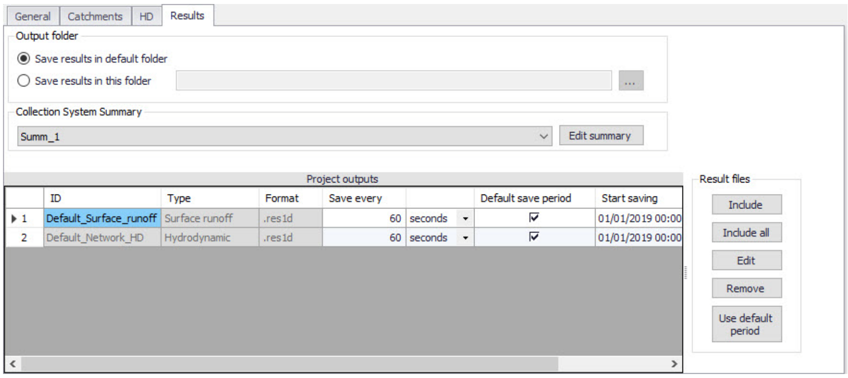


Figure 14.11 Results Tab of the Simulation Setup Editor showing multiple result items

Table 14.7 Overview of Simulation Setup Results tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Save Results in Default Folder/Save Results in this Folder [Output Folder radio buttons]	Toggles between Default folder and user-specified folder for output file location	Yes	HDOutputNo
[Input box beside 'Save Results in this Folder 'option]	Contains the path for user-specified output destination folder	Yes if 'Save Results in this Folder' activated	HDFolderPath

Table 14.7 Overview of Simulation Setup Results tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Collection System Summary dropdown menu	Specifies a MIKE1D simulation summary. User selects from the list of available summaries. Only one network summary per simulation job is possible.	Yes if including a network simulation	SummaryID
Edit Summary button	Opens the Network Summary editor with the current summary in focus. Allows for editing summary contents. If user has specified a non-existing summary ID, program automatically creates a new summary record with default contents and opens the summary editor with the new summary in focus.	Yes	-

A secondary grid in the Simulation Setup Results tab displays a list of the output files selected for the simulation setup (Figure 14.12). The grid retrieves the information from the Result Files editor (Result Specifications| Result Files).

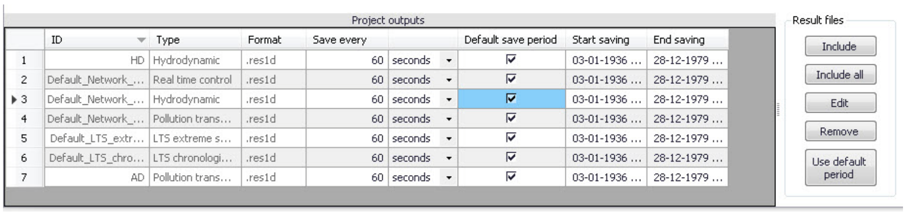


Figure 14.12 The Project Outputs secondary grid in the Results Tab

The list of outputs is controlled by the user using functional buttons to the right of the grid.

The user selects output definitions to include from among those available in the project associated with the modules included in the simulation (and specified in the Result Files editor). The options are filtered according to the contents of the actual simulation.

The list will include at least one "Default" result file definition for each module (Runoff, Network HD, etc.) containing the most usual results for the entire model domain (e.g. runoff for all catchments, discharges and water levels for all model elements, etc.). The list of result file definitions may be extended by "user-specified" output definitions.

An overview of the attributes of the Project Outputs secondary grid is shown in Table 14.8 below.

Table 14.8 Overview of the Project Outputs secondary grid attributes (Table msm_ProjectOutput)

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	MUID of the selected output file definition	Yes	OutputID
Type	Shows type and default contents of the output file (read only)	Yes	ContentsTypeNo
Format	Shows the file format (read only)	Yes	FormatNo
Save Every	Specifies results saving frequency	Yes	DtSave
[Column to the right of ‘Save Every’ column]	Specified unit for result saving frequency	Yes	DtSaveUnitNo
Default Save Period checkbox	Specify to save results for the entire simulation (check), or for user-specified period only (uncheck)	Yes	DefaultSavePeriodNo
Start Saving	Defines start date and time for saving results	Yes if user-specified save period	SaveStartDate
End Saving	Defines end date and time for saving results	if user-specified save period	SaveEndDate

The following functional buttons are available for controlling the list of outputs in the Project Outputs secondary grid:

'Include' button

Opens a list of all relevant output definitions for the simulation and allows the user to choose those which are to be added to the list.

'Include all' button

Fills the list with all relevant pre-defined output definitions found in the database. Relevance is determined by the modules included in the simulation. If the list is not empty, it only adds those outputs which are not already in the list.

'Edit' button

Opens the Result Files editor with the current output in focus.

'Remove' button

Removes selected output(s) from the list.

'Use Default Period' button

This function finds the full period available for an output item. The information is reflected in the Start Saving and End Saving secondary grid attributes. This tool may be used in defining user-specified saving periods to ensure they fall within valid periods.

14.2 Batch Simulation

If you need to run more simulations sequentially, you can choose to do so by including these to a batch simulation. This is done through the Batch Simulation editor.

The Batch Simulation editor includes functionalities allowing control and execution of batch simulations.

The 'Batch Run' button executes all simulations that have the 'Include to batch' flag set in the sequence that they are specified in the grid table. This means that multiple simulations and scenarios can be simulated in batch without user interaction.

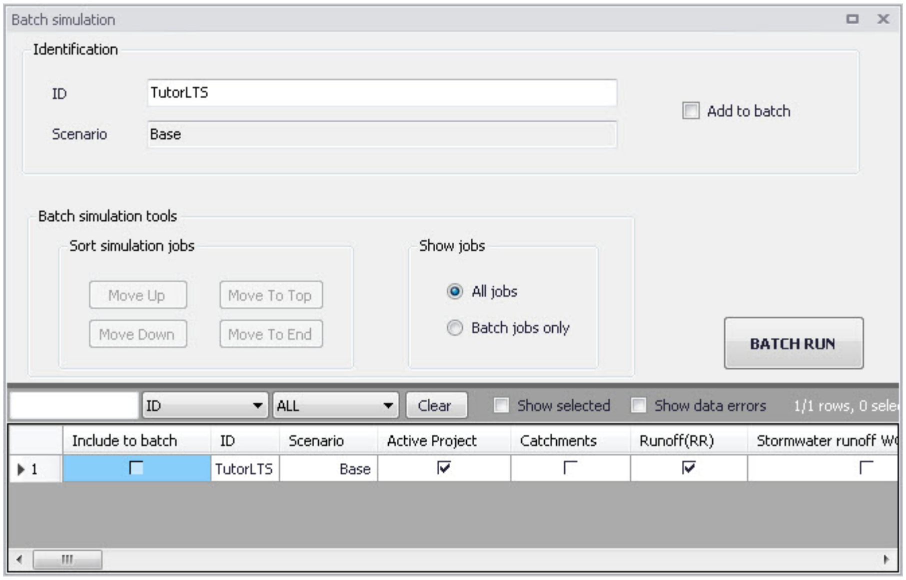


Figure 14.13 The Batch Simulation Editor

The Batch Simulation editor manages the same data from the Simulation Setup editor. The grid table shows the same entries as the grid in the Simulation Setup editor, but built-in tools allow reordering and filtering of simulation job records for batch execution.

Table 14.9 Overview of Batch Simulation editor fields (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	ID of the simulation setup	Yes	MUID
Scenario	Scenario for the simulation setup	Yes	ScenarioName
Add to Batch checkbox	Option for including a simulation setup to batch	Yes	IncludeToBatchNo

The following functionalities are available on the editor:

Move Up

Moves the active record one position up in the grid.

Move Down

Moves the active record one position down in the grid.

Move To Top

Moves the active record to the top of the table.

Move To End

Moves the active record to the bottom of the table.

‘All jobs’ and ‘Batch jobs only’ radio buttons

This filters the list of simulation jobs displayed in the table. A complete list of simulation jobs (i.e. All jobs) is shown by Default, but the display can be reduced to show only those jobs included in the batch (i.e. Batch jobs only).

‘Batch Run’ button

This starts a batch job execution following the sequence of the simulation jobs on the list. Each consecutive job must wait until the previous job has been fully completed. All user prompts are suppressed during the batch job execution, i.e. the simulations are automatically executed without user prompts.

15 Pump Emergency Storage Estimation

The 'Pump emergency storage estimation' simulations allow estimating the time and volume available prior to an overflow at a wastewater pump station asset in the event of system failure.

Sewer overflows from sewage treatment systems are considered critical. In the event of a failure at the pumping station, sufficient storage time and volume is required by the operations staff to respond and address the cause of failure, before an overflow actually occurs. Pump station failure may occur for a number of reasons including failure of power supply, switchboards, number of pumps or failure of the pump station pipework or rising main. Emergency storage is considered to include both wet well storage and storage in upstream gravity sewers.

This special analysis helps knowing the time available before overflow will occur to the environment, e.g. in order to develop a management and operations staff action plan.

For new pump stations, a minimum of 4 hours storage at Average Dry Weather Flow conditions shall usually be provided between Alarm Level and actual overflow level. Systems draining to waters deemed sensitive normally require more storage, e.g. up to 8 hours storage.

Options to be considered for emergency storage for new pump stations, in order of preference, may include:

1. Use of larger diameter wet wells
2. Online storage in larger diameter gravity sewers
3. Use of larger diameter manholes.

Offline storage structures shall not be used for new pump stations.

Where an existing pump station is to be upgraded (including the construction of a new pump station wet well to replace part or all of an existing pump station), the amount of storage required will be the same as for new stations.

Options to be considered for designing emergency storage for existing pump stations, in order of preference, may include:

1. Use of larger diameter wet wells
2. Inline storage in larger diameter gravity sewers
3. Use of larger diameter manholes
4. Offline gravity sewers

5. Offline storage tanks.

15.1 Alarm levels

Alarm levels are water levels in the pumps' wet wells or at any other relevant node, that trigger the alarm. A 'Pump emergency storage estimation' simulation uses a single set of alarms, but this set may comprise any number of alarms in individual nodes.

Identification

Each alarm set must be defined by a unique ID.

An optional description may be provided to describe the alarm set.

Local alarm levels

The table in the 'Local alarm levels' group contains the list of nodes where an alarm is defined and the corresponding alarm level, for the active set. The node ID and its alarm level may be edited either in the secondary table or in the 'Alarm settings' group.

New alarms can be added to the table using the 'Insert' button above the table. Alarm levels are independent from the 'Critical levels' defined in the nodes, but frequently can have the same notion. Therefore, the nodes' critical levels (if any) are used to define a default alarm level when inserting new nodes:

- If a critical level is defined for the selected node, the alarm level is set equal to the critical level
- If no critical level is defined for the selected node, the alarm level is set equal to its ground level minus the specified freeboard value. Note that, if weirs discharging out of the modelled network (no 'To node' defined) are connected to the node, then the alarm level is set equal to the weir crest level minus the specified freeboard.

In any case, it is possible to change the default value for any customised alarm level afterwards.

For each alarm, the 'Allow updating level' controls whether the specified alarm level can be updated by the various tools or if it should be kept unchanged. When 'Allow updating level' is selected, it is possible to reset the node's alarm level to the node's critical level.

Note: the specified freeboard is only used to compute default alarm levels when adding new nodes to the table. The freeboard value is not used as such in the simulation.

Note: sealed nodes cannot be selected to define alarms.

The table can also be populated with a list if alarms with the two following buttons:

- Insert pump critical levels: This populates the table with all pumps' wet well nodes (or only the selected ones, if a selection is active). Each inserted node is given a default alarm level calculated with the same principles as when inserting a single node, and this default value can be modified afterwards. If a pump node is already listed in the table, its alarm level is updated only if 'Allow updating level' is selected, and remains unchanged otherwise. This button is especially relevant when pumps' critical levels have been assigned beforehand, using the tool 'Set pumps critical level' .
- Insert all critical levels: this populates the table with all nodes that have critical levels defined in the Nodes editor (or only the selected ones, if a selection is active). The alarm level is here always set equal to the critical level. If a node is already listed in the table, its alarm level is updated only if 'Allow updating level' is selected, and remains unchanged otherwise.

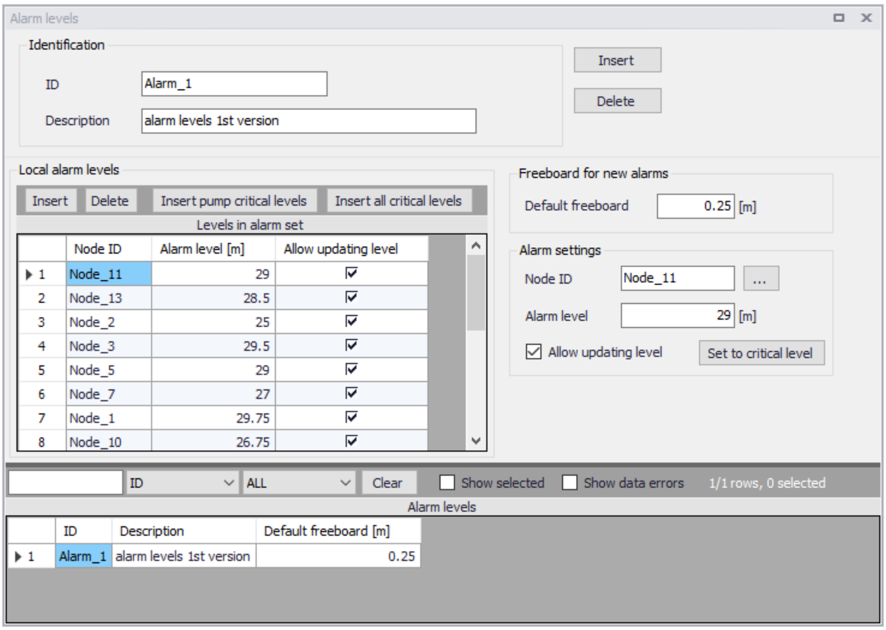


Figure 15.1 Defining alarm levels

15.2 Emergency storage estimation

15.2.1 Identification

General properties for the Pump emergency storage estimation simulations are specified under the Identification group:

- ID: Unique identifier of the simulation.
- Connections: Option description of the simulation.
- Simulation ID: The selected simulation to which the pump emergency storage simulation is attached. The simulation should be created beforehand in the 'Simulation setup' editor, and can be selected using the '…'.
- Scenario: The scenario which is run by the selected simulation ID. It is here only informative, and can only be edited from the 'Simulation setup' editor.
- Start: The date and time of the start of the selected simulation. It is here only informative, and can only be edited from the 'Simulation setup' editor.
- End: The date and time of the end of the selected simulation. It is here only informative, and can only be edited from the 'Simulation setup' editor.

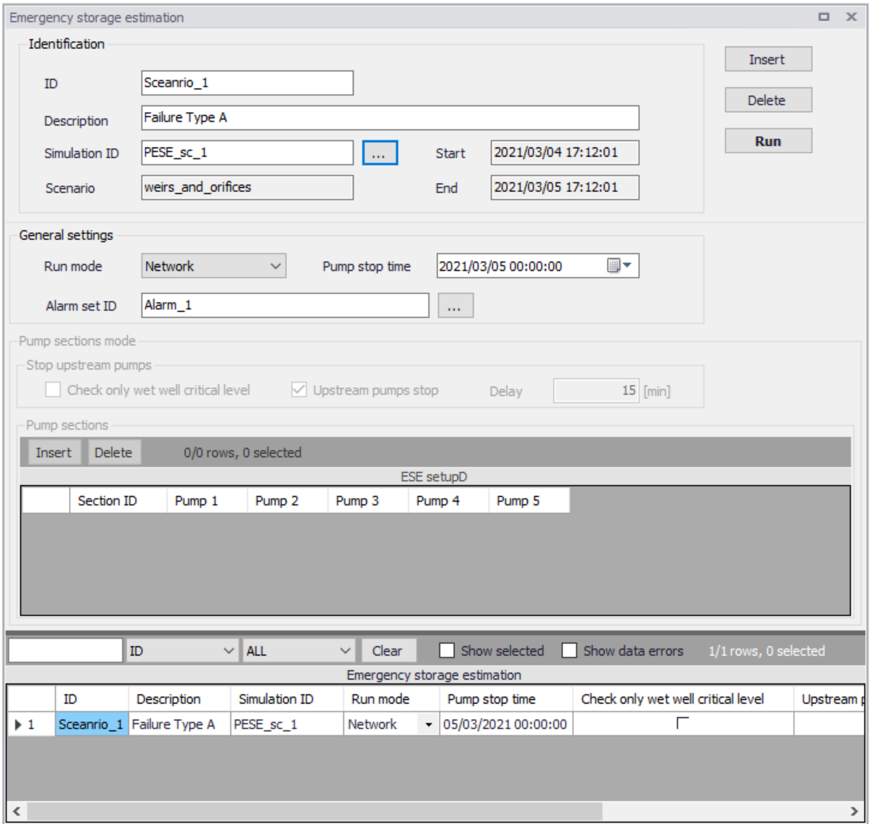


Figure 15.2 Configuring the emergency storage simulation

15.2.2 General settings

Pump emergency storage estimation simulations are defined by the main options below.

- Run mode: two modes are available
  - Network
  - Pump sections
- Pump stop time: time at which the pumps are stopped (capacity set to zero).
- Alarm set ID: The alarm set being used in the simulation is selected using the '…' button amongst the list of sets defined in the 'Alarm levels' editor.

Network

In 'Network' mode, the pump emergency storage estimation analysis is run in two automatic and consecutive simulations.

During the first simulation:

- The simulation runs under normal conditions, until the 'Pump stop time' is reached. At that time, all the pumps are stopped
- The simulation continues with the stopped pumps until alarm levels at all pumps have been exceeded, or if the simulation end time is reached
- Results of the network simulation are written to a network result file with suffix "Run1".

During the second simulation:

- The simulation starts under normal conditions as for the first simulation
- The pumped volume is recorded during the period from 'Pump stop time' until the time for reaching the specified alarm level, as simulated in the first run. This is done individually for each pump
- The simulation continues until the specified end of simulation time or until the latest time for alarm exceedance has been reached
- Results of the network simulation are written to a network result file with suffix "Run2".

Pump sections

In 'Pump sections' mode, the pump emergency storage estimation analysis is run in two automatic and consecutive simulations for each pump section. The two simulations are repeated as many times as there are pump sections specified.

The steps in the first simulation are:

- Run the simulation normally until the 'Pump stop time' is reached
- At this time, the following is done to the model for all pump stations listed in the current pump section:
  - The pump is stopped (capacity set to 0)
  - The first pump station in the list is identified as the "main" pump station for this analysis
- With these changes, the simulation continues until the alarm level is reached:
  - If the option 'Check wet well alarm level only' is selected, the alarm level check is performed only at the wet well node for the "main" pump station (the first in the list)
  - If the option 'Check wet well alarm level only' is not selected, the alarm level check is done for all pump stations in the list.
- The simulation stops once the alarm level is reached at one of the monitored points

- Results of the network simulation are written to a network result file with suffix "_PumpSectionID_Run1"

During the second simulation:

- The simulation starts under normal conditions until the user specified "Time to stop pumps" is reached
- From this time onward, the simulation continues with the identified "main" pump station active (first pump in the list). For the remaining pumps in the list there are two options available:
  – If the 'Upstream pumps stop' option is not selected, all pump stations in the list are active. This is a "normal" simulation
  – If the 'Upstream pumps stop' option is selected, all pump stations in the list except the "main" (first) one are stopped. The pumps are stopped with the specified 'Delay'
- The simulation continues until the time for reaching the alarm level is reached, as identified during the first simulation
- Results of the network simulation are written to a network result file with suffix "_PumpSectionID_Run2"

15.2.3 Pump sections mode

Options available in the 'Pump sections mode' group are only used when the run mode is set to 'Pump sections'. It comprises the following controls:

- Check only wet well critical level: Controls the alarm level check is performed only at the wet well node for the "main" pump station (the first in the list), or for all pumps in the section.
- Upstream pumps stop: Controls whether the other pumps than the main (first one) remain active or if they are stopped.
- Delay: The delay for stopping all pumps except the main one, when the 'Upstream pumps stop' option is selected.
- Pump sections table: Each row in the table adds a pumps section in the simulation. For each pumps section, it is possible to include up to five pumps in the section's list.

INDEX