MIKE+

Water Distribution

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

Access the Model type and Description editors for Water Distribution modelling under the General Settings section.

The ‘Model type’ dialogue provides an ‘at a glance’ view of which MIKE+ elements are available, and if they are activated or not. The list of available modules is:

- Water quality
- Fire flow analysis
- Network vulnerability
- Cost analysis
- Shutdown planning
- Flushing analysis
- Water hammer analysis
- Online analysis
- Optimization
- Multi-species analysis
- Autocalibration

These modules indicate which type of analysis will be modelled within the current project setup. For example, if fire flow analysis is required, hence it needs to be ‘checked’. When the module is checked, it becomes visible on the Setup tree in the left panel and can be applied to the modelled applications.

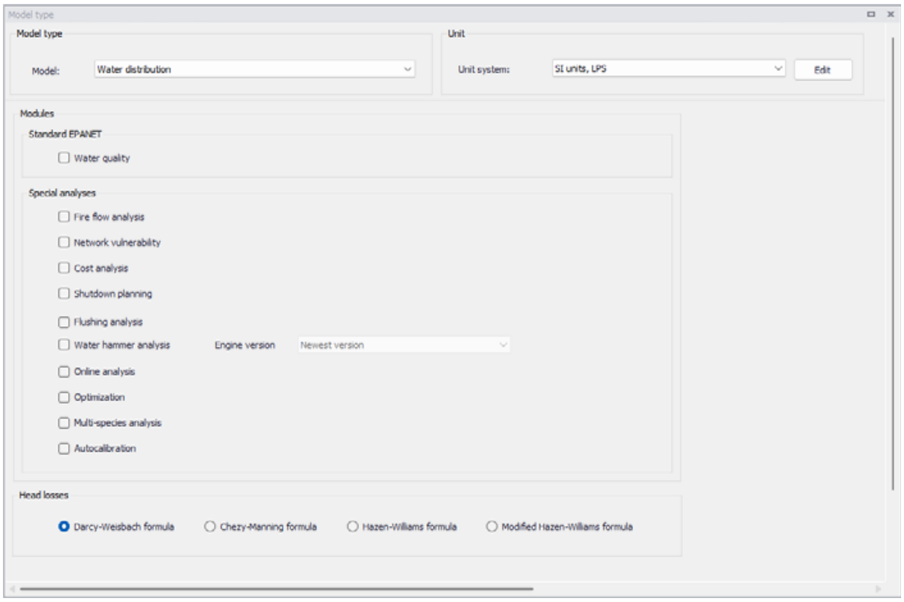


Figure 1.1 General Settings

Note that two versions of the simulation engine are available for water hammer analyses. The appropriate version must be selected in the selection list next to the 'Water hammer analysis' check box. The preferred choice should be to use the newest engine version, which offers various improvements. The former version remains available for special cases where e.g. slight differences are observed in the results from the two engines and in case it is important to maintain the same results as in former simulations.

The 'Head losses' selection controls which method is used to calculate the head losses as a function of flow rate in a pipe. It is related to the roughness coefficients in the pipe editor, where each formula has its corresponding roughness coefficient.

It has four choices:

- Darcy-Weisbach formula
- Chezy-Manning formula
- Hazen-Williams formula
- Modified Hazen-Williams formula.

Please refer to the section ‘Selecting an Appropriate Unit Environment’ in the Model Manager User guide to select units used in the project.

In addition to the ‘Model type’ dialogue, ‘General Settings’ contains a ‘Description’ editor. This editor allows addition of information about the project and a free text description for the model. It may also be used as a model build log to make notes on updates and model amendments.

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.

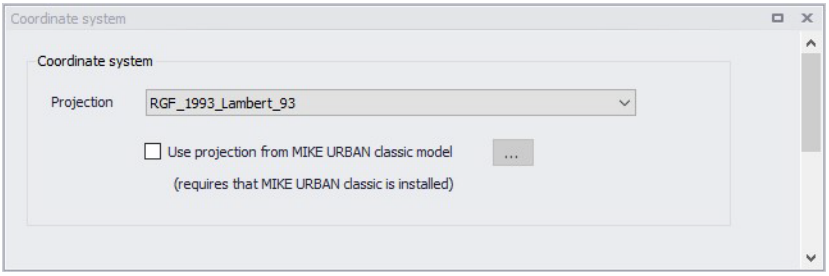


Figure 2.1 The Coordinate System dialog showing information on the projection system used in the project

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 bottom 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).

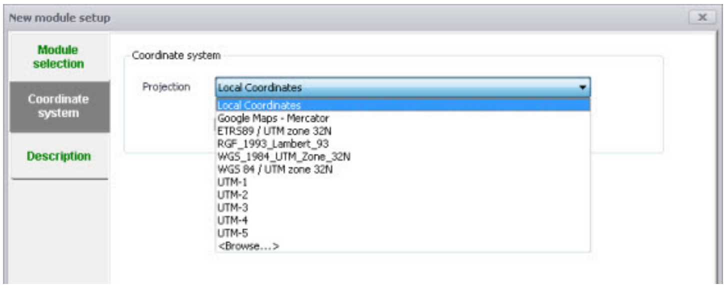


Figure 2.2 Specify the projection system for a new MIKE+ project on the New Module Setup window

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.

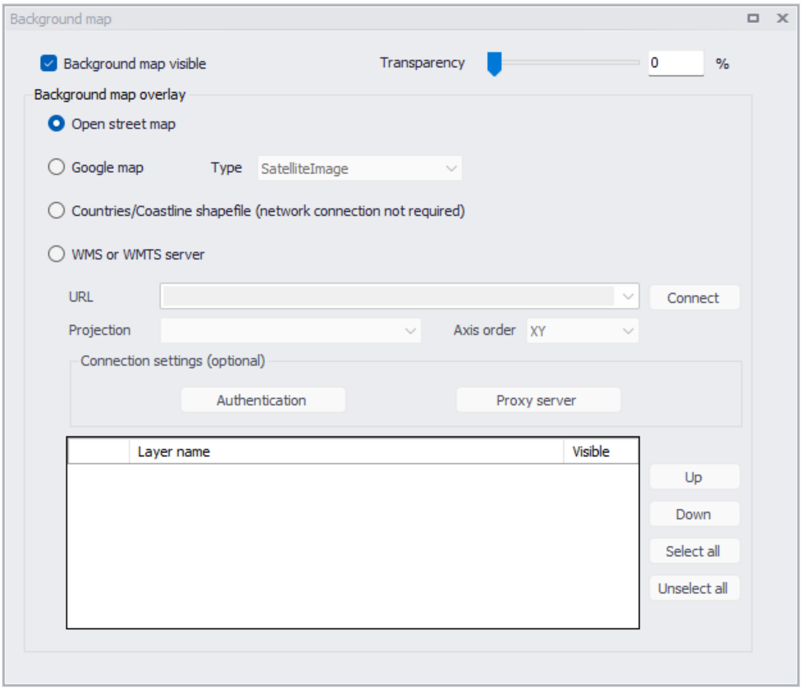


Figure 2.3 The Background Map Editor

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 or WMTS 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 or WMTS 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 server, and the one used for the model data in MIKE+ will be selected if possible. Note that displaying layers from a WMS or WMTS server requires that the MIKE+ project uses the same map projection as the WMS or WMTS layers: if the projection used in MIKE+ doesn't match any of the projections supported by the server, you will be asked to update the map projection in MIKE+. Also note that it is only possible to connect to 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 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 / WMTS server overlays (Figure 2.4).

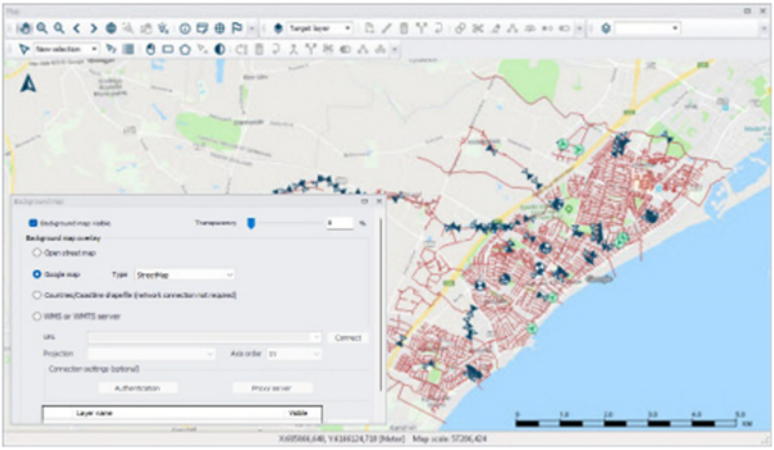


Figure 2.4 An example Google Map background on the Map View in MIKE+

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 or WMTS servers, in case of connection errors, it is possible to save the connection information to a log file. To enable this, start by opening the file "%appdata%\DHI\MIKE+\20xx\DHI.MIKEPlus.Shell.Settings.ison, and set its option 'SaveWMSLog' 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" for WMS connections and in the folder "%appdata%\DHI\MIKE+\BackgroundMapCache\WmtsRasterLayer" for WMTS connections.

3 Network Elements

3.1 Junctions

A crucial element of the water distribution network is the junction nodes, that define the interconnection between the pipes that make up the network. Junction nodes are also placed at points of water consumption or inflow, at points where specific analysis values (e.g., pressure, concentration, etc.) are desired, and at any points where pipe attributes (e.g. diameter, roughness, etc.) change.

Junction nodes are either defined graphically in the Map window using the Drawing tool in the Edit tab with Junctions selected as the Layer to edit (see Figure 3.1), or by manual data entry using the Junction Editor dialog box.

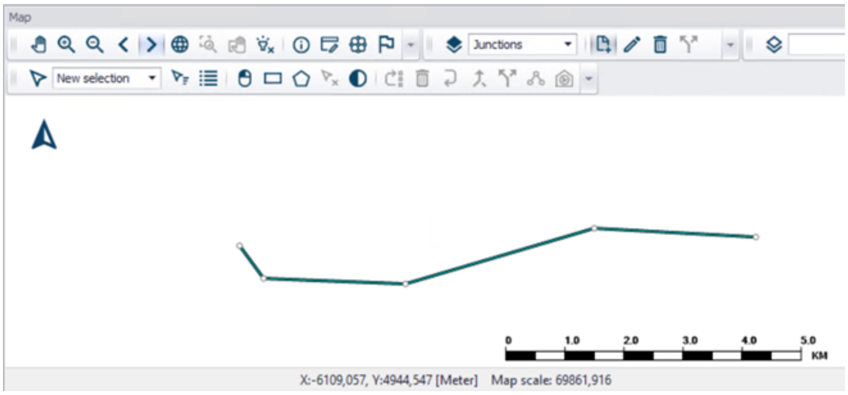


Figure 3.1 Draw Junctions

The Junction Editor allows you to define the junction's ID, location, any external demand, initial water quality conditions and a description. The Junction Editor dialog box is reached by expanding Network Elements and selecting Junctions.

Geometry

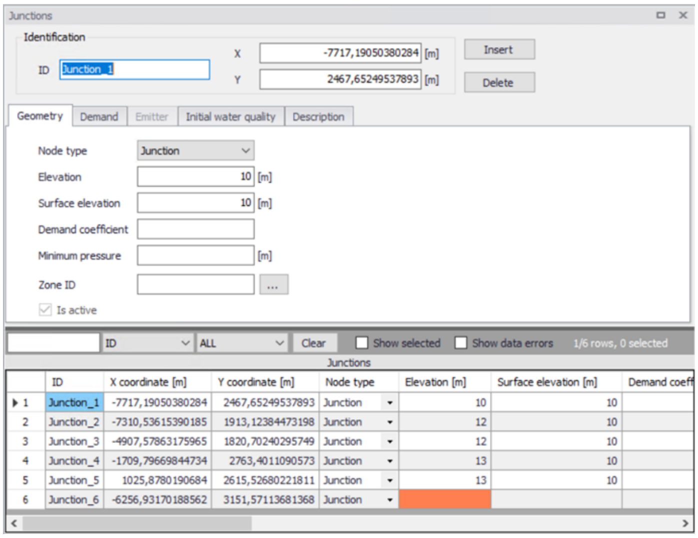


Figure 3.2 Junction Editor, Geometry tab

<Insert> will create a new Junction. <Delete> will remove the selected Junction.

ID

This data entry is used to specify an ID which uniquely identifies the junction node. The junction ID acts as a unique look up key that identifies the node from all other nodes. A node can be a junction, reservoir, or tank. Therefore, no two nodes may have the same ID. However, a node and a link (i.e., pipe, pump, or valve) can have the same ID. The node ID value can be any string value (up to 40 characters).

A new junction ID is automatically suggested by MIKE+ whenever a new junction node is placed into the list by pressing <Insert> or when defining the junction nodes graphically on the Map window using the Add Junction tool.

Coordinates

The X and Y data entries are used to define the physical (map) location of the junction node. When defining the junction nodes graphically on the Map window using the Draw tool, the X, Y location is automatically entered.

Node type

Two types of Junctions are available:

- Junction
- Emitter

Junction is used to describe normal water junctions. An emitter can be used to describe a pressure dependent discharge at the node and is described in the chapter below.

Elevation

This data entry defines the elevation above a common datum for the junction node. This value is used to determine the difference in pressure and pressure head at the node during a simulation. The default elevation is zero. Junction nodes should have their elevation specified so that pressure computations can be carried out.

Surface elevation

This data entry defines the surface elevation above a common datum for the junction node, in units of ft. or m. This value is only used to display the surface elevation in the Longitudinal Profile Plot.

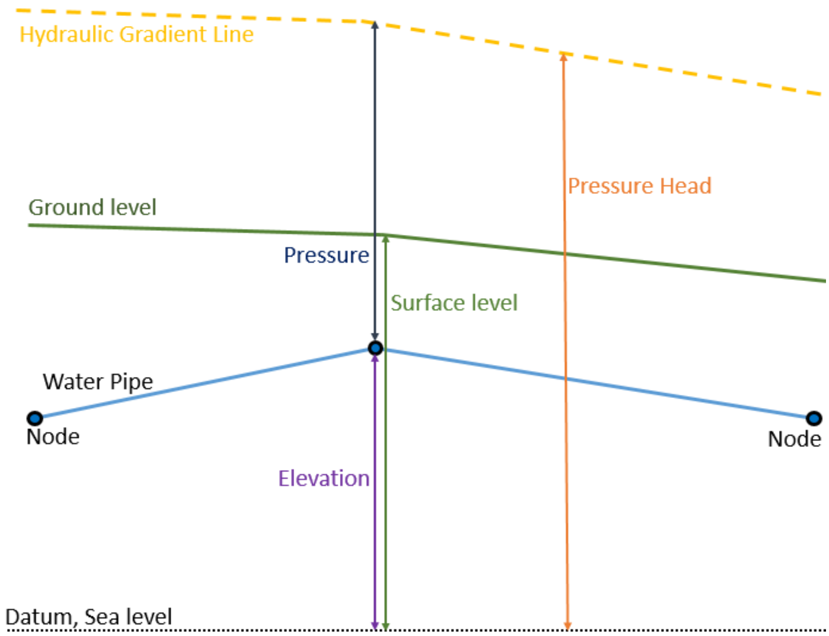


Figure 3.3 The difference in Elevation and Surface level

Minimum pressure

This data entry defines the estate height above the junction node elevation. This data entry is used to calculate Tap Pressure at the junction node and is used to verify the minimal pressure at the node.

Demand coefficient

Demand coefficient allows you to define the share from the whole network demand, which is taken by the node. This field is used only by the Demand Distribution function.

The demand distributed to a node is calculated as

$$qi = \frac{Qt}{Ct} \cdot ci \tag{3.1}$$

where:

qi = node demand
Qt = total network demand
Ct = sum of all demand coefficients
ci = node demand coefficient

Any node where the demand coefficient is not defined will get no demand from the total network demand.

Zone ID

This is an optional name for the zone to which the junction belongs. When a zone ID is specified, this zone will be listed in the 'Zones' editor. The '…' button can be used to select an existing zone.

Is active

This check box controls whether the junction will be included (when ticked) or omitted (when unticked) in the simulations. The junction is automatically omitted as soon as all connected links are also set to inactive.

Demand

The Demand tab is used to view, add or edit demands for a specified Junction. Note that all Demands in the model are stored and can be edited in the Water Demand | Multiple Demand table.

The listed demands in this tab are the items in Multiple Demands with the current Junction as “JunctionID”. The list of demands is updated if another junction is selected in the lower grid.

Junctions may have zero or any number of demands assigned to them. It is also possible to assign separate patterns to the demands assigned to a given junction.

The demand is specified as a constant. If flow is leaving the network system at this junction node, then a positive value should be specified. If an inflow into the network system occurs at this junction node then a negative value should be specified.

The amount of water leaving (or entering) the model in a specific timestep in an extended period simulation will be the junction demand value multiplied by a factor. These factor are stored in time series called patterns and assigned with a Demand pattern ID, see Tables > Patterns.

A demand for a larger part of the system can also be computed by globally defining the demand for the entire network (or a selected part of it) and then having MIKE+ distribute this demand to each of the network nodes using the Distributed Demand dialogue box. See Tools | Distributed Demand.

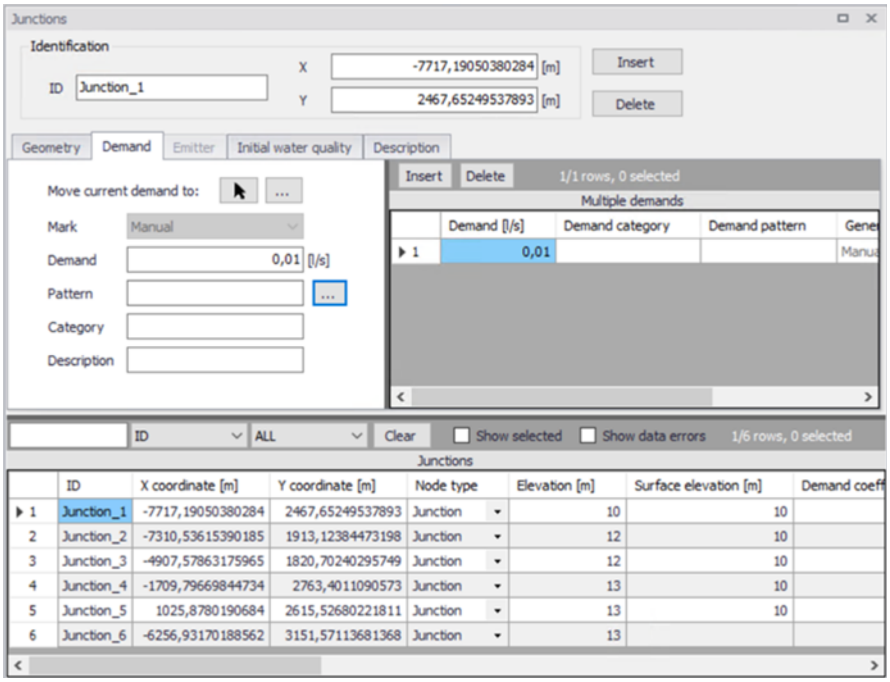


Figure 3.4 Junction editor, Demand tab

A new demand is created by clicking Insert by the list of demands in the right window.

The demand editor in the left window shows the properties of the selected demand.

Demand

The demand, specified as the flow leaving (or entering if the value is negative) in this junction.

Demand Pattern

This data entry allows you to define the ID of the demand pattern to be applied to the junction node demand values during an extended period simulation. The factor in this demand pattern will be multiplied to the defined Demand.

Demand Category

This data entry allows you to enter a description identifying the demand being entered. The demand category can be used when using the Distributed Demand tool.

Description

This data entry allows you to enter a description identifying the demand being entered.

Move Current Demand to

This allows the user to move a single Demand from the active Junction to another Junction. Either by selection from a list of Junction ID or by selecting a Junction in the map. The moved Demand will be removed from this Junction and placed at the new Junction.

Mark

Each Demand is given a Mark based on how it was created.

- Manual
- Distributed Demand
- Demand Allocation

Demands created in the Junction Editor are marked “Manual”.

Emitter

This tab contains parameters of an emitter located at the junction node. A junction is treated as an emitter if the Node Type is set to Emitter in the Geometry tab. Emitters are needed to model flow through sprinkler systems and irrigation networks. They can also be used to simulate leakage in a pipe connected to the junction if a discharge coefficient for the leading crack or joint can be estimated.

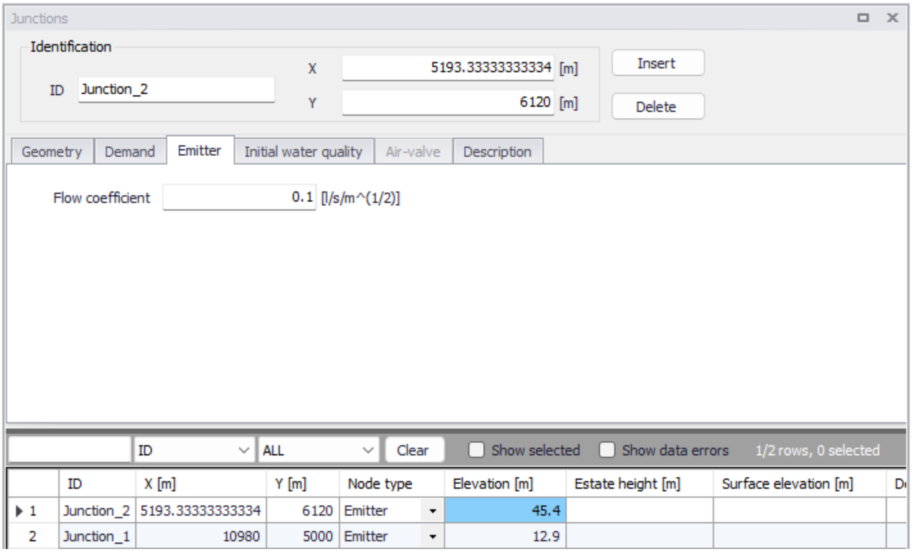


Figure 3.5 Emitter tab

Flow Coefficient

This data entry allows you to define the flow coefficient of the emitter. Flow out of the emitter equals the product of the flow coefficient and the junction pressure raised to a power.

Initial Water Quality

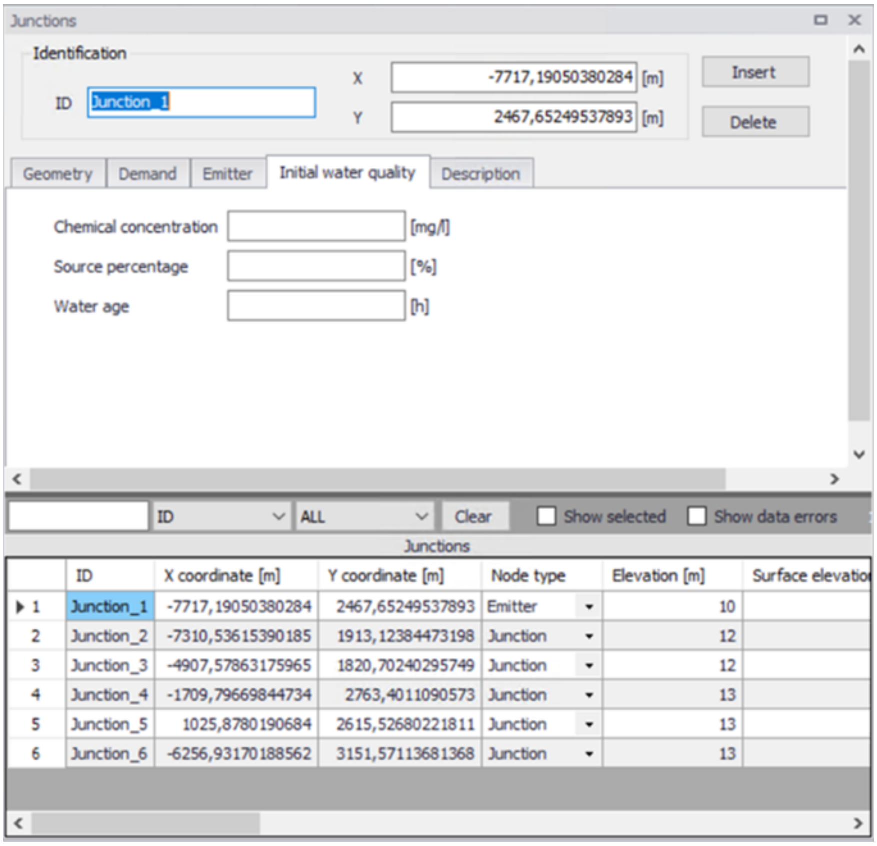


Figure 3.6 Junction editor, Initial water quality tab

The initial water quality at the start of a simulation can be assigned to individual nodes or to groups of nodes. The initial water quality can represent one of the following, depending on the type of water quality simulation.

Concentration

Initial concentration for chemical constituents in a Chemical propagation analysis.

Percentage

Initial percentage of water originating at a specified source node for Source tracing simulation.

Hour

Initial age for Water age determination.

These Initial water quality values will only be used when a Water Quality simulation of the corresponding type is started.

By default, all nodes are assigned with an initial water quality of zero.

Description

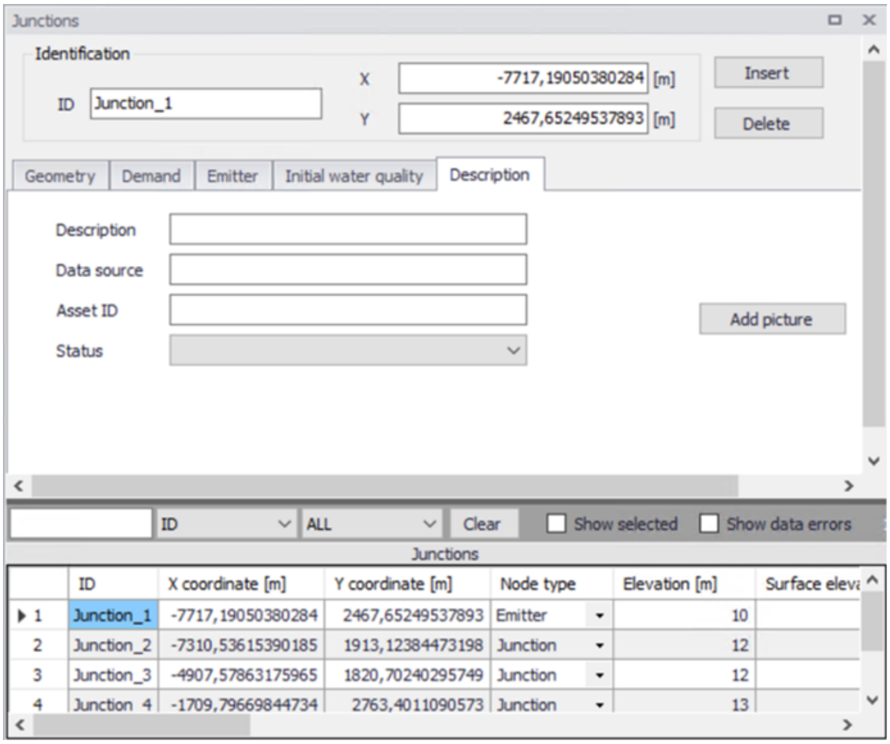


Figure 3.7 Junction editor, Description tab

Description

This data entry allows you to enter a description for the selected junction.

Add picture

The <Add picture> button allows users to add photo for individual pump. Once loaded from external source, the picture will be displayed on this tab.

Data source

This data entry is used to specify a corresponding asset data source, which identifies the Junction (such as database table or a database file name) in the asset management system.

Status

This drop down selection list data entry allows you to define whether the Junction is imported (i.e existing node was imported from the external data source), or is inserted, modified, GIS, calibrated or similar. By default, the status is undefined.

Asset ID

This data entry is used to specify a corresponding asset ID, which uniquely identifies the junction node in the asset management system (such as GIS, for example).

Attributes

Table 3.1 Junction attributes

Field	Database name	Description	Mandatory?	Default value
ID	MUID	Identifier, must be unique for all node types including Tanks etc	Yes	Labels are generated in sequential order
Node type	TypeNo	Type of node	Yes	Junction
Elevation	Elev	Elevation from datum	Yes	0
Surface elevation	Z	Surface elevation from datum at this position	No	0
Demand coefficient	DemCoeff	Coefficient for calculation of Distributed demand	No	0
Minimum pressure	MinPre	Estate hight over node elevation.	No	
Flow coefficient	Em_FlowCoeff	Flow coefficient of emitter	Yes, if Node type = Emitter	
Chemical concentration	Init_Quality_Concentration	Initial concentration for Chemical concentration simulation	No	0
Source percentage	Init_Quality_Percentage	Initial percentage from specified source in Source tracing simulation	No	0
Water age	Init_Quality_Hour	Initial age in water age simulation	No	0
Description	Description	Descriptive text	No	
Data source	DataSource	Source of data	No	
Asset ID	AssetName	ID in asset source	No	
Status	Element_S	Status or origin of data	No	

3.2 Pipes

Pipes are used to transport water from one node to another. Pipes must always begin and end at a node.

Pipes are either defined interactively on the Map window using the 'Drawing' tool on the Edit tab with Pipes selected as the Layer to edit, or by manual data entry using the Pipe Editor dialog box.

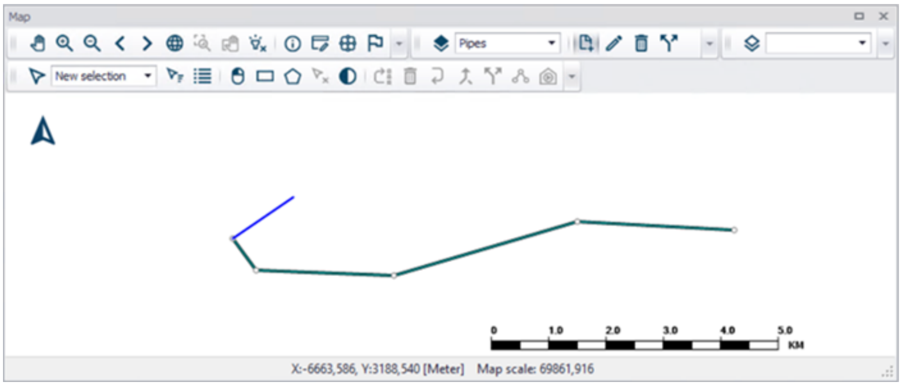


Figure 3.8 Pipes displayed in map

Geometry

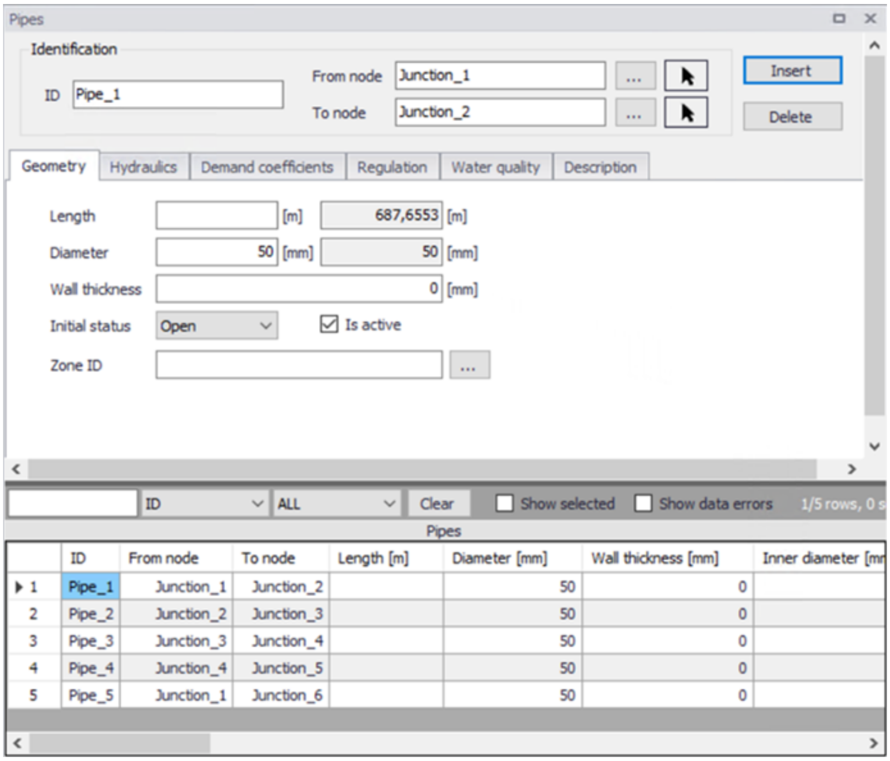


Figure 3.9 Pipe Geometry Editor

<Insert> will create a new Pipe. <Delete> will remove the selected Pipe.

ID

This data entry is used to specify an ID which uniquely identifies the pipe in the datebase. The pipe ID acts as a unique look up key that identifies this link from all other links. A link can be a pipe, valve, pump or turbine. Therefore, no two links may have the same ID. However, a node and a link (i.e., junction or reservoir) can have the same ID. The pipe ID value can be any string value (up to 40 characters).

A new pipe ID is automatically suggested by MIKE+ whenever a new pipe is placed into the list by pressing «Insert» or when defining the pipe graphically in the Map window.

From Node, To Node

These data entries define the ID of the pipe's starting (upstream) and ending (downstream) nodes. These IDs define the pipe connectivity of the network.

Choosing "..." will display the Select Node dialog box from which the user can select the appropriate node. The Node Type pull-down selection list allows the user to specify what type of node is connected to the end of the pipe. Choosing the arrow allows the user to graphically select the node from the Map window.

The order matters since the sign of the computed flow is moving from the starting node to the ending node, the computed flow value will be positive. If the computed flow is moving from the ending node to the starting node, the computed flow value will be negative.

Length

This data entry defines the pipe length, in the unit of your choice. The second (greyed out) field shows the length based upon the pipe layout. It is also possible to define a specific pipe length, independent of the pipe network layout that will be used if specified.

Diameter

This data entry defines the internal diameter of the pipe, in the unit of your choice. The second field (read-only) displays the pipe diameter as it would be used for the hydraulic analysis. The pipe diameter is automatically adjusted when the pipe wall is defined.

Wall thickness

This field is used to define the wall thickness of a pipe. The pipe diameter is automatically adjusted by the program when the pipe thickness is defined.

Initial Status

This drop down list allows the user to toggle the OPEN and CLOSED status of the pipe. Choosing CLOSED effectively removes the pipe from the network system. This is also where the user can define the presence of a check valve (CV) in the pipe. If a check valve exists, then water is only allowed to flow from the starting to ending node. This is commonly used to prevent a flow reversal through the pipe. If conditions exist for flow reversal, the valve shuts and the pipe carries no flow.

Note that you cannot set the pipe status of a pipe containing a check valve using regulation. Pipes with a check valve are initially open, and close only if flow within the pipe attempts to reverse (move from the ending downstream node to the starting upstream node).

Is active

This check box allows the user to toggle the Active status of the pipe on and off. The simulations will omit all pipes that are not active.

Zone ID

This is an optional name for the zone to which the pipe belongs. When a zone ID is specified, this zone will be listed in the 'Zones' editor. The '…' button can be used to select an existing zone.

Hydraulics

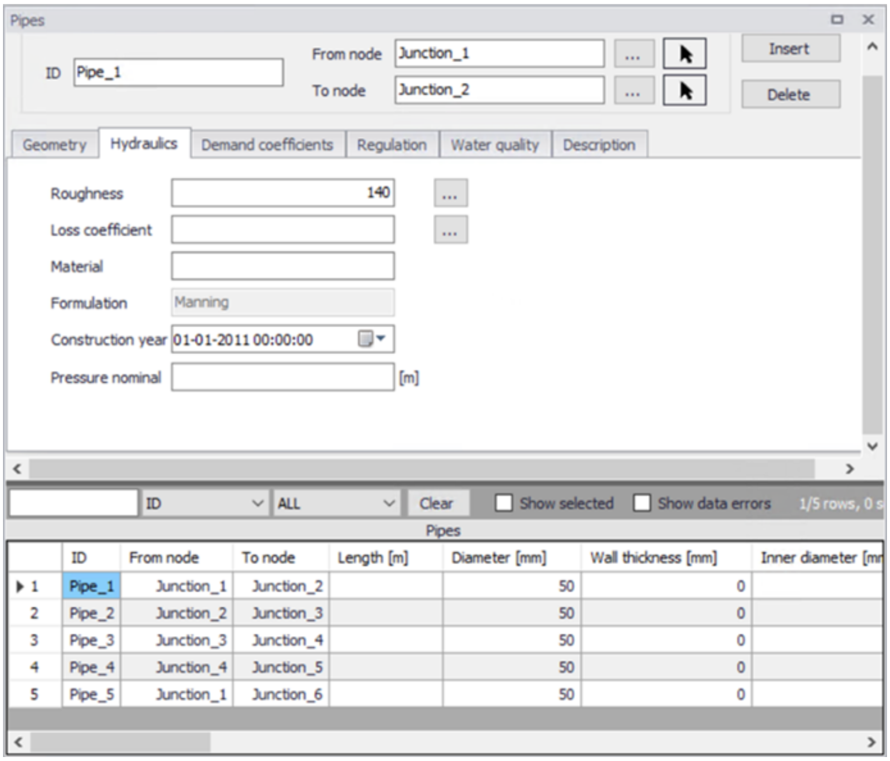


Figure 3.10 Pipe Hydraulics Editor

Roughness

This data entry defines the roughness of the interior surface of the pipe. Based upon which roughness type loss coefficient has been specified, this value is unit less for Hazen-Williams or Chezy-Manning headloss formulas, and in millifeet or mm for the Darcy-Weisbach (or Colebrook-White) formulation. Choosing “...” will display the Select Pipe Roughness Coefficient selection dialog box, allowing the user to select the appropriate roughness value to use

The roughness formulation is displayed in a field below. It can be specified by the user within the Simulation specification > Hydrodynamic simulation settings, where the Head losses setting is changed on the HD parameters tab.

Loss coefficient

This data entry defines the sum of all the minor (or local) loss coefficients for the pipe, which are unitless. Choosing "..." will display Select Minor Loss Coefficient selection dialog box, allowing the user to select the appropriate minor loss coefficient to use. If more that one minor loss component exists along the pipe, then the sum of the corresponding minor loss coefficients should be entered.

Material

This option allows the user to define the material of pipe construction. The Pipe Material is defined as a "string" a string and does not influence calculations. The friction losses in hydrodynamic calculations are based on pipe roughness, which can be globally assigned based upon the pipe material and pipe construction year, for example.

Formulation

This read only field displays the head loss setting. It can be specified by the user within the Simulation specification > Hydrodynamic simulation settings, where the Head losses setting is changed on the HD parameters tab.

Construction year

This option allows the user to define the age of the pipe. Pipe age is defined as a date. Clicking the Calendar opens a calendar dialogue where the user can browse to a date.

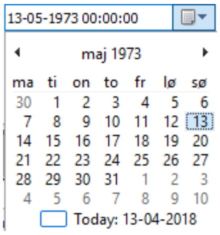


Figure 3.11 Calendar view

Demand coefficients

MIKE+ allows the user to distribute a specified water demand to the network based upon a variety of pipe properties. Three methods are available from the Distributed Demand tool (found in the Tools ribbon). This feature is useful for automatically assigning the nodal water demand to a large network, since the software will automatically proportion the total network demand based upon predefined pipe properties. These methods are used to mimic the amount of actual demand along a pipe, based upon the pipe length or predefined demand coefficients.

- Method of equal pipe lengths, distributes the demand based on pipe length and the pipe diameter.
- Method of reduced pipe length, distributes the demand based on pipe length and a user specified coefficient.
- Method of reduced Two Coefficients, distributes the demand based on two user specified coefficients.

The Method of reduced pipe length and method of two coefficients uses one or two user specified pipe coefficients. More information about these calculations are found in the chapter Distributed Demand tool.

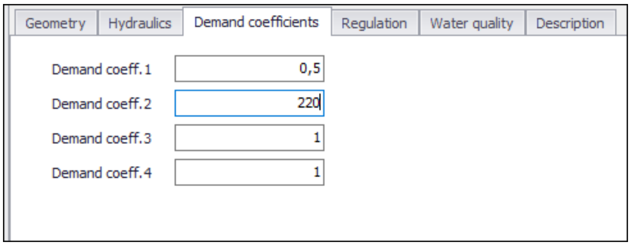


Figure 3.12 Pipe Demand Coefficients Editor

Demand coefficient 1 - 4

Fields for specifying coefficients relevant to pipe leakage. A higher number will generate a larger portion of the total demand to be distributed.

VANote that there are four fields but no more than two coefficients can be selected in a Distributed demand calculation. The coefficients that is used is specified in the Distributed demand tool.

Regulation

The regulation tab allows to set simple rules for controlling each pipe to open or close, depending on the pressure level in a node, time of day or time since simulation started.

The tab has three parts. The middle contains a grid for all rules that controls the active pipe. This window also allows to add or remove control rules for the selected pipe.

The left window is the editor for the active control rule, currently selected in the grid.

The right window displays a time series if there are rules based on Time conditions.

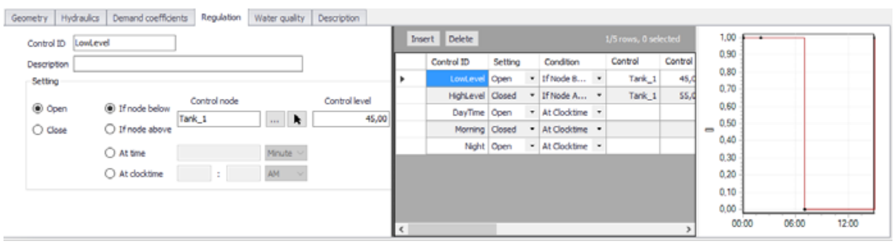


Figure 3.13 Regulation Tab

Pressing “Insert” in the middle window creates a new control rule for the selected pipe. “Delete” will remove the active control rule. The properties and settings for the active rule is displayed in the left part of the regulation tab.

Control ID

An ID for the rule is automatically generated, but could be specified by the user. Note that every Control ID for all pipes, pumps, valves and turbines in the model must be unique.

Description

This field allows users to type text to describe the Control.

Setting

The settings contain three parts:

- Action
- Type of condition
- Condition

A radio button is used to set an **Action**. A pipe can only be set to Open or Close.

A radio button is used to set **Condition type** to one type of condition that will trigger the action.

- If node below/above... This rule will execute the action if the pressure level in a specified node is above or below a specified level.
- At time... This rule will execute the action when the specified amount of time since simulation start has passed. When setting up a series of these rules there will be a time series of the setting in the right window.
- At clocktime... This rule will execute the action every day at the specified time.

The available **Condition** settings will depend on the selected condition type.

- When “If node below/above” is selected, the user must specify a node or tank ID in the first field and the threshold pressure level in the second field. Note that this is defined as the pressure at Elevation level for a node, and the pressure at Base elevation for a tank.
- When “At time” is selected, the user must specify a number and a time unit since start of simulation.
- When “At clocktime” is selected the user must specify a time of day in hours, minutes and AM/PM.

Water quality

This tab allows for each pipe to have locally defined reaction rates. Please refer to section on Water Quality reaction rates for further information.

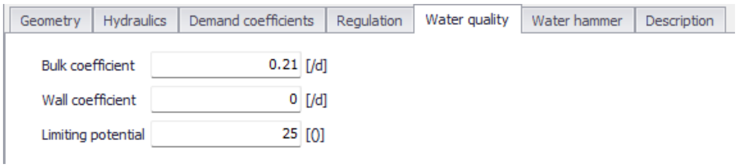


Figure 3.14 Water Quality Editor

Bulk coefficient

This data entry defines the bulk reaction rate that is applied to flow in the pipe. Units for bulk reaction rates are in 1/day.

Wall coefficient

This data entry defines the pipe wall reaction rate that is applied to flow in the pipe. Units for pipe wall reaction rates are in 1/day.

Limiting potential

This setting specifies that reaction rate is proportional to the difference between the current concentration and some limiting value. When undefined, the program will use the global limiting potential specified in water quality settings of the 'Simulation setup' editor.

Description

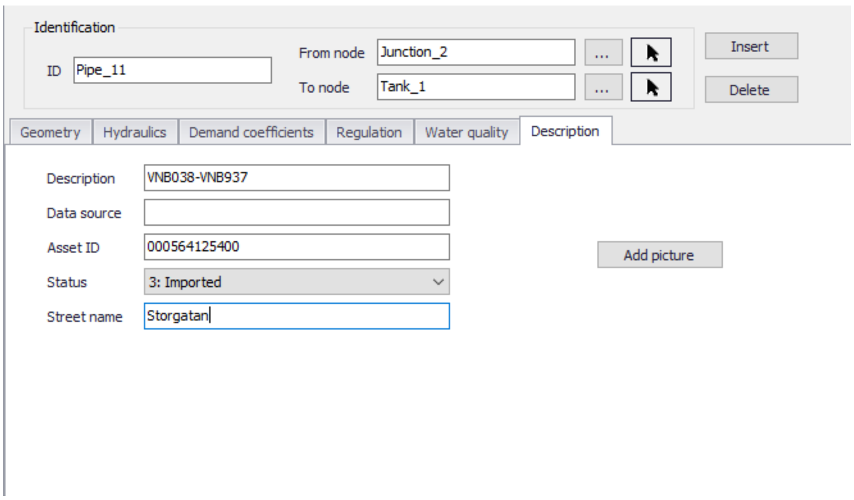


Figure 3.15 Pipe Description Editor

Description

This data entry allows you to enter a description for the selected pipe.

Add picture

The <Add picture> button allows users to add photo for a individual pipe. Once loaded from external source, the picture will be displayed on this tab.

Data source

This data entry is used to specify a corresponding asset data source, which identifies the pipe (such as database table or a database file name) in the asset management system.

Asset ID

This data entry is used to specify a corresponding asset ID, which uniquely identifies the pipe in the asset management system (such as GIS, for example).

Status

This drop down selection list data entry allows you to define whether the pipe is imported (i.e existing node was imported from the external data source), or is inserted, modified, GIS, calibrated or similar. By default, the status is undefined.

Street name

This field is used to define the street name. This is an optional field and can be used for better navigation through the pipe network and for reporting purposes.

Attributes

Table 3.2 Pipe attributes

Field	Database name	Description	Mandatory?	Default value
ID	MUID	Identifier, must be unique for all link types including valves etc	Yes	Labels are generated in sequential order
From node	FromNodeID	The from node of the pipe, defining the start	Yes	
To node	ToNodeID	The to node of the pipe, defining the end	Yes	
Length	L	Pipe length	No	
Diameter	Diameter	Diameter of pipe	Yes	50 mm
Wall thickness	Thickness	Wall thickness of pipe to calculate inner diameter	No	
Initial status	StatusNo	Sets the pipe to open, closed or check valve	Yes	Open

Table 3.2 Pipe attributes

Field	Database name	Description	Mandatory?	Default value
Is active	Enabled	Set the pipe active/inactive.		TRUE
Roughness	RCoeff	Defines the interiour surface roughness. The unit depends on the headloss formula.	Yes	
Loss coefficient	LCoeff	The sum of all minor losses within the pipe.	No	
Material	Material	Text field for pipe material. Not used in calculations.	No	
Construction year	CDate	Date to describe pipe age. Not used in calculations.	No	
Demand coeff. 1-4	Coeff1 Coeff2 Coeff3 Coeff4	Coefficient for demand distribution calculations.	No	
Bulk coefficient	Bulk_Coeff	Locally defined reaction rate in water quality calculations.	No	
Wall coefficient	Wall_Coeff	Locally defined reaction rate for water quality calculations.	No	
Description	Description	Descriptive text	No	
Data source	DataSource	Text field for data source.	No	
Asset ID	Asset	Text field to identify the pipe to the corresponding pipe in the asset management system.	No	
Street name	StreetName	Text field to define street name.	No	

3.3 Tanks

Tank Editor

Tank nodes are also placed at points in the water distribution model where a water storage tank is located. Storage tanks can be defined as tanks with the variable or fixed water level. The tank with the variable water level are modeled as tanks where the water surface level changes with time as water flows into and out of the tank. The tanks with the fixed water level represent places (reservoir) within the water distribution model where an infinite source of water (for the sake of the modeling simulation) is available. Hence, the reservoir water level remains constant during the course of the simulation.

Tank nodes are either defined interactively on the graphical Map window using the Add Tank tool (see Figure 3.16), or by manual data entry using the Tank Editor dialog box as shown in Figure 3.18. The Tank Editor allows you to define the reservoir's ID, location, properties, water quality, and description. The Tank Editor dialog box is reached by clicking **Tanks** in **Network** under Setup tree (see Figure 3.17).

Figure 3.16 The Tank editing tool

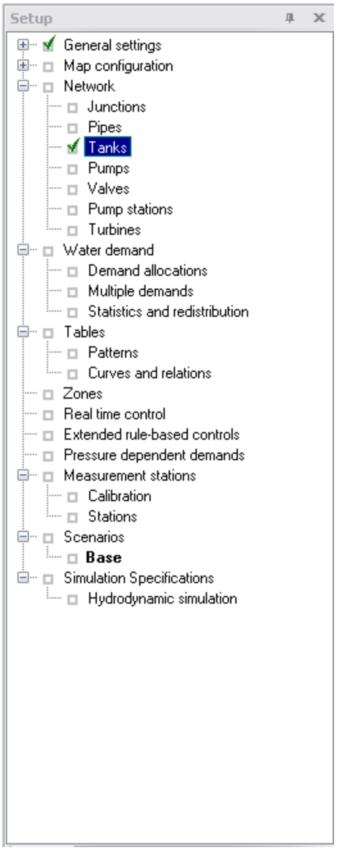


Figure 3.17   The Tank Editor dialog box is reached in Setup tree

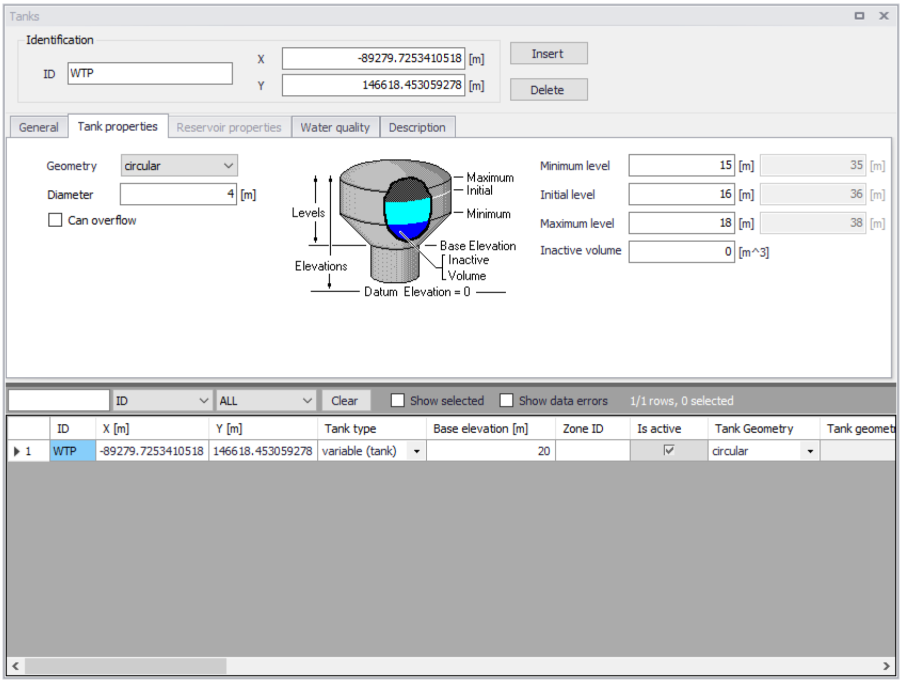


Figure 3.18 The Tank Editor allows the user to define the storage tank nodes that supply water to the distribution network

The Tank Editor contains input fields for geometry, Tank Properties, Reservoir Properties and Description.

A list of the Tank Editor data entries for Figure 3.18 follows, with a short description given for each entry.

Identification

Tank ID

This data entry is used to specify an ID which uniquely identifies the tank node. The tank ID acts as a unique lookup key that identifies the node from all other nodes. A node can be a junction, reservoir, tank, or air-chamber. Therefore, no two nodes may have the same ID.

However, a node and a link (i.e., pipe, pump, or valve) can have the same ID. The node ID value can be any string value (up to 40 characters).

A new tank ID is automatically suggested by MIKE+ whenever a new tank node is placed into the list by pressing «Insert». When defining the tank nodes graphically on the Map window Figure 3.19 using the Add Tank tool, the tank ID is automatically defined.

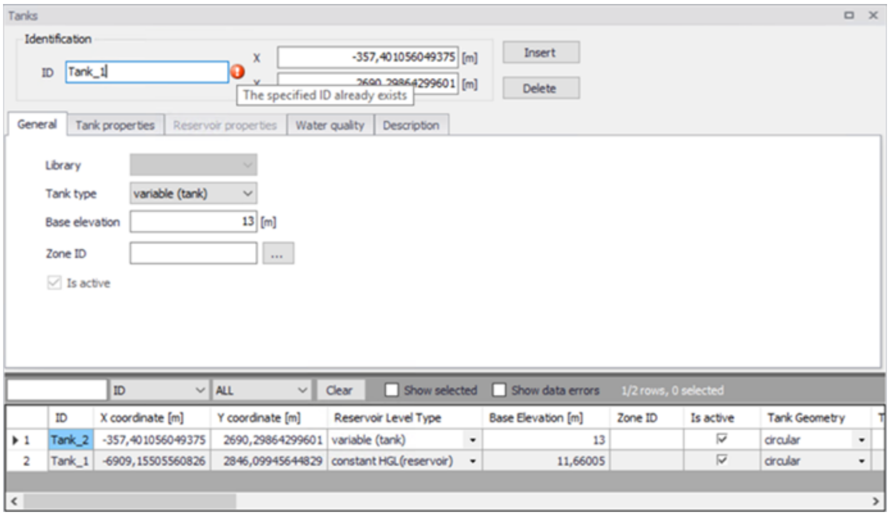


Figure 3.19 Warning message displayed when a tank has a repetitive ID

X and Y COORDINATE

The X and Y data entries are used to define the physical (map) location of the tank node, in units of ft. or m.

General

This tab gives general information of tanks as shown in Figure 3.20

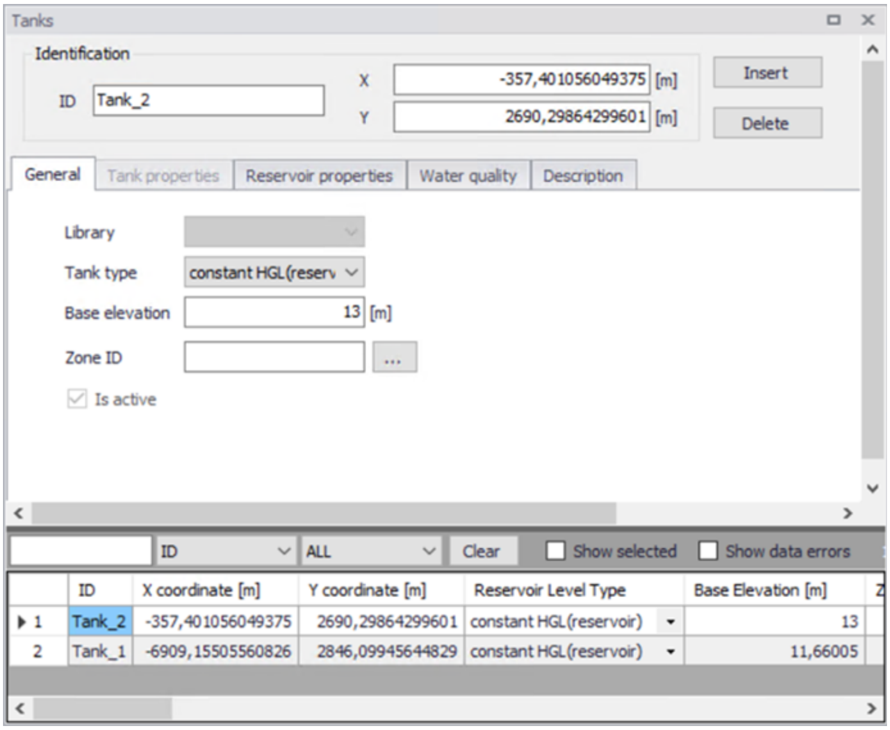


Figure 3.20 The general information of Tank

Tank Type

This drop down selection list data entry allows you to define whether the tank is modelled as reservoir (constant HGL), or is tank (variable HGL).

There are two options available:

- Constant HGL (Reservoir)
- Variable HGL (Tank)

Base Elevation (mandatory)

Base elevation defines the distance from bottom of the tank/reservoir above datum elevation.

Zone ID

This is an optional name for the zone to which the tank belongs. When a zone ID is specified, this zone will be listed in the 'Zones' editor. The '…' button can be used to select an existing zone.

Is Active

This check box controls whether the tank will be included (when ticked) or omitted (when unticked) in the simulations. The tank is automatically omitted as soon as all connected links are also set to inactive.

Tank Properties

This tab would be editable only when the tank type is “variable (tank)”, as shown in Figure 3.21

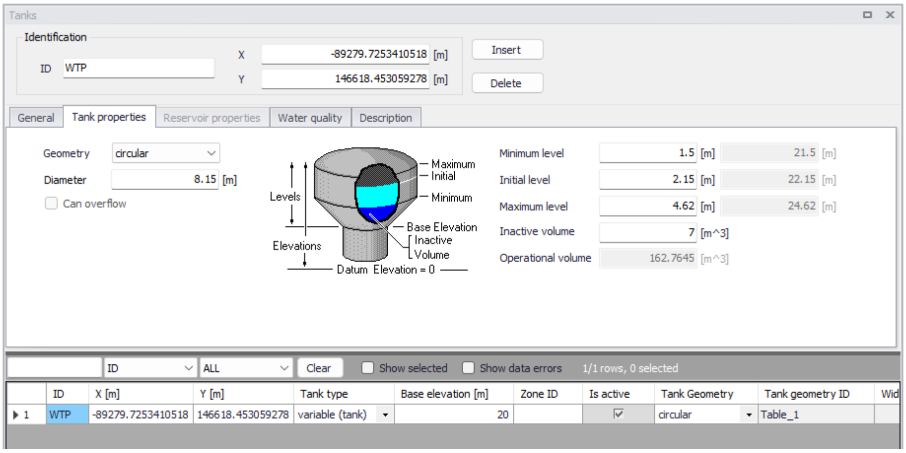


Figure 3.21 The Tank Properties

GEOMETRY (mandatory)

This drop down selection list data entry selects the type of storage tank being defined.

- Table
- Rectangular
- Circular

For different types of tank, the required geometry data is different. By default, a circular tank is defined. The elevation-volume relationship for a tank of variable geometry is needed to be defined. A Volume Curve determines how storage tank volume (Y in cubic feet or cubic meters) varies as a function of water level (X in feet or meters). It is used when it is necessary to accurately represent tanks whose cross-sectional area varies with height. The lower and upper water levels supplied for the curve must contain the lower and upper levels between which the tank operates.

GEOMETRY ID(mandatory)

The geometry ID is the ID of the relationship which determines how storage tank volume (Y in cubic feet or cubic meters) varies as a function of water level (X in feet or meters). It is used when it is necessary to accurately represent tanks whose cross-sectional area varies with height. This relationship is defined in the 'Curves and relations' editor, using the curve type 'Tank depth-volume'

DIAMETER or WIDTH and LENGTH (mandatory)
This data entry allows you to define the tank chamber size (ft or m).

CAN OVERFLOW
When this option is selected, any inflow to a full tank becomes overflow (i.e. spillage). When it's unselected, any link that would normally send flow to the tank is temporarily closed when the tank is full. This option is only available when using the EPANET 2.2 version.

MINIMUM LEVEL (mandatory)
This data entry defines the minimum level (or depth), in units of ft. or m, that the water can drop to within the storage tank. The corresponding elevation is equal to the base elevation plus the minimum level, as shown in Figure 3.22

INITIAL LEVEL (mandatory)
This data entry defines the initial water surface level (or depth), in units of ft. or m, that is used at the start of the simulation. The corresponding elevation is equal to the base elevation plus the initial level, as shown in Figure 3.22.

MAXIMUM LEVEL (mandatory)
This data entry defines the maximum level (or depth), in units of ft. or m, that the water can rise to within the storage tank. The corresponding elevation is equal to the base elevation plus the maximum level, as shown in Figure 3.22.

INACTIVE VOLUME (optional)
This data entry defines the volume of inactive water contained between the minimum level and the base elevation, in units of ft^3 or m^3, of the storage tank, as shown in Figure 3.22.

OPERATIONAL VOLUME
This volume is derived from the above geometrical properties of the tank, and represents the usable volume, which can be filled or released from the tank.

For circular and rectangular tanks, the volume equals the horizontal area of the tank times the difference between its maximum and minimum water levels.

For tanks defined using a geometry table, the operational volume is the difference between the volume of the table at the maximum level, and the volume of the table at the minimum level. Volumes at the maximum and minimum levels are obtained using a linear interpolation from the table's data.

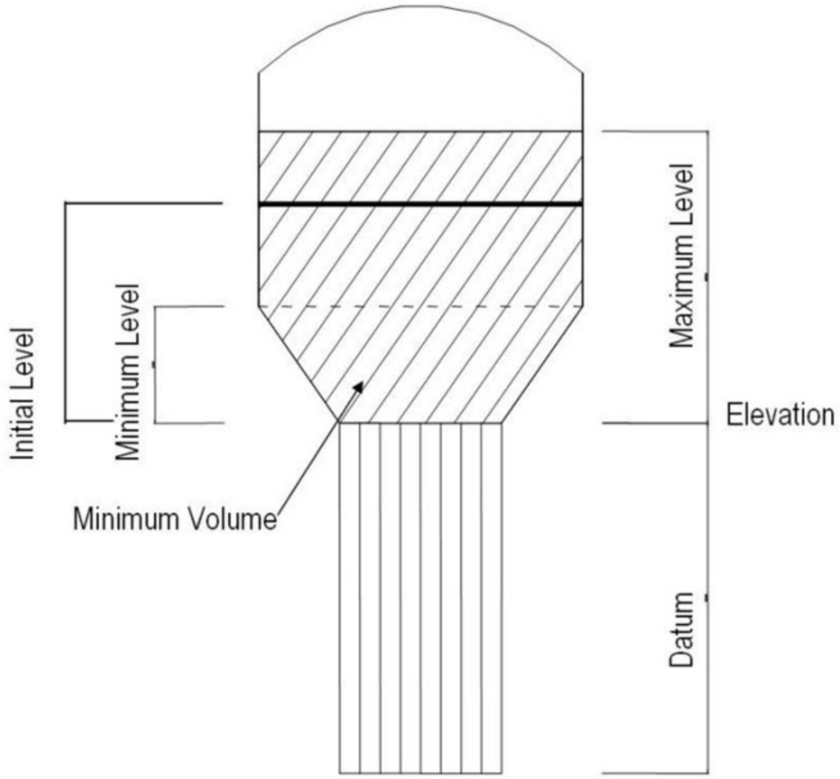


Figure 3.22 Definition of storage tank levels

Reservoir Properties

This tab would be editable only when the tank type is “Reservoir”, as shown in Figure 3.23.

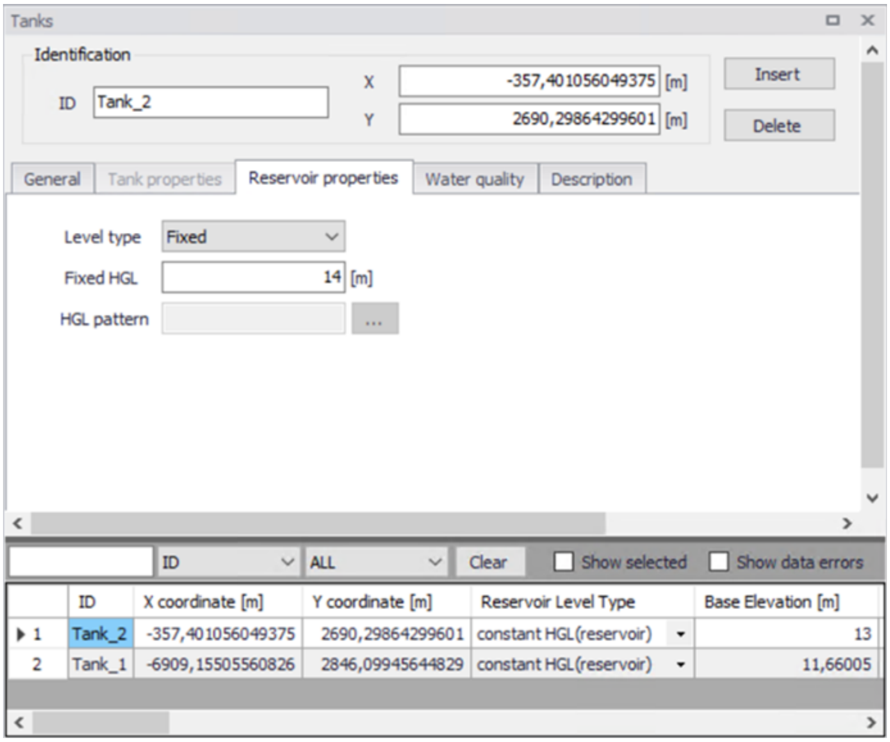


Figure 3.23 The Reservoir Properties

LEVEL TYPE

- This data determines whether the total water head of reservoir is fixed or variable. There are two options available for the level type:
- Fixed
- Pattern

FIXED HGL

This data entry allows you to define the constant water head in case that the tank is modelled as a reservoir with fixed water level. The water head is defined in ft or m.

HGL PATTERN

The ID label of a time pattern used to model time variation in the tank's (reservoir's) total head. This property is useful if the reservoir represents a tie-in to another system whose pressure varies with time.

HGL PATTERN

The ID label of a time pattern used to model time variation in the tank's (reservoir's) total head. This property is useful if the reservoir represents a tie-in to another system whose pressure varies with time.

Water Quality

This tab defines water quality parameters of tanks, as shown in Figure 3.24.

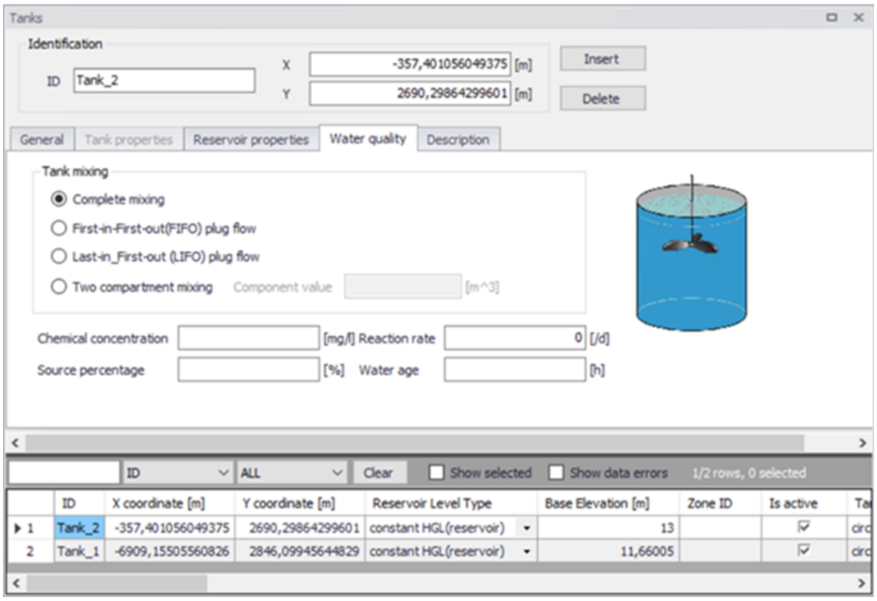


Figure 3.24 The water quality parameters of tanks

TANK MIXING (optional)

MIKE+ allows the user to choose between four different types of tank mixing, completely mixed, two compartment mixing, Last In First Out (LIFO) and First In First Out (FIFO).

The Completely mixed model assumes that all water that enters a tank is instantaneously and completely mixed with the water already in the tank. It is the simplest form of mixing behavior to assume, requires no extra parameters to describe it, and seems to apply quite well to a large number of facilities that operate in fill-and-draw fashion.

The Two-Compartment mixing model divides the available storage volume in a tank into two compartments, both of which are assumed to be completely mixed. The inlet/outlet pipes of the tank are assumed to be located in the first compartment. New water that enters the tank mixes with the water in the first compartment. If this compartment is full, then it sends its overflow to the second compartment where it completely mixes with the water already stored there. When water leaves the tank, it exits from the first compartment, which if full, receives an equivalent amount of water from the second compartment to make up the difference. The first compartment is capable of simulating short circuiting between inflow and outflow while the second compartment can represent dead zones. The user must supply a single parameter which is the fraction of the total tank volume devoted to the first compartment.

The First-In-First-Out (FIFO) Plug Flow mixing model assumes that there is no mixing of water at all during its residence time in a tank. Water parcels move through the tank in a segregated fashion where the first parcel to enter is also the first to leave. Physically speaking, this model is most appropriate for baffled tanks that operate with simultaneous inflow and outflow. There are no additional parameters needed to describe this mixing model.

The Last-In-First-Out (LIFO) Plug Flow mixing model assumes that there is no mixing between parcels of water that enter a tank. However in contrast to FIFO Plug Flow, the water parcels stack up one on top of another, where water enters and leaves the tank on the bottom. Physically speaking this type of model might apply to a tall, narrow standpipe with an inlet/outlet pipe at the bottom and a low momentum inflow. It requires no additional parameters be provided.

REACTION RATE (optional)

This data is locally defined reaction rate. It defines the rate at which constituent decays (or grows) by reaction as the constituent travels through the pipe network. Please refer to section on reaction rates for further.

CHEMICAL CONCENTRATION

This data entry is used to specify the initial water quality (chemical concentration in mg/liters) at the tank. It is used when conducting chemical concentration simulation.

SOURCE PERCENTAGE

This data entry is used to specify the initial percentage of water from the source node in percent at the tank. It is used when conducting source tracing simulation.

WATER AGE

This data entry is used to specify the initial water age of water in hour at the tank. It is used when conducting water age simulation.

Description

DESCRIPTION

This data entry allows you to enter a description identifying the tank node being entered. This description can be optionally displayed on the Map window and in reports generated by the Report Generator.

DATA SOURCE (optional)

This data entry is used to specify a corresponding asset data source, which uniquely identifies the tank node location (such as database table or a database file name) in the asset management system.

ASSET ID (optional)

This data entry is used to specify a corresponding asset tank ID, which uniquely identifies the tank node in the asset management system (such as GIS, for example).

STATUS (optional)

This drop down selection list data entry allows you to define whether the tank node is imported (i.e existing node was imported from the external data source), or is inserted, modified, GIS, calibrated or similar. By default, tank node status is undefined.

ADD PICTURE

The <Add picture> button allows users to add photo for individual tank. Once loaded from external source, the picture will be displayed on the right section in Figure 3.25.

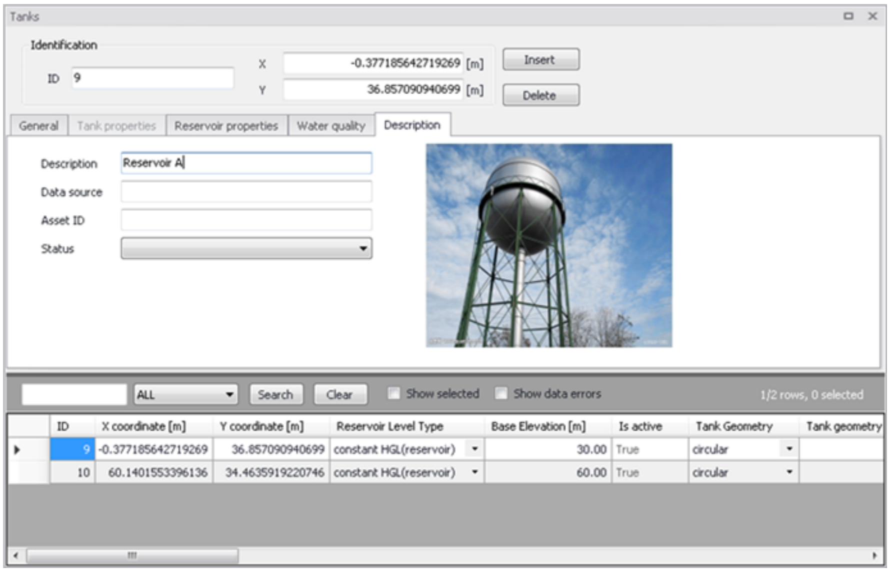


Figure 3.25 Tank Editor Picture

3.4 Pumps

Pumps are used to raise the hydraulic head of water. Pumps are represented as short links of negligible length. The simulation engine will automatically prevent flow reversal through a pump, and will issue warning messages when a pump operates outside of its normal operating range.

Pumps are either defined interactively on the graphical Map window using the Drawing tool (see Figure 3.26), or by manual data entry using the Pumps Editor dialog in Figure 3.27.

Figure 3.26 Pumps Drawing and Editing Tool

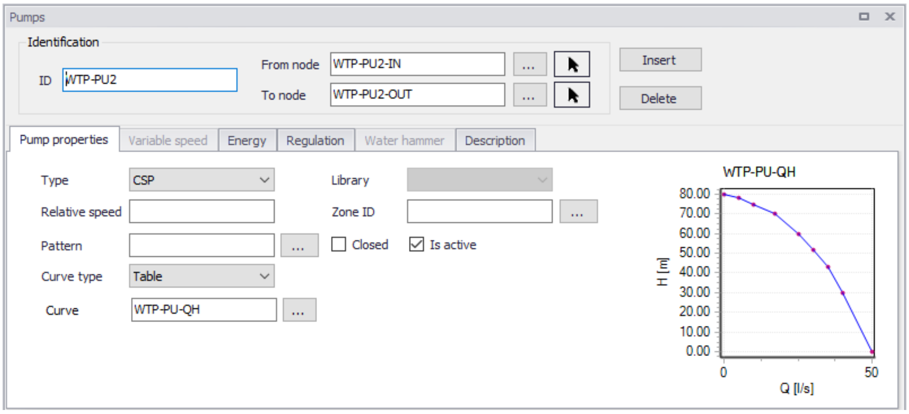


Figure 3.27 Pump Configuration Window

The Pumps Editor allows the user to define the pump´s ID, pump power curve, status, regulation, energy consumption, description, and other attributes. The Pumps Editor dialog box is reached by double clicking Pumps in Distribution Network under the Setup tree. (see Figure 3.28)

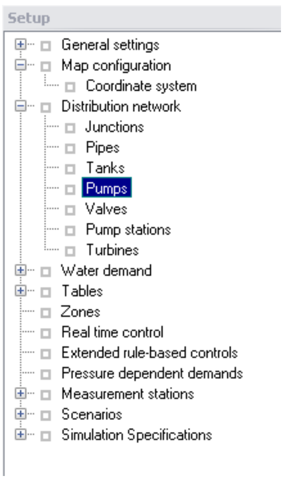


Figure 3.28 Layout of Setup Tree

Identification

Pump ID

This data entry is used to specify an ID to identify a pump link. The pump ID acts as a unique lookup key that identifies the pump link from all links.

From node, To node

These data entries define the ID of the pump's starting and ending nodes. Clicking ..., an ID selection window pops up and allows selecting the ID from a list. Clicking , it allows the user to graphically select the node from the Map window and the connection of pipe will be changed simultaneously on the map.

Pumped flow is always assumed to move from the starting node to the ending node.

Pump Properties

This tab contains the pump operating characteristics.

Type

There are two options available to define the pump types.

- VSD: variable speed drive
- CSP: constant speed pump

For VSD pump, user can control the relative speed of each pump by pressure or level control at node or by a flow control at a link. It can be set in "Variable Speed" Tab, which would only be activated for VSD pumps and would be grey otherwise.

Relative Speed

Relative Speed entry field allows the user to adjust the initial setting of the pump at the start of the simulation. For example, entering a value of 1.2 specifies that the pump operates at 1.2 times its normal speed at the start of the simulation.

Curve Type

There are four options available to define the pump specifications:

- Constant
- 1-point
- 3-point curve
- Table

Constant is used when the pump characteristic curve is unknown and a constant power output is assumed. The data entry specifies the pump power rating, in hp or kw. The default power rating is zero.

1-Point type is used for a standard pump curve with no extended flow range, where the cutoff head is 133% of the design head and the maximum flow is twice the design flow.

3-Point type can be used to describe the flow-head relationship of the pump. The Shutoff Head is the head value at zero flow. The Design Head is the standard operating head, in units of ft. or m, and are by default zero. The Design Flow is corresponding flow rate, in the user-specified units, and by default zero. The High End Head is the head at the upper end of the normal operating flow range. The High End Flow is the corresponding flow rate. The Maximum Flow is the flow rate for the extended flow range. All heads are in units of ft. or m, and flows are in the user-specified units.

The Table type is used to define a Q-H Pump Curve, created by providing either a pair of head-flow points, or four or more such points. MIKE+ creates pump curves by connecting the points with straight line segments. The Q-H pump curve must be created in the ‘Curve and relations’ editor.

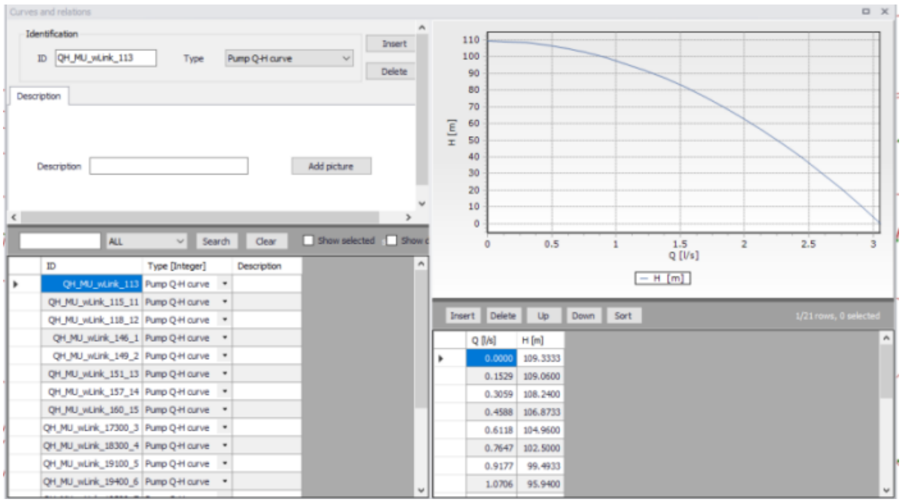


Figure 3.29 Curves and Relations Settings

Is active

This check box allows the user to toggle the Active status of the pipe on and off. The simulations will omit all pumps that are not active.

Zone ID

This is an optional name for the zone to which the pump belongs. When a zone ID is specified, this zone will be listed in the 'Zones' editor. The '…' button can be used to select an existing zone.

QH plot

The plot shows the pump's QH curve, as defined from the above properties.

It can also display the simulated operating points from a result file. See Pump Q-H Plot chapter page 468 for more information.

.Variable Speed

MIKE+ is capable of modelling VSD pumps in extended period simulations.

Control type

Three types of control can be applied:

- Downstream node control: with this control type, the variable speed is a simplified setting for pressure control at the downstream nodes of the active pump
- Remote node control: with this control type, the variable speed is controlled by the pressure in any of the node in the network.
- Link control: with this control type, the variable speed is controlled by the flow in any link in the network.

Control item

In case of a node control, the control item can be either pressure or hydraulic gradeline. In case of a link control, the control item is flow.

Control node

The selected node where the pressure controls the variable speed, when the control type is 'Remote node control'.

Control link

The selected link where the flow controls the variable speed, when the control type is 'Link control'.

Control pressure

Users can type the pressure value in meter or psi in this box. This setting refers to the downstream node of the active pump to be controlled when the control type is 'Downstream node control', or the selected node when the control type is 'Remote node control'.

Control HGL

Users can type the hydraulic grade line value in this box. This setting refers to the downstream node of the active pump to be controlled when the control type is 'Downstream node control', or the selected node when the control type is 'Remote node control'.

Control flow

Users can type the flow value in this box. This setting refers to the control link of the active pump to be controlled when the control type is 'Link control".

Curve

The ID of the selected 'Pump pressure setpoint', ‘Pump HGL setpoint’, or ‘Pump flow setpoint’-curve, specifying the setpoint value as a function of time.

The use of a curve is optional, and when no curve is selected the setpoint is constant. This option is only available when using the EPANET 2.2 version.

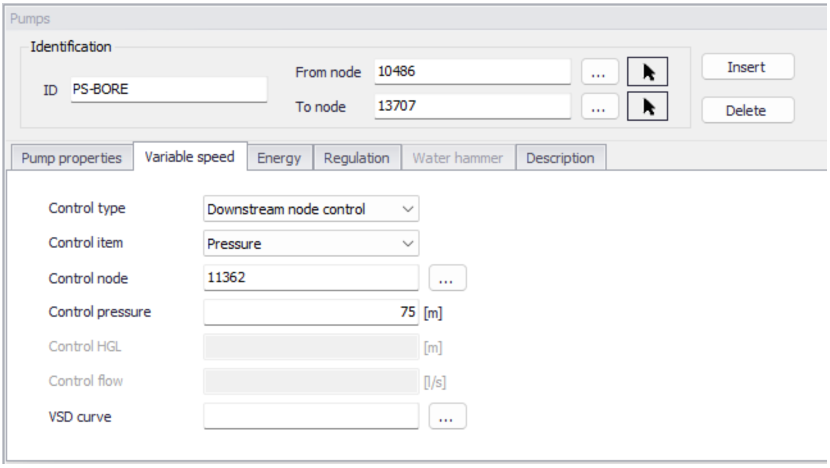


Figure 3.30 Variable Speed Setting

Energy

MIKE+ is capable of modelling the cost of operating pumps. Within the Pump Energy tab, the user can define a method for cost calculation. See Figure 3.31

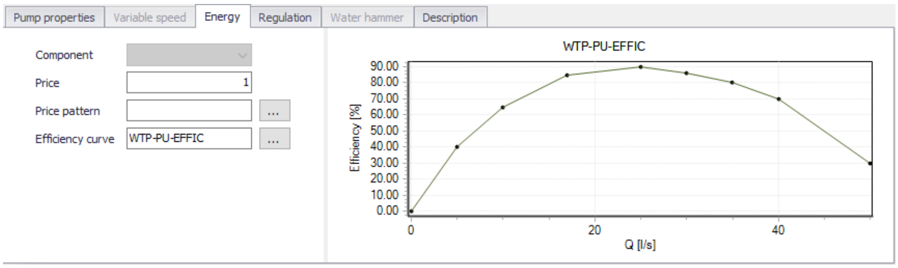


Figure 3.31 Energy Settings

Price

The user defines an energy price (e.g. $/kw-hour) to be used. In this method, MIKE+ determines the energy consumed by the pump in kw-hours and multiplies the energy consumption by the price.

Leave blank if not applicable or if the global value supplied with the Parameters in Cost Analysis will be used.

Price Pattern

The ID label of the time pattern used to describe the variation in energy price throughout the day. Each multiplier in the pattern is applied to the pump's Energy Price to determine a time-of-day pricing for the corresponding period.

Leave blank if not applicable or if the global pricing pattern specified in the project's Energy Options will be used.

Efficiency Curve

The ID label of the curve that represents the pump's wire-to-water efficiency (in percent) as a function of flow rate. This information is used only to compute energy usage. Leave blank if not applicable or if the global pump efficiency supplied with the project's Energy Options will be used.

Regulation

Settings in this 'Regulation' tab suits CSP (Constant Speed Pumps). VSD pumps can be controlled from the 'Real-Time Control' editor. Please refer to the corresponding chapter for more information.

Control ID

This is the main ID of the control.

Description

This field allows users to type text to highlight the Control that is going to be set.

Settings

The settings contain three parts:

- Action
- Condition type
- Condition

A radio button is used to set an **Action**. A pump can be set to Open, Close or a Value.

A radio button is used to set the **Condition type**, i.e. the type of condition that will trigger the action.

- If node below/above: This rule will execute the action if the pressure level in a specified node is above or below a specified level.
- At time: This rule will execute the action when the specified amount of time since simulation start has passed. When setting up a series of these rules there will be a time series of the setting in the right window.
- At clocktime: This rule will execute the action every day at the specified time.

The available **Condition** settings will depend on the selected condition type:

- When "If node below/above" is selected, the user must specify a node or tank ID in the first field and the threshold pressure level in the second field. Note that this is defined as the pressure at Elevation level for a node, and the pressure at Base elevation for a tank.
- When "At time" is selected, the user must specify a number and a time unit (hours/minutes) since start of simulation.
- When "At clocktime" is selected the user must specify a time of day in hours, minutes and AM/PM.

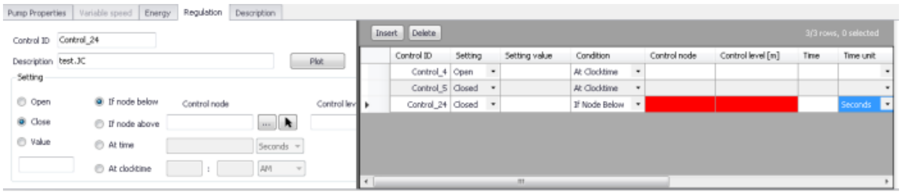


Figure 3.32 Control Settings in Regulation

Description

This data entry allows you to enter a description identifying the pump being entered. This description can be optionally displayed on the Map window and in reports generated by the Report Generator. See Figure 3.33

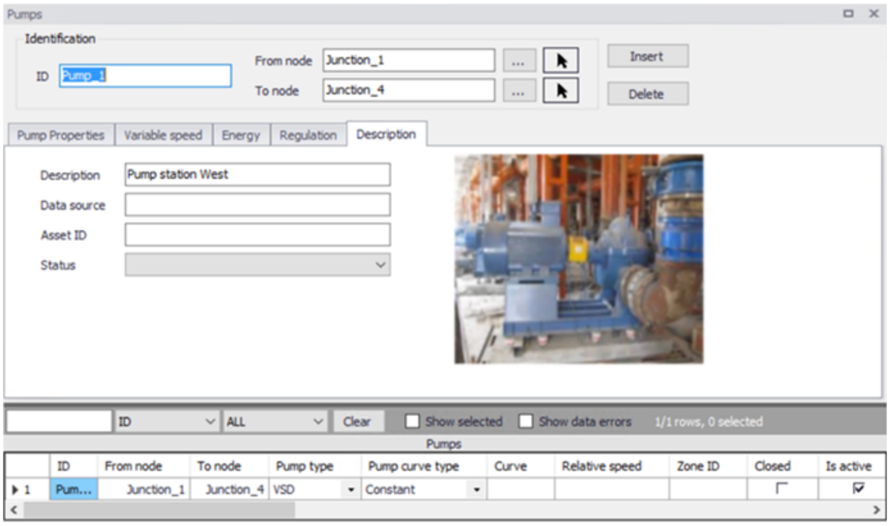


Figure 3.33 Layout of Description Settings

Data Source

This data entry is used to specify a corresponding asset data source, which uniquely identifies the pump location (such as database table or a database file name) in the asset management system.

Status

This drop down selection list data entry allows you to define whether the pump is imported (i.e existing node was imported from the external data source), or is inserted, modified, GIS, calibrated or similar. By default, pump status is undefined.

Add Picture

The <Add picture> button allows users to add photo for individual pump. Once loaded from external source, the picture will be displayed on the right section in Figure 3.34

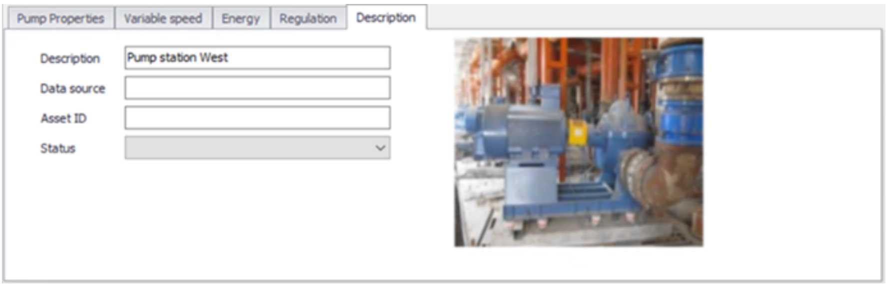


Figure 3.34 Pump Picture Displayed

Asset ID

This data entry is used to specify a corresponding asset pump ID, which uniquely identifies the junction node in the asset management system (such as GIS, for example).

3.5 Valves

Valves control the flow or pressure of water from one junction node to another. The functionality and setting of the valve is defined by it Valve Type setting. Valves are represented as links of negligible length. Note that valve pressure settings are pressures (e.g., psi or m above node elevation) and not total head (or hydraulic gradeline elevation).

Valves are either defined interactively on the Map using the 'Drawing' tool on the Edit tab with Valves selected as the Layer to edit, or by manual data entry using the Valve Editor dialog box. Valves cannot be directly connected to reservoir or storage tank nodes.

The Valve Editor allows you to define the valve's ID, type, status, nodal connectivity, description, and other attributes. The Valve Editor dialog box is reached by selecting Valves under Network.

Valve Properties

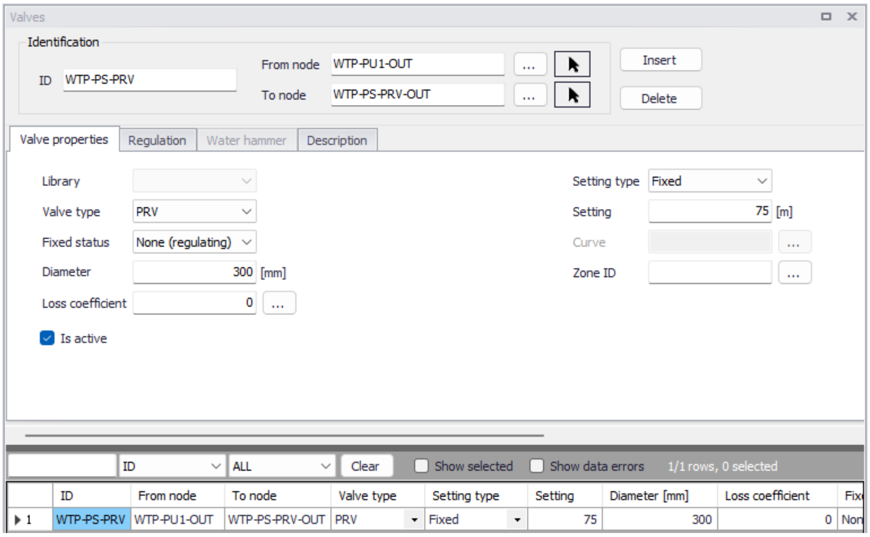


Figure 3.35 Valve properties Editor

<Insert> will create a new valve. <Delete> will remove the selected valve.

ID

This data entry is used to specify an ID which uniquely identifies the valve in the database. The valve ID acts as a unique look up key that identifies this link from all other links. A link can be a pipe, valve, pump or turbine. Therefore, no two links may have the same ID. However, a node and a link (i.e., junction or reservoir) can have the same ID. The valve ID value can be any string value (up to 40 characters).

A new valve ID is automatically suggested by MIKE+ whenever a new valve is placed into the list by pressing «Insert» or when defining the valve graphically in the Map window.

From Node, To Node

These data entries define the ID of the valve's starting (upstream) and ending (downstream) nodes. These IDs define the valve connectivity of the network.

Choosing "..." will display the Select Node dialog box from which the user can select the appropriate node. Valves cannot be directly connected to reservoir or storage tank nodes. Choosing the arrow allows the user to graphically select the node from the Map window.

Controlled flow is always assumed from the starting (upstream) node to the ending (downstream) node. Some valve types act as Check valves and does not allow flow from the To Node to the From Node. If the computed flow is moving from the ending node to the starting node, the computed flow value will be negative.

Valve type

This menu specifies the functionality of the valve. There are six different options.

PRV
A *Pressure Reducing Valve* limits the pressure at the downstream node to not exceed a preset value as long as the upstream node pressure is above the PRV setting. If the upstream pressure is below the setting, flow through the valve is unrestricted. Should the pressure at the downstream node exceed the pressure at the upstream node, the valve closes to prevent reverse flow. Note that PRVs cannot be placed directly in series. This valve requires a specified pressure (in m or ft at downstream node elevation) as setting. Pressure reducing valves can be based on the fixed pressure set-point or a set-point that is related to the actual flow, i.e. flow modulated.

PSV
A *Pressure Sustaining Valve* attempts to maintain a minimum pressure at the upstream node when the downstream node pressure is below the PSV setting. If the downstream pressure is above the setting, flow through the valve is unrestricted. Should the downstream nodal pressure exceed the upstream nodal pressure, then the valve closes to prevent reverse flow. Note that PSVs cannot be placed directly in series. This valve requires a specified pressure (in m or ft at upstream node elevation) as setting.

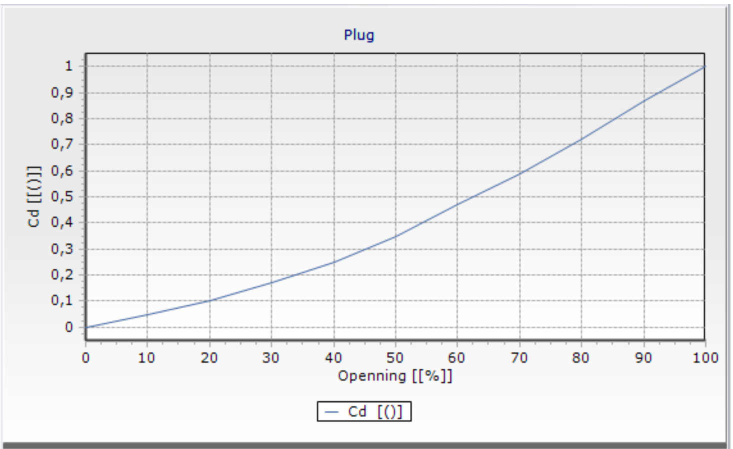
PBV
A *Pressure Breaker Valve* forces a specified pressure loss to occur across the valve. Flow can be in either direction through the valve. This valve requires a specified loss (in m or ft) as setting.

FCV
A *Flow Control Valve* limits the flow through a valve to a specified amount. The program will produce a warning message if this flow cannot be maintained with the current head at the upstream node of the valve. This valve requires a flow to be specified as setting.

TCV
A *Throttle Control Valve* is used to simulate partially closed valves by adjusting the minor head loss coefficient of the valve. This valve type requires a relationship between the degree to which the valve is closed and the resulting head loss coefficient. These are created and edited under Tables > Curves and relations. The Curve type is Valve character-

istics Cd. The curves for a few characteristic valves are available in MIKE+ as default. Other curves can usually be obtained from the valve manufacturer.

An initial opening percentage or Loss coefficient must also be specified as setting. Regulation or Rule-based control can be used to change this percentage setting during an extended period simulation, and thereby get another head loss coefficient from the curve.



Openning [[%]]	Cd [[()]]
0,0000	0,0000
10,0000	0,0500
20,0000	0,1000
30,0000	0,1700
40,0000	0,2500
50,0000	0,3500
60,0000	0,4700
70,0000	0,5900

Figure 3.36   Example of TCV curve

GPV
A *General Purpose Valve* provides the capability to model devices and situations with unique headloss - flow relationships, such as reduced pressure backflow prevention valves, turbines, and well drawdown behaviour. The valve requires a relationship curve between flow and head loss. These are created and edited under Tables > Curves and relations. The Curve type is Valve head loss.

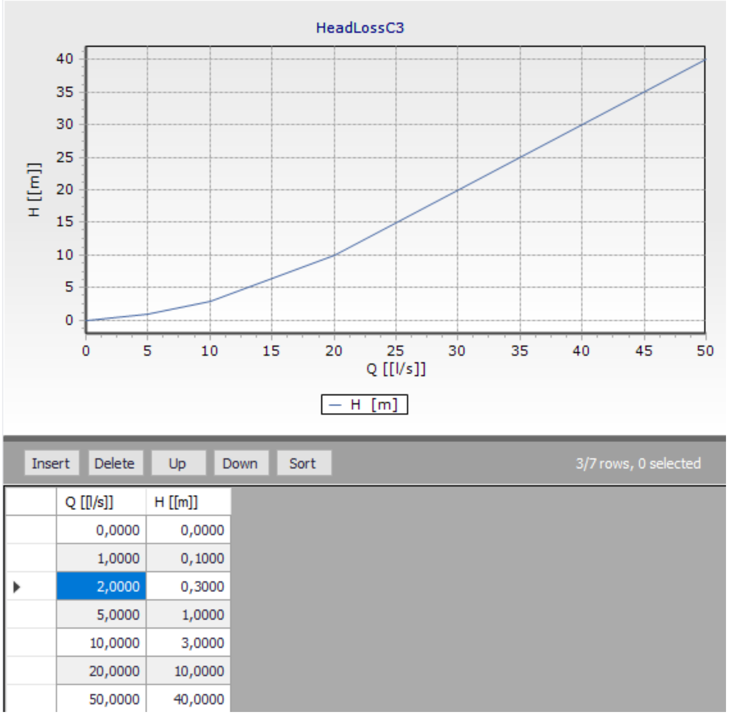


Figure 3.37 Example of GVC curve

Fixed status

This drop down list allows the user to toggle the OPEN and CLOSED status of the valve. Choosing CLOSED effectively removes the valve from the network system.

Diameter

This data entry defines the internal diameter of the valve, in the unit of your choice.

Loss coefficient

This data entry defines the sum of all the minor (or local) loss coefficients for the valve when fully opened, not including losses in TCV valve. The Loss coefficient is unitless. Choosing "..." will display Select Minor Loss Coefficient selection dialog box, allowing the user to select the appropriate minor loss coefficient to use. If more that one minor loss component exists along the valve, then the sum of the corresponding minor loss coefficients should be entered.

Is active

This check box data entry allows the user to toggle the Active status of the valve on and off. The simulations will omit all valves that are not active.

Setting type

Only available for TCV Valves or PRV Valves. Two options for the valve setting are available for TCV valves:

- Opening
- Loss coefficient

This option allows the user to choose to set a Opening % value for the valve, which is converted to a Loss coefficient using the specified Curve, or to set a Loss coefficient directly in the setting field.

Two options for the valve setting are available for PRV valves:

- Fixed
- Flow modulated

The second option allows the user to choose a pressure vs flow curve for the valve, The pressure vs flow curve is defined in the 'Curves and relations' editor.

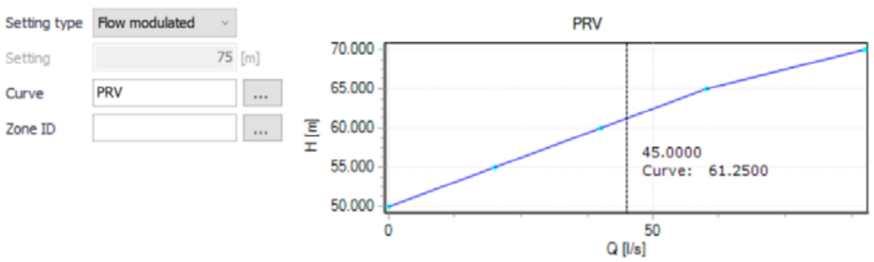


Figure 3.38 Defining a flow-modulated PRV valve

Setting

The valve setting. This data entry defines the pressure setting for PRVs, PSVs, and PBVs, whose units are in psi or m. Or, this data entry defines the flow settings (in user-defined flow units) for FCVs, or % opening or loss coefficients for TCVs.

When defining a pressure setting, the value specified is pressure at Node elevation (e.g., psi or m) and not total head (or hydraulic gradeline elevation).

Curve

The user must specify a Curve for PRV, TCV or GPV valves. [...] opens the Curve ID selector. A curve of type Valve characteristics Cd should be specified for a TCV valve and a Valve head loss curve should be specified for a GPV valve. Both curve types are generated from Tables > Curves and relations.

Zone ID

This is an optional name for the zone to which the valve belongs. When a zone ID is specified, this zone will be listed in the 'Zones' editor. The '…' button can be used to select an existing zone.

Level control

Only available for TCV valves. It allows the valve to gradually open and close based on the water level in the inlet tank. The valve's intermediate position, between the fully open and fully closed position, is determined using a valve curve and the actual water level in the tank (between the level open and close). The following settings must be specified.

- Tank ID: this is the inlet tank ID.
- Level open: this is the water level in the tank when the valve is fully open.
- Level close: this is the water level in the tank when the valve is fully close.

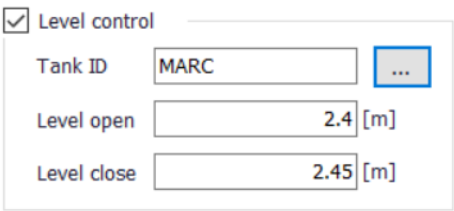


Figure 3.39 Example of level control for a TCV valve

Regulation

The regulation tab allows to set simple rules for controlling each valve, depending on the pressure level in a node or tank, time of day or time since simulation started.

The tab has three parts. The middle contains a grid for all rules that controls the active valve. This window also allows to add or remove control rules for the valve.

The left window is the editor for the active control rule, currently selected in the grid.

The right window displays a time series if there are rules based on Time conditions.

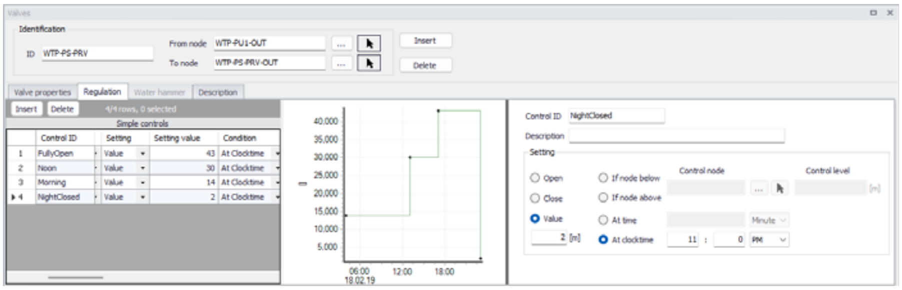


Figure 3.40 Regulation tab

Pressing “Insert” in the middle window creates a new control rule for the selected valve. “Delete” will remove the active control rule. The properties and settings for the active rule is displayed in the left part of the regulation tab.

Control ID
An ID for the rule is automatically generated, but could be specified by the user. Note that every Control ID for all pipes, pumps, valves and turbines in the model must be unique.

Description
This field allows users to type text to describe the Control.

Setting
The settings contain three parts:

- Action
- Type of condition
- Condition

A radio button is used to set an **Action**. A valve can only be set to Open, Close or a Value. The Value correspond to the Valve setting (on Valve properties tab) and changing the Value effectively means changing the setting. The function and unit depends on the valve type of the controled valve.

A radio button is used to set **Condition type** to one type of condition that will trigger the action.

- If node below/above... This rule will execute the action if the pressure level in a specified node is above or below a specified level.
- At time... This rule will execute the action when the specified amount of time since simulation start has passed. When setting up a series of these rules there will be a time series of the setting in the right window.
- At clocktime... This rule will execute the action every day at the specified time.

The available **Condition** settings will depend on the selected condition type.

- When “If node below/above” is selected, the user must specify a node or tank ID in the first field and the threshold pressure level in the second field. Note that this is defined as the pressure at Elevation level for a node, and the pressure at Base elevation for a tank.
- When “At time” is selected, the user must specify a number and a time unit (hours/minutes) since start of simulation.
- When “At clocktime” is selected the user must specify a time of day in hours, minutes and AM/PM.

Water hammer

Control valves can be simulated under hydraulic transient conditions, with the following valve types:

- TCV (throttle control valves)
- PRV (pressure reducing valves)
- PSV (pressure sustaining valves / back pressure valves)
- PBV (pressure breaker valves)
- FCV (flow control valves).

The additional settings are described below.

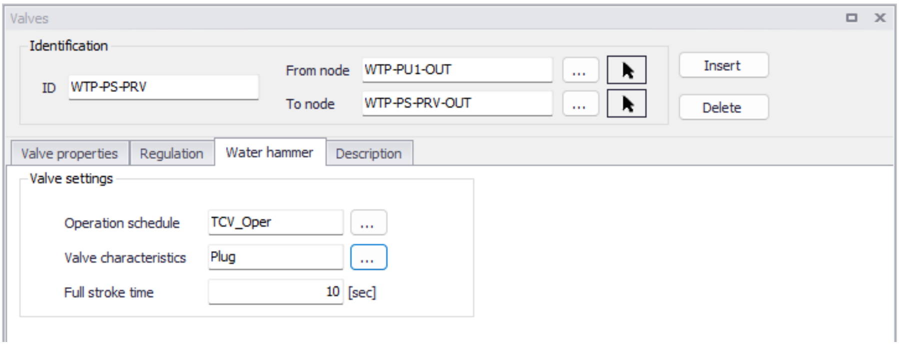


Figure 3.41 The Water hammer tab from the Valves editor

Operation schedule

This is the ID of a curve providing the relationship between the valve opening (in %) as a function of time. This curve is specified in the 'Curves and relations' editor, and must be defined with the curve type 'Valve operation schedule'. The program will use it to set the initial valve opening at the start of the simulation.

Valve characteristics

This is the ID of a curve providing the relationship between the valve coefficient Cd as a function of the valve opening (in %). This curve is specified in the 'Curves and relations' editor, and must be defined with the curve type 'Valve characteristics Cd'. The program will use it to move the valve during the transient flow to maintain the valve set point (valve settings).

Full stroke time

This is a positive float number representing the time for the valve to move from fully open to fully closed position.

Note: The 'Water hammer analysis' option must be selected in the 'Model type' editor, for this tab to be enabled in the 'Valves' editor.

Description

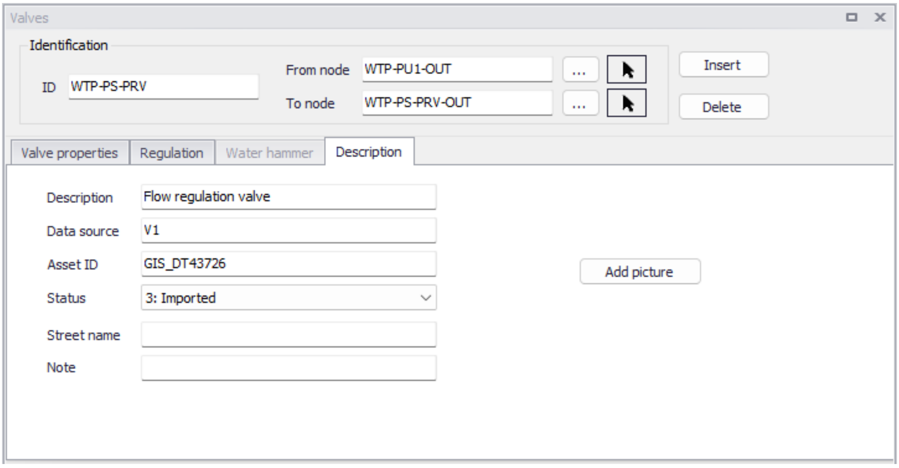


Figure 3.42 The Description tab from the Valves editor

Description

This data entry allows you to enter a description for the selected valve.

Add picture

The <Add picture> button allows users to add photo for a individual valve. Once loaded from external source, the picture will be displayed on this tab.

Data source

This data entry is used to specify a corresponding asset data source (such as database table or a database file name), in the asset management system.

Asset ID

This data entry is used to specify a corresponding asset ID, which uniquely identifies the valve in the asset management system (such as GIS, for example).

Status

This drop down selection list data entry allows you to define whether the valve is imported (i.e existing node was imported from the external data source), or is inserted, modified, GIS, calibrated or similar. By default, the status is undefined.

Street name

This field is used to define the street name. This is an optional field and can be used for better navigation through the pipe network and for reporting purposes.

Attributes

Table 3.3 Valve attributes

Field	Database name	Description	Mandatory?	Default value
ID	MUID	Identifier, must be unique for all link types including valves etc	Yes	Labels are generated in sequential order
From node	FromNodeID	The from node of the valve, defining the start	Yes	
To node	ToNodeID	The to node of the valve, defining the end	Yes	
Valve type	TypeNo	The type of valve.	Yes	PSV
Fixed status	StatusNo	Open/ close setting		None
Diameter	Diameter	Inside diameter of valve	Yes	50 mm
Loss coefficient	LossCoeff	The sum of all minor losses within the valve.	No	
Is active		Set the valve active/inactive	Yes	TRUE
Setting type	SettingNo	Setting type for TCV valve	For TCV	Loss Coefficient
Curve	HLCurveID	The Curve ID for TCV or GPV.	For TCV or GPV	
Setting	Setting	Valve setting. The unit and interpretation depends on valve type.	For PSV, PRV, PBV, FCV, and TCV	

Table 3.3 Valve attributes

Field	Database name	Description	Mandatory?	Default value
Description	Description	Descriptive text	No	
Data source	Data-Source	Text field for data source.	No	
Asset ID	Asset	Text field to identify the valve to the corresponding valve in the asset management system.	No	
Street name	Street-Name	Text field to define street name.	No	

3.6 Pump Stations

The Pump Stations editor allows to group pumps into pump stations. The purpose is to report pump energy costs per station, in the Cost Analysis.

A pump station is represented by a polygon on the map, and simply contains a list of pumps (defined in the 'Pumps' editor). New pump stations should preferably be added from the map, using the drawing tools to draw a polygon encapsulating the various pumps to be included in the station.

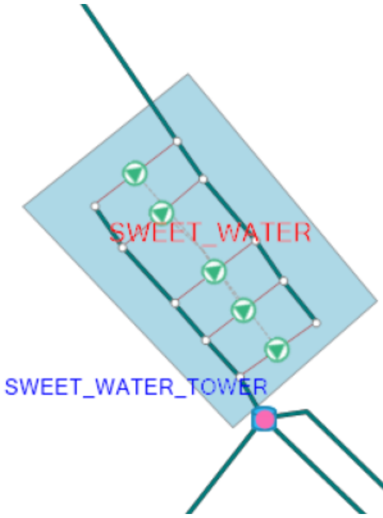


Figure 3.43 A pump station polygon grouping several pumps together

Once the pump station has been inserted, all pumps to be included in this station need to be added using the 'Insert' button. They can be removed using the 'Delete' button.

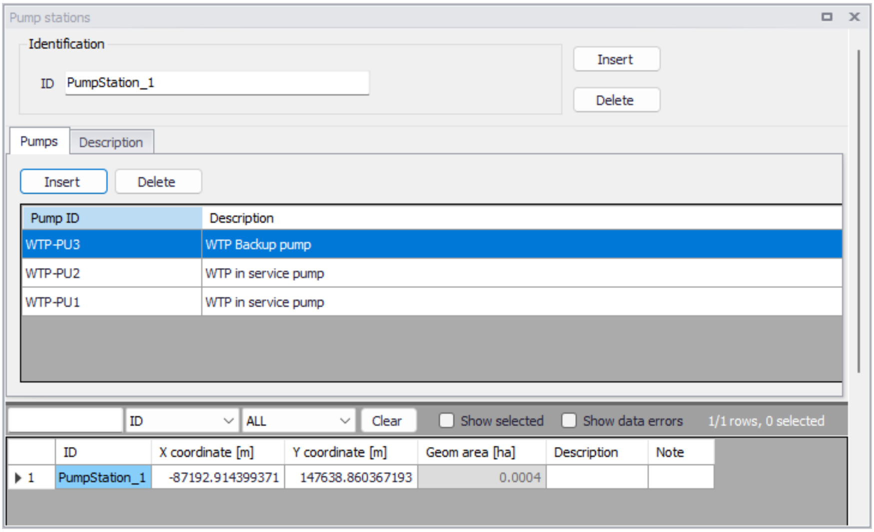


Figure 3.44 Listing the pumps belonging to a pump station

From the 'Description' tab, it is possible to add optional text descriptions of the station, and attach a picture of the station.

When running a 'Cost analysis' special analysis, its report will contain a summary per pump station. Note that, in this report, the Efficiency and the Energy/volume for the pump station are computed using only the pumps which are actually used during the simulation (i.e. with a utilization higher than 0% of the time).

3.7 Turbines

A turbine is a type of rotating equipment designed to remove energy from a fluid. For a given flow rate, turbines remove a specific amount of the fluid's energy head. Each turbine is mechanically coupled with a generator that converts rotational energy to electrical energy. Each generator's output terminal transmits electricity to the distribution grid.

Turbines are either defined interactively on the graphical Map window using the Add Turbine tool (see Figure 3.45), or by manual data entry using the Turbine Editor dialog box as shown in Figure 3.47

The Turbine Editor allows you to define the turbine ID, location, properties, energy generated, regulation and description. The Turbine Editor dialog box is reached by clicking **Turbines** in **Network** under Setup tree (see Figure 3.46).

Figure 3.45 The Turbine Editing Tool

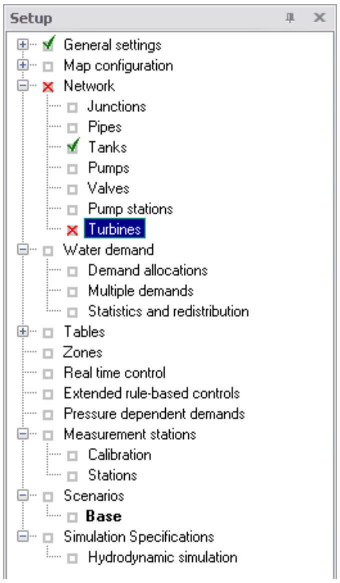


Figure 3.46 The Turbine editor in Setup Tree

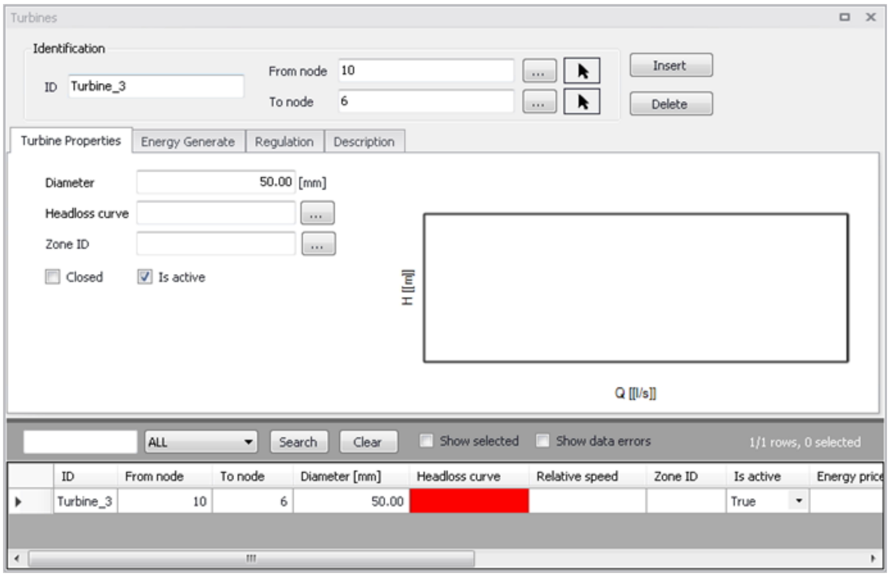


Figure 3.47 The Turbine editor allows the user to define the storage tank node that supply water to the water distribution network

A list of the Turbine editor data entries for Figure 3.46 follows, with a short description given for each entry.

Identification

Turbine ID (mandatory)

This data entry is used to specify an ID which uniquely identifies the turbine link. The turbine ID acts as a unique lookup key that identifies the link from all other links. A link can be a pipe, turbine, valve or turbine. Therefore, no two links may have the same ID. The check would be instant, when the user types an ID already used, there will be a hint beside the field and the user would not be able to type anything else (see Figure 1.4).

However, a link and a node (i.e., junction, reservoir, or tank) can have the same ID. The link ID value can be any string value (up to 40 characters).

A new turbine ID is automatically suggested by MIKE+ whenever a new turbine is placed into the list by pressing «Insert». When defining the turbine graphically on the Map window using the Add Turbine tool, the turbine ID is automatically defined.

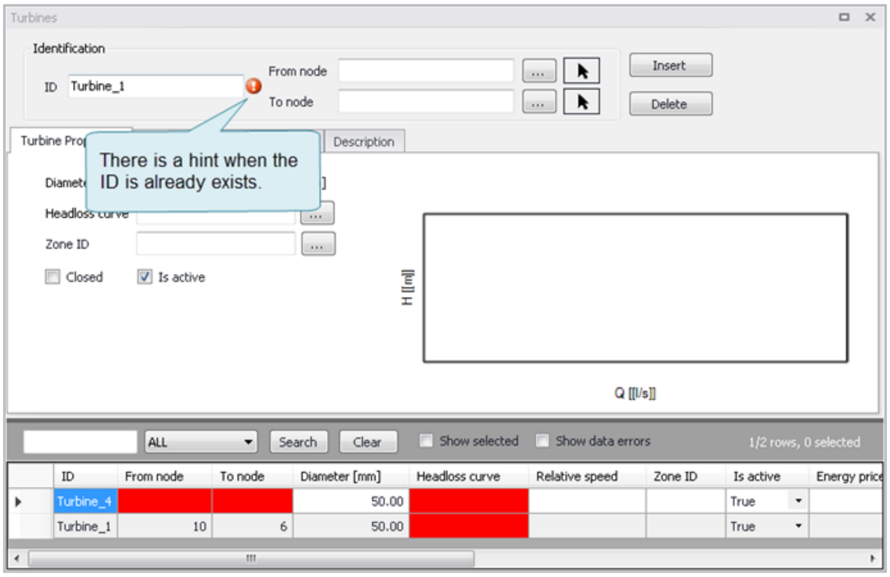


Figure 3.48 Hint when a turbine ID is repeated.

FROM NODE, TO NODE (mandatory)

These data entries define the ID of the turbine's starting (upstream) and ending (downstream) nodes. These IDs define the turbine connectivity of the network.

Clicking , ID selector window pops up and the pull-down selection list allows to specify what type of node is connected to the pump. The type selection is either Junction or Tank. Then the user can choose the appropriate node on the list below. Clicking , it allows the user to graphically select the node from the Map window and the connection of pipe will be changed simultaneously on the map.

Turbine flow is always assumed to move from the starting (upstream) node to the ending (downstream) node.

Turbine Properties

It contains input fields for Turbine Properties, Energy Generated, Regulation and Description. Detailed information of each section is shown below.

Turbine properties

This tab gives basic information of turbines, as shown in Figure 3.49.

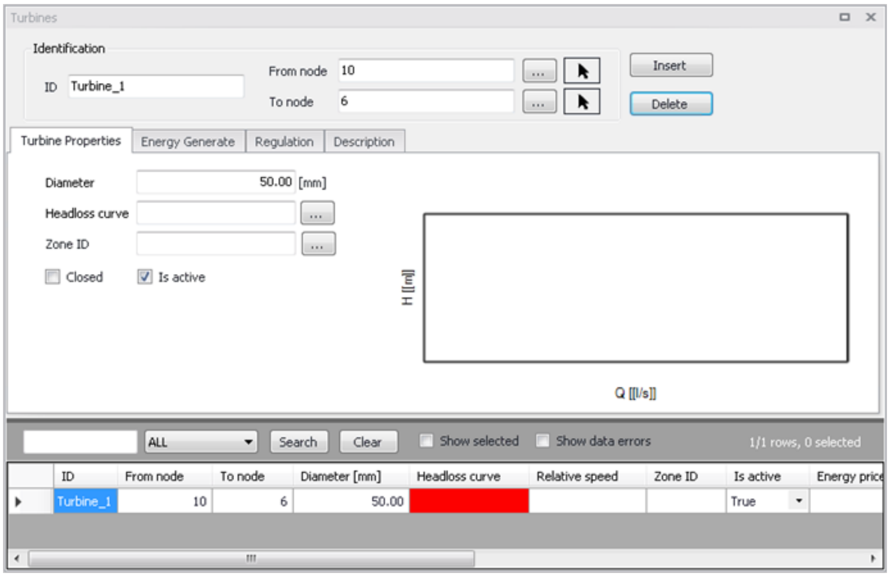


Figure 3.49 The General Information Turbine

Diameter

This data entry allows you to define the Turbine size.

Headloss Curve

A Headloss Curve is used to describe the headloss (Y in feet or meters) through a turbine as a function of flow rate (X in flow units). Clicking , headloss curve list pops up and it allows users to specify the headloss curve. The headloss curve would be plotted on the right after it defines.

Zone ID

This is an optional name for the zone to which the turbine belongs. When a zone ID is specified, this zone will be listed in the 'Zones' editor. The '…'

button can be used to select an existing zone.

Is active

It defines whether the turbine is active or not. If the turbine is active, it will be included in the simulations, otherwise it will be omitted.

Energy Generate

This tab defines the parameters for calculating how much energy or money each turbine can generate from the generator, as shown in Figure 3.50

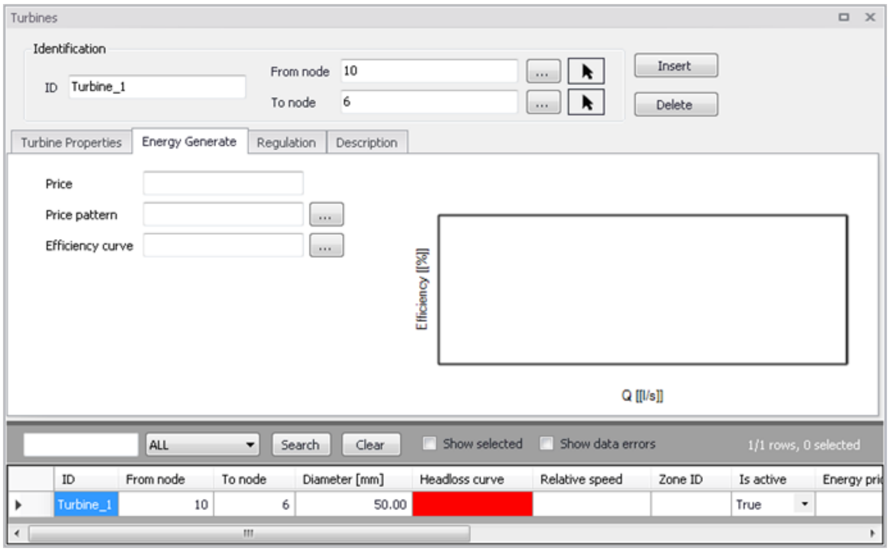


Figure 3.50 The Energy Generated of Turbines

Price

The user defines an energy price ($/kw-hour) to be used. In this method, MIKE+ determines the energy generated by the turbine in kw-hours and multiplies the energy production by the price.

Leave blank if not applicable or if the global value supplied with the Parameters in Cost Analysis will be used.

Price Pattern

It allows engineers to specify a multi-step tariff to describe the variation in energy price throughout the day. The multi-step tariff is stored as a pattern, each multiplier in the pattern is applied to the pump's Energy Price to determine a time-of-day pricing for the corresponding period.

Leave blank if not applicable or if the global pricing pattern specified in the project's Energy Options will be used.

Efficiency Curve

It allows engineers to specify a curve that used for turbine efficiency (η) as a function of flow rate (Q). This curve is used to calculate the electrical energy that turbine can extract from the water flows. The function is stated below:

$$P = \eta \, \rho \, Q \, g \, h$$

Where:

P is power in watts

η is the dimensionless efficiency of the turbine

ρ is the density of the water in kilograms per cubic metre

Q is the flow in cubic metres per second

g is the acceleration due to gravity in meters per square second

h is the height difference between inlet and outlet in meters

The curve is created in curves and relations table (see Curve and Relations section), and it could be edited through a button [...].

Regulation

This tab defines the regulation of turbine to control their operation. Turbines may change their state as storage tanks fill and empty, or pressure change throughout the network system. There are only two states of a turbine: open or close (see Figure 3.51).

For each turbine, it can create a control table to add or delete the regulation statements. The regulations would be shown in a graph next to the table.

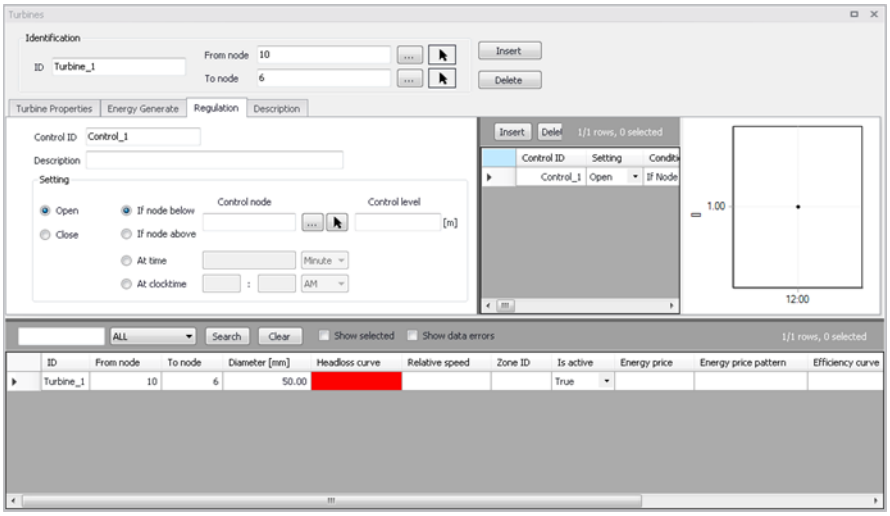


Figure 3.51 The Regulation of Turbines

A list of the data entries for Regulations follows:

Control ID
The ID of the control.

Description
An optional description of the control.

Setting
The settings contain three parts:

- Action
- Condition type
- Condition

A radio button is used to set an **Action**. A turbine can only be set to Open or Close.

A radio button is used to set the **Condition type**, i.e. the type of condition that will trigger the action.

- If node below/above: This rule will execute the action if the pressure level in a specified node is above or below a specified level.
- At time: This rule will execute the action when the specified amount of time since simulation start has passed. When setting up a series of these rules there will be a time series of the setting in the right window.
- At clocktime: This rule will execute the action every day at the specified time.

The available Condition settings will depend on the selected condition type:

- When "If node below/above" is selected, the user must specify a node or tank ID in the first field and the threshold pressure level in the second field. Note that this is defined as the pressure at Elevation level for a node, and the pressure at Base elevation for a tank.
- When "At time" is selected, the user must specify a number and a time unit (hours/minutes) since start of simulation.
- When "At clocktime" is selected the user must specify a time of day in hours, minutes and AM/PM.

Description (optional)

This data entry allows you to enter a description identifying the control rule being defined. This description can be optionally included in reports generated in MIKE+ .

Setting (mandatory)

This radio button selection entry is used to specify the OPEN or CLOSED status of the turbine being controlled. There are four types of control condition that applies the operational rule onto the turbine being controlled.

If the user selects either IF NODE BELOW or IF NODE ABOVE, then a Control Node ID and a Control Level must be specified. Choosing [...] will display the Select Node selection dialog box from which the user can select the appropriate node type and ID. Or, choosing [pointer] allows the user to graphically select the node from the Map window. Note that reservoirs are not allowed to be selected as a Control Node type.

If a junction node is selected as the controlling node, then a trigger pressure at the junction node must be specified in the Control Level data entry. If a storage tank node is selected as the controlling node, then a trigger level (not elevation) must be specified in the Control Level data entry.

If the user selects AT TIME, then a trigger time (since the start of the simulation) must be specified in the adjacent data entry field and a time unit selected from the pull-down selection list.

AT CLOCK TIME allows you to specify a trigger time, which periodically repeats each day, such as at 10.00 a.m., for example.

Description

This data entry allows you to enter a description identifying the turbines being entered. This description can be optionally displayed on the Map window and in reports generated by the Report Generator.

Data Source (optional)

This data entry is used to specify a corresponding asset data source, which uniquely identifies the turbine location (such as database table or a database file name) in the asset management system.

Asset ID (optional)

This data entry is used to specify a corresponding asset turbine ID, which uniquely identifies the turbine in the asset management system (such as GIS, for example).

Status (optional)

This drop down selection list data entry allows you to define whether the turbine is imported (i.e existing link was imported from the external data source), or is inserted, modified, GIS, calibrated or similar. By default, turbine status is undefined.

Add Picture

The <Add picture> button allows users to add photo for individual item. Once loaded from external source, the picture will be displayed on the right section in Figure 3.52.

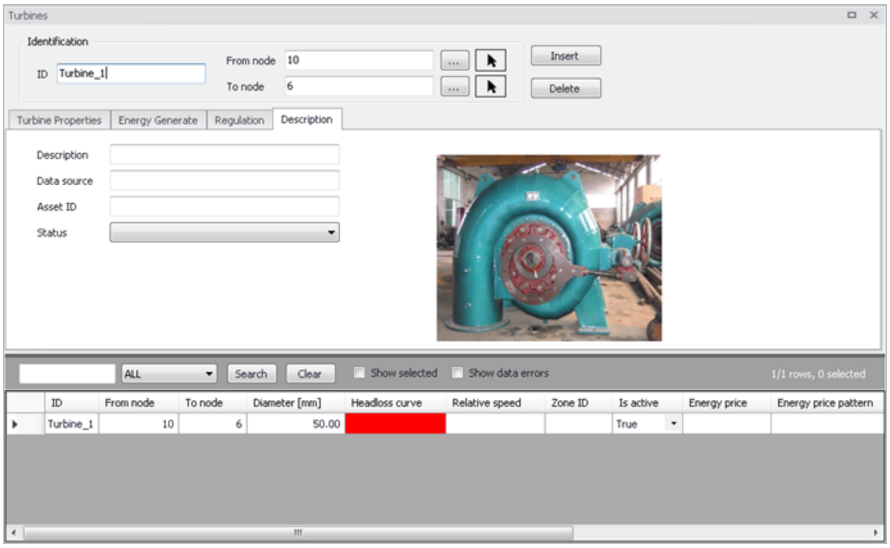


Figure 3.52 The Description of Turbines

3.8 Air-chambers

Air-chamber nodes are placed at points in the water distribution model where an air-chamber tank is located. Please note that air-chambers are used in surge protection and that they are part of the water hammer module in Special analyses. They have no hydraulic function during the EPANET-based simulations and they are treated as dead-end nodes. Air-chambers are described by their volume and the initial water level that defines the ratio between the water and pressurized air. Air-chamber nodes are either defined interactively on the graphical Map window using the Add Air-chamber tool (see Figure 3.53), or by manual data entry using the Air-chambers editor as shown in Figure 3.55.

Figure 3.53 The Air-chambers editing tools

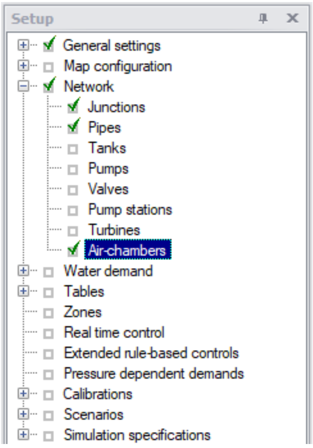


Figure 3.54 The Air-chambers editor is accessed from the Setup tree

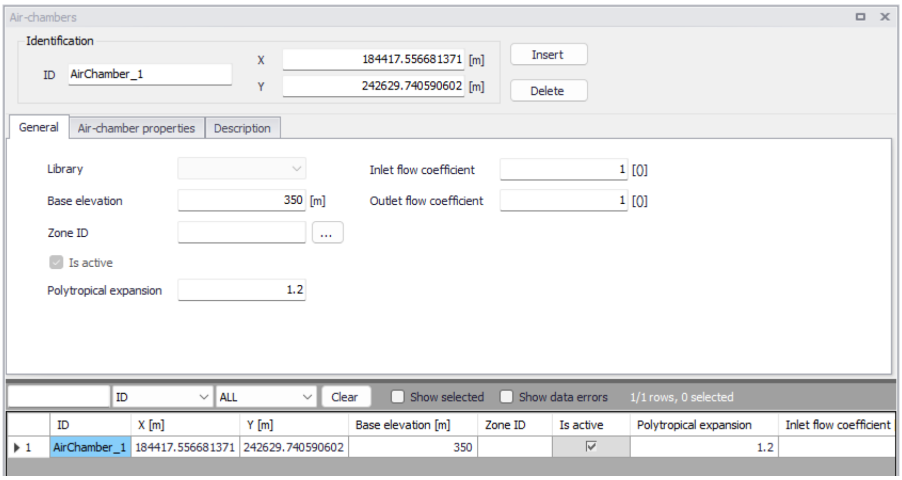


Figure 3.55 The Air-chambers editor allows the user to define the air-chamber nodes that supply water to the distribution network

The Air-chambers editor contains input fields for geometry, air-chamber's properties, and description.

A list of the Air-chambers editor data entries follows, with a short description given for each entry.

Identification

ID (mandatory)

This is used to specify an ID which uniquely identifies the air-chamber node. The air-chamber ID acts as a unique lookup key that identifies the node from all other nodes. A node can be a junction, reservoir, tank, or air-chamber. Therefore, two nodes cannot have the same ID. The ID can be any text up to 40 characters.

A new air-chamber ID is automatically suggested by MIKE+ whenever a new air-chamber node is created.

X and Y coordinates

The X and Y data entries are used to define the physical (map) location of the Air-chamber node. Their unit is controlled by the unit system used in the project.

General

This tab gives general information of air-chambers as shown in Figure 3.56.

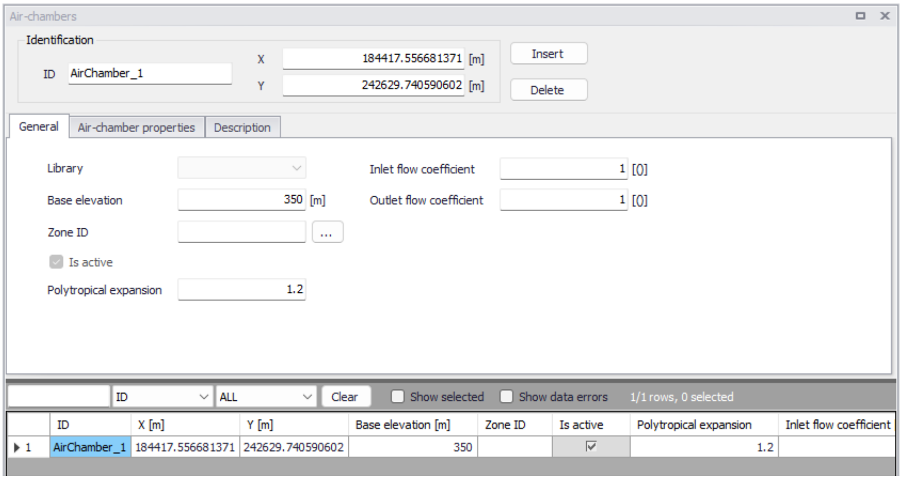


Figure 3.56 The General tab of the Air-chambers editor

Base elevation (mandatory)

Base elevation defines the distance from bottom of the air-chamber above datum elevation.

Zone ID (optional)

This is an optional name for the zone to which the air-chamber belongs. When a zone ID is specified, this zone will be listed in the 'Zones' editor. The '…' button can be used to select an existing zone.

Is active

This check box controls whether the air-chamber will be included (when ticked) or omitted (when unticked) in the simulations. The air-chamber is automatically omitted as soon as all connected links are also set to inactive.

Polytropic expansion (mandatory)

Polytropic expansion is the exponent in the polytropic gas equation (value 1.0 for an isothermal expansion, value 1.4 for adiabatic expansion) with the default value of 1.2.

Inlet flow coefficient

It represents the flow coefficient τ applied to the inlet pipe when the water flows into the air-chamber.

Outlet flow coefficient

It represents the flow coefficient τ applied to the inlet pipe when the water flows from the air-chamber.

These two flow coefficients are used in the head loss calculation as follows.

Headloss, dH:

$$dH = \xi\frac{v^2}{2g} \tag{3.2}$$

where v is the flow velocity and where ξ is the minor loss coefficient

$$\xi = \frac{1}{\tau^2} - 1 \tag{3.3}$$

Air-chamber Properties

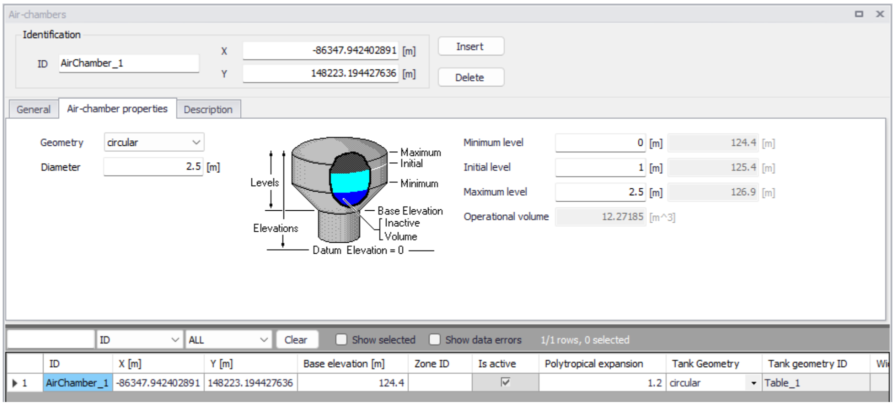


Figure 3.57 The Air-chamber properties tab

Geometry (mandatory)

This drop-down selection list selects the type of air-chamber being defined:

- Table
- Rectangular
- Circular

For different types of air-chambers, the required geometry data is different. By default, a circular air-chamber is defined. The 'Table' option is used for air-chambers whose cross-sectional area varies with height.

Diameter (mandatory)

The diameter of the air-chamber, when the geometry type is set to circular.

Length and width (mandatory)

The size of the air-chamber, when the geometry type is set to rectangular.

Geometry ID(mandatory)

When the geometry type is set to 'Table', the geometry ID holds the ID of a table containing the elevation-volume relationship. This table has to be defined in the 'Curves and relations' editor, with table type 'Tank depth-volume'. This relationship determines how the air-chamber's volume varies as a function of water level. The lower and upper water levels supplied in the table must contain the lower and upper levels between which the air-chamber operates.

Minimum level (mandatory)

This defines the minimum level (or depth), that the water can drop to within the air-chamber. The corresponding elevation is equal to the base elevation plus the minimum level.

Initial level (mandatory)
This defines the initial water surface level (or depth), that is used at the start of the simulation. The corresponding elevation is equal to the base elevation plus the initial level.

Maximum level (mandatory)
This defines the maximum level (or depth), that the water can rise to within the air-chamber. The corresponding elevation is equal to the base elevation plus the maximum level.

Operational volume
This volume is derived from the above geometrical properties of the air-chamber, and represents the usable volume, which can be filled or released from the tank.

For circular and rectangular air-chambers, the volume equals the horizontal area of the air-chamber times the difference between its maximum and minimum water levels.

For air-chambers defined using a geometry table, the operational volume is the difference between the volume of the table at the maximum level, and the volume of the table at the minimum level. Volumes at the maximum and minimum levels are obtained using a linear interpolation from the table's data.

Description

Description (optional)
This allows providing a description of the air-chamber node. This description can be optionally displayed on the Map window and in reports generated by the Report tool.

Data source (optional)
This is used to specify a corresponding asset data source, which uniquely identifies the air-chamber location (such as database table or a database file name) in the asset management system.

Asset ID (optional)
This is used to specify a corresponding asset air-chamber ID, which uniquely identifies the air-chamber node in the asset management system (such as GIS, for example).

Status (optional)
This drop-down selection list allows you to define whether the air-chamber node is imported (i.e existing node was imported from the external data source), or is inserted, modified, GIS, calibrated or similar.

Add picture

The 'Add picture' button allows to add a photo of the air-chamber. Once loaded from external source, the picture will be displayed on the right section of the editor.

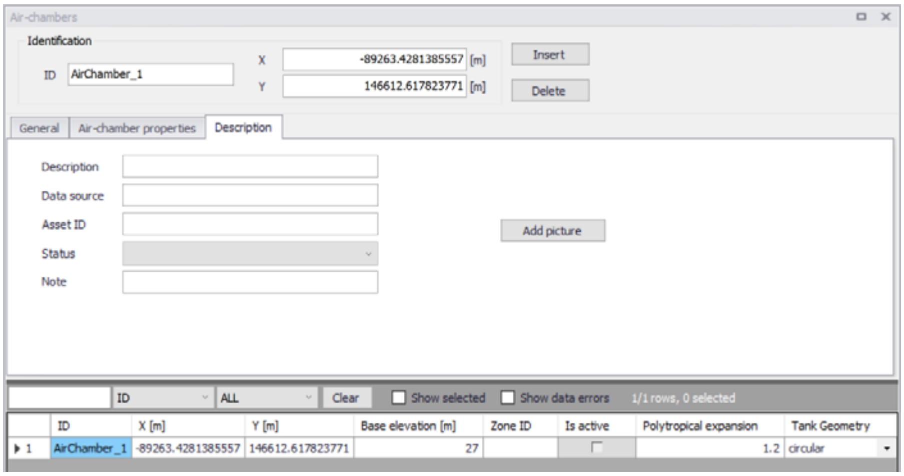


Figure 3.58 The Description tab of the Air-chambers editor

Note: Air-chambers are used in water hammer analyses. Refer to chapter "16 Water Hammer *(p. 207)*" for more information.

4 Water Demand

4.1 Network Demand

Network demand for water is assigned at junction nodes, on a node by node basis. To help develop a model, MIKE+ allows the user to automatically define the nodal demand at all of the nodes within a model, or within a pressure zone, based upon the total demand on the system or pressure zone. This section discusses how MIKE+ can automatically distribute this demand to the network system.

Typically in large network systems, the pipe network is broken up into different pressure zones (or distribution zones). Since pressure is related to ground elevation, a network system covering hilly or mountainous terrain will have more pressure zones than one covering fairly flat terrain. The section also discusses how MIKE+ defines pressure zones.

4.1.1 Zones

This editor can be used to define any type of zone. For example, pressure zones are service areas defined by the hydraulic grade-line value of the sources that supply them. A pressure zone has one or more sources of supply and may have a set of closed valves that separate it from other pressure zones.

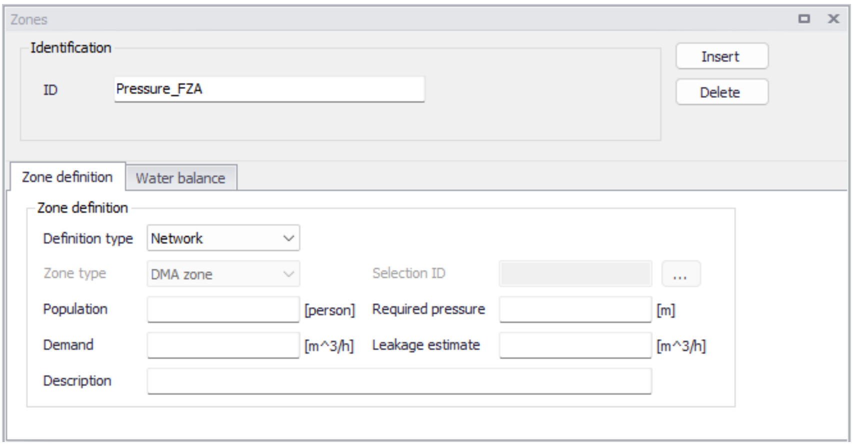


Figure 4.1 The Zones editor

Identification ID

This is the unique identifier of the zone.

If this ID is changed for a zone with a 'Network' definition type, then the Zone ID is also updated in the properties of all the network elements contained in this zone.

Insert

Pressing this button adds a new zone to the list.

Delete

Pressing this button removes the active zone from the list.

Deleting a zone with a 'Network' definition type will also clear the Zone ID specified for the network elements contained in this zone.

Zone definition

Definition type

The spatial extent of zones can be defined in two different ways:

- 'Selection': with this option, the network elements (pipes, junctions, etc.) contained in the zone are listed in a selection. A given network element can belong to several zones defined by selections.
- 'Network': with this option, the network elements defining the zone are identified by the Zone ID specified in their corresponding editors. For example, a pipe is included in a specific zone only if this Zone ID is specified in the pipe editor. A given network element can belong to only one zone defined by network properties. Network zones are synchronized with the network editors: when a new zone ID is specified for a network element, it will be listed in the list of zones; and when a zone ID is removed from all network elements, it will be removed from the list of zones.

It is recommended to use automatic 'Network' type of zone for "permanent" use in the hydraulic model, when building the model or importing GIS data. 'Selection' type of zone is suitable for testing or ad-hoc purposes when performing tests.

Zone type

This is only used for zones defined by a selection. The zone type is used to identify the distributing demand by the kind of zone established. There are five types predefined in MIKE+: DMA zone, pressure zone, demand zone, region zone, ward and other.

Selection ID

This is only used for zones defined by a selection. This data entry is used to load a previously defined selections of Network elements such as nodes and pipes for which the demand zone will be delimited.

The user can create these selection lists with the selection tools.

Demand

This data entry is used to specify the zone demand, which can be used for automatic demand distribution. To distribute the zone demand to junction nodes located within the zone, use Demand Allocation editor.

Population

This data entry is used to specify the population per zone. It is mostly used as reference data when the user define zone demands but it has no impact on the calculated demands per zone.

Required pressure

This field is used for the network energy calculations, presented in the log file and report from cost analyses. It is the required service pressure that is considered to be optimum for the pressure conditions within the zone.

This field is used to evaluate the zone energy efficiency.

Leakage estimate

This field is used for the network energy calculations, presented in the log file and report from cost analyses. It is the percentage of water consumption that refers to the leakage within the zone.

This field is used to determine energy lost due to leakages.

Description

This data entry allows you to enter a description identifying the pressure zone being defined. This description can be output in reports generated by the Report tool.

Water balance

The 'Water balance' tab can optionally be used to associate the zone with flow measurement stations, which can then be used to analyse the water consumption within the zone. The selected measurement stations should typically be located on the borders of the zone, in order to monitor the flow entering or leaving the zone. It can typically be used to:

- Estimate the volume consumed in the zone, as reported in this 'Water balance' tab.
- Estimate and assign the 'Demand' and 'Leakage estimate' values from the 'Zone definition' tab. This is achieved with the 'Compute zone demand and leakage tool'.
- Create patterns of the water demand in the zone. This is achieved with the 'Create time patterns tool'.

Time range for water balance

This controls the time period used for all water balance computations, including for the use of the 'Compute zone demand and leakage' and 'Create time patterns' tools.

Per default, the longest period covered by all flow measurements included in the zone, is used. This default period is shown at the top of the 'Time range for water balance' group. It is updated when flow measurements are added or removed from the zone.

Alternatively, it is possible to restrict the period of analysis to a shorter period. To do so, activate the 'Use custom time range' option and specify the custom start and end dates to be used.

Flow measurements

The flow measurement stations associated to the zone must be selected in this table.

Note that measurement stations are selected from the list of calibrations comparisons, defined in the 'Plots and statistics' editor. It is therefore a pre-requisite to select the measured time series in the 'Plots and statistics' editor (although no result file is needed) prior to selecting them for the zone water balance.

Click the 'Insert' button to select a measurement station to add to the zone, from the 'Plots and statistics' list. Click the 'Delete' button to remove it from the list.

For flow measurement station, the table provides the following information and control:

- Plot ID: the ID of the plot, as specified in the 'Plots and statistics' editor.
- Unit: the unit of measured flow time series, as specified in the 'Plots and statistics' editor.
- Positive flow: this indicates whether the positive values in the time series represent water flowing into the zone or leaving out of the zone. Time series marked as "Into zone" are added to the water balance, while time series marked as "Out of zone" are subtracted from the water balance.
- Volume: this column shows the computed accumulated volume from the time series. The period used to compute this accumulated volume is the one specified in the 'Time range for water balance' group.

Note that two types of flow measurement data can be selected in the 'Plots and statistics' editor:

- Measured discharge / flow time series, providing instantaneous values in flow units (e.g. l/s or gal/min).
- Measured volume time series, providing values of volume per time step, in volume units (e.g. liter or gallon).

Volume balance

The 'Volume balance' value shown above the table is the estimated volume consumed in the zone, during the period of the analysis.

It is computed as the sum of all volumes from measurement stations flowing into the zone, minus the sum of all volumes from measurement stations flowing out of the zone.

4.1.2 Demand allocation

An alternative way to generate junction node demands can be developed based on connecting the consumption data to the appropriate nodes or pipes and aggregating their set point demand values to the junction demands. This simplifies the demand development process and allows to import consumption data from the consumption database systems and connect it to junctions based on X, Y geographical coordinates.

The Demand Allocations editor is reached from the TOC under the Water demand layer group. It is used to store and edit the consumption data defined as consumption points, so that they can be connected to the appropriate network junctions or pipes.

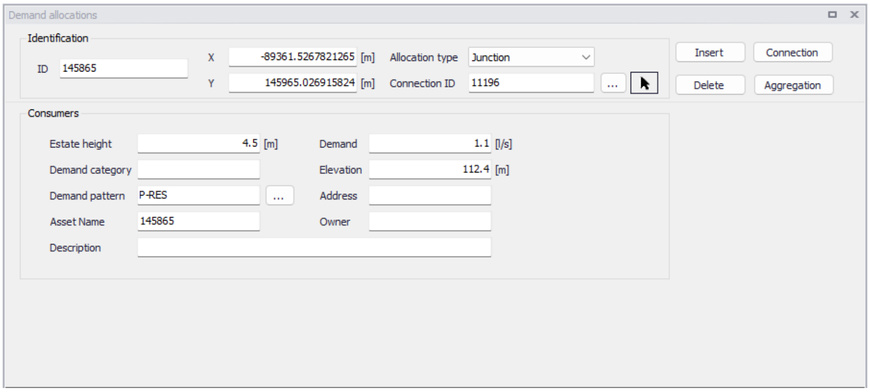


Figure 4.2 Demand Allocations editor

A list of the Demand Allocations parameters follows, with a short description of each entry.

ID

This data entry is used to specify an ID which uniquely identifies the demand point. The ID acts as a unique lookup key that identifies the demand point from all other demand points. The ID value can be any string value (up to 40 characters).It is recommended that this reference ID corresponds to the asset ID, which uniquely identifies the demand point in the customer information or billing database system.

XY coordinates

The geographical coordinates of the demand point.

Allocation type
This is used to specify if the connected demand is loaded to either a junction or a pipe.

Connection ID
The ID of the junction or pipe to which the demand is connected.

Demand
This data entry is used to specify the demand value, which will be then used in the process of the demand aggregation. This demand set point value can be imported from the external database systems (such as CIS Customer Information System, for example) or it can be developed from the minimum, average, or maximum demand values.

Elevation
This data entry defines the elevation above a common datum for the demand allocation point, in units of feet or meters. The default value is zero.

Estimated height
The estimated heigh is used for the computation of service pressure. The purpose of this field is to store the information used for service pressure calculation. The service pressure is commonly defined as pressure above the roof of the house or building. Hence service pressure will allow the user to store the information about such height. (In the results items this pressure will be computed as “Pressure” - “Service height”).

Description
This data entry allows you to enter a description identifying the consumption point defined. This description can be output in reports generated by the Report tool

Demand Category
This data entry is used to specify the demand category such as residential, commercial, or leakage, for example. The demand category can be then used in the process of demand aggregation when demands belonging to the same junction node or a pipe are aggregated based on their demand category.

Address
This data entry alllows you to enter an address identifying the consumption point being defined. This field can be output in the reports generated by the Report tool

Demand Pattern
This data entry allows you to define the ID of the demand pattern to be applied to the junction node demand values during an extended period simulation.

This demand pattern will be applied to the defined baseline demand. If a groundwater well is associated with this node, then a demand pattern should not be assigned—otherwise the groundwater extraction rate will be adjusted according to the assigned demand pattern. By default this data entry is blank and default demand pattern ID of 1 is assigned.

Asset name

The optional name which uniquely identifies the demand point in the customer information or billing database system.

Owner

This data entry allows you to enter an owner name identifying the consumption point being defined. This field can be output in reports generated by the report tool.

Connection button

This button opens the Connection Tool, which allows to connect customer demands to model junctions.

Aggregation button

This button opens the Aggregation tool , which allows to develop total junction demands based on demand connections.

4.1.3 Multiple demand

Junction node demands can be edited either in the junctions editor for each particular node or in the Multiple demand editor, which allows the user to display and edit all multiple demands. The Multiple Demand Editor dialog is opened from Water Demand | Multiple Demands in the Setup tree.

Junction ID (mandatory)

The Junction ID identifies the selected Junctions which multiple demands are assigned to.

Demand (mandatory)

The demand field shows all the values that are assigned to junctions with multiple demands. The demand values must be manually entered in the demand field.

Description (optional)

This data entry allows you to enter a description identifying the multiple demand being defined. This description can be output in reports generated by the reporting tool.

Demand Category (optional)

Pattern category is not editable but is automatically displayed based on the category defined in the pattern Editor for the particular Pattern ID. It is possible to import and export multiple demands from the ASCII text files, which allows easy data exchange with other programs.

Demand Pattern (optional)

This data entry allows you to define the ID of the demand pattern to be applied to the junction node demand values during an extended period simulation.

This demand pattern will be applied to the defined baseline demand. If a groundwater well is associated with this node, then a demand pattern should not be assigned—otherwise the groundwater extraction rate will be adjusted according to the assigned demand pattern. By default this data entry is blank and default demand pattern ID of 1 is assigned.

Is active

This check box allows the user to toggle the Active status of the demand on and off. The simulations will omit all demands that are not active. Demands which are not active are also omitted in the ‘Statistics and redistribution’ table.

4.1.4 Statistics and redistribution

MIKE+ can generate statistical information for junction node demands. Demand statistics can be computed for each pressure zone as well as for the complete network. Additionally, this editor allows the user to redistribute node demands by changing the calculated statistical results.

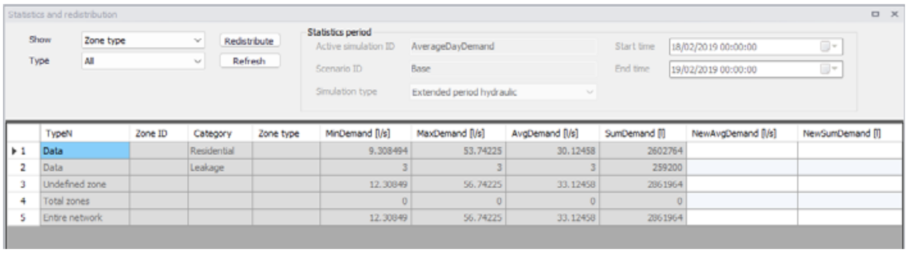


Figure 4.3 Statistics and Distribution

When the average demand for the 'Total zones' and the 'Entire network' are different, the corresponding values will be highlighted in the table. This can indicate that some parts of the network are included in multiple zones, or that some are not included in any zone. Users should investigate the cause of this difference, and check if the result is acceptable or not. Having parts of the network included in multiple zones may e.g. be a problem while redistributing the demand.

A list of the Statistics information for the above figure follows, with a brief description given for each item.

Show

This list located above the table controls whether the table show data for zones with a 'Network' definition type only, for zones with a 'Selection' definition type only, or for the entire network.

Type

The 'Type' list above the table controls the type of data shown in the table:

- When showing network zones, the type controls whether the table processes links (i.e. pipes, pumps, valves and turbines) or nodes (junctions, tanks and air-chambers)
- When showing selection-based zones, the type controls the type of zone (DMA zone, pressure zone, demand zone, region zone, ward, other).

Statistics period

Statistics shown in the table are computed for the settings (especially Start and End times) defined in the active simulation, in the 'Simulation setup' editor. These simulations settings are shown in the 'Statistics period' group, for clarity.

TypeN

This column in the table is used to distinguish between *data* (demand statistics for the selected category), *zone* (demand statistics for pressure zone), and *network* (demand statistics for the whole network).

Zone

Zone ID identifies the zone for which the demand statistics is generated.

Category

This data entry identifies the category within the current pressure zone for which the demand statistics is generated.

Minimum Demand

This data entry represents the minimum demand per category per zone. The minimum demand is calculated as minimum demand at junction nodes at specific time level.

Maximum Demand

This data entry represents the maximum demand per category per zone. The maximum demand is calculated as maximum demand at junction nodes

Average Demand

This data entry represents the average demand per category per zone. The average demand is calculated as average demand at junction nodes during the entire simulation.

Sum Demand
This data entry represents the total demand per category per zone. The total demand is calculated as total demand at junction nodes during the entire simulation period.

New Average Demand
This data entry allows the user to specify the new total demand for selected category, zone or a network. All corresponding junction node demands will automatically be adjusted (scaled) in order to fit the new total demand value.

New Sum Demand
This data entry allows the user to specify the new total demand for selected category, zone, or a network. All corresponding junction node demands will automatically be adjusted (scaled) in order to fit the new total demand value.

Refresh
Select *Refresh* to re-generate the demand statistics.

Redistribute
Select Redistribute to redistribute node demands based on the new values of Average or Total demand (zone or network). This powerful feature provides the user with the option of specifying the new zone or a network demand from within the Demand Statistics window and redistribute the node demand accordingly. The process of the demand distribution is based on using the existing node demands as coefficients - weights to calculate the new demand values.

4.1.5 Compute zone demand and leakage tool

When flow measurement stations are associated with a network zone, in the 'Water balance' tab from the 'Zones' editor, the 'Compute zone demand and leakage' tool can estimate the water demand and/or leakage within the zone. The computed values are then updated in the 'Zones' editor.

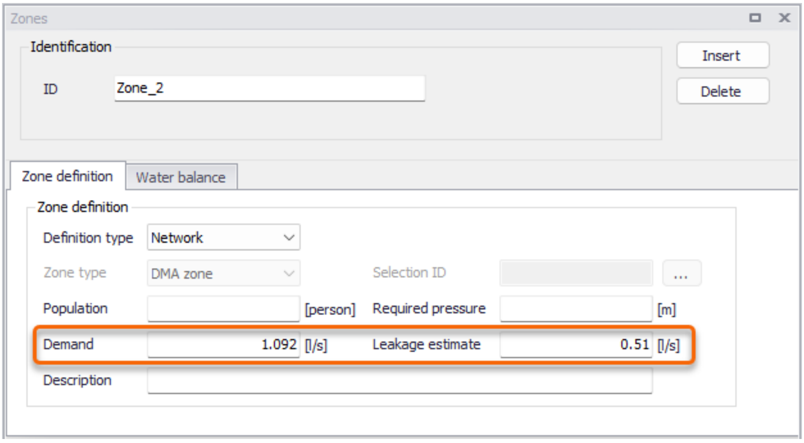


Figure 4.4 Zone demand and leakage values which can be computed with the 'Compute zone demand and leakage' tool

This tool is accessed from the 'WD toolbox' group in the 'WD network' tab of the ribbon, from the list of demand tools.

Note that the tool cannot work with zones defined with a 'Selection' definition type.

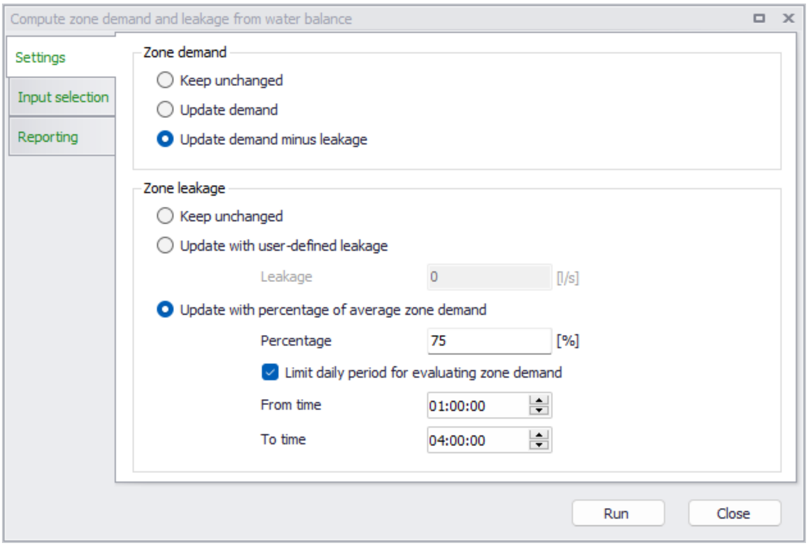


Figure 4.5 The 'Compute zone demand and leakage' tool's settings

Zone demand

The selection from the 'Zone demand' group controls the action performed for the demand:

- Keep unchanged: this does not perform any change to the demand values defined in the 'Zones' editor. It should be selected if the goal is to compute the leakage only.
- Update demand: this updates the Demand value in the 'Zones' editor, with the computed average demand of the zone. This average demand is computed as the accumulated volume of all measurement stations flowing into the zone minus the accumulated volume of all stations flowing out of the zone, divided by the duration of the analysis. The period of time used for the analysis is the period specified in the 'Water balance' tab of the 'Zones' editor, which may vary for each of the processed zones. This analysis period is the longest period covered by all measurement time series associated to the zone, unless a custom period has been specified.
- Update demand minus leakage: this updates the Demand value in the 'Zones' editor with the same method as for the 'Update demand' option, but subtracting the estimated leakage. The method for estimating the leakage is selected in the 'Zone leakage' group.

Zone leakage

The selection from the 'Zone leakage' group controls the action performed for the leakage:

- Keep unchanged: this does not perform any change to the leakage estimate values defined in the 'Zones' editor. It should be selected if the goal is to compute the demand only.
- Update with user-defined leakage: this assigns the specified leakage value to all processed zones.
- Update with percentage of average demand: this method computes a leakage value for each zone, being the specified percentage of the zone demand (as computed with the 'Update demand' method). Two options are available:
  - If 'Limit daily period for evaluating zone demand' is inactive, the leakage is the percentage of the average demand estimated from the entire period of the analysis (specified in the 'Water balance' tab of the 'Zones' editor, for each zone individually).
  - If 'Limit daily period for evaluating zone demand' is active, the method is the same except that the average demand is estimated only for the specified times of the day. This typically allows to estimate the leakage as a percentage of the flow at night.

Input selection

The tool can process either:

- All zones: the demand and/or leakage estimate values are updated for all zones with 'Network' definition type, defined in the Zones editor.

- Selected zones: the demand and/or leakage estimate values are updated only for selected zones with 'Network' definition type.
- Single zone: the demand and/or leakage estimate values are updated only for the selected zone ID.

4.1.6 Create time patterns tool

When flow measurement stations are associated with a network zone, in the 'Water balance' tab from the 'Zones' editor, the 'Create time patterns' tool can compute the pattern of the water demand within the zone. The computed pattern is then saved in the 'Patterns' editor.

This tool is accessed from the 'WD toolbox' group in the 'WD network' tab of the ribbon, from the list of demand tools.

Note that the tool cannot work with zones defined with a 'Selection' definition type.

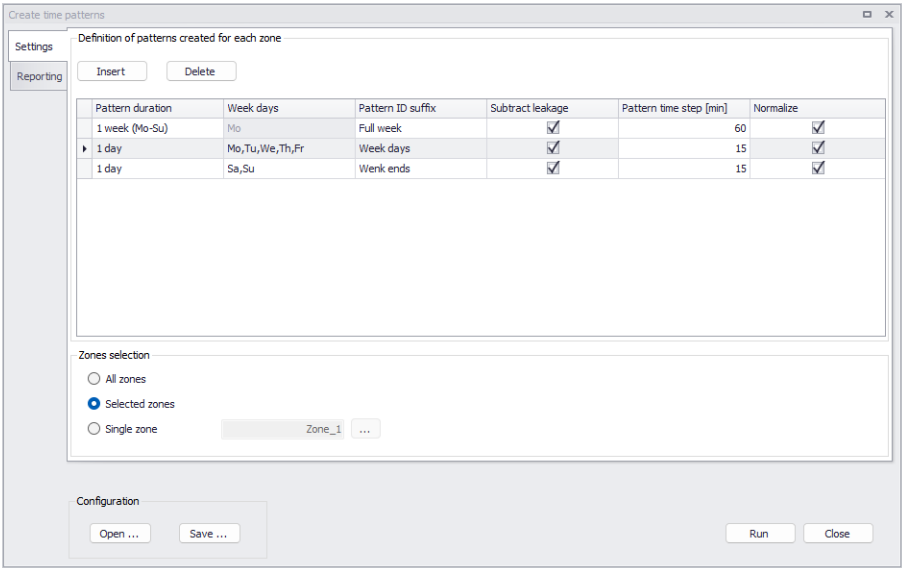


Figure 4.6 The 'Create time patterns' tool's settings

Patterns are derived from the demand in the zone. The total demand in a zone is derived from the flow measurement time series associated with the zone. This total demand is defined as the sum of all time series measuring flows entering into the zone minus the sum of all time series measuring flows out of the zone. Optionally, a leakage can be subtracted from this total demand before computing the pattern, so that the resulting pattern reflects the actual demand only.

Definition of patterns

The upper table in the 'Settings' tab controls the number and the types of patterns to be created for each processed zone. It is therefore possible to create several patterns per zone, typically for different parts of the week.

Click the 'Insert' button to add a new pattern to the list, or 'Delete' to remove the selected one.

For each pattern in the list, the following properties must be specified:

- Pattern duration: this is the duration of the created pattern, which is either one day or one week. In the latter case, several definitions of the week's start/end are offered (Monday-Sunday, Sunday-Saturday, Saturday-Friday).
- Week days: when the pattern duration is set to one day, this list controls which days of the week are taken into account for the analysis. Multiple days can be selected. During the analysis, only the flow measurements from the selected days are used to compute the pattern.
- Pattern ID suffix: created patterns are saved in the 'Patterns' editor. Each pattern is named with the zone ID it refers to, supplemented by the specified suffix.
- Subtract leakage: when this option is inactive, the pattern is computed from the total demand in the zone. When this option is active, the leakage in the zone is subtracted from the total demand time series, before computing the pattern. This subtracted leakage value is the constant 'Leakage estimate' value, specified in the 'Zones' editor. This leakage value must therefore be specified prior to computing patterns. It can be estimated with the 'Compute zone demand and leakage tool'.
- Pattern time step: this is the time step length of the created pattern.
- Normalize: when this option is active, the created pattern provides multiplier values, varying as a function of time. These multipliers represent the demand value at a specific time step divided by the average demand throughout the duration of the pattern. When this option is inactive, the computed values are not divided by the average demand, and therefore represent actual flow values instead of multipliers.

The period of time used for the analysis is the period specified in the 'Water balance' tab of the 'Zones' editor, which may vary for each of the processed zones. This analysis period is the longest period covered by all measurement time series associated to the zone, unless a custom period has been specified.

The different steps performed by the tool to compute a pattern are as follows:

1. Derive the total demand time series from flow measurements associated with the zone, for the period of analysis defined in the 'Zones' editor. This demand time series is defined as the sum of all time series measuring flows entering into the zone minus the sum of all time series measuring

flows out of the zone. Measured time series values are interpolated as necessary to provide values at the specified pattern time steps.

2. Split this demand time series into multiple daily or weekly time series. For 1-week patterns, each weekly time series starts at the specified start day of the week (Monday, Sunday or Saturday), for a duration of seven days. For 1-day patterns, each daily time series covers one of the selected week days. Daily or weekly time series starting before the start of the analysis, or ending after the end of the analysis, are discarded (i.e. partial daily or weekly time series are ignored).
3. Transform all the daily or weekly time series with a relative time axis (i.e. ignore dates) and compute the average time series.
4. If 'Subtract leakage' is active, subtract the constant leakage estimate at all time steps of the average time series.
5. If 'Normalize' is active, compute the average value of the obtained average time series, and divide all time steps' values by their average value.

Zones selection

The tool can process either:

- All zones: requested patterns are created for all zones with 'Network' definition type, defined in the Zones editor.
- Selected zones: requested patterns are created only for selected zones with 'Network' definition type.
- Single zone: requested patterns are created only for the selected zone ID.

Configuration

- Save: saves the current settings in the tool to a file for later re-use.
- Open: loads the settings from a file created during a previous use of the tool.

4.1.7 Distributed demands tool

For large network systems, assigning demand data can be a very tedious job. Since many times the total demand is known for a particular network pressure zone or the entire network system, MIKE+ provides the capability to distribute this total demand among the applicable junction nodes.

This tool is used to automatically assign the demands at the appropriate junction nodes. The Distributed Demands Tool can be accessed from the 'WD toolbox' group in the 'WD network' tab of the ribbon.

Pipe demand coefficients

MIKE+ computes the water demands for each network system based on a user specified water demand, the water demand specified per Zone ID. There are two methods to compute the water demands the *Method of Pipe Lengths* and the *Method of Two Coefficients.* This is useful when assigning the nodal water demand for a large network, since the software will automatically proportion the total network demand based upon one of these two methods. These methods are used to mimic the amount of actual demand along a pipe, based upon the pipe length or a pre-defined demand coefficient.

The Distributed Demands dialog, as shown in Figure 4.7 is used to automatically assign the demands at the appropriate junction nodes. The Distributed Demands tool is reached from the 'Demand tools' button in the 'WD network' ribbon tab.

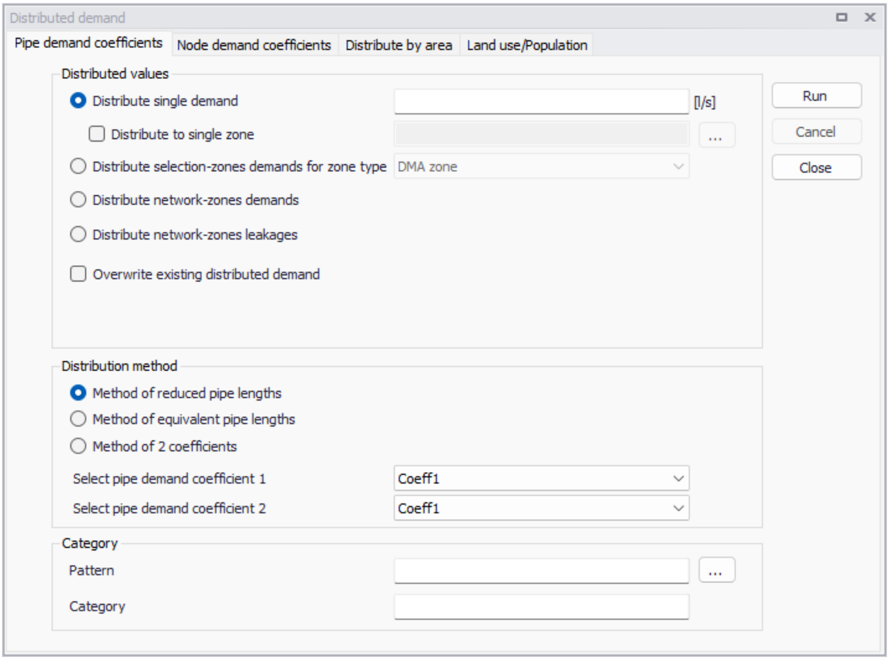


Figure 4.7 Distributed Demands Pipe Demands Coefficients

A list of the entries for the Distributed Demands Tool follows.

Distribute single demand

This option is used to specify the network demand for a particular network pressure zone or the entire network system.

Note that this total demand represents the total demand regardless into which multiple junction demand is distributed. Multiple demands are specified in the junction Editor or in the Multiple Demand Editor.

Distribute to single zone

When this check box is active, the specified water demand value is distributed to a single zone. This zone is selected in the field next to the check box.

The zone ID can also be selected from the list using the '…' button. When this check box is unticked, the specified water demand is distributed to the entire water distribution network.

Distribute selection-zones demands for zone type

This option distributes the specified water demands to zones defined with a selection, and with the zone type selected in the drop-down list. All relevant zones are then listed in the table underneath, where the water demand going to be distributed for each of the zone can be seen. These water demand values are specified in the 'Zones' editor in the 'Demand' field, where they can be adjusted before running the tool, if required.

Distribute network-zones demands

This option distributes the specified water demands to zones defined as network zone (i.e. with the zone ID specified in the network features). All relevant zones are then listed in the table underneath, where the water demand going to be distributed for each of the zone can be seen. These water demand values are specified in the 'Zones' editor in the 'Demand' field, where they can be adjusted before running the tool, if required.

Distribute network-zones leakages

This option distributes the specified leakage values to zones defined as network zone (i.e. with the zone ID specified in the network features). All relevant zones are then listed in the table underneath, where the leakage going to be distributed for each of the zone can be seen. These leakage values are specified in the 'Zones' editor in the 'Leakage estimate' field, where they can be adjusted before running the tool, if required. Leakages are then computed as an additional demand at junctions in the zone.

Overwrite existing distributed demand

When the tool is run, new junction demands are added to the 'Multiple demands' editor, with a 'Mark' indicating that the demand has been computed by the 'Distributed demand' tool.

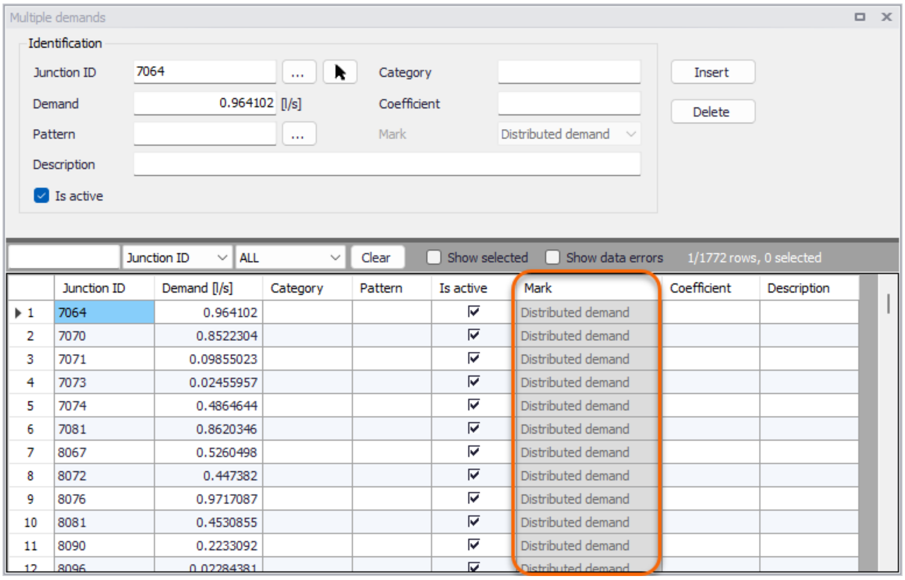


Figure 4.8 Mark added by the 'Distributed demand' tool in the 'Multiple demands' editor

When the ‘Override existing distributed demand’ check box is active, the tool will remove the existing demands / leakages (with the same mark) in the junctions first before distributing the new ones. If unticked, the new computed demands / leakages will be added to existing ones.

Method of Two Coefficients or Method of Reduced Pipe Lengths or Method of Equivalent Pipe Lengths

MIKE+ allows the user to compute the nodal water demands based upon the total network demand using two methods: the *Method of Pipe Lengths* and the *Method of Two Coefficients*.

Selecting the Method of Two Coefficients, MIKE+ computes the total water demand assigned to each pipe (which is then split between the starting and ending nodes) as:

$$q_{pi} = \frac{(Q)k_{1i}k_{2i}}{\Sigma(k_{1i}k_{2i})} \tag{4.1}$$

Selecting the Method of Reduced Pipe Lengths, MIKE+ computes the total water demand assigned toe each pipe (which is split between the starting and ending nodes) as:

$$q_{pi} = \frac{(Q)l_ik_{li})}{\Sigma(k_{li}l_i)} \tag{4.2}$$

Selecting the Method of Equivalent Pipe Lengths, MIKE+ computes the total water demand assigned to each pipe (which is then split between the starting and ending nodes) as:

$$q_{pi} = \frac{(Q)l_i k_{Di})}{\Sigma(k_{Di} l_i)} \tag{4.3}$$

where:

q_{pi} = Total water demand applied to the pipe, split between the two end nodes
Q = total network water
l_i = Pipe length
k_{1i}, k_{2i} = pipe demand coefficients

k_{Di} = pipe demand coefficient is calculated by the program as a factor, calculated as pipe diameter/diameter_normal (where diameter normal is 150mm or 6 inch). This helps to scale the pipes based on their diameter i.e. perimeter, this method is recommended when the distributed demand corresponds to the amount of leakage.

These demand coefficients are defined for each pipe using the Pipe editor. The computed demands, which are assigned once <<Compute>> is selected, are stored at each individual node. There demands are stored in the Junction Editor. Selecting <<Cancel>> will cancel the computation of demand distribution. Selecting <<Close>> will close the tool dialog.

Select Pipe Demand Coefficients 1,2

This list box data allows the user to specify the demand coefficient, which will be used for demand coefficient 1 or 2. There are four possible pipe demand coefficients, which can be defined for each pipe.

Category

This entry let the user select the category type identifying the consumption point being defined.

Pattern

This data entry allows the user to define the ID of the demand pattern to be applied to the distribute demand.

Node demand coefficients

Node demand coefficient allows the user for each node to define the share from the whole network demand, which is taken by that node. The total network demand is then distributed to the corresponding junction nodes by Demand Distribution function.

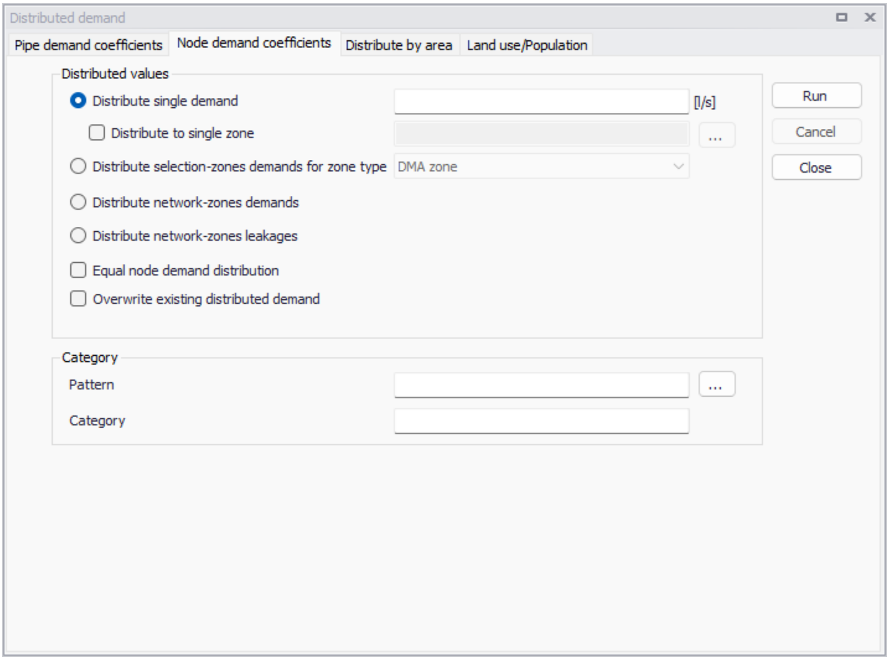


Figure 4.9 Node Demands Coefficients

This option will only assign demand to nodes with Demand Coefficient applied (different from 0 or NULL). In the case of an equal distribution, the node demand coefficients have to be equal and different from zero.

The user can distribute the water demand by node demand coefficient. Most of the distribution settings are identical to those used for the Pipe demand coefficients method (page 99).

Equal node demand distribution

This check box allows the user to distribute the network (or zone) demand equally to each node within the zone or network

$$q_{ni} = \frac{Q}{N} \tag{4.4}$$

Where:

Q = Total network water demand (or zone demand)
q_{ni} = calculated demand at each junction node
N = junction nodes count with the selected zone or a total network

Pattern

This data entry allows the user to define the ID of the demand pattern to be applied to the distribute demand.

Category

This entry let the user select the category type, which will be used as a target demand for the distributed demand. If the multiple demand with the specified category does not exist, the program will create it and it will override the existing values in case such demand category already exist for each node used in the demand distribution.

Distribute by area

Distribute by Area allows the user to distribute demands by the ratio of service area of each node. The service area can be defined by Thiessen polygon method or other external source of data, i.e: a feature shape file. External layers needs to be imported to the map previously to be used by the tool.

The user will specify the water demand to be distributed amongst the service areas through the "*Total network water demand*".

Service Area Layer

This field will points at the shape files loaded to the map that can be used as area layers or the generated Thiessen polygon layers.

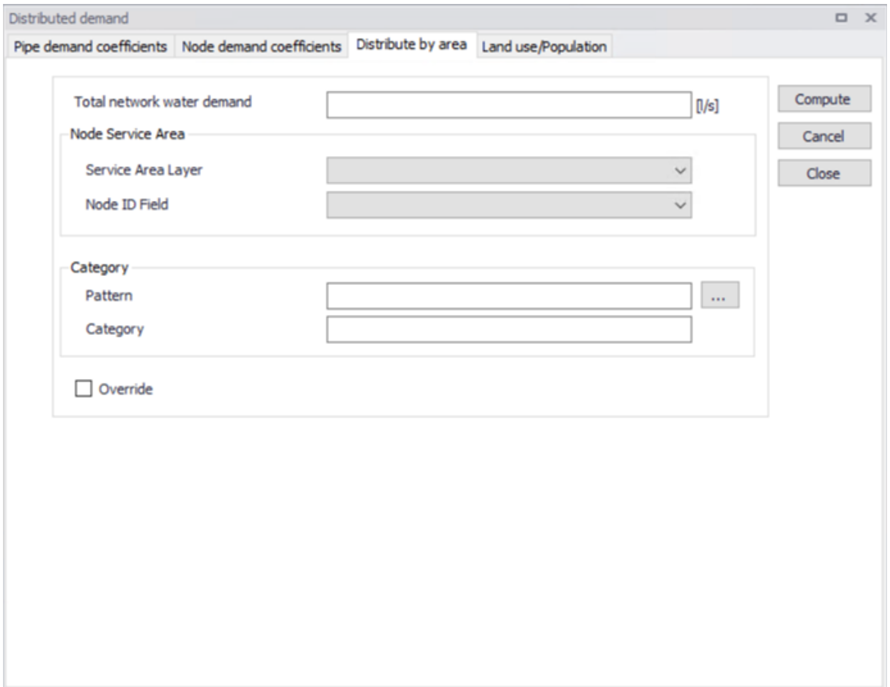


Figure 4.10 Distributed Demands Area

The Node ID field defines the Junction ID for each area. The pattern and category can also be specified here.

Land use/Population

The water demands can be distributed by means of the Land use/Population specified for the service area of each node. Similarly to the distribution by Area the user can use Thiessen Polygons or external layers such as feature shape files.

Selecting "Land use" will enable the user to select an external layer of population and the Type Fied.

The user must define the unit demand of each type in the "Use Type" table, this data regards the water demand per hectare (ha) per day. Further the engine will compute the demand for each node and distribute to them.

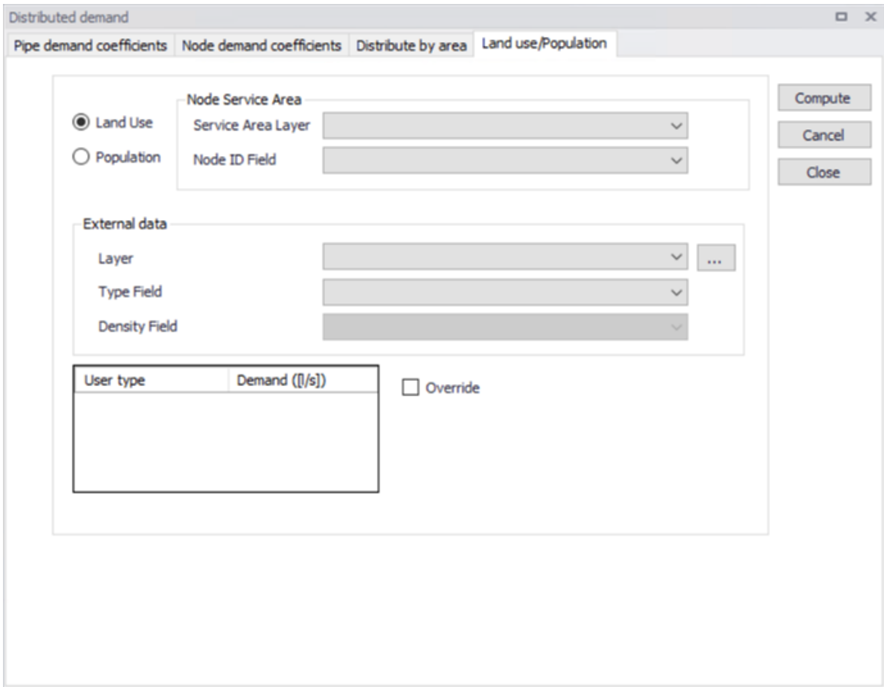


Figure 4.11 Distributed Demands Land use

Selecting "Population" will enable also the option to select the Density Field. By choosing this option the user must specify an external layer of population and identify the field of population types. It is required to identify which field in the shape layer contains the population data. Similarly to Land Use it is required to specify the unit demand of people per each type in the table (water demand per capita per day). The engine will compute the water demand for each node and distribute to them.

4.1.8 Aggregation tool

Demand aggregation allows to use customer connections to model junctions and to develop node demands. Node demands can be developed from consumption points in two essential methods:

- Assigning: this will create 1:1 relation, i.e. 1 allocation point = 1 node demand
- Aggregation: this will create 1: N relation, i.e. multiple allocation points = 1 node demand.

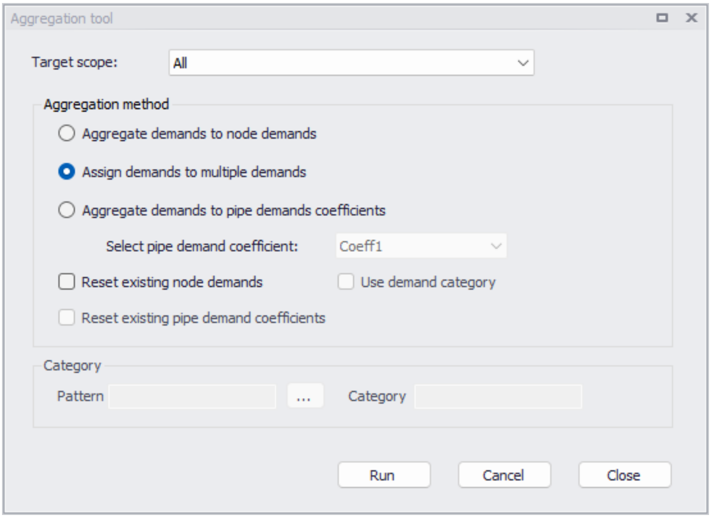


Figure 4.12 The demand aggregation tool

Target scope

It is possible to use all demands (all consumers) or only the selected ones (e.g. new consumers), using this list.

Aggregation method

The following methods can be applied:

- Aggregate demands to node demands: the program will aggregate data from multiple consumer points (assigned to the same node) and create a new node demand.
- Assign demands to multiple demands: the program will create a new multiple node demands for each consumer point.
- Aggregate demands to pipe demand coefficients: the program will aggregate data from multiple consumer points (assigned to the same pipe) and enter the total value into a pipe demand coefficient.
  - Select pipe demand coefficient: selected pipe demand coefficient

The following options can additionally be applied:

- Reset existing node demands: the program will remove all existing multiple demands, which are marked as created by the 'Demand allocation' tool. Multiple demands with a mark set to 'Manual' or 'Distributed demand' will be kept unchanged.
- Reset existing pipe demand coefficients: the program will remove data from the selected pipe demand coefficient in all pipes.
- Use demand category: the program will aggregate demand based on the category. For example, if there are 5 consumer points to be aggregated to the same node and 3 are residential and 2 are commercial, the program will create 2 new node demands, one for residential (where 3 residential consumer points are aggregated into one) and one for commercial (where 2 residential consumer points are aggregated into one).

Category

Pattern: when a pattern is selected, the program will use the pattern name for multiple demands that will be created in this process.

Category: when a category name is provided, the program will use this category name for multiple demands that will be created in this process.

5 Tables

The tables group comprehends the setting of Time Patterns, Curves and Relation and the information relevant for Engineering tables.

5.1 Time Patterns

MIKE+ uses EPANET as its numerical engine for hydrodynamic and water quality simulations. This engine assumes that water usage rates, external water supply rates and constituent source concentrations at nodes remain constant over a fixed period of time, but that these quantities can change from one time period to another. The default time period interval is one hour, but this can be set to any value. The value of any of these quantities in a time period equals a baseline value multiplied by a time pattern factor for that period. Following it is illustrated a pattern of factors that might apply to daily water demands, where each demand period is of an hour duration. Different patterns can be assigned to individual nodes or groups of nodes.

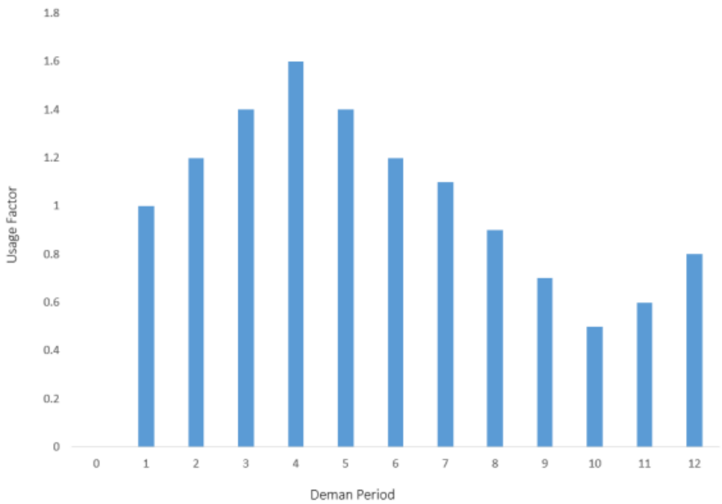


Figure 5.1 Time pattern for water usage

5.1.1 Diurnal Patterns

The diurnal profiles are used to define a series of multipliers (multiplication factors applied to a baseline value of junction node demand, constituent source concentration, storage water level). The diurnal profiles settings can be accessed from the Patterns tab.

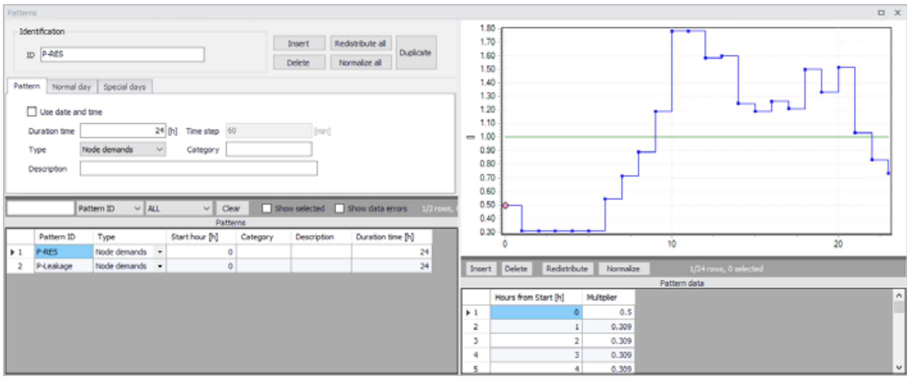


Figure 5.2 Diurnal Pattern

Pattern ID

This data entry is used to specify the ID of component being defined. The pattern ID value can be any string value (up to 40 characters). There is no limit to the number of demand patterns that can be defined.

Use date and time

‘Use date and time’ sets whether the absolute date and time will be used for each and every multiplier, instead of the default format 'Hours from start'.

Time Step

The pattern time step specifies the length of time between each pattern change (i.e., the period of time over which water demands and constituent source strengths remain constant). To change the pattern time step, use the field *Pattern time step* specified in the ‘Simulation setup’ editor.

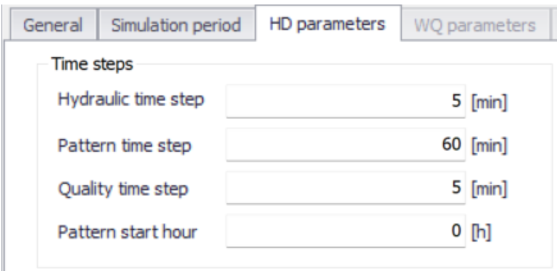


Figure 5.3 Pattern time step in the ‘Simulation setup’ editor

Duration time

The duration is used when redistributing the pattern data: in that case, the duration time and time step control the number of records in the pattern data table.

Type

The available types are *Node demands, Water quality, Tank water, Energy Price, Pump relative speed* or *Undefined. The selected type controls how and where the pattern can be applied.*

Category

An optional user-defined category description.

Description

This data entry allows the user to enter a category that further define the demand pattern. For example, a demand might have the description of a residential, and a category of either high density, medium- density or low density to further define what is meant by residential.

Redistribute

The 'Redistribute' buttons fills the pattern data with a fixed time interval corresponding to the pattern time step, and with the specified duration. Use this button to fill the pattern table with initial values, or when the pattern time step or duration was changed and you want to adjust the pattern accordingly.

Normalize

The 'Normalize' button adjusts the pattern multipliers so that their average value equals “1”. Note, that this will change the pattern values (multipliers) but it will not change their count. Normalization of patterns is useful when you want to create a typical daily demand patterns.

5.1.2 Normal Day

Normal days are understand as usual schedule days in which the consume of water shall not vary significantly. The contends for days are weekly defined days; Monday, Tuesday, Wednesday, Thursday, Friday, Saturday and Sunday. For months the items are: January, February, March, April, May, June, July, August, September, October, November and December.

There is an option to add factors to specific days, default value is one (1). When the factor is modified, MIKE+ will find the multipliers that correspond to the “Day” or “Month” based on “hours from simulation” and the Simulation Day set in time settings, further this factor will be applied to those multipliers.

5.1.3 Special Days

The user can define factors for special days (e.g. Holidays) in which the water demand pattern varies from normal days. These days can be defined in the special days table. The factor specified for *Special Days* has a higher priority than the day and month factors in *Normal Days.* When the date meets holiday, pattern would use the factor in the special days table instead of using ones of normal days.

5.2 Curves and Relations

The user define data curves and their X, Y coordinate points in the Curves and Relation editing group. The following curves can be used to represent relations:

- Pump Efficiency. Efficiency versus flow for pumps
- Valve Head Loss. Head Loss versus flow for GPV General Purpose Valve.
- Pump Q-H Curve. Head versus flow for pumps.
- Tank Depth-Volume Curve. Volume versus depth for tanks.
- Water Source Price. Production water costs versus produced volume.
- Transient Q-Boundary. Inflow/outflow at the boundary node versus time (only for Water Hammer Analysis).
- Transient H-Boundary. Hydraulic Grade Line at the boundary node versus time (only for Water Hammer Analysis).
- Valve Operation Schedule. Valve opening versus time (only for Water Hammer Analysis).
- Valve Characteristics Cd. Valve flow coefficient Cd versus valve opening
- Valve Characteristics Kv. Valve flow coefficient Kv versus valve opening
- Dual-acting characteristics. Volume of air versus pressure difference (only for Water Hammer Analysis)
- Pump Operational Schedule. Pump speed versus time (only for Water Hammer Analysis).
- Pump Torque. Pump torque versus flow (only for Water Hammer Analysis).
- Motor Torque. Motor torque versus pump speed (only for Water Hammer Analysis).
- PID Set Point Value Curve. Set point setting versus fraction of a day (only for RTC Real-time Control Analysis).

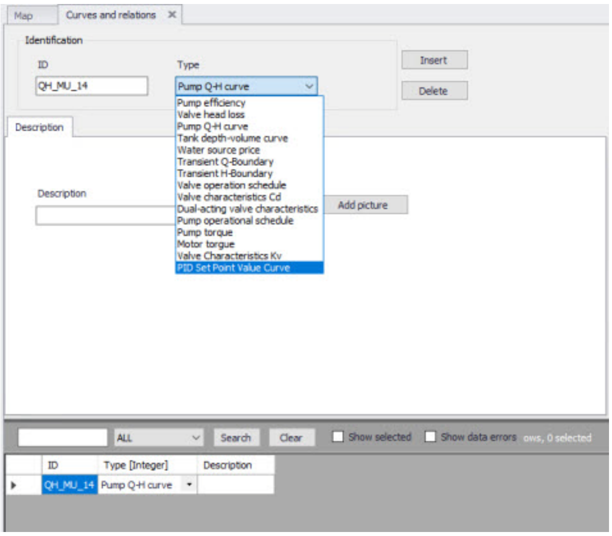
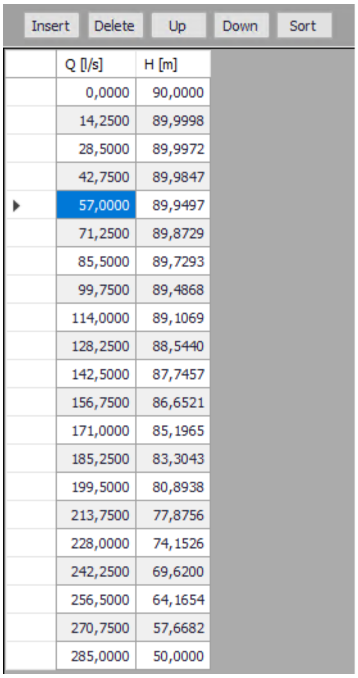


Figure 5.4 Curve and Relations, Identification

The user is capable to insert different types of curves (previously described).The values to be included in each specific curve can be edited in the grid editor, where entries can be added, deleted, reordered and sort.



Q [l/s]	H [m]
0,0000	90,0000
14,2500	89,9998
28,5000	89,9972
42,7500	89,9847
57,0000	89,9497
71,2500	89,8729
85,5000	89,7293
99,7500	89,4868
114,0000	89,1069
128,2500	88,5440
142,5000	87,7457
156,7500	86,6521
171,0000	85,1965
185,2500	83,3043
199,5000	80,8938
213,7500	77,8756
228,0000	74,1526
242,2500	69,6200
256,5000	64,1654
270,7500	57,6682
285,0000	50,0000

Figure 5.5 Curves and Relations, Definition of data curves

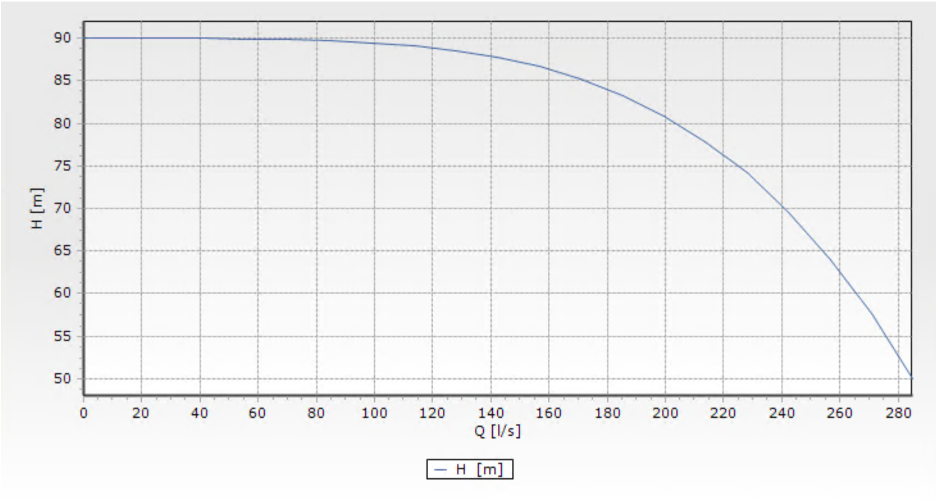


Figure 5.6 Preview of a defined curve in Curve Editor

The points of a curve must be entered in order of increasing X- values (lowest to highest).

Data rows in any of the curve's definitions must not have 2 or more lines with the same "X" value.

6 Control

6.1 Real Time Control

Real time control provides the following operations:

- Variable pump speed to maintain pressure or level or flow/velocity set-points
- Movement of valves to maintain pressure or level or flow/velocity set-points

The purpose of this kind of control is to provide generic way of moving valves and changing pump speed other than what is done using IF-THEN-ELSE rules or VSD pump control. IF-THEN-ELSE rules control valves and pumps instantly (at the time step) and may result, in some cases, in oscillating solutions in between time steps. Or, in case of VSD pump control, they may not be able to operate more than 1 pump within the same zone due to interference of the algorithm with other pumps. The presented real-time control provides an independent mechanism that can be used to determine or simulate pump or valve operations in a physical system.

The real-time control provides two algorithms:

- Linear control
- PID (Proportional – Integral – Derivative) control

Linear control is a mechanism that will increase or decrease the control setting based on the actual value of the measured process variable versus the set-point. The position of a control valve will be increased or decreased and the pump speed will be increased or decreased. The increase and decrease rates as well as the maximum and minimum settings are pre-defined.

PID (Proportional – Integral – Derivative) control is a control loop feedback mechanism (controller) commonly used in industrial control systems. A PID controller continuously calculates an error value e(t) as the difference between a desired set point and a measured process variable. The controller attempts to minimize the error over time by adjustment of a control variable u(t), such as the position of a control valve or a pump speed to a new value determined by a weighted sum:

$$u(t) = K_p e(t) + K_i \int_0^t ((e(t)\Delta t)) + \left(\frac{Kd\Delta e(t)}{\Delta t}\right) \tag{6.1}$$

Where Kp, K_i, and K_d are all non-negative coefficients for the proportional, integral, and derivative terms. In this model:

- K_p accounts for present values of the error. For example, if the error is large and positive, the control output will also be large and positive.
- K_i accounts for past values of the error. For example, if the current output is not sufficiently strong, error will accumulate over time, and the controller will respond by applying a stronger action.
- K_d accounts for possible future values of the error, based on its current rate of change.

6.1.1 Setup

A list of the Real Time Control dialog box data entries for Figure 6.1 follows, with a short description given for each entry.

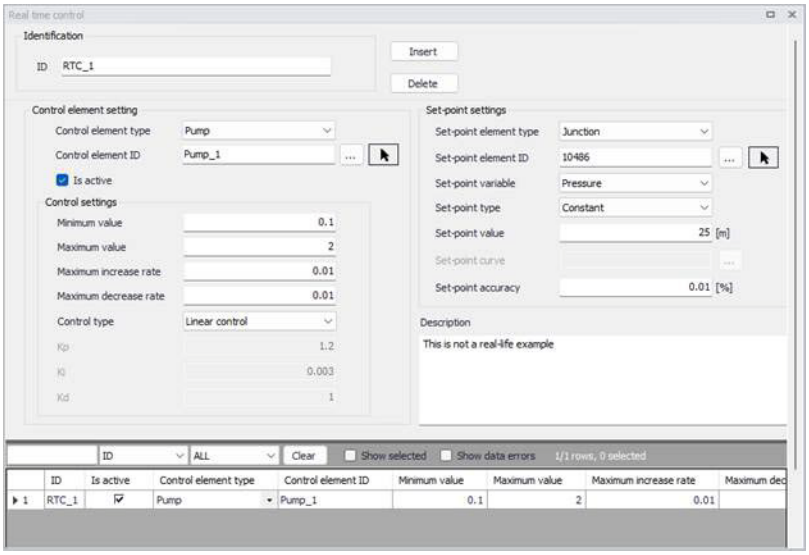


Figure 6.1 The Real Time Control dialog box is used to the analysis parameters

Identification ID

This data entry allows you to define the ID (name) of the real-time control.

Control element settings

Control element type

This data entry allows you to define the type of a controlled element: pump, or a TCV valve.

Controlled element ID

This data entry allows you to define the ID of the controlled element. You can select the element from a list by clicking the … button.

Is active
This check box allows the user to toggle the Active status of the control on and off. The simulations will omit all controls that are not active.

Minimum value
This data entry allows you to define the minimum control value (relative pump speed or valve opening percentage).

Maximum value
This data entry allows you to define the maximum control value (relative pump speed or valve opening percentage).

Maximum increase rate
This data entry allows you to define the maximum rate at which the variable can increase. In units of the control variable (relative pump speed or valve opening percentage per minute.

Maximum decrease rate
This data entry allows you to define the maximum rate at which the variable can decrease. In units of the control variable (relative pump speed or valve opening in %) per minute.

Control type
This data entry allows you to choose between the linear control or PID control (Proportional-integral-derivative control).

Kp
In case of PID control, this data entry allows you to define the proportional constant (proportional gain).

Ki
In case of PID control, this data entry allows you to define integral constant (integral gain).

Kd
In case of PID control, this data entry allows you to define derivative constant (derivative gain).

Set-point settings

Set point element type
This data entry allows you to select the type of the set point element such as a tank or a junction node, or a pipe.

Set point element ID
This data entry allows you to define the ID of the set point node.

Set point variable

This data entry allows you to define type of the set-point variable. In case of a tank or a junction (set-point element type) the set-point variable can be “Grade – hydraulic gradeline” or “Level” (in case of tanks), or “Pressure” in case of a junction node. In case of a pipe (set-point element type) the set-point variable can be “Flow” or “Velocity”.

Set-point type

This data entry allows you to define the type of a set-point value a “Constant” or “Variable”.

Set point value

In case of a constant set-point, this data entry allows you to define the set-point value.

Set point curve

In case of a variable set-point, this data entry allows you to define the curve ID defining how the set point value changes in time. Note, that the curve (table) definition is done in the Curves Editor.

Set-point accuracy

This data entry allows you to define the accuracy of the algorithm (control) in percentage of the set-point.

Description

This data entry allows you to specify a user defined description of the control entry.

Remarks

The program is using the following predefined valve characteristics tau curves in case of TCV valve control:

- Quick opening valve (gate)
- Globe valve
- Plug valve
- Butterfly valve
- Ball valve
- Equal percentage 5%

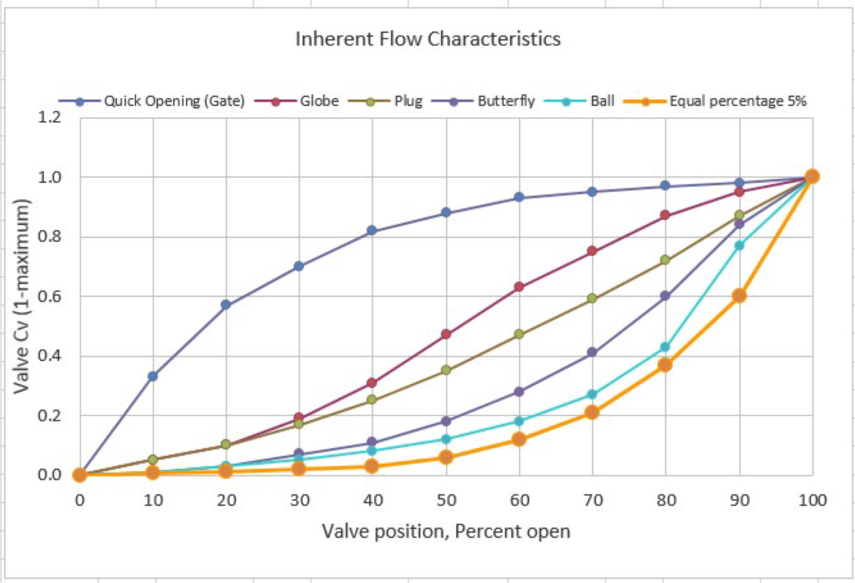


Figure 6.2 Default valve characteristics

6.2 Extended Rule-Based Controls

Rule-Based Controls allow link status and settings to be based on a combination of conditions that might exist in the network over an extended period simulation. Rule based controls will be either in the form of an action clause or a condition clause. The Rule Based Controls Editor dialog box is reached by selecting Extended Period Rule Based Controls from the Table of Content.

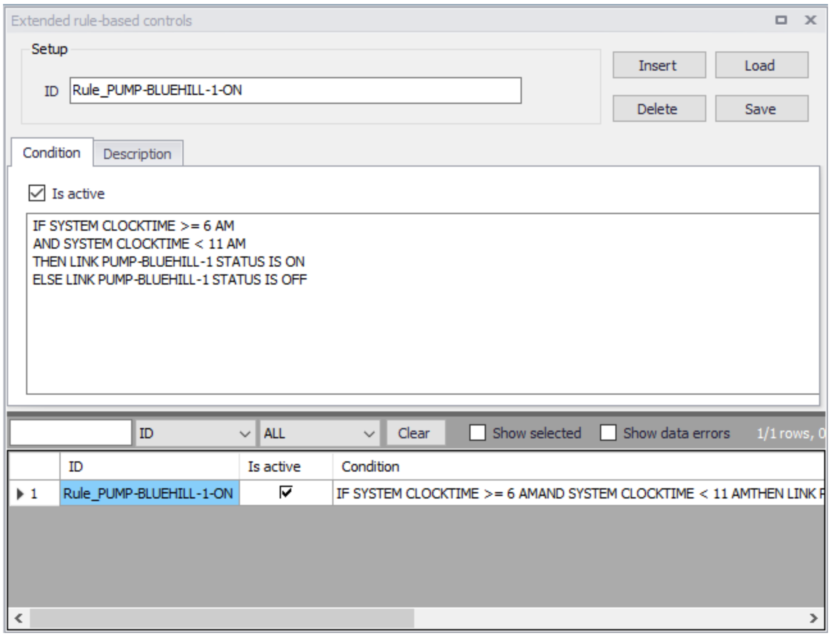


Figure 6.3 The Extended rule-based control editor

Insert

Insert a new rule.

Delete

Delete a rule.

Load

Load rules from a text file (*).

Save

Save rules into a text file (*).

(*) You can edit the rules within the ASCII file and import them back to MIKE+ by selecting Load from ASCII file. This is convenient in cases when you use Excel or other tools to create the list of rules for the model.

6.2.1 Format of rule

Each rule is a series of statements of the form:

IF condition_1
AND condition_2
OR condition_3
AND condition_4
etc.

THEN action_1
AND action_2
etc.

ELSE action_3
AND action_4
etc.

PRIORITY value
where:
conditon_n = a Condition clause
action_n = an Action clause
priority = a priority value (e.g., a number from 1 to 5)

Remarks

1. Keywords IF, AND, OR, THEN, ELSE, PRIORITY must always start at a new line.
2. Only the RULE, IF and THEN portions of a rule are required; the other portions are optional. The "RULE" portion is automatically created from the contents of Rule ID and Description fields. The portions "IF" (i.e. condition clause) and "THEN" (i.e. action clause) must be provided by the user.
3. When mixing AND and OR clauses, the OR operator has higher precedence than AND, i.e.,

   IF A or B and C
   is equivalent to
   IF (A or B) and C.

   If the interpretation was meant to be
   IF A or (B and C)
   then this can be expressed using two rules as in
   IF A THEN ...
   IF B and C THEN ...
4. The PRIORITY value is used to determine which rule applies when two or more rules require that conflicting actions be taken on a link. A rule without a priority value always has a lower priority than one with a value. For two rules with the same priority value, the rule that appears first is given the higher priority.
5. The decimal separator for numerical values must be a point.

Condition clause

A *condition clause* in a Rule-Based Control takes the form of:

object id attribute relation value

where
object = a category of network object
id = the object's ID label
attribute = an attribute or property of the object
relation = a relational operator
value = an attribute value

Some example conditional clauses are:
JUNCTION 23 PRESSURE > 20
TANK T200FILLTIME BELOW 3.5
LINK 44 STATUS IS OPEN
SYSTEM DEMAND >= 1500
SYSTEM CLOCKTIME = 7:30 AM

The Object keyword can be any of the following:

NODE LINK SYSTEM
JUNCTION PIPE
RESERVOIR PUMP
TANK VALVE

When SYSTEM is used in a condition no ID is supplied.

The following *attributes* can be used with Node-type objects:

DEMAND
HEAD
PRESSURE

The following *attributes* can be used with Tanks:

LEVEL
FILLTIME (hours needed to fill a tank)
DRAINTIME (hours needed to empty a tank)

These *attributes* can be used with Link-Type objects:

FLOW
STATUS (OPEN, CLOSED, or ACTIVE)
SETTING (Pump speed or Valve setting)
LIKE (See Multiple Pumps, Valves for more details)

The SYSTEM object can use the following *attributes*:

DEMAND (total system demand)
TIME (hours from the start of the simulation)
CLOCKTIME (24-hour clock time with AM or PM appended)

Relation operators consist of the following:

= IS
<> NOT
< BELOW
> ABOVE
<=> =

Action clause

An *action clause* in a Rule-Based Control takes the form of:

object id STATUS/SETTING IS value

where

object = LINK, PIPE, PUMP, or VALVE keyword
id = the object's ID label
value = a status condition (OPEN or CLOSED), pump speed setting, or valve setting

Some example action clauses are:

LINK 23 STATUS IS CLOSED
PUMP P100 SETTING IS 1.5
VALVE 123 SETTING IS 90

LIKE

A special case of action clause is the LIKE setting.:

Setting Value = A (another link setting) B (multiplier) C (increment)

The default values for the B (multiplier) and C (increment) are B=1, C=0 and they do not need to be provided.

The setting value is calculated as:
Setting Value = Setting Value (link A) * B + C

See Multiple Pumps, Valves for more details.

6.2.2 Multiple Pumps, Valves

Note that the LIKE setting allows you to control multiple pumps or valves in efficient way. It is possible to set a pump speed to x% of another pump, for example. Such an option could also be used when the new value is the value of the object itself. (Set a pump speed to increase with 20%, or valve to open 10% and so on).

Example 1:

IF SYSTEM CLOCKTIME = 8 AM
THEN PUMP 3 SETTING LIKE PUMP 4 1 0

The pump 3 setting will be set equal to the settings of the pump 4 (multiplier= 1 and increment = 0) at time 8 am.

Example 2:
IF SYSTEM CLOCKTIME = 8 AM
THEN PUMP 3 SETTING LIKE PUMP 4 1.10 0

The pump 3 setting will be higher by 10% than the settings of the pump 4 (multiplier = 1.10 and increment = 0) at time 8 am.

Example 3:
IF SYSTEM TIME >= 12 AM
THEN VALVE 10 SETTING LIKE VALVE 20 1.0 -10

The valve 10 setting (PRV setting, for example) will be lower by 10 (pressure units) than the settings of the valve 20 (multiplier = 1 and increment = -10) at any time (12 AM is the simulation start).

Example 4:
IF SYSTEM TIME >= 12 AM
THEN PUMP 3 STATUS LIKE PUMP 4

The pump 3 status (OPEN, CLOSED) will be set equal to the status of the pump 4 at any time (12 AM is the simulation start).

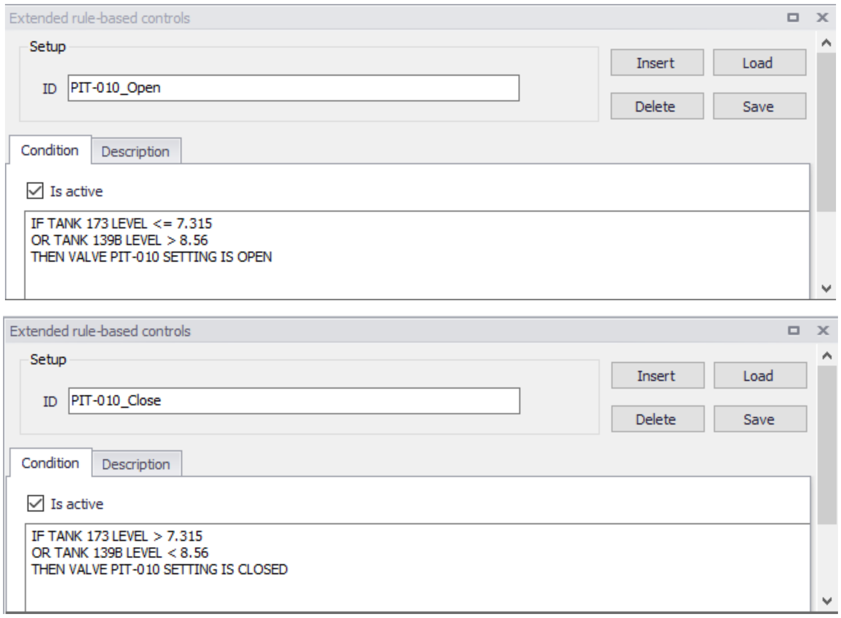
Example 5:
IF PUMP 3 SETTING LIKE PUMP 4
THEN …

If pump 3 setting is equal to the settings of the pump 4 (default multiplier = 1 and increment = 0) then …

6.2.3 Controls Examples

Control of a valve

This set of rules opens and closes a valve based on the water level in a storage tank.



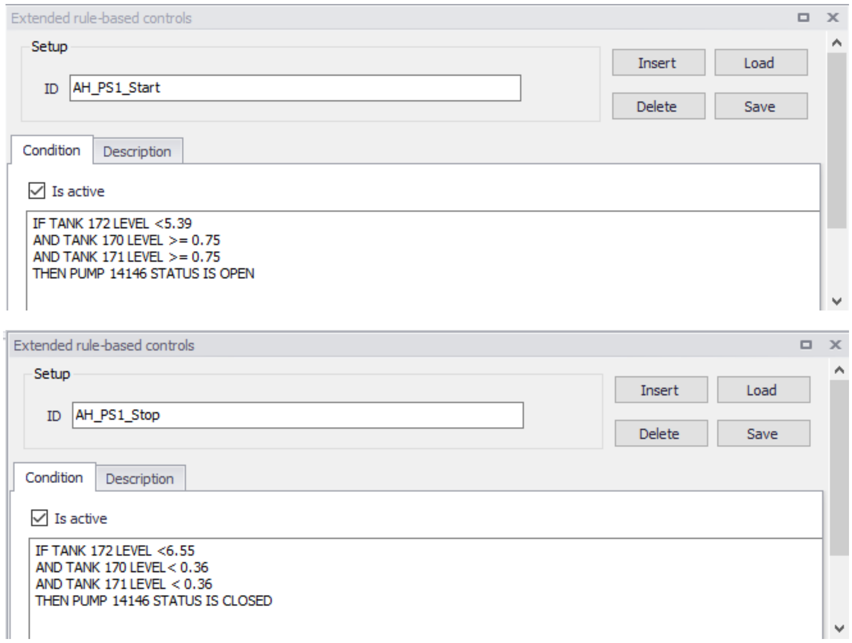

```
RULE   PIT-010_Open;    BUNKER RD NORTH CV OPEN
IF TANK 173 LEVEL <= 7.315
OR TANK 139B LEVEL > 8.56
THEN VALVE PIT-010 SETTING IS OPEN

RULE   PIT-010_Close;   BUNKER RD NORTH CV CLOSE
IF TANK 173 LEVEL > 7.315
OR TANK 139B LEVEL < 8.56
THEN VALVE PIT-010 SETTING IS CLOSED
```

Figure 6.4 Rules in EPANET *.inp file

Control of a pump

This set of rules opens and closes a pump based on the water level in a storage tank.




```
RULE   AH_PS1_Stop;     ALEXHILL PS1 - AUTOMODE1
IF TANK 172 LEVEL <6.55
AND TANK 170 LEVEL< 0.36
AND TANK 171 LEVEL < 0.36
THEN PUMP 14146 STATUS IS CLOSED

RULE   AH_PS1_Start;    ALEXHILL PS1 - AUTOMODE1
IF TANK 172 LEVEL <5.39
AND TANK 170 LEVEL >= 0.75
AND TANK 171 LEVEL >= 0.75
THEN PUMP 14146 STATUS IS OPEN
```

Figure 6.5 Rules in EPANET *.INP file

6.3 Regulation Overview

Typically during an extended period simulation, the pipes, pumps, turbines and valves contained in a network will change their status (i.e. open or close) as storage tanks fill and empty and pressures change throughout the network. Also, for a steady state simulation, network components may change their state as the analysis model iterates to a valid solution.

The 'Regulation overview' editor can be used to specify these operational controls on the network.

The following situations are examples of applications of such operational controls:

- A pipe can be opened at a given time (based upon the beginning of the network simulation). This type of operational control has no effect in a steady state simulation.
- A pump can be turned on or off depending on the water level in a specified tank.
- A valve can be opened or closed based upon the pressure in an adjacent node.

Control rules are added using the 'Insert' button and removed using the 'Delete' button. For each rule, the following Identification parameters can be specified:

Control ID
The ID of the control.

Description
An optional description of the control.

Link type
The selection of link type to which the control applies. It can either be a pipe, a pump, a valve or a turbine.

Link ID
The ID of the link to which the control applies. Use the '…' button to access a filtered list of valid IDs.

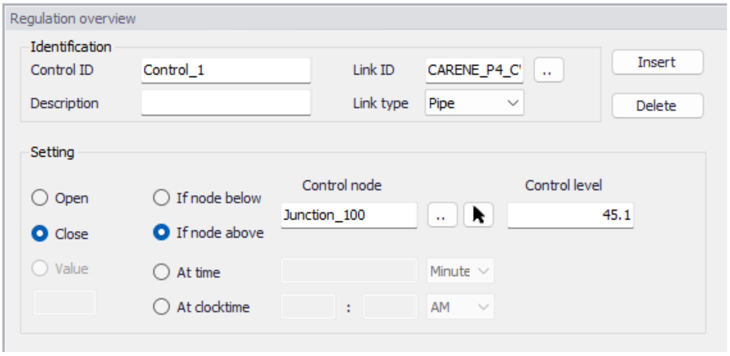


Figure 6.6 The Regulation overview editor

These control rules can also be visualized and edited respectively in the Pipes, Pumps, Valves and Turbines editors, in their 'Regulation' tab and for the link ID they are assigned to. There is no difference between editing the rules in one or the other editor.

For a description of the regulation settings, please refer to the Pipes, Pumps, Valves or Turbines editor description chapters.

7 Pressure Dependent Demands

Traditionally, water demands are defined prior to the simulation and thus independent of the actual pressure. With the Pressure Dependent Demands, the Wagner equation can be used to adjust the node demands based on the available pressure.

Pressure Dependent Demands Analysis is an alternative computational method based on pressure driven analysis comparing to the traditional demand driven analysis. Node demands are automatically adjusted based on the available pressure. This approach can be used to model intermittent water supply, low pressure situations, and it is also suitable for modelling system shut- down and maintenance.

There are three formulations of the demand versus pressure relation that can be used in computation: Wagner, Tucciarelli, and Fujiwara equation. They all adjust the node demand based on the available pressure.

Wagner equation [1]:

$$Q_{new} = Q_{original}\left(\frac{P_{actual} - P_{minimum}}{P_{required} - P_{minimum}}\right)^{\frac{1}{n}}$$

Tucciarelli equation [2]:

$$Q_{new} = Q_{original}\left(\sin\left(\pi\frac{P_{actual}}{2P_{required}}\right)^2\right)$$

Fuijiwara equation [3]:

$$Q_{new} = Q_{original}\left(\left(\frac{(P_{actual} - P_{minimum})^2(3P_{required} - 2P_{actual} - P_{minimum})}{(P_{required} - P_{minimum})^3}\right)\right)$$

[1] J. Wagner, U. Shamir, D. H. Marks (1988) "Water distribution reliability: Simulation Methods." J Water Resour Plan Manage Div Vol. 114.3: 253-275
[2] T. Tucciarelli, A. Criminisi, D. Termini (1999) Leak Analysis in Pipeline Systems by Means of Optimal Valve Regulation. Journal of Hydraulic Engineering 125(3): 277-285.
[3] O. Fujiwara and T. Ganesharajah (1993) Reliability assessment of water supply systems with storage and distribution networks. Water Resources Res 29.8: 2917-2924. 10.1029/93WR00857

where:

- Q_{new} = adjusted node demand
- P_{actual} = actual pressure
- $P_{required}$ = required pressure (such as e.g. 15 m), node demand is equal to the original demand if the pressure (such as e.g. 5m), node demand is 0 if the pressure drops below the minimum pressure
- n = coefficient with recommend values between 1.5 2.0 (2.0 is recommended by Wagner)

Note that nodes with negative demand i.e. inflow nodes are excluded from the above equation.

Note that to run the pressure dependent demands analysis, you need to select *Run* from within the Pressure Dependent Demands dialog box.

7.1 Settings

A list of the Pressure dependent demands data entries for Figure 7.1 follows, with a short description given for each entry.

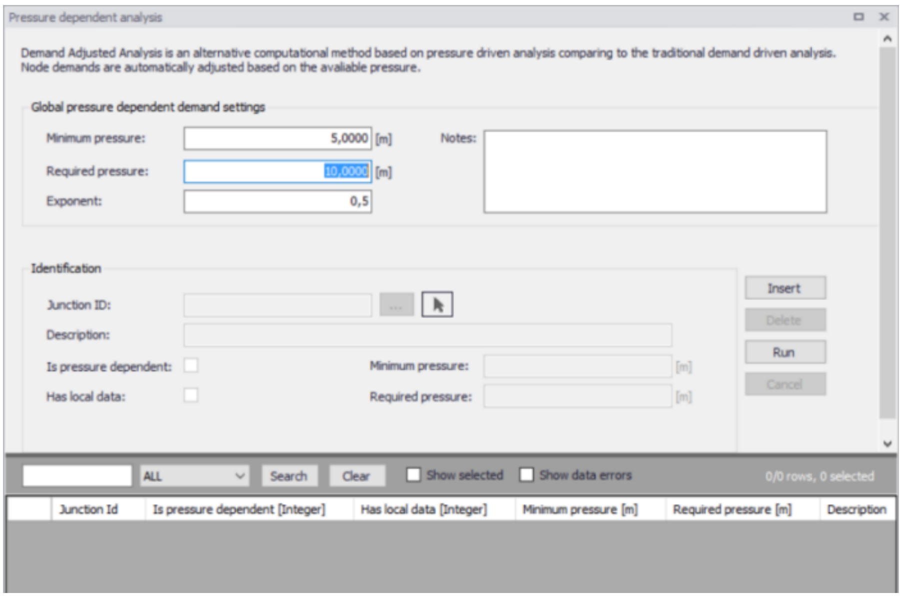


Figure 7.1 The Pressure dependent demands dialog box is used to the analysis parameters

Minimum pressure

This data entry allows you to define the minimum pressure (such as 5m), node demand is 0 if the actual computed pressure drops below the minimum pressure

Required pressure
This data entry allows you to define the required pressure (such as 10m), node demand is equal to the original demand if the actual computed pressure is above the required pressure.

Formula
This data entry allows you to select the equation that will be used to compute pressure dependent demands. The options are Wagner equation,

Wagner exponent
This data entry allows you to define the coefficient "n" for the exponent in Wagner equation (exponent = 1/n).

Global nodes are pressure dependent
This data entry allows you to activate pressure dependent demands for all nodes unless they are locally changed using the “has local data” option. Similarly, if you only want several specific nodes (demands) to be pressure dependent, unselect this data check box and use local data to define pressure dependent nodes.

Notes
This data entry allows you to enter any notes or further descriptions.

Insert/delete
Allows you to add or remove local data.

Junction ID
This data entry allows you to define the local node. Use “…” to select the junction node from the list or use the arrow “↑” to select the junction node from the Map.

Description
This data entry allows you to provide user defined description.

Is pressure dependent
This data entry allows you to define the local node as either “pressure dependent” or “not pressure dependent”.

Has local data
This data entry allows you to define if the local node shares the global pressure settings or whether it will use its own pressure settings (local data).

Minimum pressure (local data)
This data entry allows you to define the minimum pressure that will apply only to the local node.

Required pressure (local data)
This data entry allows you to define required pressure that will apply only to the local node.

7.2 Running simulation

Select “Run” from within the Pressure dependent demands dialog box in order to run the simulation. The simulation progress will be displayed in the application status window. The simulation can be interrupted (cancelled) by pressing “Esc”.

7.3 Pressure dependent demand results

Results of the Pressure dependent demands simulations can be displayed as results for the standard hydraulic simulation. However, there are several additional results items that can be used in data display:

Nodes

- Demand (pressure depended requested)
- Demand (pressure depended supplied)
- Demand (pressure depended deficit)
- Demand (pressure depended supplied percentage)

8 Water Quality

8.1 Water Quality Simulation

MIKE+ allows you to perform water quality simulations. In order to perform a water quality simulation, an extended period simulation must also be specified. Defining an extended period simulation was discussed in the previous section.

The following sections describe how to perform a particular type of water quality simulation, and the various water quality editors used to define each type of water quality simulation.

Water quality simulation is normally hidden in the Setup Tree on the left but it can be brought up by double clicking “Model type” in “General settings” (see Figure 8.1), and check the box of “Water quality” in Figure 8.2

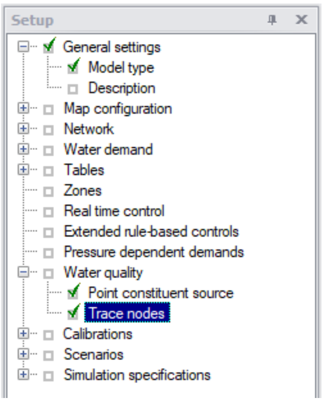


Figure 8.1 Water Quality setup tree layout

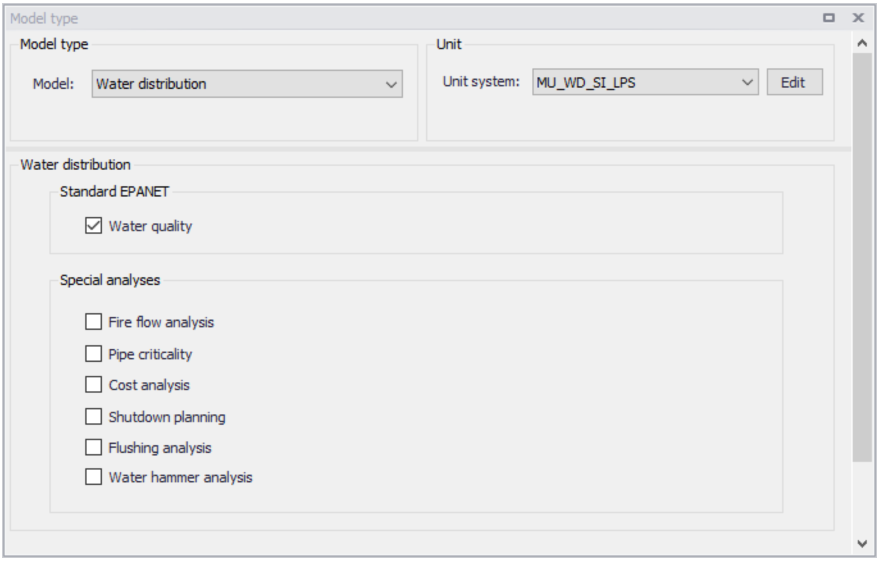


Figure 8.2 Layout of Models

Note that MIKE+ can only perform one type of water quality analysis during a simulation: Point Constituent Source or Trace Nodes.

8.1.1 Point Constituent Source

The Point Constituent Source Editor, as shown in Figure 8.3, allows you to specify at which nodes an external chemical constituent enters the network system. At least one node in the network (with its ‘Is active’ box selected) must be specified as a point source of chemical constituent when performing a chemical concentration analysis. There are three sections in this configuration window.

Identification

After clicking “Insert” button, the field of ID, Node type and Node ID will be enabled as well as next field of Source type, Concentration and Cyclic profile ID.

ID

This entry is automatically filled by default value “Source_x”. Users can edit the text to rename the source name.

Node ID

This data entry is used to define the ID of the node the point constituent is being assigned to. Users can select the appropriate node type and ID from the node list or on the map.

Node Type

This pull-down selection list allows the user to select what type of node (i.e., junction, reservoir, or tank) the point constituent is being specified for.

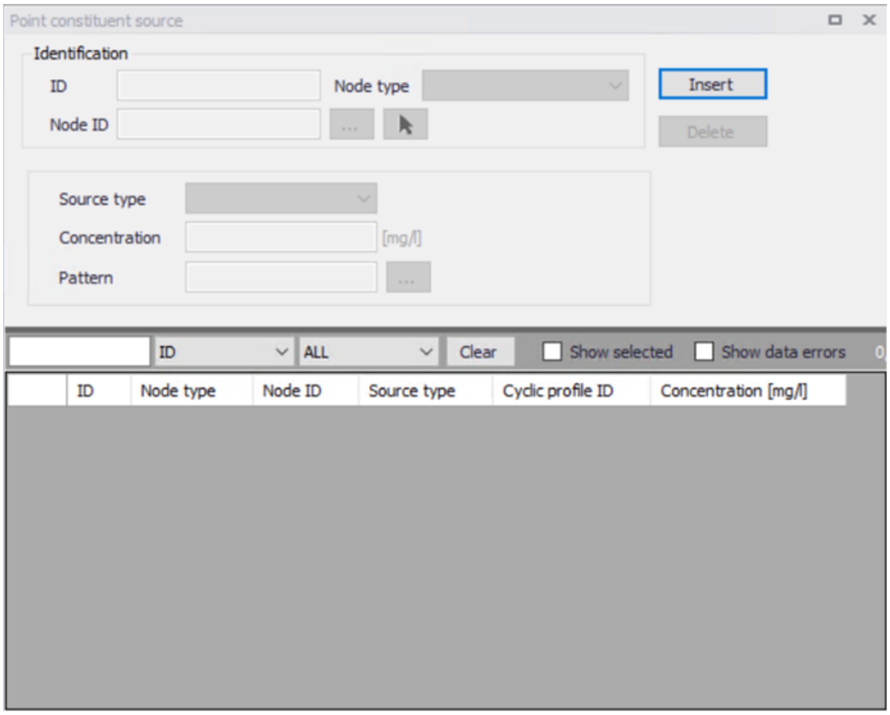


Figure 8.3 Layout of Point Constituent Source

Source Type

Water quality sources are nodes where the quality of external flow entering the network is specified. They can represent the main treatment works, a well-head or satellite treatment facility, or an unwanted contaminant intrusion.

Source quality can be made to vary over time by assigning it a time pattern. MIKE+ can model the following types of sources (see Figure 8.7):

A **concentration source** fixes the concentration of any external inflow entering the network at a node, such as flow from a reservoir or from a negative demand placed at a junction.

A **mass booster source** adds a fixed mass flow to that entering the node from other points in the network.

A **flow paced booster source** adds a fixed concentration to that resulting from the mixing of all inflow to the node from other points in the network.

A **setpoint booster source** fixes the concentration of any flow leaving the node (as long as the concentration resulting from all inflow to the node is below the setpoint).

The concentration-type source is best used for nodes that represent source water supplies or treatment works (e.g., reservoirs or nodes assigned a negative demand). The booster-type source is best used to model direct injection of a tracer or additional disinfectant into the network or to model a contaminant intrusion

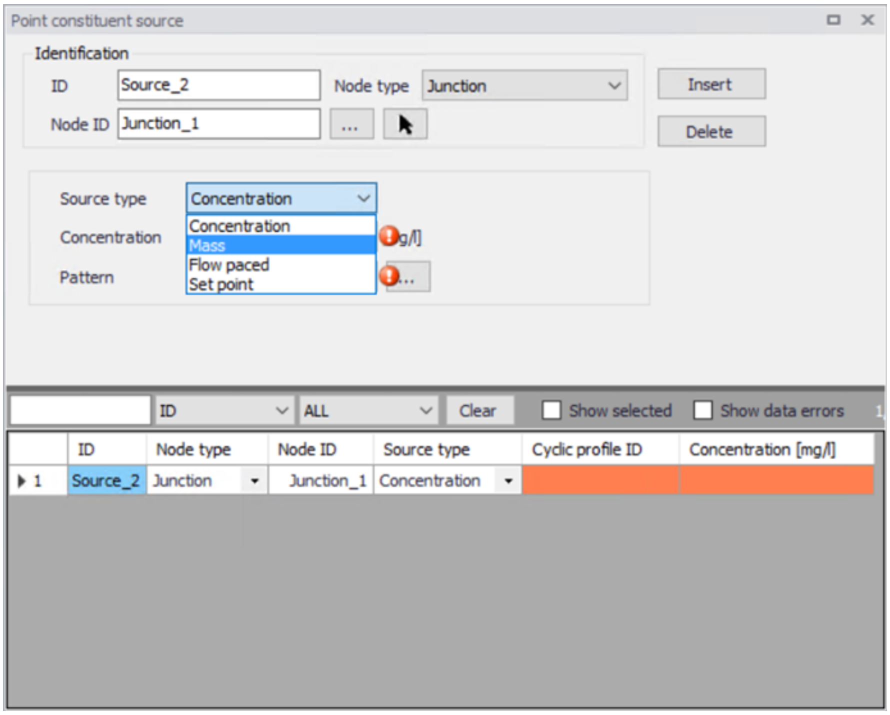


Figure 8.4 Source Type Options

CONCENTRATION

This data entry is used to specify the baseline concentration (in mg/liter) of the constituent entering the node as an external source.

PATTERN ID

This data entry allows you to define the ID of the constituent pattern to be applied to the specified baseline concentration entering the node. If a pattern ID is omitted for the specified source node, then there is no variation in the source strength of the constituent.

Select button [...] allows users to display the Select Pattern selection dialog box (Figure 1.5), where the appropriate pattern ID can be selected.

Point constituent source concentration time patterns are similar in concept to demand patterns. Each concentration time pattern consists of a set of multipliers that are multiplied to the specified baseline concentration over the extended period simulation. This allows the user to model changes in the amount of constituent applied at a node over an extended period simulation. See the section on Cyclic Profiles for further information on time patterns.

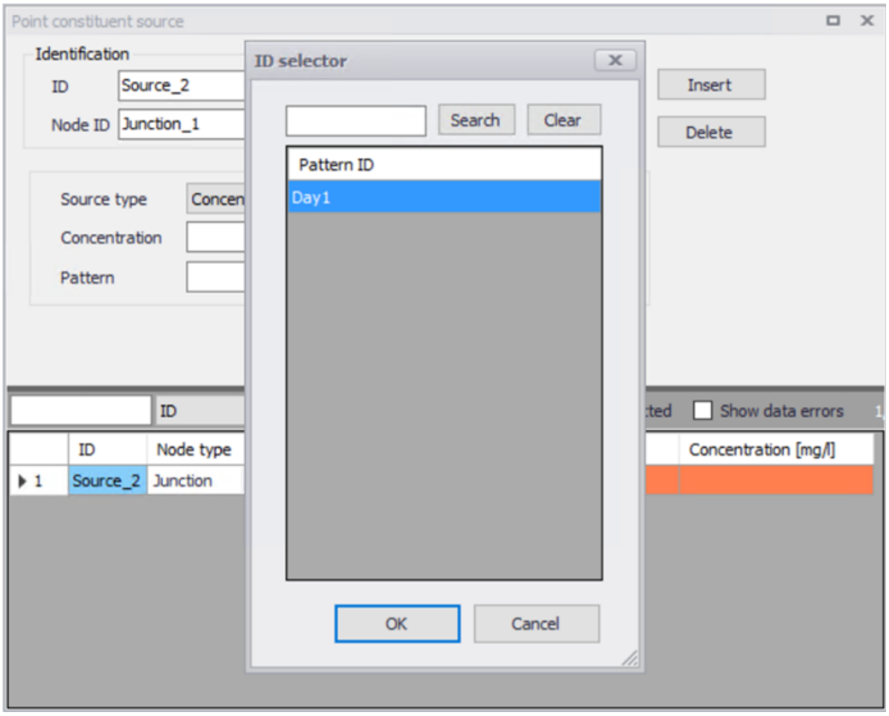


Figure 8.5 ID Selector Window

Table

The table contains all the detailed information of Point Constituent Source items. They can be edited or deleted by the user once they are selected in the table. If there was none item in this table, all nodes would be assigned with an initial water quality of zero by default. See Figure 1.6.

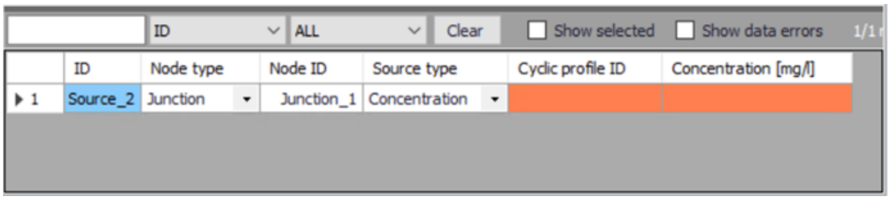


Figure 8.6 Detail Table of Point Constituent Source Settings

8.2 Multiple trace node analysis

Tracing nodes allows the user to track overt time what percent of water reaching any node in the network had its origin from a specified node (i.e. junction, tank or reservoir). This tool is useful for analysing a network distribution system that draws water from two or more different raw water supplies. Then it is possible to show to which extend water from a given source blends with that from another source, and how the spatial pattern of this blending changes through the simulation time.

The Trace Nodes Editor contains two sections, one for identification of the point source constituent and the editing table.

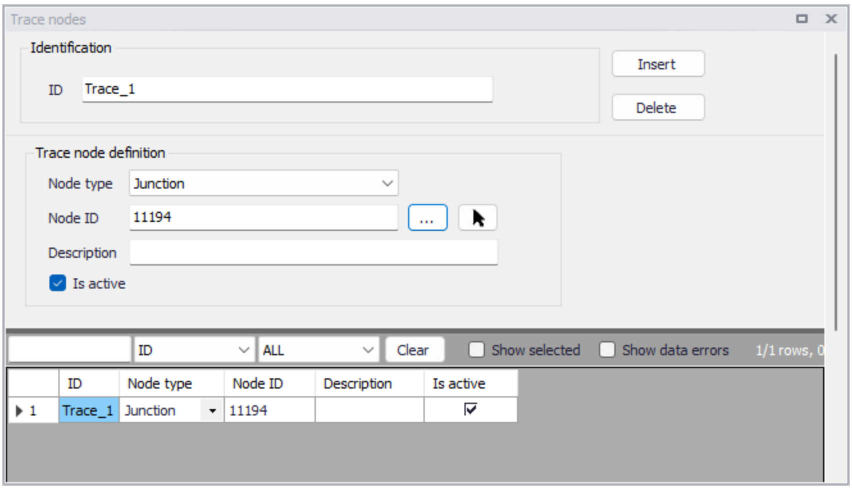


Figure 8.7 Point constituent source editor

Is active

This check box allows the user to toggle the Active status of the trace node on and off. The simulations will omit all trace nodes that are not active.

Identification ID

The identification field allows a maximum number of 40 characters. This entry is automatically filled by default value “Trace_x”. Users can edit the text to rename the trace name

Node Type

The trace node can be identify by means of it’s type of node through the pull-down selection lists which displays contends such as junction and tank.

Node ID

This data entry is used to define the MUID of the node the trace node is assigned to. It is possible to select the node ID from the node list or by selecting directly on the map.

Description

This field allows users to enter a description identifying the simulation defined. The description can be output in reports.

Multiple tracing blending

When there are more than two trace nodes defined the source tracking simulation will be run on a one-by-one basis for each water source, all results are further combined and the fraction from water source is computed and presented in the map, as follow:

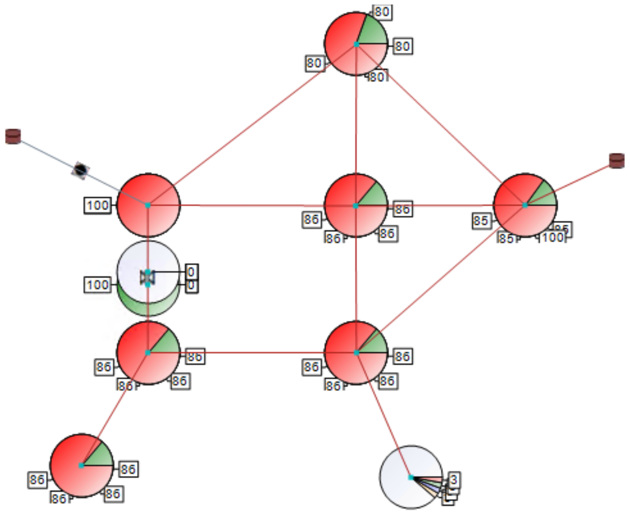


Figure 8.8 Multiple source blending (detail from the supply zone, colour shows water from different water sources)

9 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. 192)* for more information.

9.1 Simulation Setup

There are six primary tabs in the Simulation setup editor:

- General
- Simulation period
- HD parameters
- WQ parameters
- Water hammer
- HD output

A project simulation is added using the “Insert” button. Each simulation requires a unique simulation ID to be specified.

The table under tabs shows the project simulation details including ID, hydraulic parameters, etc. The parameters can be edited by double clicking the editable cells. The table also contains an 'Active simulation' check box: this is used to select a default simulation ID, which will be used in case the simulation is started from the 'Simulation' ribbon, or for example when executing the simulation from a command line for automating the simulation runs.

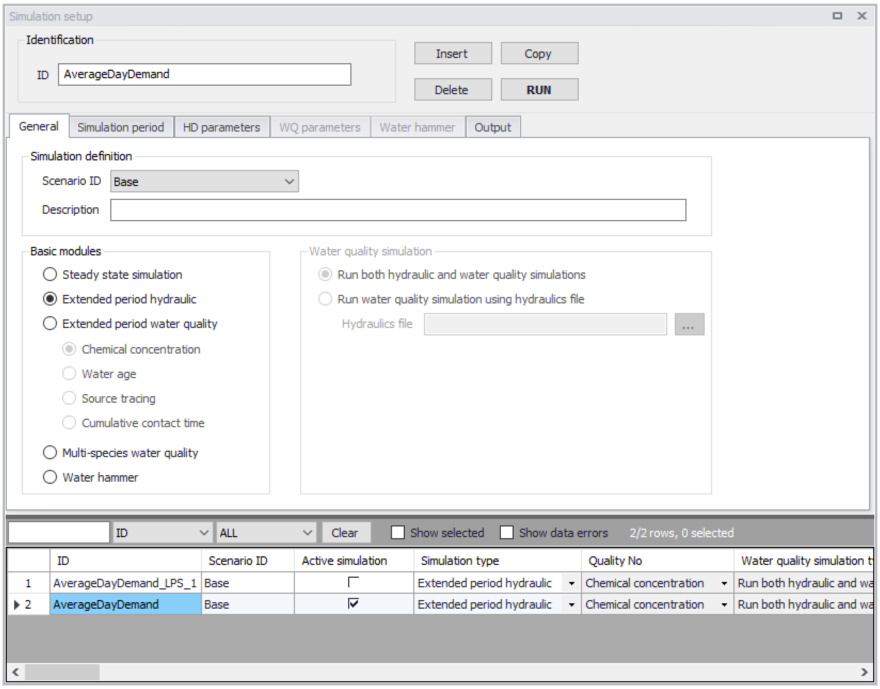


Figure 9.1 Layout of the Simulation setup editor

General

A description of all options available in the General tab follows.

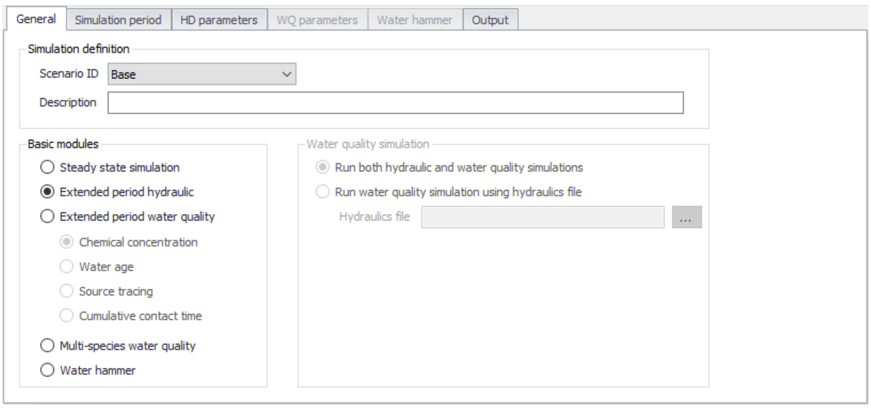


Figure 9.2 The General tab.

Simulation definition

Scenario ID: select the scenario which is to be executed in the active simulation ID.

Description: the description field is an optional field with further details about the simulation.

Basic modules

There are five types of modules in this field:

- Steady state simulation: EPANET-based steady state analysis
- Extended Period Hydraulics: EPANET-based extended period simulation.
- Extended Period Water Quality: Chemical concentration (compute chemical concentration), Water Age (compute water age), Source Tracing (trace flow from a specific node).
- Multi-species water quality: EPANET MSX-based analysis of multiple, interacting chemical species
- Water hammer

Water quality modules are normally disabled but users can enable them by opening ‘Model type’ under General settings and activating the appropriate modules.

When a water quality module is selected, two simulation modes are available: you can either run both hydraulics and water quality simulations at the same time, or you can run only the water quality simulation using an "hydraulics" file resulting from an earlier hydraulics simulation. The latter helps reducing simulation times when the hydraulic simulation takes a long time and does not need to be repeated while running the water quality scenarios. The input hydraulics file is saved from the hydraulics simulation by selecting 'Save hydraulics file' in the 'Output' tab.

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 unique MUID.

'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.

9.1.1 Simulation Period

This section is able to set the simulation duration. It has two ways:

- Define the simulation start time and simulation end time

- Define the duration directly

A text field box to define the starting time of the simulation and the other to establish the end time of it. It can be defined that the start and end of the simulation directly from the text box by typing the date or by means of using the pop up calendar window (accessible through the arrow on the right corner of the box).

The default start time is current date and time on the computer while the end time is 1 day after the default start time.

Alternatively once the initial date of the simulation has been set the user can define the duration of simulation in the duration section using the day, hour, minutes and second fields.

The default duration of simulation is 1 single day.

A Gantt graphic calendar visualization of the simulation period is presented on the right as additional aid to comprehend the extension and make corrections if needed. Right-click on the Gantt chart, a selection list of options to view the chart would pop up. Users can direct to a specific date, choose their view type preference and adjust time scales.

"Reset time period" button allows to clear all settings and reset the time and duration to default values.

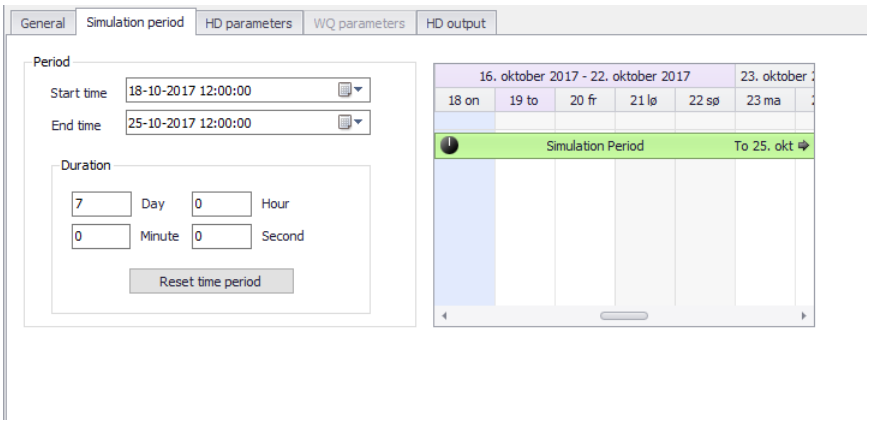


Figure 9.3 Layout of Simulation Period Settings Tab

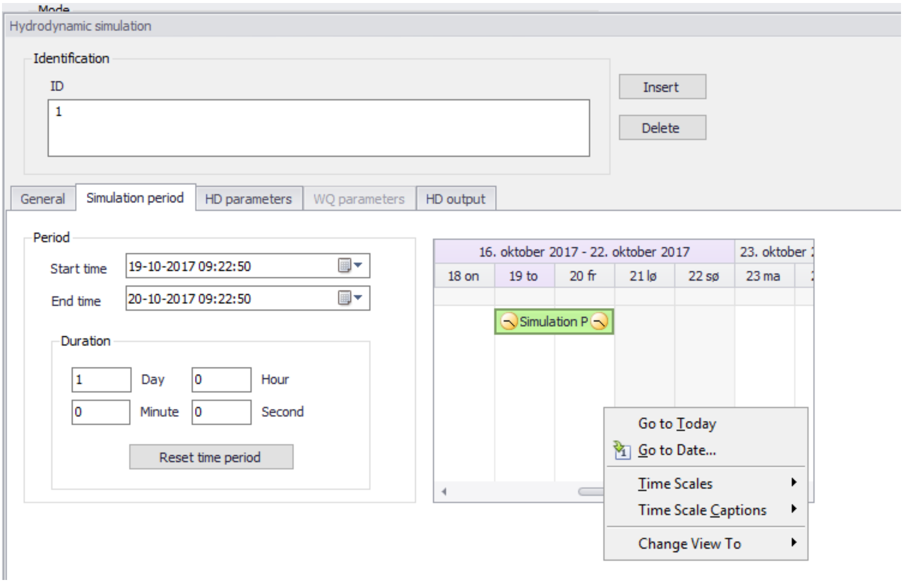


Figure 9.4 Layout of Gantt Chart View Option in Simulation Period tab

9.1.2 HD parameters

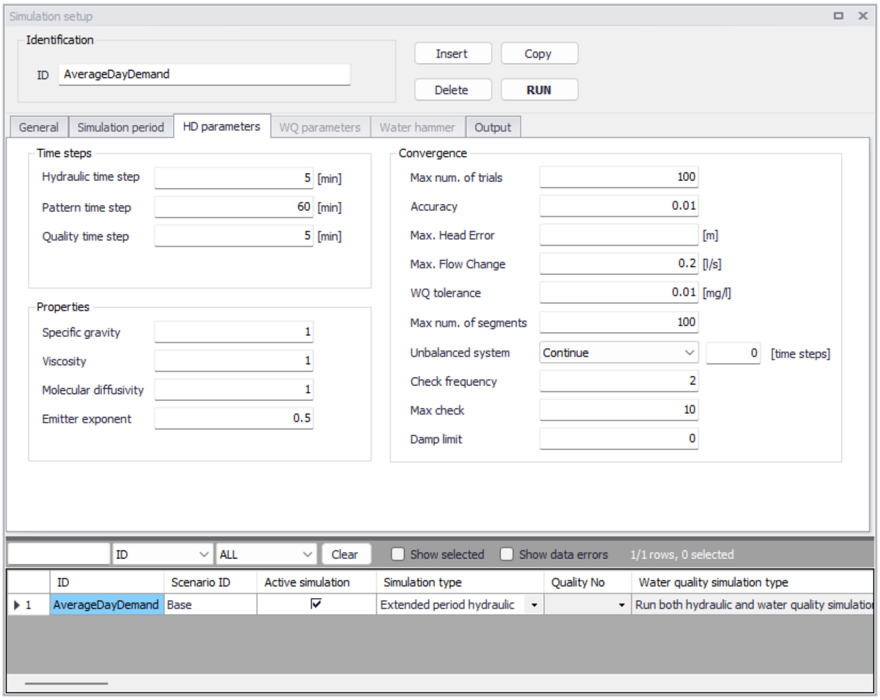


Figure 9.5 Layout of HD Parameters Tab

Time Step

This section defines the time step of each simulation run, such as hydraulic time step, pattern time step and quality time step.

Hydraulic time step

The time step (sometimes called the time interval), which is used to model the simulation in steps, that is how often a new hydraulic computation of the pipe network system is to be computed. This is typically 5 minutes by default.

Pattern time step

This data entry is optional, and specifies the length of time between each pattern change (e.g., the period of time over which water demands and constituent source strengths remain constant). If necessary, MIKE+ will adjust the specified Hydraulic Time Step so that it is not greater than the specified Pattern Time Step. The default value is 5 minutes.

Quality time step

This data entry is used for water quality analysis, and specifies the time step to be used to track water quality changes in the pipe network system. If this entry is left blank, the program then uses an internally computed time step based upon the smallest time of travel through any pipe in the network. The default value is 5 minutes.

Properties

These data entries allow you to determine the hydraulic and water quality behaviour of the pipe network should be analysed.

Specific Gravity

This data entry specifies the specific gravity of the fluid at the temperature condition being simulated. This data entry allows fluids other than water to be simulated. Gravity is the weight per unit volume of the fluid being modelled relative to water. Specific gravity is the ratio of the density of the fluid being modelled to that of water at 4 deg. C. (unitless).

Viscosity

This data entry specifies the kinematic viscosity of the fluid at the temperature condition being simulated. The units of viscosity are ft2/sec (or m2/sec for SI units). The viscosity is the kinematic viscosity of the fluid being modelled relative to that of water at 20 deg. C (1.0 centistoke). The default value is 1.0.

Molecular Diffusivity

This data entry specifies the molecular diffusivity of the chemical being tracked. The diffusivity is the molecular diffusivity of the chemical being analysed relative to that of chlorine in water. The default value is 1.0. Diffusivity is only used when mass transfer limitations are considered in pipe wall reactions. A value of 0 will cause MIKE+ to ignore mass transfer limitations.

Emitter Exponent

Power to which pressure is raised when computing the flow through an emitter device. The textbook value for nozzles and sprinklers is 0.5. This may not apply to pipe leakage.

Convergence

This section allow you to determine how the hydraulic and water quality behaviour of the pipe network should be analysed.

Maximum numbers of trials

Accuracy

Convergence criterion used to signal that a solution has been found to the nonlinear equations that govern network hydraulics. Trials end when the sum of all flow changes divided by the sum of all link flows is less than this number. Suggested value is 0.001.

Max Head Error

Convergence criterion requiring that the head loss computed by the head loss formula compared to the difference in nodal heads across each link be less than the specified value. When the value is 0 or empty, the criterion is ignored. This criterion is only available when using the EPANET 2.2 version.

Max. Flow Change

Convergence criterion requiring that the largest absolute flow change between the current and previous solutions be less than the specified value. When the value is 0 or empty, the criterion is ignored. This criterion is only available when using the EPANET 2.2 version.

Water Quality Tolerance

Smallest change in quality that will cause a new parcel of water to be created in a pipe. A typical setting might be 0.01 for chemicals measured in mg/L as well as water age and source tracing. The Quality Tolerance determines when the quality of one parcel of water is essentially the same as another parcel. For chemical analysis this might be the detection limit of the procedure used to measure the chemical, adjusted by a suitable factor of safety. Using too large a value for this tolerance might affect simulation accuracy. Using too small a value will affect computational efficiency.

Maximum Number of Segments

Maximum number of segments, which could be generated for a pipe during the water quality analysis. The default is left as blank.

Unbalanced System

Action to take if a hydraulic solution is not found within the maximum number of trials. Choices are STOP to stop the simulation at this point or CONTINUE to use extra trials, with no link status changes allowed, in an attempt to achieve convergence. For the CONTINUE option, the number of extra trials must be specified.

CheckFreq

This sets the number of solution trials that pass during hydraulic balancing before the status of pumps, check valves, flow control valves and pipes connected to tanks are once again updated. The default value is 2, meaning that status checks are made every other trial. A value equal to the maximum number of trials would mean that status checks are made only after a system has converged. (Whenever a status change occurs the trials must continue since the current solution may not be balanced.) The frequency of status checks on pressure reducing and pressure sustaining valves (PRVs and PSVs) is determined by the DampLimit option (see below).

MaxCheck

MAXCHECK is the number of solution trials after which periodic status checks on pumps, check valves flow control valves and pipes connected to tanks are discontinued. Instead, a status check is made only after convergence is achieved. The default value is 10, meaning that after 10 trials, instead of checking status every CheckFreq trials, status is checked only at convergence.

DampLimit

This is the accuracy value at which solution damping and status checks on PRVs and PSVs should begin. Damping limits all flow changes to 60% of what they would otherwise be as future trials unfold. The default is 0 which indicates that no damping should be used and that status checks on control valves are made at every iteration. Damping might be needed on networks that have trouble converging, in which case a limit of 0.01 is suggested.

9.1.3 WQ parameters

The WQ parameters tab allows to specify the rate at which a constituent decays (or grows) by reaction as the constituent travels through the pipe network. It can be enabled only when the water quality module is ticked in ‘Model type’ editor.

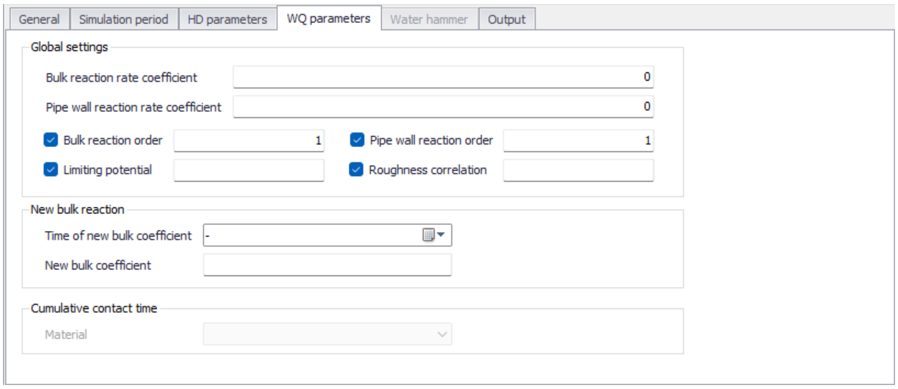


Figure 9.6 Layout of WQ Parameters Tab

Global Settings

Bulk Reaction Rate Coefficient

This data entry defines the bulk reaction rate that is applied to all flow in the pipe network system. Units for bulk reaction rates are in days–1. Note that this reaction rate coefficient is applied globally to the entire pipe network.

Pipe Wall Reaction Rate Coefficient

This data entry defines the pipe wall reaction rate that is applied to all flow in the pipe network system. Units for pipe wall reaction rates are in ft/day (or m/day). Note that this reaction rate coefficient is applied globally to the entire pipe network.

One method that can be used to compare the relative magnitude of the pipe wall reaction rate with the bulk reaction rate is to divide the pipe wall reaction rate coefficient by the hydraulic radius of the pipe (i.e., 1/2 the pipe radius). The resulting quantity will have the same units as the bulk reaction rate coefficient, days–1.

Bulk Reaction Order

Power to which concentration is raised when computing a bulk flow reaction rate. Use 1 for first-order reactions, 2 for second-order reactions, etc. Use any negative number for Michaelis-Menton kinetics. If no global or pipe-specific bulk reaction coefficients are assigned then this option is ignored.

Pipe Wall Reaction Order

Power to which concentration is raised when computing a bulk flow reaction rate. Choices are 1 for first-order reactions or 0 for constant rate reactions. If no global or pipe-specific wall reaction coefficients are assigned then this option is ignored.

Limiting Potential

This setting specifies that reaction rates are proportional to the difference between the current concentration and some limiting potential value. This value can be locally overwritten by limiting potential values specified in water quality settings of the 'Pipes' editor.

Roughness Correlation

This setting will make all default pipe wall reaction coefficients be related to pipe roughness in the following manner.

New Bulk Reaction

At a certain time level, the bulk coefficient will change to a new value. This section defines the new value of bulk coefficient and the time the new bulk coefficient would start. After the start time, the simulation would apply the new bulk coefficient for calculation.

9.1.4 Output

In this section, users can select from the following to store simulation results:

- Use default folder and file name: save outputs in the folder containing the MIKE+ project
- Use this folder: save outputs in a custom folder but with a default file name
- Use this folder and file name: save outputs in a custom folder and with a custom file name

Report time step: time interval between times at which computed results are reported. Normal default is 1 hour.

Report start time: time that the report starts. For example, if the report start time is 5 hours, the report would start 5 hours later from the simulation start time.

Statistics: Type of statistical processing used to summarize the results of an extended period simulation. Choices are:

- Without Statistics (results reported at each reporting time step)
- Average (time-averaged results reported)
- Minimum (minimum valve results reported)
- Maximum (maximum valve results reported)
- Range (difference between maximum and minimum)

Report raw results: when this option is selected, the hydraulic results will be reported "as computed" regardless of the physical meaning of the values. In some cases, this may result in showing very large negative pressures "infinitely high" and flows in pipes where it is not possible to supply water due to negative pressures.

Save hydraulics file: when this option is selected, a hydraulics file is saved from the hydraulics simulation, for later use as input for a decoupled water quality simulation. This option is only available for 'Steady state simulation' and for 'Extended period hydraulics'.

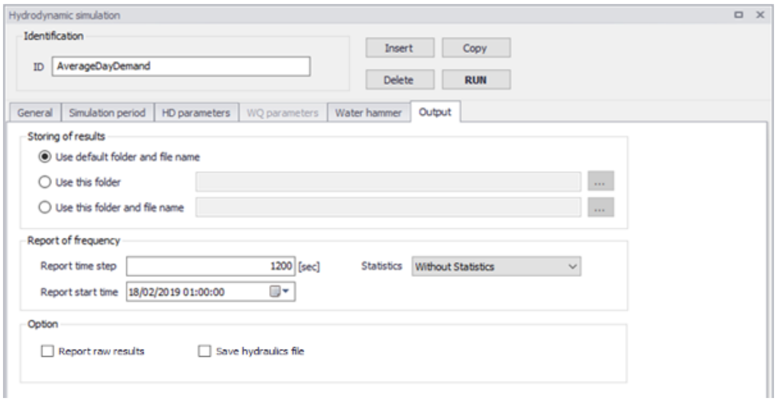


Figure 9.7 Layout of Output Setup

A predefined list of result items is saved in the result files during the simulation. This list is supplemented by a number of "derived items", which are computed in memory when the result files are loaded in MIKE+. The following table lists these two types of result items.

Table 9.1 Summary of result items available in result files and as derived items

Items saved in result files	Derived items
Node water demand (outflow, inflow) Node head (hydraulic grade line) Node pressure Node water quality (water age, source trace, chemical) Tank water demand (outflow, inflow) Tank head (water level elevation) Tank pressure (water depth) Tank water quality (water age, source trace, chemical) Link (pipe) flow Link (pipe) velocity (absolute) Link (pipe) headloss per 1000 Link (pipe) water quality Link (pipe) status code (open, closed, check valve) Link (pipe) setting (friction) Link (pipe) reaction rate Link (pipe) friction factor Valve flow Valve velocity Valve headloss per 1000 Valve water quality Valve status code (closed, open, active) Valve setting Valve reaction rate Valve friction factor Pump flow Pump velocity Pump headloss per 1000 Pump water quality Pump status code (closed, open) Pump setting (relative speed) Pump reaction rate Pump friction factor	Link (pipe) flow (absolute): absolute value of the flow in the pipe. Link (pipe) pressure gradient: Difference in pressure grade line between beginning and end nodes of the pipe. Valve flow (absolute): absolute value of the flow in the valve. Valve pressure gradient: Difference in pressure grade line between beginning and end nodes of the valve. Pump flow (absolute): absolute value of the flow in the pump. Pump pressure gradient: Difference in pressure grade line between beginning and end nodes of the pump. Link Accumulated flow: Volume accumulated over time, computed from the link flow time series. The accumulated value increases when the flow is positive and decreases when the flow is negative. Link Accumulated Flow (Absolute): Volume accumulated over time, computed from the absolute link flow time series. The accumulated value always increases, even when the flow changes direction.

Refer to the Results Presentation chapter (in MIKE+ Model Manager User Guide) for mode details about loading results in MIKE+.

9.2 Batch Runs

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 'Add to batch' flag set in the sequence that they are specified in the table. This means that multiple simulations and scenarios can be simulated in batch without user interaction.

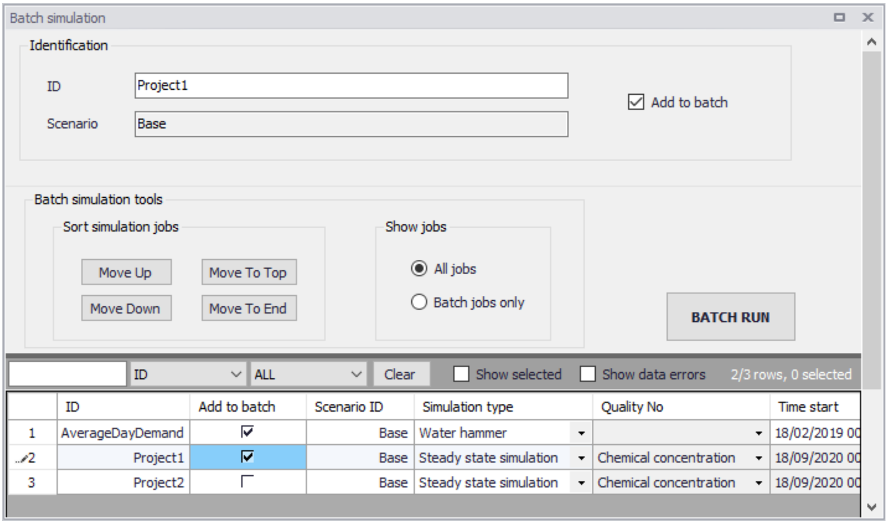


Figure 9.8 The Batch Simulation Editor

The Batch Simulation editor manages the same data from the Simulation Setup editor. The table shows the same entries as the grid in the Simulation Setup editor, but built-in tools allow reordering and filtering of simulations for batch execution.

Table 9.2 Overview of Batch Simulation editor fields (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in data structure
ID	ID of the hydrodynamic simulation	Yes	MUID

Table 9.2 Overview of Batch Simulation editor fields (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in data structure
Scenario	Scenario for the hydro-dynamic simulation	Yes	Scenario ID
Add to batch check box	Option for including a hydrodynamic simula-tion 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.

10 Water Distribution Toolbox

10.1 Sustainability Analysis

Sustainability Analysis is not a simulation but a way of reporting results in a way that will help the user understand possible problems in the model.

The tool helps understand WD simulation results and analyze them for possible problems, anomalies, critical areas, and similar. Various predefined thematic maps are available including:

- Unit headloss to determine pipe size problems
- Reverse flows to identify possible water quality issues
- Service pressures
- Pressure, velocity, and other anomalies

10.1.1 Sustainability Analysis Dialog

The Sustainability Analysis tool can be opened from the ‘WD analyses’ tools from the ‘Results’ tab of the ribbon.

You can define the following settings:

Result file
Result file name

Flow threshold
Minimum flow criteria used for reverse flow calculation.

Map layers
Select what layers will be added to the Map:

- Service pressure
- Unit headloss
- Pipe flow

Report
Select sections included for reporting:

- Storage tanks
- Pumps
- Unit headloss
- Service pressure
- Pipe flow
- Report each time level (pipes and nodes): Please note that selecting each time level can result in excessive processing time

'Create' button

Perform the sustainability analysis

'Report' button

Create a report

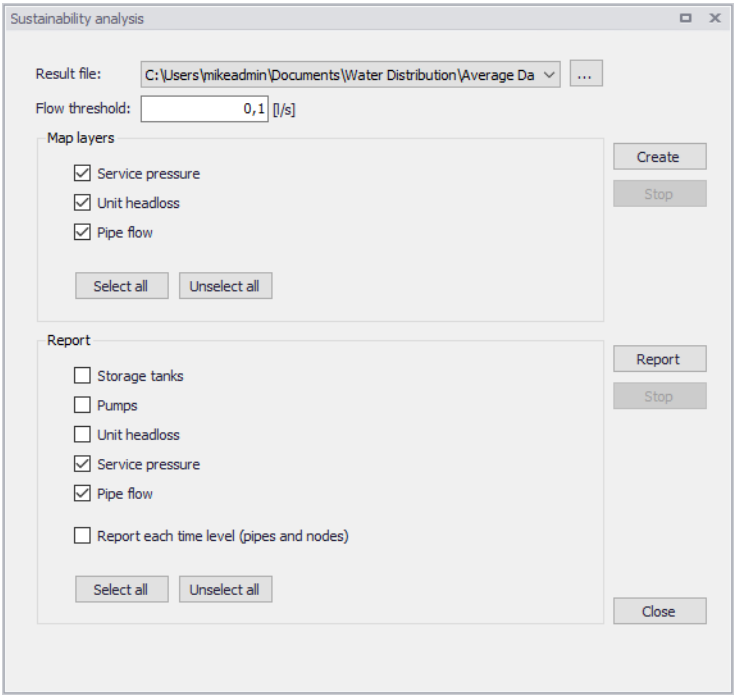


Figure 10.1 The Sustainability Analysis dialog

10.1.2 Results Presentation

Results from running the tool are added to the Map (Figure 10.2).

The tool provides a detailed analysis of the simulation results, and it will create the following layers. The symbology of each layer is predefined, but may be changed via the Symbology Settings editor from the Layers and Symbols panel.

- Unit head loss: Link head loss per 1000 (see Figure 10.2)
- Service pressures
  - Pressure anomalies e.g. pressure too low or too high
  - Pressure range: Pressure distribution
  - Pressure fluctuation: Difference between the minimum and maximum pressure at every node during the simulation.
- Pipe flows

- Reverse flow: The layer will show how many times the flow direction has changed in every pipe. Note that a threshold value needs to be specified e.g. "0.1" , meaning that if the absolute flow is smaller than that, the pipe is not considered for reporting, i.e. the flow must be smaller than -0.1 and greater than 0.1 to be considered as a pipe with flow.
- Flow velocity: Velocity distribution
- Flow velocity fluctuation

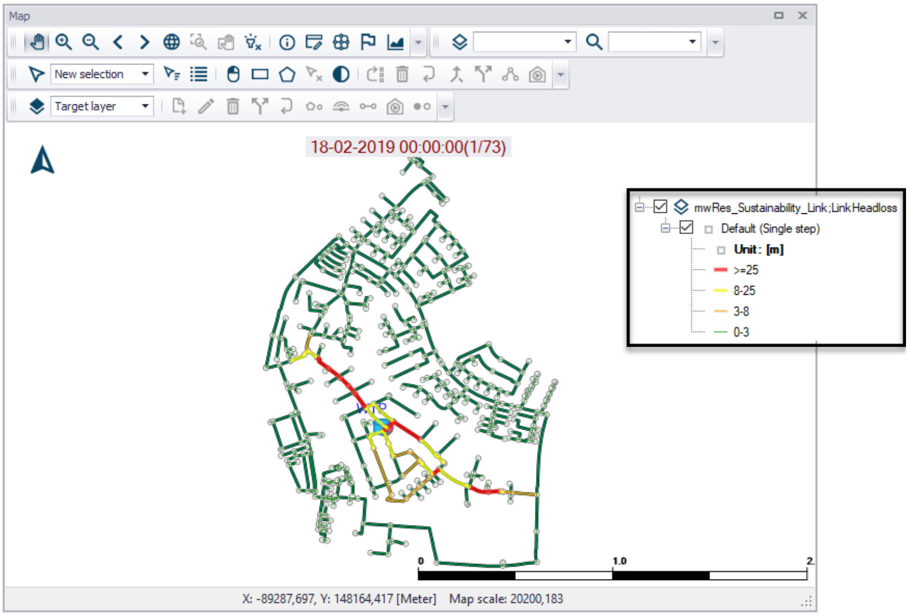


Figure 10.2 Example Sustainability Analysis Unit Head Loss results on the Map

10.1.3 Reports

In addition to the Map layers, detailed reports can be generated to understand the operation of pumps, storage tanks, and other facilities (Figure 10.3). Click on the ‘Report’ button on the Sustainability Analysis dialog (Figure 10.1) to generate a report.

The Sustainability Analysis report uses a pre-set template and may contain the following information:

- Storage tanks: Reports tanks that are either drained or overflows during the simulation.
- Reports if the tanks are balanced within the simulation. Balanced tank is a tank where the water level at the beginning of the simulation is the same as at the end of the simulation.
- Pumps: Reports pump that are operated near their maximum capacity

- Unit Headloss: Reports pipes with too high unit head loss
- Service Pressures: Reports excessive pressures
- Pipe flows: Reports reverse flows
- Flow velocity: Reports excessive flow velocity

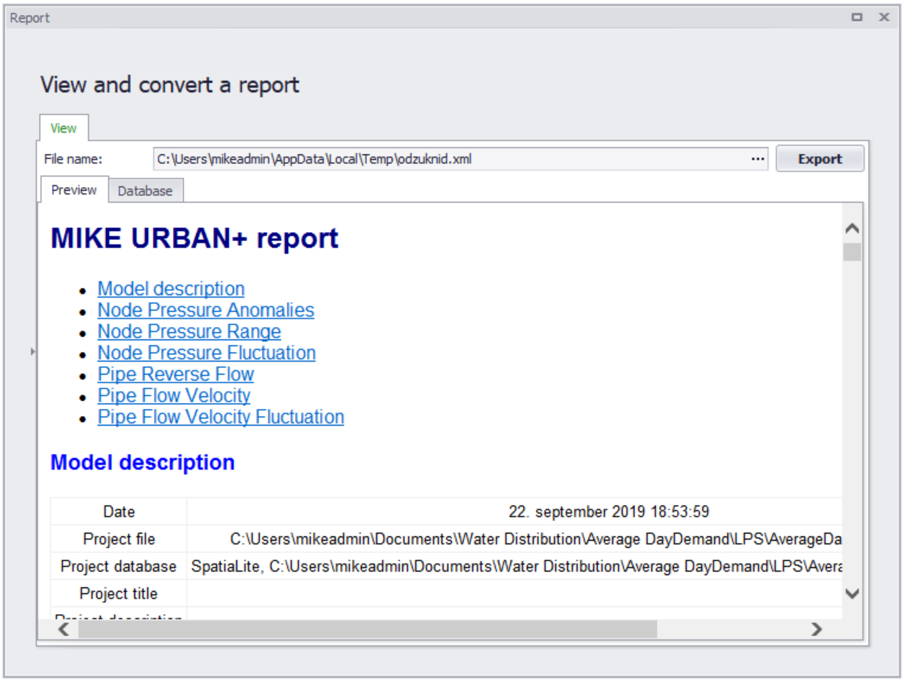


Figure 10.3 Example Sustainability Analysis Report

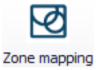
10.2 Zone Mapping

10.2.1 Create zones from network separators

This tool creates zones based on the network topology and geometry, closed pipes, closed valves, and pumps.

Defining zones helps to visualise how different network parts are hydraulically interconnected and where the HGL line breaks. It helps understand the hydraulic behaviour of the network prior to running the hydraulic simulation, and also helps detect possible errors in the network connectivity.

Launch the ‘Create zones from network separators’ tool from the WD Network ribbon.



On the ‘Create zones from network separators’ dialog, define the breakdown rules of the network, the separators, and the rules for merging small groups of pipes into one big group:

- Separators: Separators are links, which will be used to separate one zone from another. Separators can be:
  - Closed link: any type of closed link will separate zones, e.g. pipes, valves or pumps.
  - Pump: all pumps will act as separator, regardless of their opened / closed status.
  - Vale types (PRV, PSV, PBV, FCV, TCV, GPV) : all selected valve types will act as separator, regardless of their status (regulating / opened / closed).
  - Check valve
- Merge zones smaller than: In case that there are many small zones (a typical example would be small pipes located in pumping stations and storage tanks), they will be all merged into the same zone for graphical display instead of creating a separate zone for each of them.
- Save zones: The user can decide whether to save the zones to the zone editor or not. A zone category should be defined, then it would automatically create all zones named after the category, e.g. Zone_mapping_1.

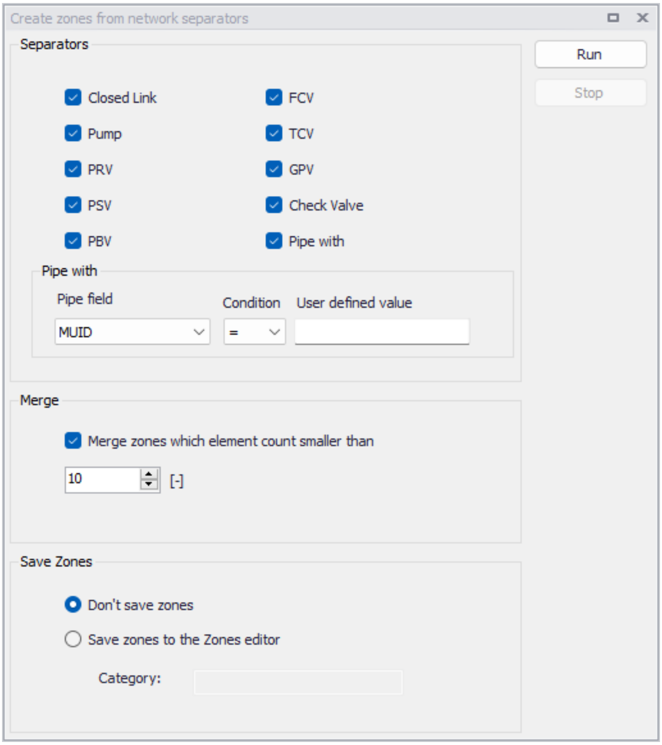


Figure 10.4 The 'Create zones from network separators' dialog

Run the analysis using the 'Run' button. This produces a result layer, which is automatically loaded on the Map (Figure 10.5).

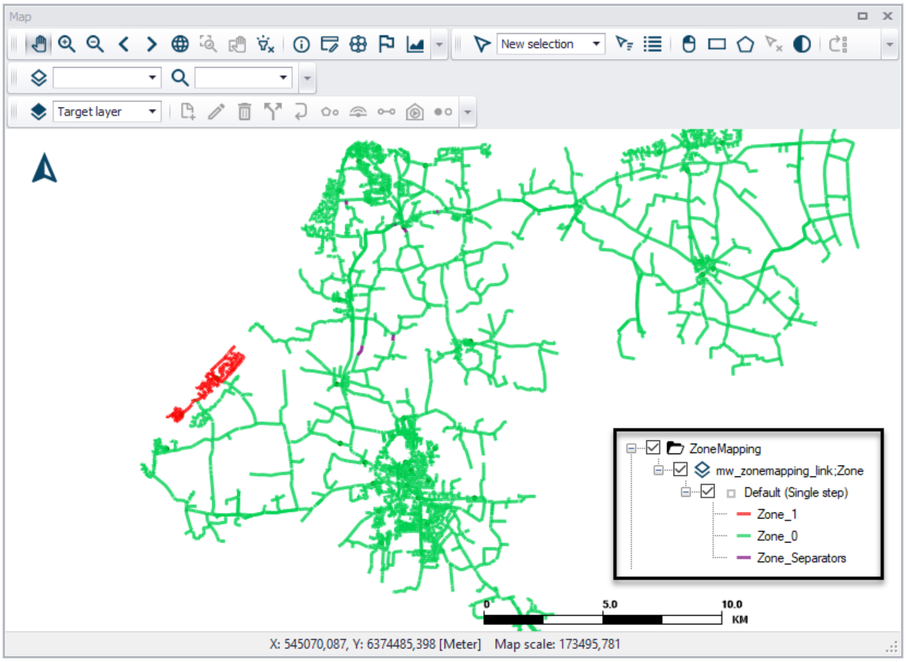


Figure 10.5 Example Zone Mapping result plotted on the Map

Running the tool also automatically generates a report with information of different zones, such as the number of links in each zone, the number of separators and merged zones (Figure 10.6).

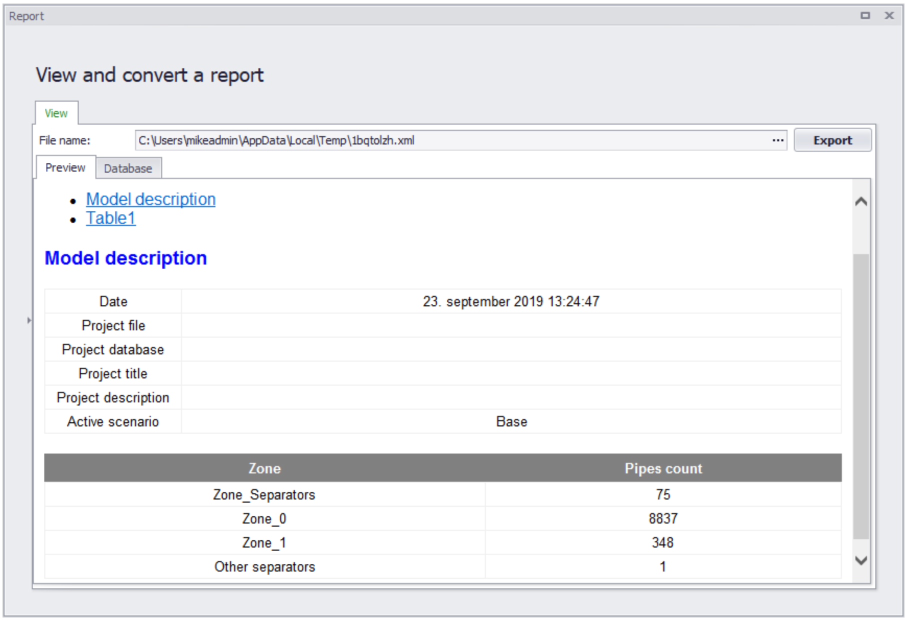


Figure 10.6 Example Zone Mapping report

10.2.2 Create zones from GIS layer

This tool automates the creation of new zones using a polygon shape file to read the zones' extents from.

Defining zones helps to visualise how different network parts are hydraulically interconnected and where the HGL line breaks. It helps understand the hydraulic behaviour of the network prior to running the hydraulic simulation, and also helps detect possible errors in the network connectivity.

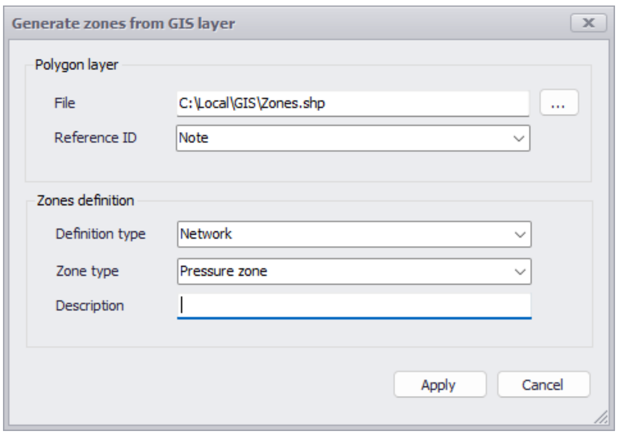


Figure 10.7 Creating new zones from a GIS layer

10.3 Valve Criticality

The Valve Criticality tool allows analysis of a valve from the valve layer to determine which valves need to be closed in order to replace the selected valve. This tool helps you to understand the important of isolation valves and assists you in the valve maintenance and replacement program.

Launch the Valve Criticality tool from the WD Analysis Toolbox on the WD Network ribbon.

On the Valve Criticality dialog that appears, define the layers containing your pipe network and valves. Valve criticality can operate in two modes:

- Interactive mode: Allows you to inspect valves one by one by pointing and clicking on a valve.
- Automatic mode: Allows you to run the tool for selected valves and store the results in the database.

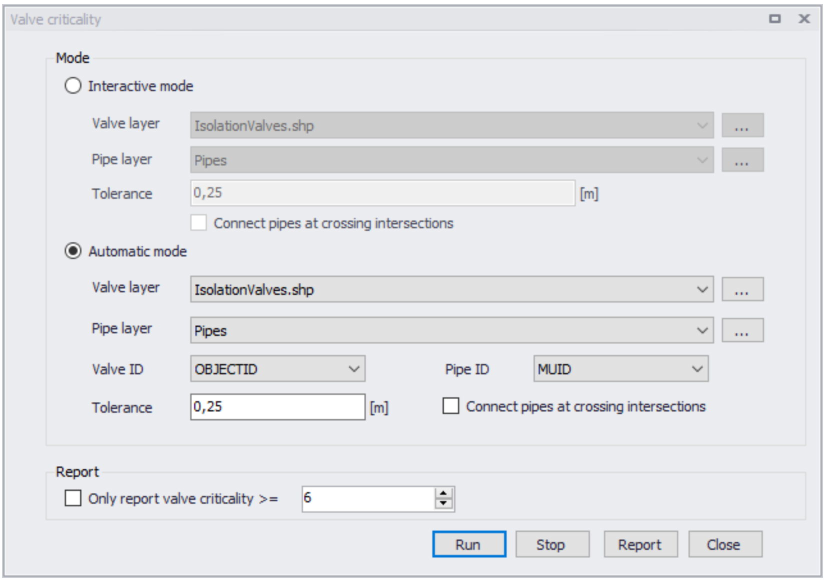


Figure 10.8 The Valve Criticality dialog

10.3.1 Settings

A list of the Valve Criticality data entries for Figure 16.1 follows, with a short description given for each entry.

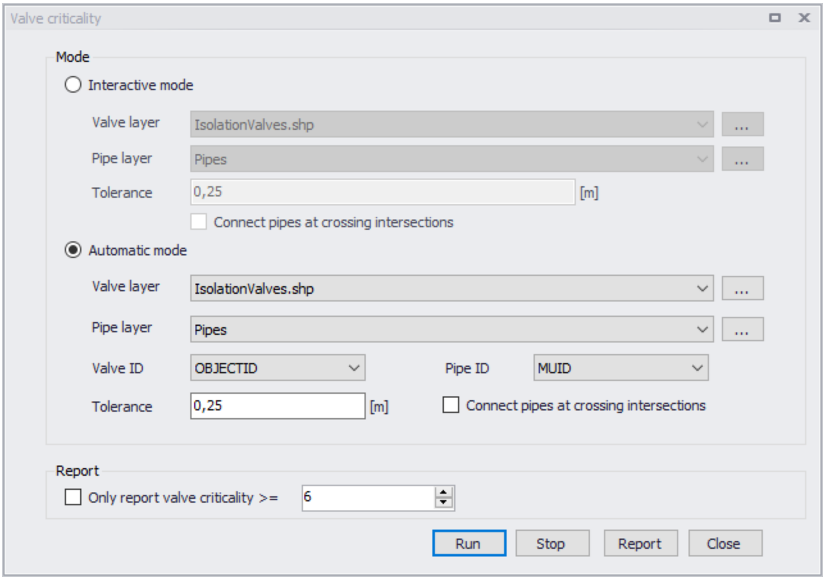


Figure 10.9 The Valve Criticality dialog box is used to define the analysis parameters

Interactive mode

This mode allows you to run the valve criticality analysis in interactive mode when you click the valve in the Map and the program finds substitute valves.

Automatic mode

This mode allows you to run the valve criticality analysis in automatic mode for any number of selected valves.

Valve layer

This data entry allows you to select the GIS layer with valves (typically isolation valves) that will be used in the valve criticality analysis. Please note, that in order to select valve layer in this data entry, the valve layer needs to be already added to the Map layers beforehand.

Pipe layer

This data entry allows you to select the layer with pipes. It could be a model pipes layer or GIS pipe layer. Please note, that in order to select a GIS pipe layer in this data entry, the layer needs to be already added to the Map layers beforehand.

Tolerance

This data entry allows you to define the spatial tolerance that will be used to track the pipe network connectivity.

Connect pipes at crossing intersections

When the pipe layer is defined from a GIS layer, this option allows you to define whether the pipe network connectivity will consider pipes connected whenever they cross each other. In case of a pipe layer from the hydraulic model this option would not be used because connecting pipes require a junction node at their cross connection. However, in case of a GIS layer this option could be required in order to track the connectivity.

Valve node ID

This data entry allows you to define the valve identification ID that will be used by the program for reporting purposes.

Pipe node ID

This data entry allows you to define the pipe identification ID that will be used by the program for reporting purposes.

Running analysis

Click the "Run" button in order to run the analysis. In case of "interactive" analysis the program will allow you select the valve from the Map window. In case of "automatic" analysis, the program will start analysing all valves and the simulation progress will be displayed in the application status window. The analysis can be interrupted (cancelled) by pressing the 'Esc' key or the 'Stop' button.

10.3.2 Valve criticality results

Results of the Valve Criticality tool are added to the Map where valves are assigned symbols based on the number of substitute valves (Figure 10.10).

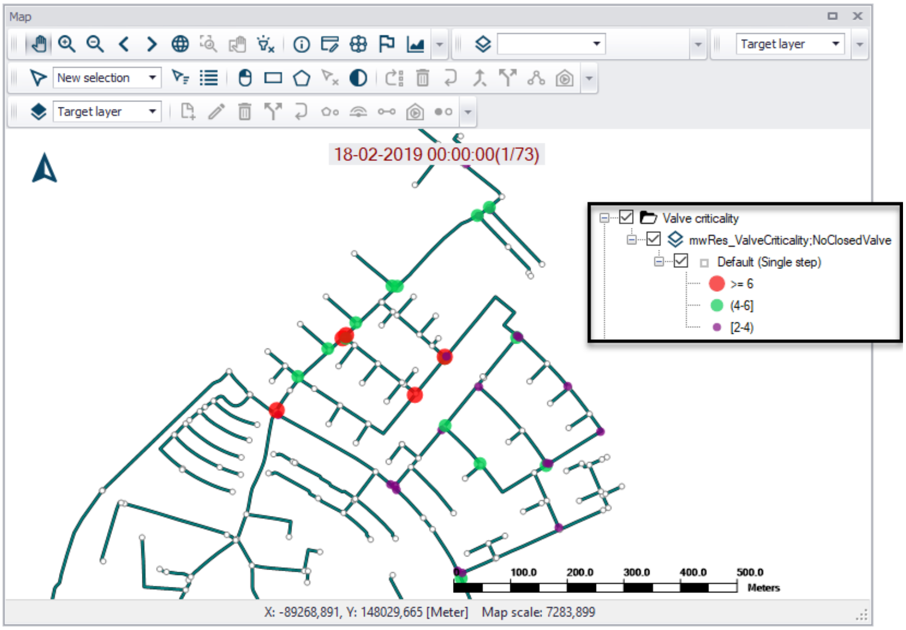


Figure 10.10 Example Valve Criticality result plot on the map sizing and coloring valves according to importance/criticality

Valve criticality results include:

- **No of closed valves**: Number of other valves that need to be closed in order to replace the malfunctioning (selected) valve
- **Closed valves**: List of such valves
- **Closed pipes**: List of pipes that are contained within the pipe network area isolated by valves
- **Sum length of pipe**: Length of such pipe network

Click on the ‘Report’ button on the dialog to generate a report on these Valve Criticality results (Figure 10.11).

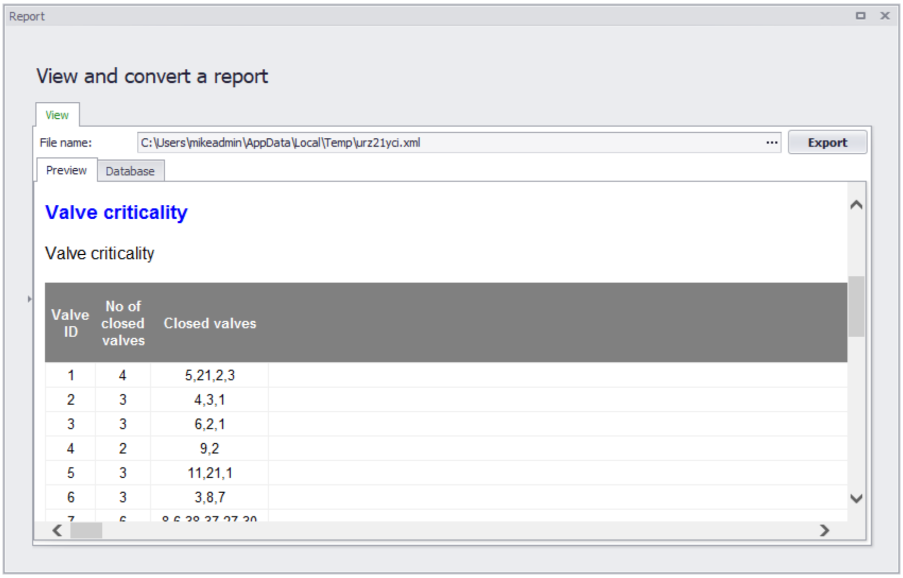


Figure 10.11 Example Valve Criticality report

10.4 Alarms and Violations

The Alarms and Violations tool provides a way to impose user-defined checks for Water Distribution model results. It allows for quick examination of the performance of elements that are important to the WD system, or of particular interest to the user.

Launch the Alarms and Violations tool from the Results ribbon. The tool is only available for Water Distribution models.

Alarms and violations allows definition of critical values for various result items anywhere within the model network, such as maximum velocity, minimum or maximum pressure, low or high level, and high water age, and let the hydraulic model evaluate them based on the actual simulation results.

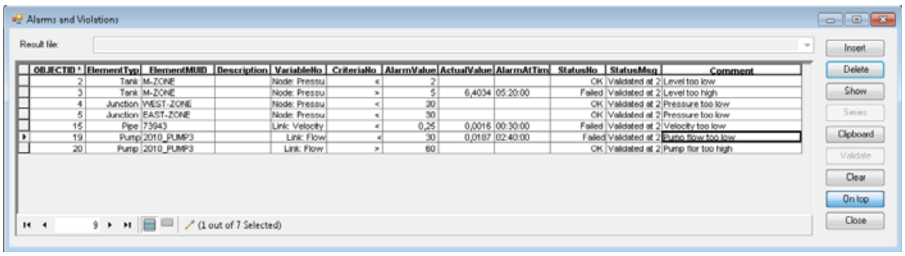


Figure 10.12 The Alarms and Violations dialog

The dialog table (Figure 10.12)allows you to define or display the following parameters:

- **Element Type**: Select from the following types:
  - Junction
  - Tank
  - Pipe
  - Pump
  - Valve
- **Element ID**: The MUID of the element. Use a ‘*’ to apply the criterion to all elements of a selected type.
- **Description**: Optional user-defined description
- **Variable No**: Select the result item from available items according to element type.
- **Criteria**: Basic math operator (i.e. <, <=, =, etc.) for building the criterion.
- **Alarm Value**: Set the critical (alarm) value.
- **Actual Value**: Validation result showing the highest or lowest of the actual values resulting in the alarm.
- **Alarm at Time**: Validation result showing the time of the simulation corresponding to the "Actual value".
- **Status**: Status of the validation, "OK" (No violations) or "Failed" (With violations).
- **Status Message**: Information on the results validation (i.e. Date/time of validation)
- **Comment**: User-defined comment

Note that the alarm will be triggered if the criterion is fulfilled i.e. if the criterion is defined for a "Tank level < 2" then the alarm will be reported once the computed level is below "2".

Button functionalities on the dialog include:

Insert

Adds a new record to the table.

Delete

Deletes the highlighted record from the table.

Highlight

Highlight specified elements on the Map. (see Figure 10.13)

Clear highlight

Removes highlight of specified elements on the Map.

TS plot

Displays the result time series for the selected alarm item (Figure 10.14). A tabular view of the time series results is also available from the TS Plot window (Figure 10.15).

Clipboard

Copies the content of the table into a clipboard.

Validate

Performs the results validation for the selected results file.

Clear

Clear results from the last validation.

Duplicate

Makes a copy of a selected alarm setup record.

Note that the above mechanism allows you to define certain checks for specific locations (e.g. junctions and pipes), as well as for all locations when the '*' is used instead of the MUID for the Network ID column.

For example, the criterion "Junction * pressure > 100" will map all junction nodes where the pressure was more than 100 during the simulation.

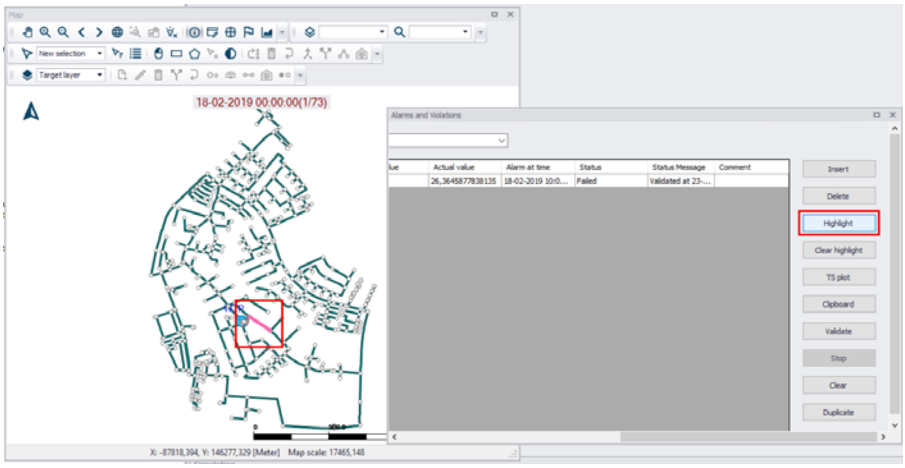


Figure 10.13 Highlight elements on the map using the 'Highlight' button on the dialog

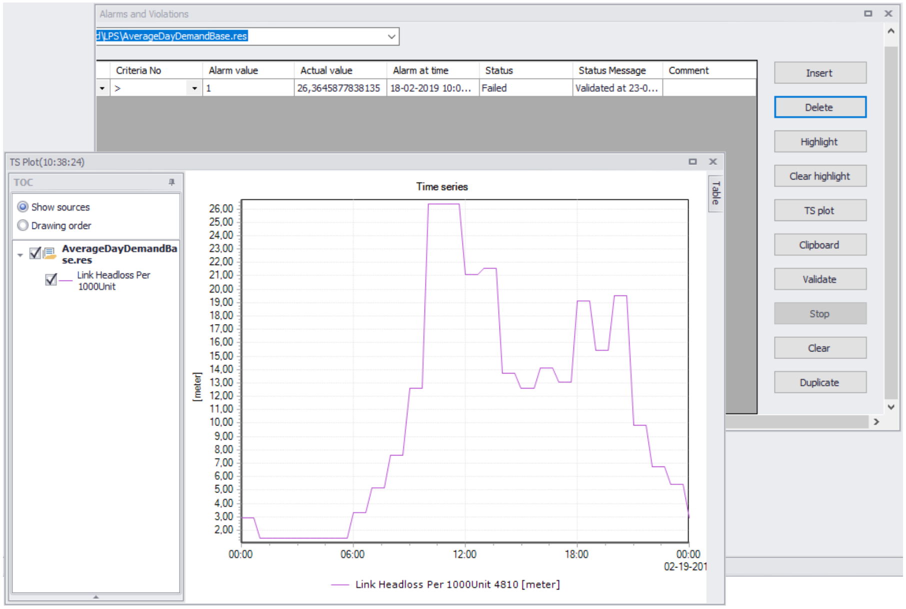


Figure 10.14 Example time series plot generated from the 'TS plot' button on the dialog

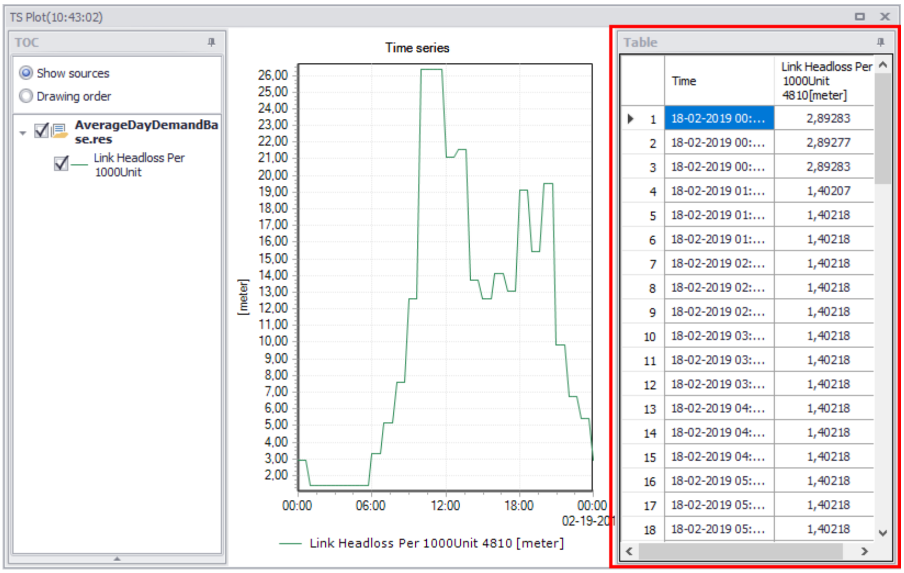


Figure 10.15 A tabular view of time series results is also possible from the TS Plot window

11 Fire Flow Analysis

The Fire Flow Analysis module allows you to calculate the available flow for the design pressure or to calculate the residual pressure for the design flow. Fire flow requirements are one of the most common design requirements when designing the new or evaluating the existing water supply and water distribution system.

A fire flow is the maximum flow rate available at a specific minimum pressure, typically 20 psi (15m). There are three basic ways to model a fire flow:

- Specify a design fire flow rate and compute the available fire flow pressure.
- Specify a design fire flow pressure and compute the available fire flow rate.
- Compute both hydrant discharge and residual pressure for a free discharge hydrant.

The Fire Flow Analysis dialog box is reached by selecting ‘Model type’ from General Settings from the Table of Contents and then by selecting the ‘Fire flow analysis’ option. The "Fire flow analysis" entry will be added into the Table of Content of the model "Setup" under "Special Analyses" group. Note that to run the fire flow analysis, you need to select *Run* from within the Fire Flow Analysis dialog box.

A unique feature of MIKE+ is its capability of computing a fire flow for fire hydrants that are not part of the hydraulic model and that can be specified by using a reference GIS file.

A list of the Fire Flow Analysis dialog box data entries for Figure 11.1 follows, with a short description given for each entry. Note, that it is possible to insert multiple fire flow analysis, each with its own settings, and then run the selected fire flow simulation by selecting it from the list. This is convenient when you need to investigate fire flow capacity of the network under different conditions.

Insert
This button is used to insert a new Fire flow analysis into the list.

Delete
This button is used to delete a Fire flow analysis from the list.

Run
This button is used to run the simulation for the active Fire flow analysis.

Stop
This button is used to stop the fire flow analysis that is currently running.

Report

This button is used to generate a report from the fire flow simulation.

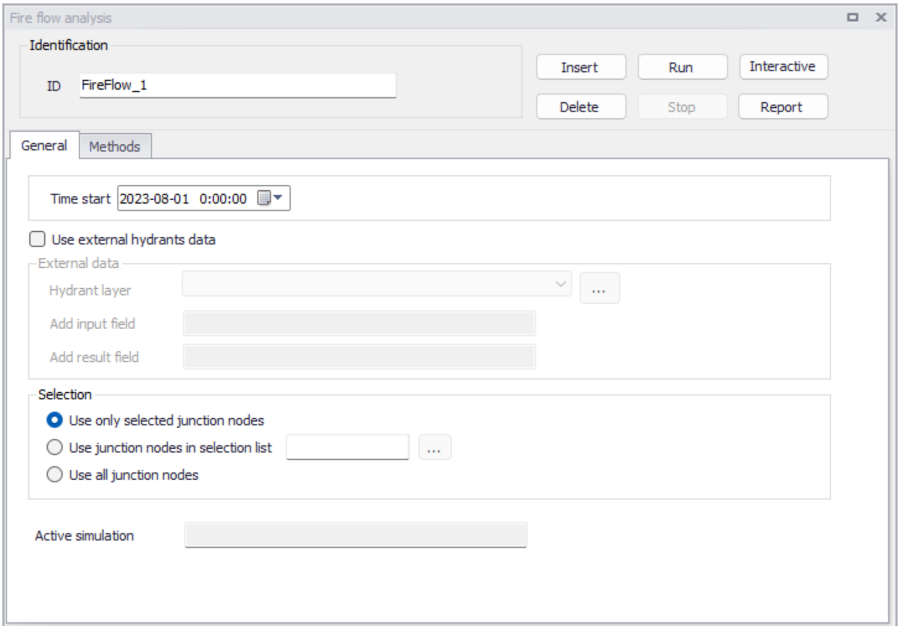


Figure 11.1 The Fire Flow dialog box is used to specify fire flow analysis parameters

Interactive

Using the 'Interactive' button it is possible to run the fire flow simulation interactively with simplified data entry for a specific node.

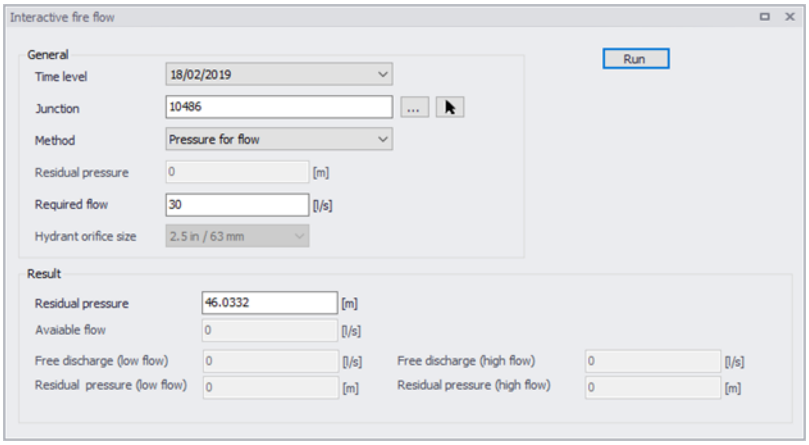


Figure 11.2 The Fire Flow dialog box used in Interactive simulation

Time level

This data entry is used to define the time (time level) when the fire flow will be simulated. If you select e.g. 9:00 AM, the program will run the standard hydraulic simulation from the beginning of the simulation to the time level corresponding to 9:00 AM and then the fire flow will be computed.

Junction

This data entry defines the node used for the fire flow simulation. It is possible to select the node from the list of all junction nodes by selecting "…" button or click the node in the Map by using the "arrow" button.

Method

Three methods are available:

- Pressure for flow: specify a design fire flow rate and compute the corresponding available fire flow pressure.
- Flow for pressure: specify a design fire flow pressure and compute the corresponding available fire flow rate.
- Free discharge hydrant: compute both hydrant discharge and residual pressure for a free discharge hydrant.

Residual pressure

This data entry allows you to define the required design pressure that will be used in the calculation when you select the method "Flow for pressure".

Required flow

This data entry allows you to define the required design flow that will be used in the calculation when you select the method "Pressure for flow".

Hydrant orifice size

This is used to select the size of the hydrant orifice, among the two available sizes:

- 2.5 in / 63 mm
- 4.5 in / 115 mm

Run

This button computes the fire flow results according to the selected method. For the 'Free discharge hydrant' method, the program runs the fire flow simulation and displays the hydrant discharge and residual pressure. The program is using experimental data to estimate the hydrant discharge coefficient based on the hydrant orifice size and it provides low and high hydrant discharge values rather than one exact value. The computed flow corresponds to the "free flow" from the hydrant orifice without pumping.

11.1 General

Time start

This data entry Is used to define the time (time level) when the fire flow will be simulated. If you select e.g. 9:00 AM, the program will run the standard hydraulic simulation from the beginning of the simulation to the time level corresponding to 9:00 AM and then the fire flow will be computed.

Use external hydrants data

This data entry is used to define an external file with fire hydrants and use them in the simulation instead of selecting model nodes.

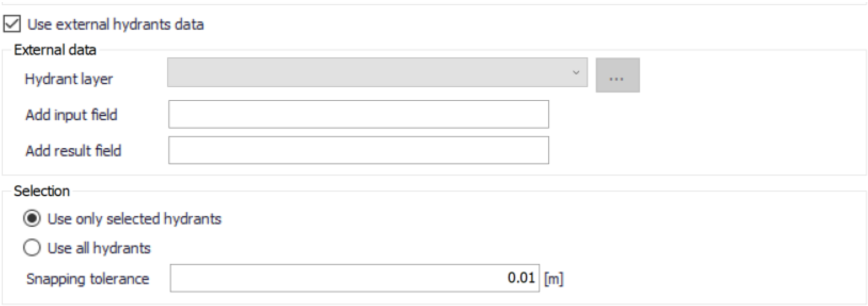


Figure 11.3 The Fire Flow dialog box using external hydrants data

Hydrant layer

This data entry allows you to select a shapefile with fire hydrants. The program will use these hydrants for the fire simulation and it will find the nearest node in the hydraulic model, run the simulation, and write the results into the hydrant layer fields.

Input field

This data entry allows you to specify a field from the hydrant layer where the program will write the input value (required pressure or required flow) for which the simulation was done.

Results field

This data entry allows you to specify a field from the hydrant layer where the program will write the computed value (residual pressure or available flow).

Selection

When external hydrant data are not used, this radio button selection allows you to define whether the fire flow analysis is performed for the selected junction nodes, for junction nodes within the selected selection list, or for all junction nodes. Use the next data entry to specify a selection list with junction node. Note, that the selection list can be defined using “Selection Manager” from the Map ribbon menu.

When external hydrant data are used, this radio button selection allows you to define whether the fire flow analysis is performed for selected hydrants only (from the layer with reference shapefile) or for all hydrants in the reference shapefile. A Snapping tolerance defines the spatial tolerance that will be used to find the node in the hydraulic model that is the nearest junction node to the hydrant.

11.2 Methods

From this tab, it is possible to run the analysis in automatic mode for a number of selected nodes.

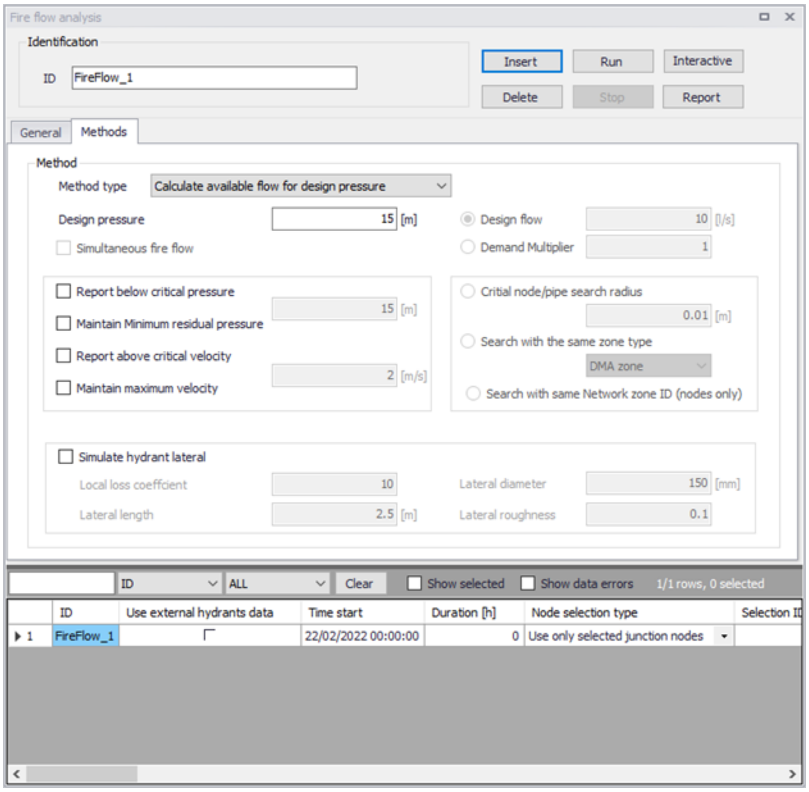


Figure 11.4 The Fire Flow dialog box is used to specify the fire flow analysis parameters

Method type

This selection allows you to select the fire flow analysis type:

- Calculate available flow for design pressure
- Calculate available pressure for design flow
- Calculate Q-H curve

It is possible to specify a design fire flow rate and compute the available fire flow pressure or to specify a design fire flow pressure and compute the available fire flow rate. In addition to this, it is also possible to calculate the Q-H curve for the selected junction node.

Design fire pressure

This data entry is used to define the design (required) residual fire pressure for which the available fire flow will be calculated.

Design fire flow

This data entry is used to define the design (required) fire flow for which the available (residual) fire pressure will be calculated. Note, that this design fire flow is added to the existing node demand(s). In other words, if the design fire flow is 6 l/s and the node has e.g. a residential category demand of 0.25 l/s, the total flow (demand) out of the node during the fire flow analysis will be 6 + 0.25 = 6.25 l/s.

Demand multiplier

This data entry allows you to specify the node demand multiplier which will be used to define the required fire flow by multiplying the node demand that is defined for the respective node in the Multiple demand editor.

Simultaneous fire flow

This data entry allows you to run the fire flow simulation (*Calculate available pressure for design flow* mode) simultaneously i.e. all selected nodes flowing at the same time. If this option is not activated, the fire flow simulation is executed for all selected nodes in a sequential manner, i.e. one node at the time. A typical use of this option would be to run the fire flow simulation for simultaneously for 3 selected nodes.

Report below critical pressure

This data entry allows you to report nodes, where the minimum residual pressure during the fire flow simulation is less than the critical pressure. The critical node pressure is entered into the field next to it.

Maintain minimum residual pressure

This data entry allows the program to make corrections to the computed fire flow and reduce the amount of available flow in order to maintain the minimum residual pressure in critical nodes.

Report above critical velocity

This data entry allows you to report pipes, where the maximum velocity during the fire flow simulation is greater than the critical velocity. The critical pipe velocity is entered into the field next to it.

Maintain maximum velocity

This data entry allows the program to make corrections to the computed fire flow and reduce the amount of available flow in order to maintain the maximum velocity in critical pipes.

Critical node/pipe search radius

This data entry allows you to define the search node and pipe radius, which will be used to identify nodes where the computed residual pressure is less than a critical pressure or pipes, where the computed velocity is bigger than a critical pipe velocity.

Search within the same zone type

This data entry allows you to define that the search for the nodes and pipes should happen within the same pressure zone, where the zone is defined by a selection. In this case, you do not need to define the search radius and the program will search for all nodes or pipes within the same zone. Note, that this search option provides more accurate results than search based on the radius.

Search within the same Network zone ID (nodes only)

This option allows you to define that the search for the nodes should happen within the same pressure zone, where the zone is defined by the network properties (using the Zone ID specified in junctions, tanks and air-chambers). In this case, you do not need to define the search radius and the program will search for all nodes within the same zone. Note, that this search option provides more accurate results than search based on the radius.

Simulate hydrant lateral

This data entry allows you to model a fire hydrant lateral (connecting) pipe at the junction node. Note that the fire flow results may significantly change with or without such pipe.

Local loss coefficient

This data entry allows you to define a local loss coefficient (sum of) representing all local losses at the fire hydrant lateral pipe. A typical value is 6-10.

Lateral diameter

This data entry allows you to define a diameter of the fire hydrant lateral pipe.

Lateral length

This data entry allows you to define a length of the fire hydrant lateral pipe.

Lateral roughness

This data entry allows you to define a roughness coefficient of the fire hydrant lateral pipe.

11.3 Running Simulations

Select "Run" to run the fire flow simulation. The program will run the fire flow simulation based on the data specification and the simulation progress will be displayed in the status panel where you can see the currently executed node and results. The results CSV file with the fire flow results is created during the fire flow simulation and can be used to browse the results outside of MIKE+ or it can be loaded into MIKE+ for results processing. The program also creates a Log file that contains additional details from the fire flow simulation. The simulation can be interrupted (cancelled) by pressing "Esc" .

11.4 Fire Flow Results

Results of the fire flow simulations can be displayed in different ways. Chapters below describe how to display these results in tables or on a map. Besides this, the program also creates a *.log file in the project directory that contains additional details about the fire flow simulation.

Tabular Results

The simulated fire flow results for all simulation modes are written into the output CSV file. The CSV file is a comma separated text file in a format that is suited for opening into Microsoft Excel, for example. The tabular results can be displayed in MIKE+ in result tables.

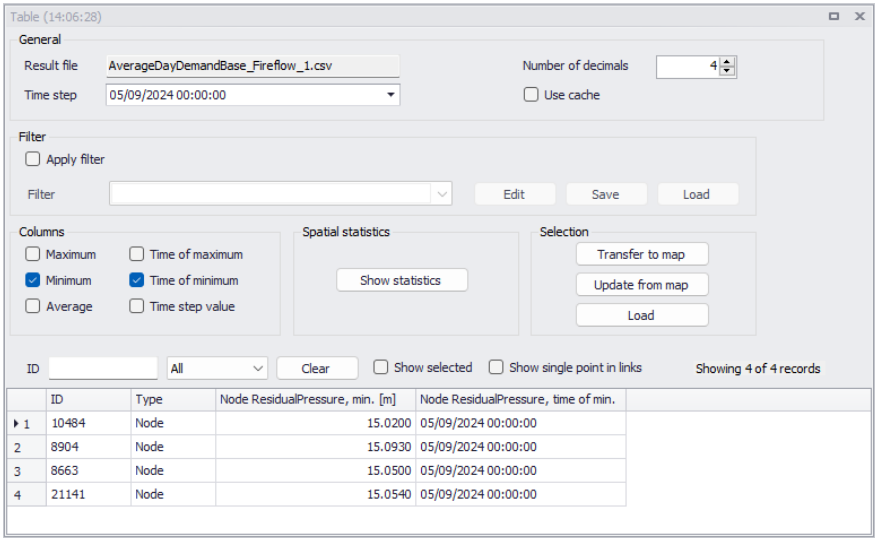
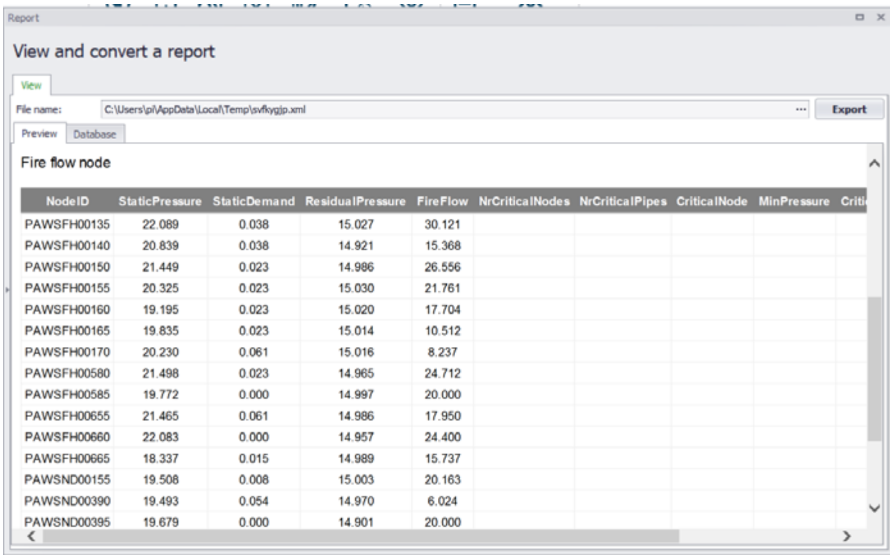


Figure 11.5 Example tabulated Fire Flow Analysis results

They can also be displayed directly from within the Fire Flow Analysis editor box by selecting “Report”.



NodeID	StaticPressure	StaticDemand	ResidualPressure	FireFlow	NrCriticalNodes	NrCriticalPipes	CriticalNode	MinPressure	Criti
PAWSFH00135	22.089	0.038	15.027	30.121					
PAWSFH00140	20.839	0.038	14.921	15.368					
PAWSFH00150	21.449	0.023	14.986	26.556					
PAWSFH00155	20.325	0.023	15.030	21.761					
PAWSFH00160	19.195	0.023	15.020	17.704					
PAWSFH00165	19.835	0.023	15.014	10.512					
PAWSFH00170	20.230	0.061	15.016	8.237					
PAWSFH00580	21.498	0.023	14.965	24.712					
PAWSFH00585	19.772	0.000	14.997	20.000					
PAWSFH00655	21.465	0.061	14.986	17.950					
PAWSFH00660	22.083	0.000	14.957	24.400					
PAWSFH00665	18.337	0.015	14.989	15.737					
PAWSND00155	19.508	0.008	15.003	20.163					
PAWSND00390	19.493	0.054	14.970	6.024					
PAWSND00395	19.679	0.000	14.901	20.000					

Figure 11.6 Fire flow results report

Thematic Maps

The simulated fire flow results can be displayed using the colour coded maps. Select Layers and Add Layer and select one of the fire flow result items to create a colour coded map with the fire flow results.

List of available fire flow result items:

- Node static pressure: steady state pressure at the fire flow node
- Node static demand: steady state demand at the fire flow node
- Node residual pressure: simulated or given residual pressure during the fire flow simulation at the fire flow node
- Node fire flow: simulated or given fire flow at the node
- Number of critical nodes: the number n means at how many nodes the residual pressure was below the critical pressure
- Number of critical pipes: the number n means at how many pipes the velocity was above the critical velocity
- Critical node: node with the minimum residual pressure below the critical pressure
- Minimum pressure: minimum residual pressure reported for a critical node that is below the critical pressure
- Critical pipe: pipe with the maximum velocity above the critical velocity
- Maximum velocity: maximum velocity reported for a critical pipe that is above the critical velocity
- Status: error code:

- 0: No errors
- 1: Static pressure is already below the residual pressure, no flow available
- 2: Cannot find upper flow limit, no flow will be computed
- 3: Cannot iterate flow for pressure, no flow will be computed
- 4: No fire flow available at this residual pressure
- 5: Node does not exist, no flow will be computed
- 6: No flow available at this residual pressure and velocity
- 7: Residual pressure is negative for the required fire flow

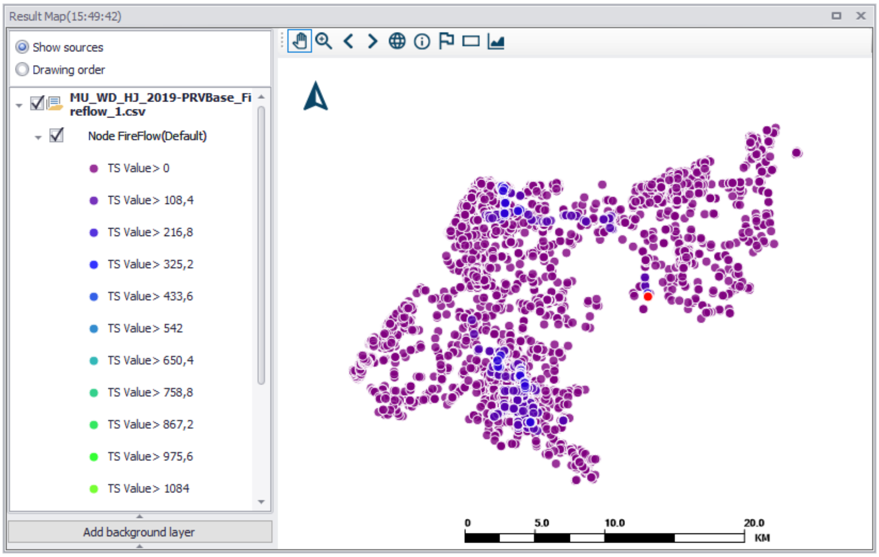


Figure 11.7 Example Fire Flow Analysis result map plot of Node fire flow

Q-H plots

When applying the method to compute a Q-H curve, the Fire Flow analysis saves the relationship between the hydrant flow and the residual pressure in a .csv file. This table can be shown in a Q-H plot window. Refer to Hydrant Q-H Plot (p. 469) for more details.

12 Cost Analysis

Cost analysis allows you to review energy consumption results on more details, create tabular outputs, and great graphs of pump/turbine utilization, average power consumption/production, and energy costs.

12.1 Settings

In the 'Simulation' group, the ID of the associated simulation (from the 'Simulation setup' editor) which is used as the base of the cost analysis, must be selected. Then the following parameters must be specified in the 'Settings' tab:

- Currency: The currency name. It can be selected from the list of available currencies. If not available from that list, select 'Other' to enter a custom currency name.
- Global Price: Average cost per kW/hour for the pump / turbine.
- Price Pattern ID: ID label of time pattern describing how energy price varies with time for the pump / turbine.
- Pump/Turbine Efficiency: The single perfect efficiency of the pump/turbine.
- Demand Charge: Added cost per maximum kW usage during the simulation period for the pump.
- Carbon emission factor: this optional entry is used for computing carbon emissions related to pump energy in mass of CO_2 per kWh. Hence the user can define the amount of carbon emissions per unit of energy usage.

The engine combines the hydraulic results of pumps/turbines and their general parameters to calculate the energy cost of each pump and turbine, as well as the statistical data.

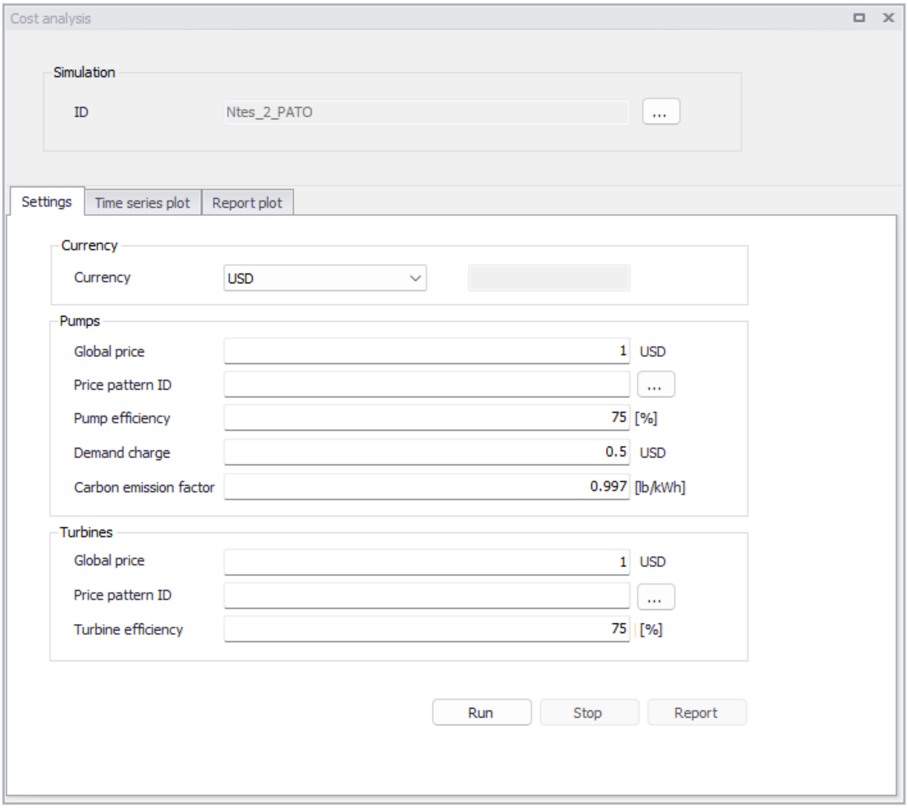


Figure 12.1 The Cost Analysis window

12.2 Time Series Plot

The Time Series Plot can display the time series of energy consumption or generated, energy cost, efficiency and average energy per million cubic meters (or gallons) of each pump and turbine. The result data table is also accessible on the right side of the panel.

- Efficiency: efficiency of pump/ turbine with time (%)
- Energy per volume: Accumulated power consumption/production (kWh) per millions gallons (or cubic meters).
- Power used: Energy consumption during a pump operation over time
- Power generated: Energy production during a turbine operation over time (negative to represent generated energy)
- Energy Cost: The accumulated cost of energy consumption or generated of the pump/turbine operation over time (negative to represent generated energy)

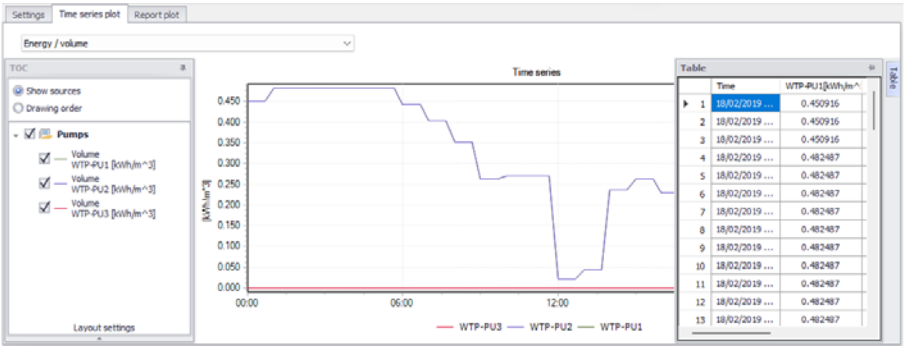


Figure 12.2 Cost Analysis Time Series plot

12.3 Report plot

The report plot displays the statistical data of pump / turbine utilization, average efficiency, energy / volume, average power consumption, peak power consumption, and cost per day.

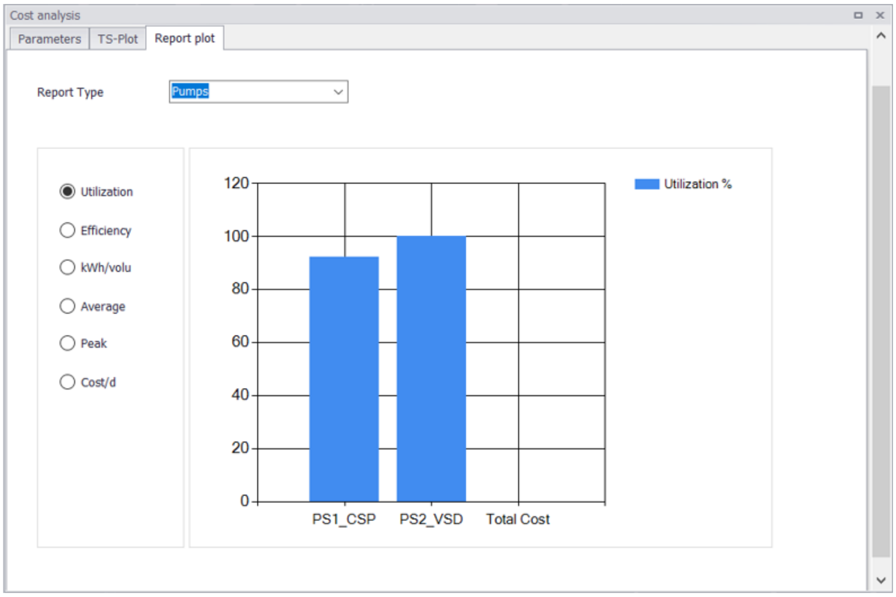


Figure 12.3 Cost Analysis report plot

The following fields are calculated and reported:

- Utilization: percent utilization i.e. percent of the time that the pump / turbine was operating
- Efficiency: Average efficiency of the pump / turbine
- Energy per volume: Energy consumption pumped or turbine production

- Average power: Average rate of energy usage if the pump or turbine power generated
- Peak power: Peak rate of energy usage of the pump operation
- Cost/day: total cost of the pump / turbine operation per day

12.4 Report

This type of report presents all the statistical energy data of all pumps and turbines, including system energy performance indicators.

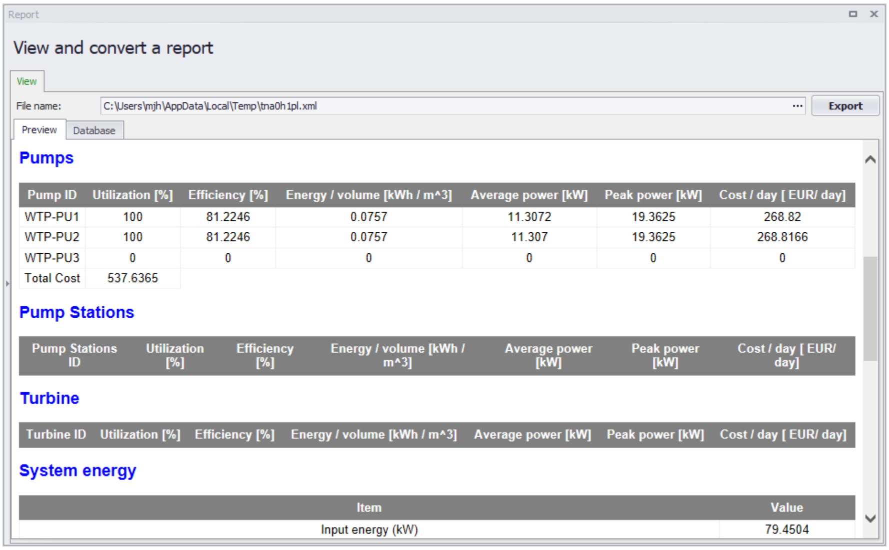


Figure 12.4 The Energy Report

The report has the following sections:

- Model description
- Pumps
- Pump stations
- Turbines
- System energy
- Sum

Model description

This section contains information about the model file, project title and descriptions and name of the active scenario.

Pumps

This section contains the following outputs for each pump:

- Utilization: average pump utilization (%)
- Efficiency: average pump efficiency (%)
- Energy/volume: energy per volume (kWh/volume)
- Average power: average pump power (kW)
- Peak power: pump peak power (kW)
- Cost/day: average costs per day (currency)

Pump stations

If pump stations have been defined in the network model, the same outputs as for individual pumps will be reported at the station level.

Turbines

This section contains the following outputs for each turbine:

- Utilization: average turbine utilization (%)
- Efficiency: average turbine efficiency (%)
- Energy/volume: energy per volume (kWh/volume)
- Average power: average turbine power (kW)
- Peak power: turbine peak power (kW)
- Cost/day: average costs per day (currency)

System energy

This section contains the following performance indicators for the whole system (model):

- Input energy: total input energy (natural and pumps) (kW)
- Natural input energy: external energy supplied by reservoirs or external tanks (kW)
- Pump shaft input energy: flow rate pumped by station and the head of pumps (kW)
- Energy delivered to users: based on supplied pressure and flow (kW)
- Friction energy on pipes: energy lost due to friction in pipes (kW)
- Friction energy on valves: energy dissipated on valves (kW)
- Output energy: total output energy (delivered to users). Energy lost due to water losses is not accounted for (kW).
- Dissipated energy: total dissipated energy (kW)
- Pump motor input energy: pump motor input energy (kW)
- Shaft energy per injected volume: pump energy per injected volume (kWh/volume)
- Shaft energy per consumed volume: pump energy per consumed volume (kWh/volume)

- Excess of supplied energy: the ratio between the real energy entering the system and the minimum useful energy (based on minimum required service pressure of 35m or 50 psi) (-)
- Network energy efficiency: ratio between energy injected vs delivered to users (consumed) (-)
- Standard compliance: the ratio between the energy delivered to users and the minimum required useful energy (based on minimum required service pressure of 35m or 50 psi) (-)
- Carbon emission: the total mass of CO_2 emitted for pumps
- Carbon emission per day: the average mass of CO_2 emitted for pumps per day.

Sum

This section contains the following information:

- Energy: total pump or turbine energy (kwh)
- Energy cost: total pump or turbine energy costs (currency)
- Daily energy cost: average daily energy costs (currency)
- Volume: accumulated pump or turbine volume (volume unit)
- Unit energy use: unit energy use (kwh/volume)
- Unit energy cost: unit energy costs (cost/volume)
- Peak demand cost: peak demand costs for pumps (currency)
- Run duration: pump or turbine (motor) hours (hours)

13 Network vulnerability

Network vulnerability modelling is required to predict the water distribution system response to pipe breaks situations, planned reconstructions, and other scenarios of limited water supply. Network vulnerability allows also the develop a pipe ranking based on the importance for the water supply and such importance can be then taken into account for the planning of pipe rehabilitation and reconstructions.

The Todini Index

The Todini index is a system relative aggregated measure defining how close a water distribution network operates compared to its minimum required level.

The Todini index (TI) is defined in Eq. (13.1):

$$TI = \frac{\sum_{j=1}^{n_n} d_j (h_j - h_{aj})}{\sum_{i=1}^{n_0} q_I h_I + \left(\frac{1}{\gamma_W}\right) \sum_{k=1}^{n_p} P_k - \sum_{j=1}^{n_n} d_j h_{aj}} \tag{13.1}$$

where: n_n = number of nodes in the network, d_j = demand at node j, h_j = hydraulic head at node j, h_{aj} = required minimum hydraulic head at node j, n_0 = number of reservoirs in the systems, q_i = outflow from reservoir i, h_i = hydraulic head at reservoir I, np = number of pumps in the network, P_k = power of pump k, and γ_w = water specific weight.

The Todini index is computed at a given time level. When computing it for an extended period simulation, the reported index value is the average of the Todini index computed at each time level.

The Connectivity Index

The Connectivity Index is the probability that all nodes in the system are connected to at least one source.

The Node reachability Index

The Reachability Index is the probability that a given node in the system is connected to at least one source.

The Pipe criticality Index

The pipe criticality is determined based on evaluation of several performance indicators including:

- Pipe flow criteria (PI-1)

- Service pressure criteria (PI-2)
- Water demand criteria (PI-3)
- Pipe length criteria (PI-4)
- User defined criteria (PI-5)

The combined pipe criticality is computed as an average of all above performance indicators, i.e.

$C_{(pipe\ i)}$ = Average $(P1+P2+P3+P4)_{(pipe\ i)}$, where

- Pipe flow criteria (P1) is computed as water (in flow units) that cannot be delivered through the pipe. The value of 1 corresponds to the total flow.
- Service pressure criteria (P2) is computed as number of nodes, where the service pressure is below the required level e.g. 15 m or 20 psi, for example. The value of 1 corresponds to the total number of nodes.
- Water demand criteria (P3) is computed as the value of total water demand that cannot be delivered in nodes due to insufficient service pressure. The value of 1 corresponds to the total water demand consumption.
- Pipe length criteria (P4) is computed as a total length of pipes where the pressure is below the required service level. The value of 1 corresponds to the total pipe length. Similarly, it is possible to use the number of population disconnected from the water supply or number of disconnected residences or houses.
- P5 is computed as a total (sum of) “criteria” of pipes where the pressure is below the required service level

User defined performance indicator P5 can be used by selecting any numerical field that is defined in the Pipe Editor. If the pipe demand coefficient 1, for example, contains values corresponding to the number of connected customers then the P5 indicator will report number of connected users that are affected by the particular pipe unavailability.

The user defined performance indicator “P5” is an optional parameter and it is therefore not included in the combined indicator “C” where only default indicators P1, P2, P3, and P4 are accounted for.

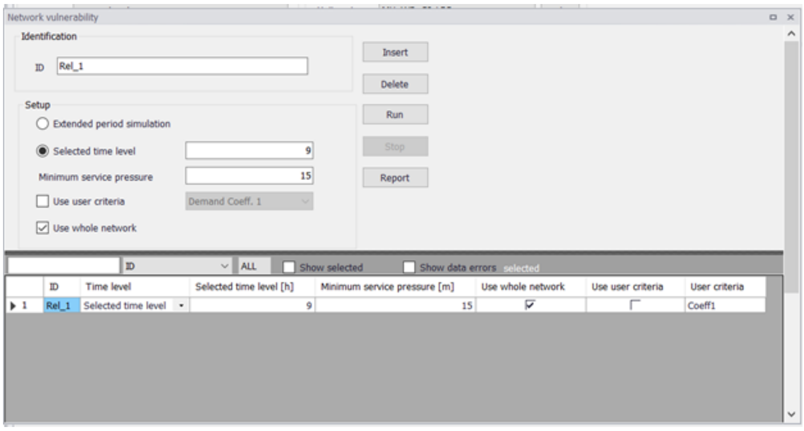


Figure 13.1 Network vulnerability defined criteria

Selected time level is entered in hours or a fraction of hours from the simulation start, e.g. entering "9" will run the network vulnerability at 9 AM for the simulation start at 12 AM.

Select “Use user defined criteria:” and then select the pipe field that holds the data used for the criteria evaluation, e.g. “Demand coefficient 1” in case that you want to include only such pipes in the analysis.

The Network Vulnerability Analysis dialog box is reached by selecting ‘Model type’ from General Settings from the Table of Contents and then by selecting the ‘Network vulnerability’ option. Note that to run the network vulnerability analysis, you need to select *Run* from within the Network Vulnerability Analysis dialog box.

The results of the Network Vulnerability analysis can be viewed as follows:

- Map layers for Node reachability and Pipe criticality
- Report with tabular results for Todini index, Node reachability and Pipe criticality

13.1 Setup

A list of the Network Vulnerability dialog box data entries for Figure 13.2 follows, with a short description given for each entry.

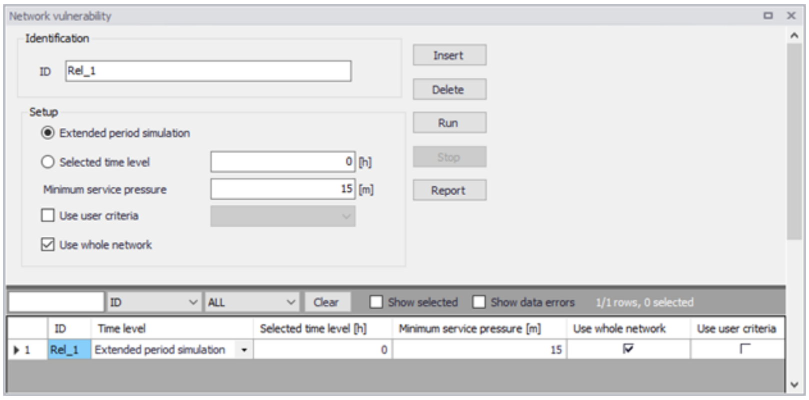


Figure 13.2 The Network vulnerability dialog

Selected time level

This data entry allows you to select the time level from the extended period simulation that will be used to compute the network vulnerability.

Extended period simulation

This data entry allows you to compute the network vulnerability for all time levels i.e. for the entire duration of the extended period simulation. Please note, that this option may lead to extensive simulation times. If this option is selected, the performance indicators PI1-PI4 will be based in results over the entire simulation and may, for example, contain results of water demand deficiencies caused by storage tanks that were drained due to closed pipes.

Minimum service pressure

This data entry specifies then minimum acceptable service pressure within the network e.g. 15m or 20 psi that is required for uninterrupted water supply.

Use whole network

This data entry allows you to run the network vulnerability for all pipes within the model or for pipes within selected pressure zone(s).

13.2 Running Simulations

Select “Run” from within the network vulnerability dialog box in order to run the simulation. The simulation progress will be displayed in the application status window. The simulation can be interrupted (cancelled) by pressing “Esc”.

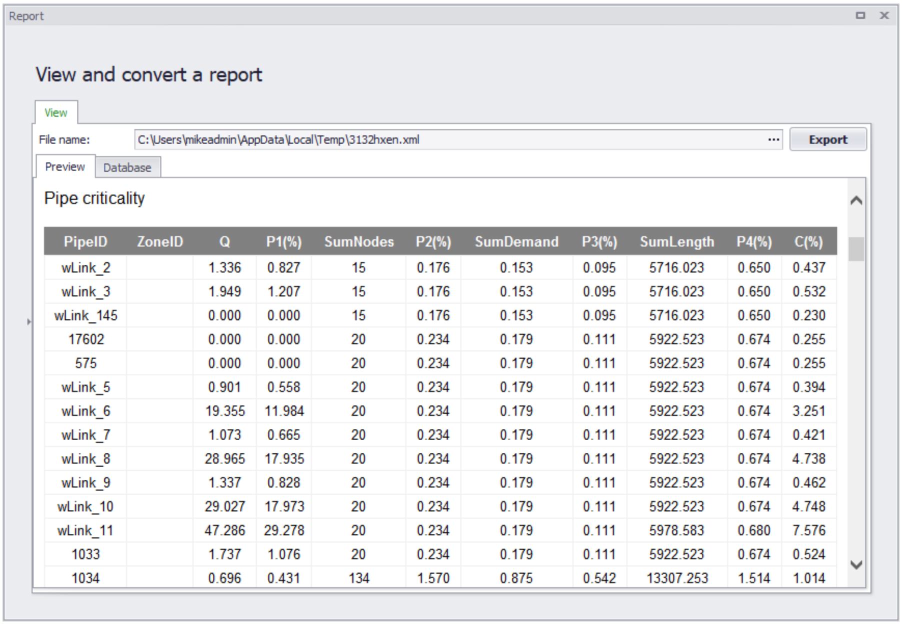
13.3 Network Vulnerability Results

Results of the network vulnerability simulations can be displayed in different ways. They are saved in a .CSV file, which is a comma separated text file in a format that is suitable for importing into Microsoft Excel, for example.

Tabular Results

The simulated pipe criticality results file can be displayed in a results table.

The tabular results can also be displayed directly from within the network vulnerability dialog using the “Report” button.



PipeID	ZoneID	Q	P1(%)	SumNodes	P2(%)	SumDemand	P3(%)	SumLength	P4(%)	C(%)
wLink_2		1.336	0.827	15	0.176	0.153	0.095	5716.023	0.650	0.437
wLink_3		1.949	1.207	15	0.176	0.153	0.095	5716.023	0.650	0.532
wLink_145		0.000	0.000	15	0.176	0.153	0.095	5716.023	0.650	0.230
17602		0.000	0.000	20	0.234	0.179	0.111	5922.523	0.674	0.255
575		0.000	0.000	20	0.234	0.179	0.111	5922.523	0.674	0.255
wLink_5		0.901	0.558	20	0.234	0.179	0.111	5922.523	0.674	0.394
wLink_6		19.355	11.984	20	0.234	0.179	0.111	5922.523	0.674	3.251
wLink_7		1.073	0.665	20	0.234	0.179	0.111	5922.523	0.674	0.421
wLink_8		28.965	17.935	20	0.234	0.179	0.111	5922.523	0.674	4.738
wLink_9		1.337	0.828	20	0.234	0.179	0.111	5922.523	0.674	0.462
wLink_10		29.027	17.973	20	0.234	0.179	0.111	5922.523	0.674	4.748
wLink_11		47.286	29.278	20	0.234	0.179	0.111	5978.583	0.680	7.576
1033		1.737	1.076	20	0.234	0.179	0.111	5922.523	0.674	0.524
1034		0.696	0.431	134	1.570	0.875	0.542	13307.253	1.514	1.014

Figure 13.3 Example Network Vulnerability report

Thematic Maps

The simulated network vulnerability results can be displayed using the colour coded maps.

List of available network vulnerability result items:

- Q: flow per pipe that was not delivered (flow units or volume units in case of extended period simulation for all time levels)
- P1: performance indicator P1 (-)
- SumNodes: number of nodes where the service pressure is insufficient
- P2: performance indicator P2 (-)
- SumDemand: demand or total water volume in case of extended period simulation for all time levels)
- P3: performance indicator P3 (-)
- SumLength: total pipe length where the service pressure is insufficient
- P4: performance indicator P4 (-)
- C: performance indicator C (-)

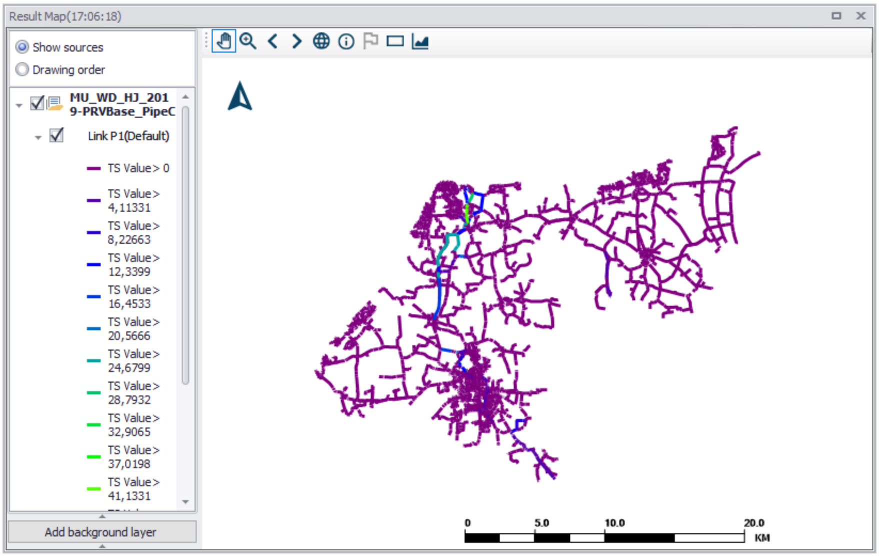


Figure 13.4 Example Network vulnerability map result showing Link P1 results

14 Shutdown Planning

Shutdown planning is designed to determine impact of pipe maintenance on the water supply conditions. It helps the user to define the shutdown, find isolation valves, run hydraulic simulations, and evaluate simulation results. Shutdown planning includes these steps:

- Planning shutdown
- Close pipes for selected isolation valves
- Analyse shutdown
- Generate shutdown results
- Generate shutdown report

The Shutdown Planning dialog box is reached by selecting ‘Modul type’ from General Settings from the Table of Contents and then by selecting the ‘Shutdown Planning’ option. Note that to run the shutdown planning analysis, you need to select *Run* from within the Shutdown Planning dialog box.

14.1 Settings

A list of the Shutdown Planning data entries for Figure 14.1 follows, with a short description given for each entry.

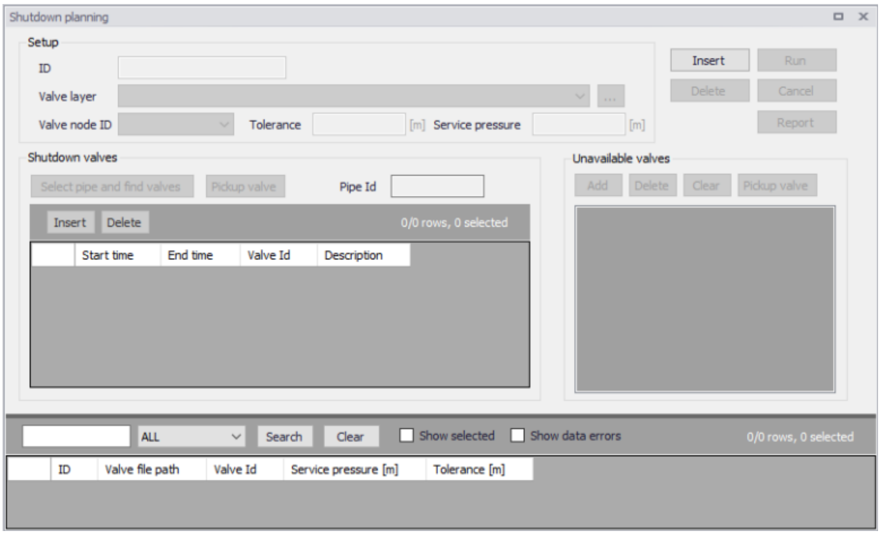


Figure 14.1 The Shutdown Planning dialog box is used to define the analysis parameters

14.1.1 Setup

ID
This data entry allows you to identify the shutdown analysis. You can define multiple shutdown planning analyses and they will be displayed in the main grid at the bottom of the Shutdown Planning dialog box.

Valve layer
This data entry allows you to select the GIS layer with valves (typically isolation valves) that will be used in the valve criticality analysis. Please note, that in order to select valve layer in this data entry, the valve layer needs to be already added to the Map layers

Valve node ID
This data entry allows you to define the ID field used for reporting GIS valves.

Tolerance
This data entry allows you to define the spatial tolerance that will be used to track the pipe network connectivity.

14.1.2 Shutdown valves

Select pipe and find valves
This command allows you to define the pipe that you want to isolate by clicking the pipe in the Map. Once the pipe is selected, the program will find valves that need to be closed in order to isolate the selected pipe. The list of valves is displayed in the grid and the pipe ID is displayed in the Pipe ID field.

Pickup valve
Pickup valve allows you to select a valve (manually) from the Map. Once selected, the valve will be added into the table with valves.

Pipe ID
ID of a pipe that is selected for the shutdown analysis.

Start time
Start time is the time when the selected valve will be closed during the shutdown planning.

End time
End time is the time when the selected valve will be re-opened during the shutdown planning.

Insert
Insert a new line (record) into the table with valves,

Delete
Delete a line (record) from the table with valves.

14.1.3 Unavailable valves

Add
In case that one of the valves that were identified by the program as required in order to isolate a pipe is unavailable (e.g. malfunctioning or not physically available), this command allows you to define such a valve or valves and the program will find substitute valve when you click “Pickup valves”.

Delete
Delete a line (record) from the table with unavailable valves.

Clear
Delete all lines (records) from the table with unavailable valves.

Pickup valves
This command allows the program will find substitute valves for valves that are selected as “unavailable”.

14.1.4 Commands

Insert
Create (insert) a new shutdown planning analysis.

Delete
Delete active shutdown planning analysis.

Run
Run the hydraulic simulation to analyse the pressure and flow conditions during the shutdown planning analysis.

Report
Generate a report from the shutdown planning analysis.

14.2 Running simulation

Select “Run” from within the shutdown planning analysis dialog box in order to run the simulation. The simulation progress will be displayed in the application status window. The simulation can be interrupted (cancelled) by pressing “Esc”.

14.3 Shutdown planning results

Results of the shutdown planning analysis simulations can be displayed as results for the standard hydraulic simulation.

From a Shutdown Analysis result layer on the map, symbology settings may be used to highlight the shutdown area - e.g. pipes with zero flow displayed in red. The same may be done for nodes where the pressure is below the service pressure.

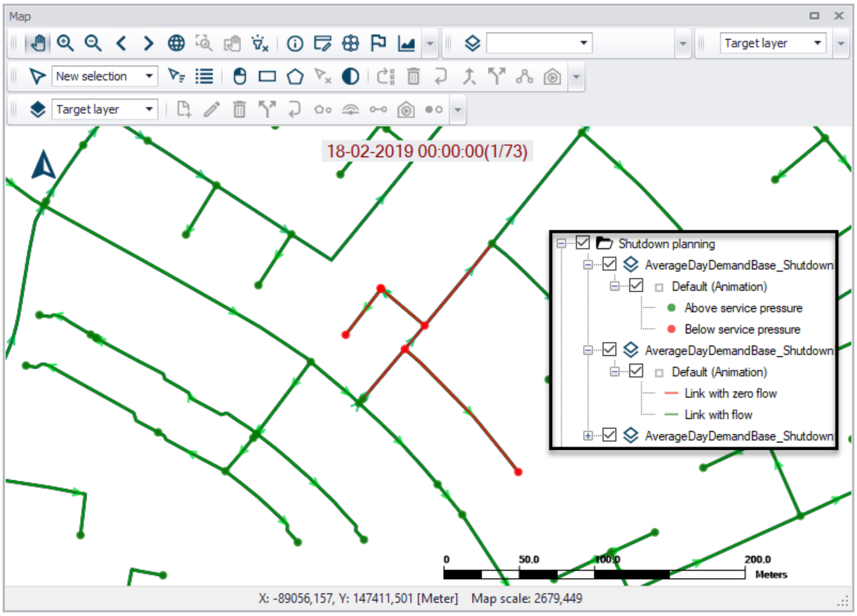


Figure 14.2 Example map of Shutdown Planning results highlighting pipes with zero flow in the shutdown scenario (in red)

In addition to the map display, it is possible to get a report for shutdown planning using the 'Report' button on the Shutdown Analysis dialog. It can generate a report on the valves that closed, pipes with zero flow and the list of demand allocation that have insufficient pressure.

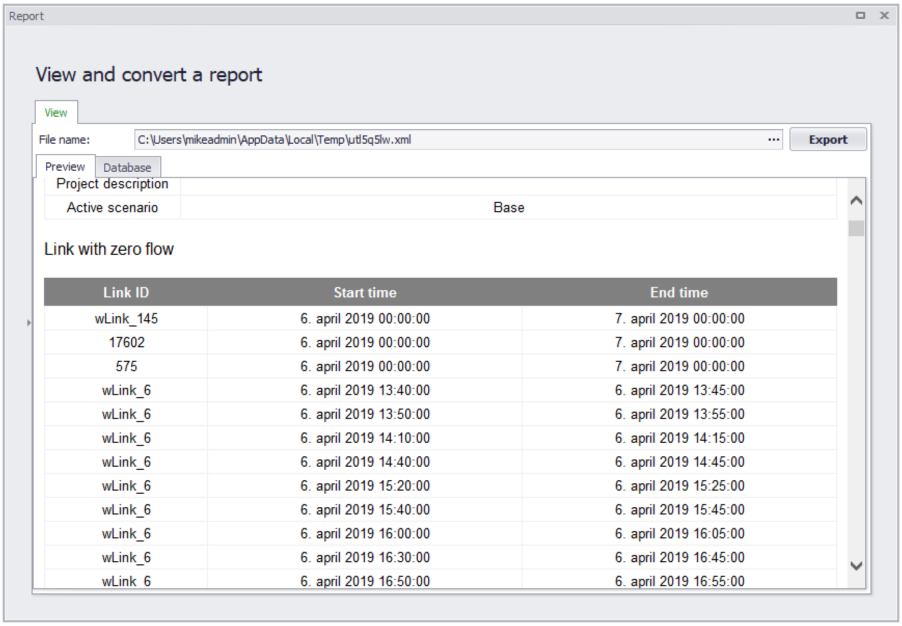


Figure 14.3 Example Shutdown Analysis report

15 Flushing analysis

Flushing of pipelines is a common practice used by water utilities to clean pipelines in their water distribution systems. The conventional way to flush pipelines is just to open selected fire hydrants successively and let them flow until the flowing water appears clean. Unidirectional flushing (UDF), which is a more effective way to flush pipelines. involves closing or opening selected valves to direct flow through target pipes in order to achieve higher velocities for the same hydrant flows. The set of valves that need to be operated and hydrant that is opened is called a flushing sequence.

The flushing analysis can be used in two modes:

- Conventional flushing
- Unidirectional flushing

Conventional flushing can run in a batch mode when the program will simulate the flushing successively for every selected outlet. For every outlet, the program will compute the actual flushing time required to flush (exchange) the water in the selected pipelines. The actual flushing time can be extended by a safety factor that multiplies the minimum required flushing time. The idle time in between switching the outlet nodes can also be specified.

Unidirectional flushing will not run in a batch node, it will run for one specific outlet node but it will allow the user to close additional number of valves (pipes) in order to maximize the flushing result. The program can assist in finding sections valves need to be closed and then it will determine pipes that need to be closed for the simulation.

Flushing velocity i.e. maximum velocity as well as the change in the velocity achieved during the flushing sequence is very important for the success of cleaning pipes. These are some of the recommended values:

- 0.9 m/sec or 3 ft/sec removes sediment and lowers disinfectant demand
- 1.5 m/sec or 5 ft/sec removes biofilm and promotes scouring
- 3.7 m/sec or 12 ft/sec removes sand from inverted siphons

It is encouraged to achieve a 1.5 m/sec or 5 ft/sec on every flush.

The Flushing Analysis dialog box is reached by selecting ‘Model type’ from General Settings from the Table of Contents and then by selecting the ‘Flushing analysis’ option. Note that to run the flushing analysis, you need to select *Run* from within the Flushing Analysis dialog box.

15.1 Settings

A list of the Flushing settings data entries for Figure 15.1 follows, with a short description given for each entry.

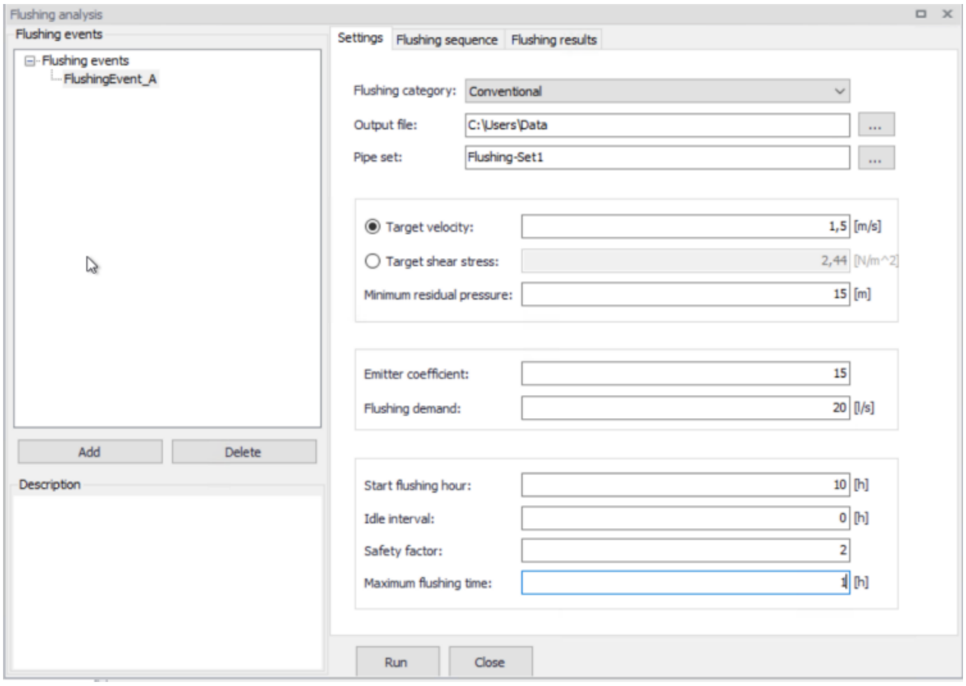


Figure 15.1 The Flushing settings box is used to the analysis parameters

Flushing events

This data entry allows you to add flushing events that are further specified by data in the right-hand side of the dialog. You can add and delete flushing events using “Add” and “Delete”.

Flushing category

Select from conventional and UDF unidirectional flushing.

Output file

This data entry allows you to specify where the output report with flushing analysis results will be stored.

Pipe set

This data entry allows you to select a pipe set with pipes that will be used in flushing i.e. pipe to be flushed. In order to define the pipe set, use Selection from the main application menu and create a selection list.

Target velocity

This entry allows you to define the target velocity that will be used to quantify the success of flushing event.

Target shear stress

This entry allows you to define the target shear stress that will be used to quantify the success of flushing event.

Minimum residual pressure

This data entry allows you to define the minimum residual pressure within the flushed pipes during the flushing event. If the actual (computed) residual pressure would be smaller than the minimum residual pressure, the program will report it.

Emitter coefficient

This data entry allows you to define the emitter coefficient that will be inserted by the program to the outlet node and used in flushing. The program will change the outlet node to an emitter node when you select this option. Hydrant flows may be specified directly in flow units or as an emitter coefficient. For standard North American hydrants that comply with AWWA Standard C502 or C503, the emitter coefficient would be 150-180 gpm/psi$^{0.5}$ (11-14 l/s/m$^{0.5}$) for the 2.5 in (63 mm) outlet and 380-510 gpm/psi$^{0.5}$ (30-40 l/s/m$^{0.5}$) for the 4.5 in (115 mm) outlet depending on the model of hydrant, size of barrel and length of barrel. In terms of flow units, free discharge from a hydrant can vary from 500 to 1500 gpm (32-95 l/s) depending primarily on the strength of the distribution system at that point. Note, that the emitter coefficient needs to be entered in flow units matching the model flow units.

Flushing demand

This data entry allows you to define the demand that will be inserted by the program to the outlet node and used in flushing.

Start flushing hour

This data entry allows you to define the start time of the flushing event.

Safety factor

This data entry allows you to prolong (extent) duration of the flushing. The program computes the flushing time by tracking the volume of water that was initially contained in the flushed pipes and how much of that volume was replaced by fresh water from the start node (source node). The safety factor bigger than 1 allows you to prolong the flushing above the minimum flushing time.

Maximum flushing time

This data entry allows you to define the maximum duration of the flushing event. The simulation will stop when the maximum flushing time is reached regardless whether pipes were completely flushed or not.

15.2 Flushing sequence

A list of the Flushing sequence data entries for Figure 15.2 follows, with a short description given for each entry.

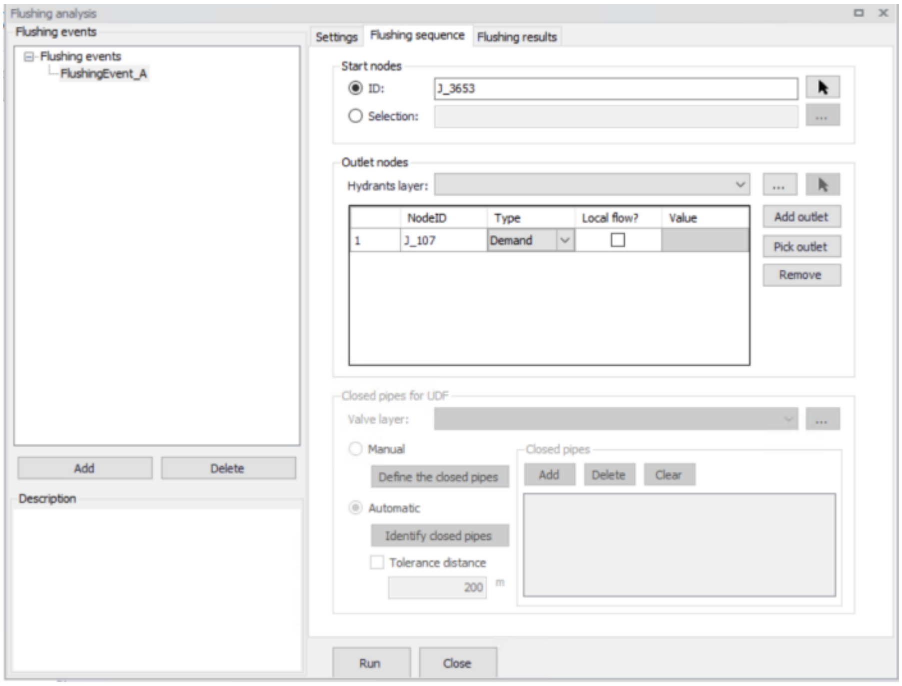


Figure 15.2 The Flushing sequence box is used to the analysis parameters

Start nodes

This data entry allows you to define the source of fresh water, this could be the starting node of the first pipe in the flushing sequence or this could be any other node in the network. The selected node will be used by the program for accounting for water that is flushed in the selected pipes during the flushing event. Note, that it is possible to select multiple nodes in case that the pipes selected for flushing are receiving water from different entry points.

Outlet nodes

This data entry allows you to define the outlet node that is used to flush the water out of the selected pipelines. The amount of water that is leaving the system through this node is determined by the program based on the flushing demand or emitter coefficient defined in Settings. Note, that every outlet can inherit flushing demand or emitter coefficient from the general settings or that these entries are defined specifically for the outlet. If you wish to use different settings per outlet, select “local flow” and enter the required value for the flushing demand or emitter coefficient.

Closed pipes for UDF (unidirectional flushing)

This data entry allows you to define pipes that will be closed by the program during the unidirectional flushing. In order to select such pipes, use “Add”, “Delete”, and “Clear”.

Valve layer

This data entry allows you to automatically identify pipes that will be closed by the program during the unidirectional flushing by selecting isolation valves from the Valve layer that will be closed in the physical system. In order to select the shapefile with such valves, use “Valve layer” selection and locate the data source with valves. Next, define the tolerance distance that will be used by the program to find the pipe that is nearest to the selected isolation valve.

15.3 Running simulation

Select “Run” from within the Flushing dialog box in order to run the simulation. The simulation progress will be displayed in the application status window. The simulation can be interrupted (cancelled) by pressing “Esc”.

15.4 Flushing Results

Results of the flushing simulations can be displayed in different ways. They are written into an output .csv file. This .csv file is a comma-separated text file in a format that is suitable for importing into Microsoft Excel, for example.

Tabular Results

The tabular results can be displayed directly from within the Flushing Analysis editor, in the “Flushing results” tab.

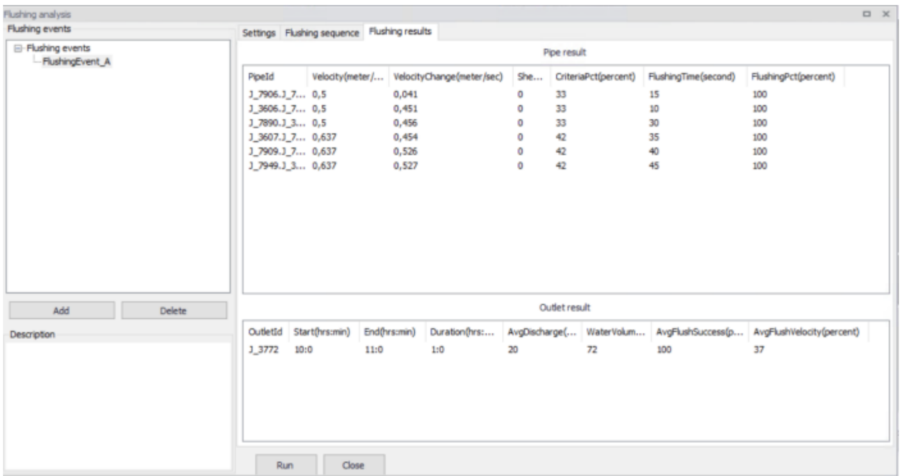


Figure 15.3 Flushing analysis results

Report

Generate a report document from Flushing Analysis results using the 'Report' button in the Flushing Analysis editor. It summarizes values for various result items in formatted tables.

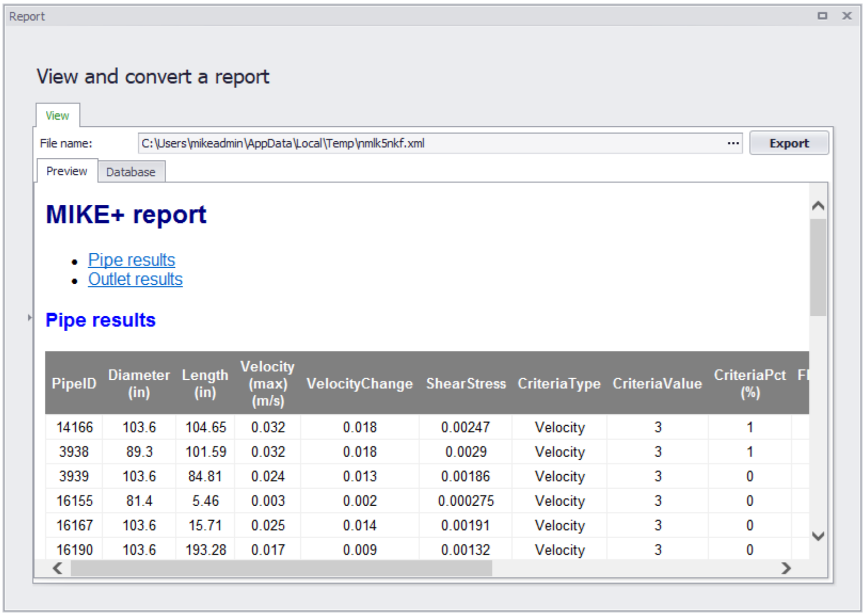


Figure 15.4 Example Flushing Analysis report

Thematic Maps

The simulated flushing results can be displayed using the colour coded maps. Select Layers and Add Layer and select one of the flushing result items to create a colour coded map with the fire flow results.

List of available flushing result items:

Pipes

- Pipe ID: unique pipe identifier.
- Velocity (max): maximum velocity reached during the flushing event in the pipe
- Velocity change: difference between the flow velocity before flushing and the maximum velocity during the flushing event.
- Shear stress (max): maximum shear stress reached during the flushing event in the pipe.

- Criteria percentage (%): the value indicates how well the flushing criteria was fulfilled during the simulation. Value of 75%, for example, would mean that if the required velocity was e.g.1.5 m/s then the actual maximum velocity reached during the flushing was 75% of that value, i.e. 0.75 * 1.5 = 1.125 m/s.
- Flushing Time(min): the program computes the minimum time required to fully replace the pipeline volume be a fresh water from the flushing source. This time can only be computed in case that it was actually possible to replace 100% of the pipeline volume. In case that the volume of replaced water in the pipeline was not 100%, the minimum flushing time is not computed and the value is set to “-1”.
- Flushing percentage (%): the value represents the % of water the water that was replaced in the pipeline during the flushing. Value of 85%, for example, would mean that 85% of the pipeline volume was replaced by a fresh water originating from the source of flushing.
- Comment: description indicates the flushing success e.g. pipeline flushed, pipeline flushed but criteria not reached, pipeline not flushed.

Outlets

- Outlet ID: unique node identifier.
- Start (hrs:min): start of a flushing event is calculated from the start of the whole flushing sequence and from the idle interval in between flushing events.
- End (hrs:min): end of a flushing event is calculated from the start time and duration of a flushing sequence.
- Duration (hrs:min): duration of a flushing event. The duration of a flushing even is computed from the minimum flushing time and a safety factor. In case that the maximum flushing duration was reached, the duration is equal to the maximum flushing duration.
- Average discharge (flow units): average flow in a pipe during the flushing event
- Water volume (volume units): volume of water that was discharge (flushed) from the outlet during the flushing event.
- Average flushing success (%): average flushing success from pipes i.e. a percentage indicating of how well the pipe is flushed weighted by a pipe length.
- Average flushing velocity (%) : average flushing velocity from pipes.

16 Water Hammer

Water Hammer (a part of the WD-Tools module) simulates transient (unsteady) flow in any fully pressurized system carrying liquids. MIKE+ Water Hammer provides a cost effective tool for engineers seeking fast answers to questions about rapid operation of piping systems. Water hammer is based on the high-order implicit scheme solving the continuity and momentum equation using the finite difference method. The initial conditions are modeled using MIKE+ standard water distribution module.

Water Hammer allows you to model:

- Sudden changes in flows and pressures
- Pump start-up and pump trip-off
- Valve operations
- Power or equipment failure events
- Surge protection

16.1 Water Hammer Calculation

MIKE+ Water Hammer computes hydraulic transients in pipe networks. The computations are based on the continuity equation:

$$\frac{\partial Q}{\partial x} + \frac{gA}{a^2}\frac{\partial H}{\partial t} = 0 \tag{16.1}$$

and the equation of motion:

$$\frac{\partial Q}{\partial t} + gA\frac{\partial H}{\partial x} + \frac{f}{2DA}Q\left|Q\right| = 0 \tag{16.2}$$

in which Q is the discharge, H - the piezometric head above arbitrary datum, f - the Darcy-Weisbach friction factor, D - the internal pipe diameter, A - the cross-sectional area of the pipe, g - gravitational acceleration, a - wave speed, x - distance along the pipe axis and t - time.

In the governing equations the acceleration terms which are very small compared to the other terms have been disregarded.

The general expression for the wave speed (only important for water hammer computations) presented by Halliwell (1963) has been used

$$a = \sqrt{\frac{K}{\rho\left[1 + \left(K/E\right)\psi\right]}} \tag{16.3}$$

in which E is the Young's modulus of elasticity of the conduit walls, K - the bulk modulus of the fluid, r - the density of the fluid and y - a non dimensional parameter. For more details see Section 5.

An implicit finite difference scheme described by Verwey and Yu (1993) has been implemented for water distribution, slow transient and water hammer simulations. The scheme uses only two adjacent grid points in space on a non-staggered grid and is defined on three time levels. The elimination of the most important phase error allows the simulation of both water hammer and slow transients.

16.2 Theoretical Background

The following section describes the MIKE+ water hammer numerical engine.

16.2.1 Description of Water Hammer Model

The water hammer computation is based on the Continuity equation

$$\frac{\partial Q}{\partial x} + \frac{gA}{a^2}\frac{\partial H}{\partial t} = 0 \tag{16.4}$$

and the Momentum equation

$$\frac{\partial Q}{\partial t} + gA\frac{\partial H}{\partial x} + \frac{f}{2DA}Q\left|Q\right| = 0 \tag{16.5}$$

where Q is the discharge, H - the piezometric head above arbitrary datum, f - the Darcy-Weisbach friction factor, D - internal pipe diameter, A - the area of pipe, g - the gravitational acceleration, a - the wave speed, x - the distance along pipe axis and t - the time.

In the governing equations the acceleration terms which are very small compared to the other terms have been disregarded.

Wave Speed

For pure liquids Halliwell (1963) presented the general expression for the wave speed

$$a = \sqrt{\frac{K}{\rho[1 + (K/E)\psi]}} \tag{16.6}$$

in which E is the Young's modulus of elasticity of the conduit walls, K is the bulk modulus of the fluid, ρ is the density of the fluid and ψ is a nondimensional parameter.

Rigid Conduit

$$\psi = 0 \tag{16.7}$$

Thick-Walled Elastic Conduit (D/e<=10)

- anchoring at both ends = full restraint

$$\psi = 2\,(1 + \nu)\left(\frac{R_o^2 + R_i^2}{R_o^2 - R_i^2} - \frac{2\,\nu R_i^2}{R_o^2 - R_i^2}\right) \tag{16.8}$$

in which ν is the Poison's ratio, R_o is an external diameter, R_i is an internal diameter.

- upstream anchoring = upper restraint

$$\psi = 2\left(\frac{R_o^2 + 1.5\,R_i^2}{R_o^2 - R_i^2} + \frac{\nu\left(R_o^2 - 3\,R_i^2\right)}{R_o^2 - R_i^2}\right) \tag{16.9}$$

- frequent expansion joints = expansion joints

$$\psi = 2\left(\frac{R_o^2 + R_i^2}{R_o^2 - R_i^2} + \nu\right) \tag{16.10}$$

Thin-Walled Elastic Conduit (D/e>10)

- anchoring at both ends = full restraint

$$\psi = \frac{D}{e}\left(1 - \nu^2\right) \tag{16.11}$$

in which D is the conduit diameter and e is the wall thickness

- upstream anchoring = upper restraint

$$\psi = \frac{D}{e}\left(1 - 0.5\,\nu\right) \tag{16.12}$$

- frequent expansion joints = expansion joints

$$\psi = \frac{D}{e} \tag{16.13}$$

Tunnels Through Solid Rock, Parmakian 1963

- Unlined tunnel

$$\psi = 1 \quad E = G \tag{16.14}$$

where G is the modulus of rigidity of the rock.

- Steel - lined tunnel

$$\psi = \frac{DE}{GD + Ee} \tag{16.15}$$

in which e is the thickness of the steel liner and E is the modulus of elasticity of steel.

Reinforced Concrete Pipe

This pipe can be replaced by an equivalent steel pipe having equivalent thickness.

$$e_e = E_r\, e_c + \frac{A_s}{L_s} \tag{16.16}$$

in which e_c is the thickness of the concrete pipe, A_s - the cross-sectional area of steel bars, L_s - the spacing of steel bars, E_r - the ratio of the modulus of elasticity of concrete to steel (0.06 - 0.1), but 0.05 for cracks.

Diagrams

The following diagrams can be used in order to estimate the wave speed.

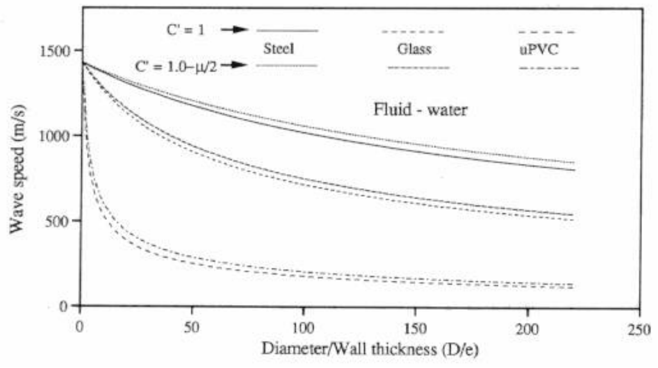


Figure 16.1 Fluid water

Values of Young's Modulus of Elasticity and Poisson's Ratio for a range of common materials are available in the following table.

Table 16.1 Values of Young's Modulus of Elasticity and Poisson's Ratio for a range of common materials

Material	Young’s Modulus (10E9 N/m2)	Poisson’s Ration (-)
Aluminum	70	0.3
Cast Iron	80-110	0.25
Concrete	20-30	0.1-0.3
Copper	107-130	0.34
Glass	68	0.24

Table 16.1 Values of Young's Modulus of Elasticity and Poisson's Ratio for a range of common materials

Material	Young's Modulus (10E9 N/m2)	Poisson's Ration (-)
GRP	50	0.35
Polyethylene	3.1	-
PTFE Plastic	0.35	-
PVC Plastic	2.4-2.8	-
Reinforced Concrete	30-60	0.15
Rubber	0.7-7.0	0.46-0.49
Steel	200-24	0.3
Titanium	103.4	0.34

Typical values of Bulk Modulus:

- K = 2.05 x 10E9 N/m2 for water
- K = 1.62 x 10E9 N/m2 for oil.

16.3 Numerical Scheme and Algorithm

The numerical solution is based on the approach suggested by Verwey and Yu (1993). An implicit, space-compact finite difference scheme has been implemented for simulation in pipe networks including a variety of control elements. The same numerical scheme can be used for simulation of both hydraulic transients and water distribution problems. The inertia terms in the governing equations can be manipulated to produce relatively fast convergence for steady state problems.

The implicit finite difference formulation is based on a non-staggered grid in time and space, where at each grid point the independent variables Q and H are to be computed. The friction term in the governing equations has been expressed as

$$\frac{f}{2DA}\left|Q\right|Q \approx \frac{1}{2}\frac{f}{2DA}\left(\left|Q_{j-1}^{n}\right|Q_{j-1}^{n+1} + \left|Q_{j}^{n}\right|Q_{j}^{n+1}\right) \tag{16.17}$$

The coefficients for the water hammer model have been derived and have the following form:

16.3.1 Coefficients for the numerical scheme

$$\alpha = \frac{gA}{a^2} \tag{16.18}$$

$$fric = \frac{\lambda}{2AD} \tag{16.19}$$

$$Cr = \frac{a\ \Delta t}{\Delta x} \tag{16.20}$$

$$\alpha_c = \left(6\psi^2 - 6\psi + 1\right) + Cr^2\left(6\theta - 6\theta^2 - 1\right)\frac{\Delta x^2}{3a^2\Delta t^2} \tag{16.21}$$

$$\alpha_m = \left(6\psi^2 - 6\psi + 1\right) + Cr^2\left(6\theta - 6\theta^2 - 1\right)\frac{gA\,\Delta x^2}{3a^2\Delta t^2} \tag{16.22}$$

$$A1 = \alpha\left(1 - \psi\right)\Delta x \tag{16.23}$$

$$B1 = -\theta\ \Delta t + \alpha_c \tag{16.24}$$

$$C1 = \alpha\ \psi\ \Delta x \tag{16.25}$$

$$D1 = \theta\ \Delta t - \alpha_c \tag{16.26}$$

$$\begin{aligned} E1 = {} & -(1-\theta)\Delta t(Q_{j-1}^n + Q_{j-1}^{n-1} - Q_j^n - Q_j^{n-1}) + \theta\Delta t(Q_{j-1}^n - Q_j^n) + \\ & + \alpha_c(Q_j^{n-1} - 2Q_j^n - Q_{j-1}^{n-1} + 2Q_{j-1}^n) + \alpha_c(1-\psi)\Delta x H_{j-1}^{n-1} + \alpha_c\psi\Delta x H_j^{n-1} \end{aligned} \tag{16.27}$$

$$A2 = -g\ a\ \theta\ \Delta t + \alpha_m \tag{16.28}$$

$$B2 = (1-\psi)\Delta x + fric\,\Delta t\Delta x\left|Q_{j-1}^{n}\right| \tag{16.29}$$

$$C2 = g\;a\;\theta\;\;\Delta t - \alpha_m \tag{16.30}$$

$$D2 = \psi\Delta x + fric\Delta t\Delta x\left|Q_j^n\right| \tag{16.31}$$

$$\begin{aligned}E2 = {} & (1-\psi)\Delta x Q_{j-1}^{n-1} + \psi\Delta x Q_j^{n-1} + gA(1-\theta)\Delta t\left(H_{j-1}^{n} - H_j^n + H_{j-1}^{n-1} - H_j^{n-1}\right) + \\ & + gA\theta\Delta t\left(H_{j-1}^n - H_j^n\right) + \alpha_m\left(H_j^{n-1} - 2H_j^n - H_{j-1}^{n-1} + 2H_{j-1}^n\right)\end{aligned} \tag{16.32}$$

16.3.2 Looped network solution algorithm

The main algorithm generates a set of grid points using a finite difference scheme, see Cunge, Holly, Verwey (1980). The grid is introduced in time and space, where at every point the values of H and Q are defined as the unknown variables. Between the two successive grid points in time and space both the continuity and the momentum equation are applied. Together with the necessary boundary data, a sufficient number of equations are obtained to solve H and Q at every grid point.

The general form of the governing equations is

$$A1_j H_{j-1}^{n+1} + B1_j Q_{j-1}^{n+1} + C1_j H_j^{n+1} + D1_j Q_j^{n+1} = E1_j \tag{16.33}$$

$$A2_j H_{j-1}^{n+1} + B2_j Q_{j-1}^{n+1} + C2_j H_j^{n+1} + D2_j Q_j^{n+1} = E2_j \tag{16.34}$$

where coefficients A1,B1,C1,D1,E1 for the continuity equation and A2,B2,C2,D2, E2 for the momentum equation are derived from the high-order scheme.

The looped algorithm is based on the fact that a looped network contains elements known as nodes which represent the confluence of several flow paths, some of which originate from other nodes, some from boundary points. A system of simultaneous linear equations is developed where the piezometric head changes at each node are the only unknowns. Solution of this system by any matrix elimination technique yields the piezometric heads at each node.

Suppose that there are three links, 2-1,2-3 and 2-4 and that there are b grid points along branch 2-3 and c grid points along a link 2-4, see Figure 16.2.

For any computational grid point, equations (16.35), (16.36) may be written as

$$H_i = L1_i H_I + M1_i H_{jj} + N1_i \tag{16.35}$$

$$Q_i = L2_i H_I + M2_i H_{jj} + N2_i \tag{16.36}$$

where L,M,N are functions of coefficients A, B, C, D, E, found through a double sweep elimination

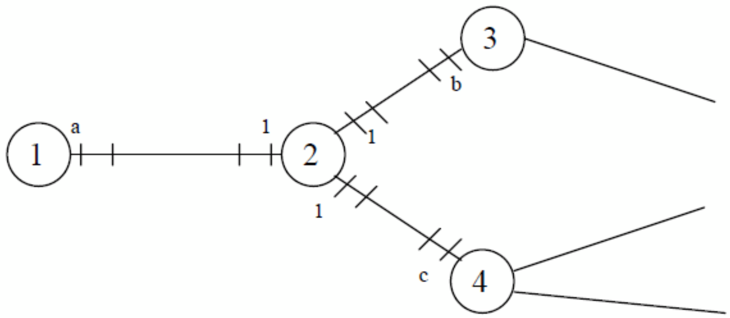


Figure 16.2 Part of a looped pipe network

These equations express the partial dependence of the unknown variables Q and H at any grid point in a branch on the value of H in the two adjacent nodes.

At internal nodes a compatibility condition must be satisfied. The simplest condition is node continuity and common piezometric head.

$$\sum_{k=1}^{m} Q_{1k}^{n+1} = 0 \tag{16.37}$$

$$h_{11}^{n+1} = h_{12}^{n+1} = \mathrm{K} = h_{1k}^{n+1} = \mathrm{K} = h_{1m}^{n+1} \tag{16.38}$$

where n+1 indicates the (n+1)Dt time level in the solution, k is the index of the links emanating from node 2, and m is the number of such links. These relations can be written for each from M nodes, and this leads to a system of M linear equations having as unknowns the piezometric head changes H at each node.

$$[S]\,\{h\} = \{T_L\} \tag{16.39}$$

where [S] is a coefficient's matrix, M x M elements, {h} is a vector of unknowns, M elements; {T(L)} is a vector of the free terms.

This system of linear equations may be solved by any matrix inversion techniques. Once the increments of piezometric head H are known at the nodes, it is possible to recompute Q(i) and H(i) values for all intermediate grid points through equations (16.37) and (16.38).

The looped algorithm may be described by the following steps:

- The coefficients of the high-order scheme discretize the governing equations between two successive grid points on a branch.
- The local elimination method is used to express Q and H grid point values on each branch in terms of H at the branch ends (nodes).
- One equation for each node leads to the system of linear equations that is solved by the matrix elimination method.
- Substitutions inside the branches yield the Q(i) and H(i) values for all intermediate grid points from the known values of H at the branch ends.

16.3.3 Hydraulic structures

The implementation of a hydraulic structure in the domain of the solution may be solved by replacement of the governing equations by another set of equations that characterise the particular hydraulic structure. Every time the main algorithm comes to the location of such a structure, it must switch between the governing equations. Any hydraulic structure can be implemented into such a numerical scheme in the following way. The hydraulic structure is placed between the two successive grid points, and we can assume that.

Another way of implementing a hydraulic structure is to handle it in the similar way to a node. The hydraulic structures are not located between the two successive grid points but in the node. Instead of modifying coefficients A,B,C,D, and E, we increase the number of linear equations.

The main algorithm is designed in such a way, that, after a process of linearisation and discretization of the governing equations, it solves them on a prescribed set of grid points using an appropriate numerical scheme. If a hydraulic structure is present in the domain of the solution, the algorithm must replace the governing equations by other equations defining a hydraulic structure in order to provide the numerical solution. Various hydraulic structures can be coupled together, e.g., the closing of one valve can determine the operating of another valve. In cases where this link exists between hydraulic structures, communication must be maintained and controlled by this main algorithm. This message has to be attached to the object in such a way that it represents the reality. Object-oriented design has been applied to create a safer interface to the numerical algorithm, since the low level operations that remain the same are hidden inside objects.

16.4 Water Hammer Calculations

Water hammer simulates transient (unsteady) flow in any fully pressurized system carrying liquids. MIKE+ Water Distribution Water Hammer provides a cost effective tool for engineers seeking fast answers to questions about rapid operation of piping systems. Water hammer is based on the high-order implicit scheme solving the continuity and momentum equation using the finite difference method. The initial conditions are modeled using MIKE+ Water Distribution standard water distribution module.

Water Hammer allows you to model:

- Sudden changes in flows and pressures
- Pump start-up and pump trip-off
- Valve operations
- Power or equipment failure events
- Surge protection

Water hammer data preparation

allows you to create all the input files interactively and save them for computation. Data Preparation is integrated into editors used for any water distribution model setup and addition data entries used only for transient modeling are enabled when the model type is set to "Water hammer". The data preparation provides interactive data input, editing and error checking. Graphical facilities enable the display of data on a plan plot and use the Query-By Examples (QBE) facilities of the database. .

The present version will handle any number of pipes, nodes, and loops in complex networks with various components.

Water hammer result presentation

enables you to view results generated from the calculation as thematic maps, time series graphs, profile plots, or as text in ASCII format. The results can be displayed using different plots, namely time series plots of the variables, time series plots of the variables for the current time in the longitudinal profile, and colour-coded plan plots. The last two choices can be used for a time animation. The use of colour-coded plan plots allows you to define what numerical ranges of variables between grid points correspond to a particular colour. Zoom facilities enable to magnify interesting portions of drawings.

16.4.1 Running water hammer simulations

In order to be able to start water hammer simulations you have to prepare the steady state model and obtain satisfactory results. In the next step, you need to specify *Water Hammer* Analysis type in the Setup - General Settings -

Modules dialog, enter data into fields used for transient analysis, define boundary conditions and computational parameters.

Initial conditions

Initial conditions are computed with the use of the steady state analysis. The results of the initial state are saved in the file as H, Q values at the beginning and end of the pipes respectively and in the vicinity of hydraulic structures such as valves, pumps, etc. There is a direct connection between the result file from initial conditions and the water hammer execution, in spite of the fact that the two models use different computational grids and different numerical engines.

Boundary conditions

There are in principle two types of boundary conditions, namely the piezometric head, H, above a specified datum, e.g., in tanks, and the discharge, Q, e.g., water demand. Both H and Q are given under selected names as time series in the Curve and Relations Editor and stored in the database. These boundary conditions may be assigned to any node in the network. Boundary for each time step is assigned from given time series specified by the user. If time step used by water hammer computation is smaller than appropriate neighbouring values in boundary conditions time series then linear interpolation is applied. There are nodes of the following types: H - boundary, Q - boundary, compatibility and structure (hydraulic component) description. It should be pointed out that time patterns, used in the Steady State Model, are ignored by the Water Hammer simulations.

For the Initial State for Water Hammer Model, the water level and/or discharges are constant in time. The boundary conditions using time series must be specified for a sufficiently long time interval.

Computational parameters

General parameters consist of fluid density, fluid bulk modulus, absolute temperature, vapour pressure and gravitational acceleration. The most important numerical parameter is a time step. Since a numerical solution must be stable and as accurate as possible, you have to choose a proper value of D_t. The stability condition is given by the Courant number

$$Cr = \frac{a\Delta t}{\Delta x} \tag{16.40}$$

in which a is the wave speed and Dx is the distance between two successive grid points. In principle, an implicit, space-compact scheme is unconditionally stable, with exact solutions generated for the Courant numbers Cr = 0.5 and Cr = 1.0, respectively. The scheme enables us to vary the Courant number over pipes while maintaining its high accuracy. Accurate results are produced in the range 0< Cr < 1.1. You should try to maintain the Courant numbers below unity, but as close as possible to Cr = 1. If you select the menu item Geometry and Branch, you can control the values of Courant numbers. The question how to choose the time step is dictated by the nature of the hydraulic transient itself and by the shortest pipes in the system. The time step can vary from the order of 10-3 to 10+1 seconds. The time steps must be small enough in order to describe very fast changes of variables. It is recommended to start with the shortest pipe section and to calculate the time step, considering Cr = 1. Pipe sections with high Courant numbers are numerically treated in MIKE+ Water Distribution as rigid pipelines. This simplification enables a user to deal with a very short pipe section which would not be important within the water hammer simulation. Once the time step has been prescribed, you have to input the simulation time. MIKE+ Water Distribution calculates a number of time levels, which you need to prescribe in the Check level item. In the Project parameters' window you can also change throughout network whether you intend to use a friction factor and/or an absolute roughness.

The last group of parameters is referred to as advanced parameters. For an experienced user there is direct access to a weighting coefficient q which has a default value of 0.5. For special cases you can use weak forward centring of the scheme and hence activate the diffusive part of the truncation error, see Verwey and Yu (1993).

16.4.2 Definition of network layout

An example of a topological representation of a network is shown in Fig.3.1. The solution domain consists of branches connected one to another by means of nodes. Grid points are generated along branches and they represent the place where we are looking for the solution of the governing equations. Different hydraulic structures can be included later at selected places in the network.

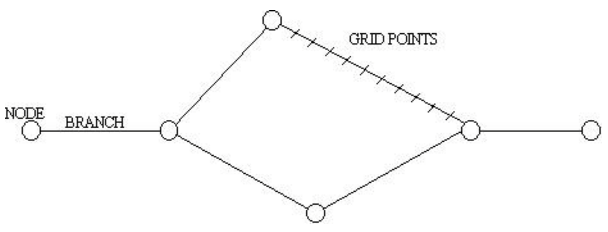


Figure 16.3 Definition Network Layout

For model construction, we can define a range of model elements such as nodes, branches, grid points and hydraulic structures.

Branches

can be used to represent pipes of constant properties. In the pipe network, branches may include hydraulic elements, for example, valves, pumps. Nodes represent the applicable boundary conditions at the end of branches.

Nodes

are elements that represent free branch ends, branch connections or a specific storage. At nodes with one simple pipe connected, boundary conditions are usually defined by specifying the values of piezometric head or discharge as a constant value or as a function of time. Flow continuity and a piezometric level compatibility is assumed at nodes connecting several branches together.

Generally, there are these three different types of nodal boundary conditions:

- H (pressure (m) is given).
- Q (discharge (l/s) is given).
- Compatibility (common H).

Other types of nodes can be given as:

Table 16.2 Node boundary conditions

Node Type	Meaning	Variable
H-Boundary	Given HGL	H=f(t)
Q-Boundary	Given Demand (inflow/outflow)	Q=f(t)
Continuity	Continuity	None
Junction node without demand	Continuity	None
Junction node with demand	Given Demand	Q=const
Tank	Calculated HGL	H=f(t), H=const
Air-Chamber	Calculated HGL	H=f(t)
Vented Air-Chamber	Calculated HGL	H=f(t)
Air-Valve	Calculated HGL	H=f(t)
Emitter	Calculated (pressure dependent) Demand	Q=f(t)

Shaded VARIABLE types are set automatically by the program.

Grid points

are generated automatically by Water Hammer along the branches and they represent the computational grid where the values of piezometric head and discharge are solved and the input and/or output data are required. The program generates grid points based on the hydraulic time step entered by the user, wave speed given for every pipe and Courant number criterion. The system requires a different computational grid for steady state and water hammer computations.

Computational grid and hydraulics structures

The hydraulic components are located either in nodes or on branches. An example of grid-generation in a water distribution application with a valve illustrates the procedure of implementation of the hydraulic components. For water hammer applications the grid is defined as a function of the length of the pipe elements, the wave speed of water hammer and the speed of system operation.

Specific pipe data

Input of pipes is the same as in the case of steady state analysis. Then you have to specify the wave speed. Wave speed (celerity of the pressure wave) is the only one specific (and mandatory) parameter for the water hammer calculations:

- Wave speed: the sonic velocity is also the speed at which the pressure waves generated by water hammer travel in the pip (m/s or ft/s)

In case of a pipe with a check valve, the following fields need to be defined:

- check valve time to open: time interval to open the valve from closed position (sec)
- check valve time to close: time interval to close the valve from open position (sec)
- check valve cracking pressure: pressure that is required to open the valve (m or psi)
- check valve minimum velocity: velocity that is required to keep the valve open (m/s or ft/s)
- check valve is regulating: initial position of the valve i.e. if the valve is initially in the closed position (Yes/No)
- check valve idle interval: time interval that is needed before the valve close or open again (sec)
- check valve can reopen: setting that defines if the valve can re-open after it gets closed (Yes/No)

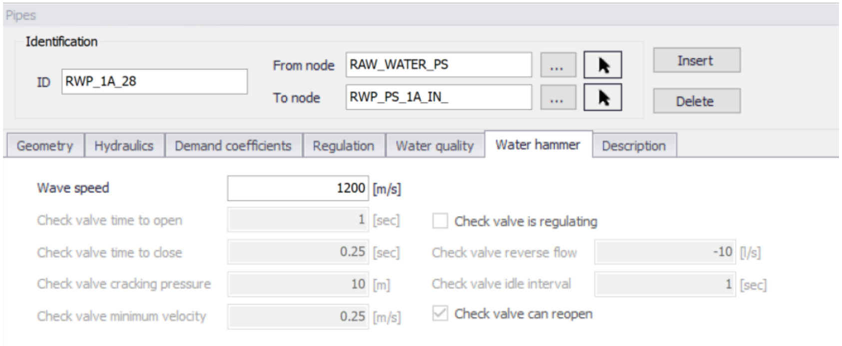


Figure 16.4 Pipe editor with water hammer settings

Junction node demands

Until specified as Water Hammer Boundary conditions, node demands are kept constant through out the water hammer simulation period. Junction demands i.e. multiple demands and their patterns - diurnal curves are use to calculate the steady state i.e. initial conditions for water hammer and they are kept on the same value for the water hammer analysis. Node elevation must be defined for every node.

Control rules

Simple Control Rules and Rule Based Controls are ignored during the water hammer analysis. Valve opening and pump scheduling is handled directly by the specific valve and pump data in Pumps and Valves editors and in Curves and Relations Editor.

Specific pump data

Input of pumps is the same as in the case of steady state analysis. Then you have to specify rated rotational pump speed and its schedule - time series of the rotational pump speed versus time.

Pumps may be located inside pipeline systems (booster pumps) or they may be connected to a suction well. Pumps are frequently used for various pipeline systems, and may operate during hydraulic transients with constant pump speed. Alternatively, the pump speed can decrease and/or increase depending on pump shut-down and/or pump start-up. The greatest difficulties come from hydraulic transient flows caused by turbopumps, since they may work in four quadrants. Four quadrants pumps are currently not supported.

There are in principle four dependent variables describing any state of a pump, namely:

- discharge - Q (m3/s)
- total dynamic head (tdh) - H (m)

- rotational speed - N (rpm)
- shaft torque - T (N.m)

The total dynamic head is defined as follows:

$$tdH = H = \left(\frac{V_d^2}{2g} + \frac{p_d}{\rho g} + z_d \right) - \left(\frac{V_s^2}{2g} + \frac{p_s}{\rho g} + z_s \right) \quad (16.41)$$

where the subscripts, d and s denote the discharge and suction flanges, respectively. Power input P (kW) is defined as:

$$P = \frac{\rho g Q H}{\eta} = T \omega = \frac{T 2 \pi N}{60} \quad (16.42)$$

where h is the pump efficiency and T (N.m) is the torque which may be calculated from this equation.

Manufacturers may provide pump performance characteristics using other variables, e.g., {H, Q, N, P}, {H, Q, N, h}. If the pump operates only in the first quadrant, the typical pump characteristics {H, Q, N, h} for a given rotational speed of a centrifugal pump are shown in Figure 16.5. The H - Q curve should be a monotonously decreasing function and then it is called a stable pump curve. The H - Q performance curve for a pump operating at constant rated speed may be approximated as:

H = b + aQ2

where b is the shut-off head and a is determined for maximum efficiency of the pump.

If the pump characteristics does not satisfy parabolic relation large errors may be produced in transient method and in all computation modules if the pump discharge is out of the Q-H curve.

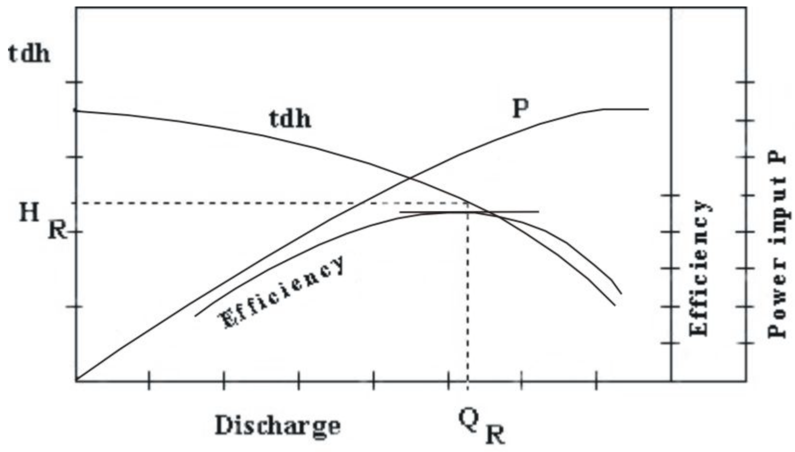


Figure 16.5 Q - H Curve

Another performance characteristic curve which should be specified by the manufacturer is the net positive suction head (NPSH). The absolute pressure at the inlet flange of the pump should be above NPSH in order to avoid cavitation.

By applying the principles of dimensional analysis, the following relationships can be written for a pump operating at two different speeds N1, N2

$$\frac{Q_1}{Q_2} = \frac{N_1}{N_2} \quad \frac{H_1}{H_2} = \left(\frac{N_1}{N_2}\right)^2 \quad \frac{P_1}{P_2} = \left(\frac{N_1}{N_2}\right)^3 \qquad (16.43)$$

Subscripts 1 and 2 are only for corresponding points on an affinity law parabola. The affinity laws for discharge and head are accurate for all types of centrifugal pumps. However, large errors may be produced using the affinity law for a power requirement. It is recommended to compute P from head, discharge and efficiency and not from affinity laws.

Many of the important transient analyses situations are caused by start-up and shutdown of pumps. For a pump power failure the change in rotational speed of the pump depends upon the unbalanced torque applied

$$P = \frac{\rho g Q H}{\eta} = I_w = \frac{T 2 \pi N}{60} \qquad (16.44)$$

where I_w (N.m.s) is combined moment of inertia and D_t (s) is time step used for the calculation.

Pump start-up can be described by similar equation

$$\Delta N = \frac{(\ Tm - T\ )\Delta t 30}{I_{\varpi} \pi} \tag{16.45}$$

where, Tm (N.m) is the motor torque.

The relation between the pump speed and the total pump dynamic head is described by the following equation:

$$tdH_t = \frac{tdH_{100\%}}{N_{100\%}^2} N_t^2 \tag{16.46}$$

where, index (100%) represents the 100% of the pump rated speed and the time index t represents the actual value of tdH and N during the analysis.

Three different modes can be used in the transient flow analysis:

1. pump is controlled by a pump operation schedule (N-time) curve
2. pump is controlled by a pump operation schedule until time of the simulation is equal time of the power failure, then pump shutdown is applied and pump remains stopped till the end of the computation run.
3. pump is primarily stopped (N equals zero) until time of pump start-up is reached, then pump start-up equation is applied.

Moment of Inertia, resistance of a rotating body to the change of its rotational speed, sometimes called rotational inertia. In linear motion, inertial mass is the measure of the resistance of a body to a change in its state of rest or uniform motion in a straight line. In rotational motion, moment of inertia is the measure of the resistance of a body to a change in its rate of rotation.The laws of motion of rotating objects are equivalent to the laws of motion for objects moving in a line, with moment of inertia replacing mass, angular acceleration replacing linear acceleration, and so on.

Force = mass x acceleration (F = ma) (linear motion)

Torque = moment of inertia x angular acceleration (T = Ia) (rotational motion)

The moment of inertia of a body can be calculated by dividing the object up into many small elements each with mass, m. If each element is a distance, ri, from the axis of rotation, the moment of inertia of the body is given by:

$$I\omega \sum_{i=1}^{n} mr_j \tag{16.47}$$

The moment of inertia of a body depends on the axis about which the body is rotated. If two axes of rotation have different distributions of mass around them, then the body will have different moments of inertia for each of these axes.

Torque, a twisting effort applied to an object that tends to make the object turn about its axis of rotation. The magnitude of a torque is equal to the magnitude of the applied force multiplied by the distance between the object's axis of rotation and the point where the force is applied. In many ways, torque is the rotational analogue to force. Just as a force applied to an object tends to change the linear rate of motion of the object, a torque applied to an object tends to change the object's rate of rotational motion.

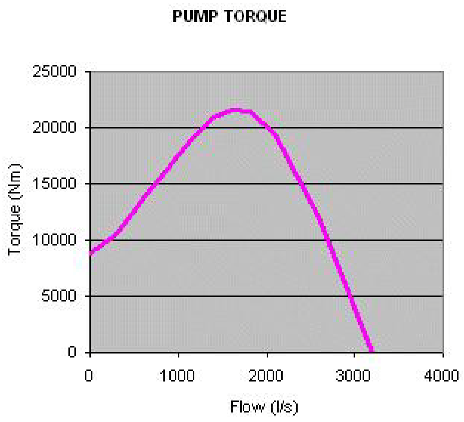


Figure 16.6 Pump torque curve

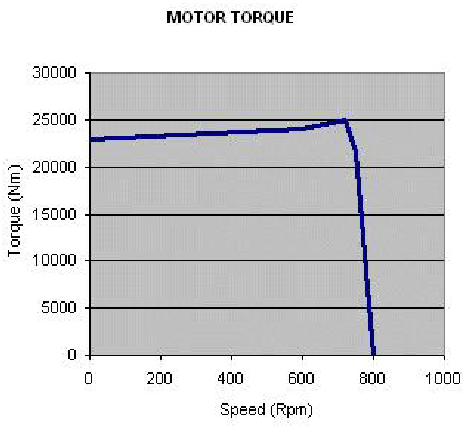


Figure 16.7 Motor torque curve

The following fields need to be entered in addition to water distribution modeling:

- Operation type:
  – Pump schedule: pump operation is defined by a pump speed vs time curve
  – Pump trip off: calculated pump trip off based on pump's data
  – Pump start up: calculated pump start up based on pump's data
- Operation schedule: pump operation defined as pump speed vs time curve that is used before the pump starts up or fails.
- Rotational pump speed: pump speed (rpm)
- Moment of inertia: pump moment of inertia (kg m^2 or lb ft^2)
- Pump torque: pump torque (Nm = m^2 kg s^{-2} or lb ft)
- Motor torque: motor torque (Nm= m^2 kg s^{-2} or lb ft)
- Pump start up time: time when the pump starts up (sec)
- Pump trip off time: time when the pump trips off (sec)

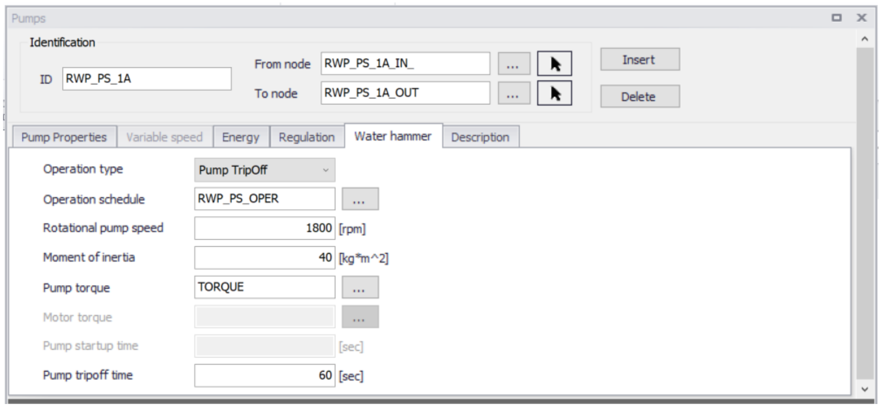


Figure 16.8 Pump editor with water hammer settings

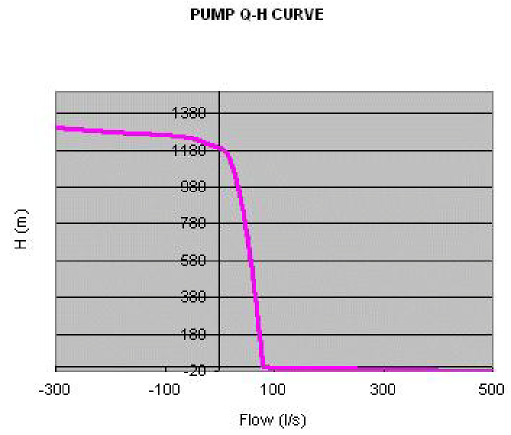


Figure 16.9 Pump Q-H curve defined for 100% rated speed

Specific valve data

Input of valves is the same as in the case of steady state analysis. Then you have to specify valve characteristic curve (in case of TCV valves) and valve schedule - relation between valve opening versus time.

The relationship between the flow *Q* and the head drop Δ*H* is expressed using a discharge coefficient *Cd* for:

In-line valve

$$Q = Cd \cdot Av \cdot \sqrt{2g\Delta H} \quad \text{where} \quad Cd = \frac{1}{\sqrt{\xi}} \tag{16.48}$$

where *Av* is the valve area and ξ is the valve minor loss coefficient.

Free-discharge valve

$$Q = Cd \cdot Av \cdot \sqrt{2g\Delta H} \quad \text{where} \quad Cd = \frac{1}{\sqrt{\xi + 1}} \tag{16.49}$$

where ξ is the valve minor loss coefficient for a free-discharge valve.

Values of the discharge coefficients as functions of the relative valve opening (which is the ratio of valve and pipe area) have to be specified in the in Curve Editor. Typical representative data is of the following form

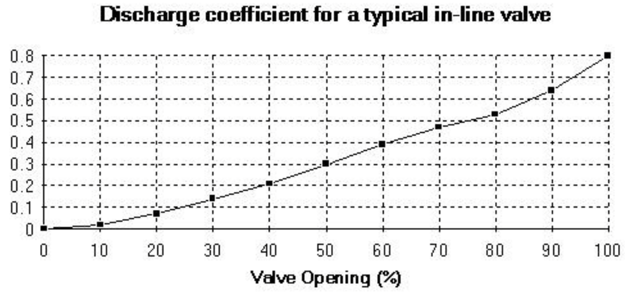


Figure 16.10 Discharge coefficient for a typical in-line valve

Remarks:

TCV Throttle Control Valves can also be used as Isolation Valves for example for isolation of a pipe section in case of repair, isolation of a pump, etc.

The following fields need to be entered in addition to water distribution modeling:

- Operation schedule: valve opening (stroke position) vs time
- Valve characteristics: valve Cd or Kv coefficient vs time

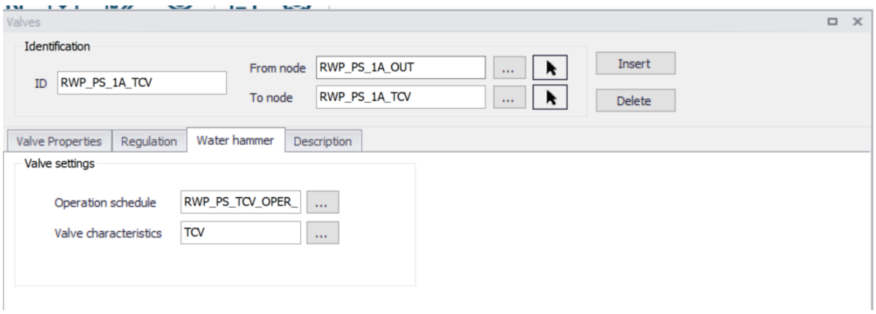


Figure 16.11 Valve editor with water hammer settings

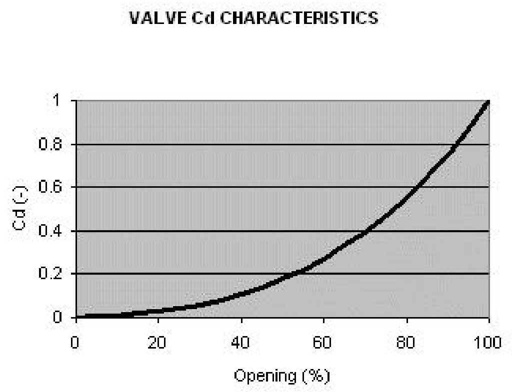


Figure 16.12 Valve Cd Characteristics

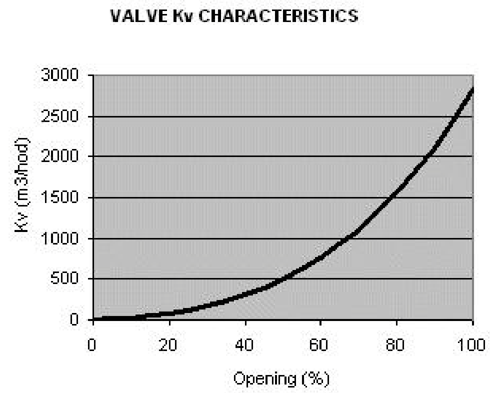


Figure 16.13 Valve Kv Characteristics (Example)

The relation between Cd and Kv valve coefficients is given by the following equation:

$$Cd = \frac{Kv}{3600A\sqrt{20g}} \tag{16.50}$$

The relation between Cd valve coefficient and x minor loss coefficient is given by the following equation:

$$Cd = \frac{1}{\sqrt{\xi - 1}} \tag{16.51}$$

or, for an in-line valve:

$$Cd = \frac{1}{\sqrt{\xi}} \tag{16.52}$$

Note, that the valve minor loss coefficient used for the steady state analysis must correspond the initial valve opening used for the water hammer analysis.

Specific project options settings

Analysis type included fast transient flow analysis. Currently, only SI units with LPS are allowed for the transient flow analysis along with Darcy-Weisbach friction expression. Specific numeric parameters, such as theta - used to centre the high order finite difference scheme in time, default value of 0.5, and others can be defined.

Specific time settings

Running the fast transient analysis requires entering specific time setting, namely hydraulic time step and duration of the analysis. Pressure waves travels with a high speed in the pressurized pipe networks; wave speed in steel pipes is app. 1,200 m/s. In order to maintain Courant number criterion, dt - time step has to be very small number such as dt = 0.1s.

$$Cr = 1 = a\frac{\Delta t}{\Delta tx} \tag{16.53}$$

in which a - wave speed, dt - time step, dx - grid step, Cr - Courant number, a non dimensional parameter.

Specific curves data

The following curve types below are available in the 'Curves and relations' editor, for use in Water Hammer simulations:

Table 16.3 Curve data

Curve type	Description
HGL transient boundary	Define how HGL changes in time
Q transient boundary	Define how flow changes in time (positive value-outflow, negative value-inflow)
Valve schedule	Define valve opening and closing as a function of time

Table 16.3 Curve data

Curve type	Description
Valve characteristic	Flow coefficient versus valve opening
Pump schedule	Define pump starting and closing as a function of time
Pump torque	Pump torque versus flow
Motor torque	Motor torque versus pump rotational speed
Dual-acting valve characteristic	Air discharge versus gauge pressure

16.4.3 List of components

List of supported components

The following components are supported by the Water Hammer simulations.

Table 16.4 List of supported components

Component	Remark
Tank	Supported
Pump	Supported
Pressure reducing valve PRV	Not supported (*)
Pressure sustaining valve PSV	Not supported (*)
Pressure breaker valve PBV	Not supported (*)
Flow control valve FCV	Not supported (*)
Throttle control valve TCV	Supported
Closed pipes	Supported
Pipes with check valves CV	Supported
Node demands	Multiple junction demands including their patterns are kept constant during water hammer analysis.
Emitter	Supported

(*) replace the valve with a throttle control valve TCV and use the steady state valve opening (stroke position) as the initial valve opening in the valve operation schedule curve used in water hammer setup.

List of unsupported components

The following components are not supported by the Water Hammer simulations.

Table 16.5 List of unsupported components

Component	Remark
General purpose valve GPV	Not supported
Simple control rules	Not supported
Rule base controls	Not supported
Patterns	Demand and Reservoir patterns need to be entered as Transient Boundary Conditions

List of additional components

Several additional network components are used in Water Hammer simulations comparing to EPANET based simulation. These components (structures) are classified according to their location either in nodes or on branches.

Table 16.6 List of new components

Component	Remark
Air Chamber	Supported
Vented air-chamber	Supported
Air Valve	Supported

16.4.4 Components located in nodes

One of the most frequently used components of water distribution networks are tanks. Depending on their geometry, the tanks are classified as rectangular tanks, circular tanks, or tanks with the Depth-Volume curves. Tanks are entered in the same way as in the case of steady state or extended period analysis.

Tanks

Surge Tanks have been widely used for hydroelectric systems in order to protect the low-pressure supply tunnel. They may also sometimes be suitable for water supply schemes. There are various types of Surge Tanks. The schematic presentation of common Surge Tanks is the same as mentioned above for Tanks.

The governing equations describing their hydraulic behavior are the dynamic equation and the continuity equation. Losses are disregarded at the junction, but are taken into account for pipes. Parameters characterizing the Surge Tank are:

Parameters:

- Node ID.
- Maximum water depth above datum.
- Starting water depth for computation.
- Tank bottom level.
- Tank Type:
  - Rectangular tank: [a] [b] right prism rectangular tank, the base with sides a, b
  - Circular tank: vertical cylinder with diameter D
  - Variable: depth versus volume curve

Air-Chambers

Air Chambers contain compressed air which prevent very low minimum pressures in the pipeline and hence column separation. They are frequently used behind the pumps in water supply pipelines. Mostly they are cylindrical with a vertical and/or horizontal axis. A horizontal cylinder may be preferred for a very long pipeline when a large volume of air is required. The analysis is similar for both cases, but the computation of the volume of air in a horizontal cylinder is more difficult. Figure 16.15 illustrates an Air Chamber with a vertical cylindrical tank.

The hydraulic behavior of an Air Chamber is described by the relation between air pressure, its volume and continuity equation. It is assumed that the enclosed air follows the polytropic relation for a perfect gas

$$C = H_{air} * \forall_{air}^{\kappa} \tag{16.54}$$

in which H_{air} and $\forall_{air}$ are the absolute pressure head and the volume of the enclosed air, κ is the exponent in the polytropic gas equation (κ = 1.0 for an isothermal expansion, κ = 1.4 for adiabatic expansion). The orifice losses are different for the inflow and outflow from the chamber.

The following fields need to be entered in addition to water distribution modeling:

- Polytropic expansion: the exponent in the polytropic gas equation (default value κ=1.2).

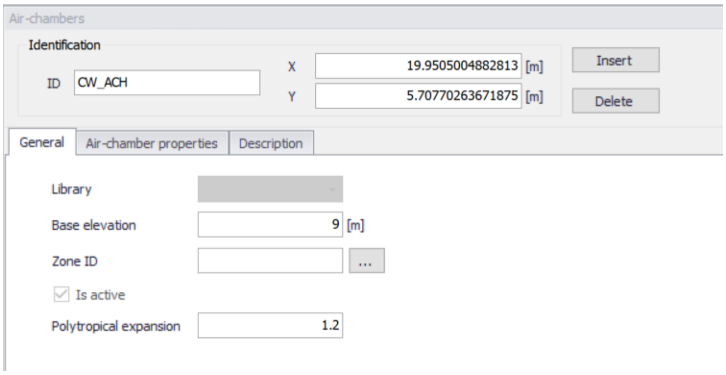


Figure 16.14 Air chamber editor with water hammer settings

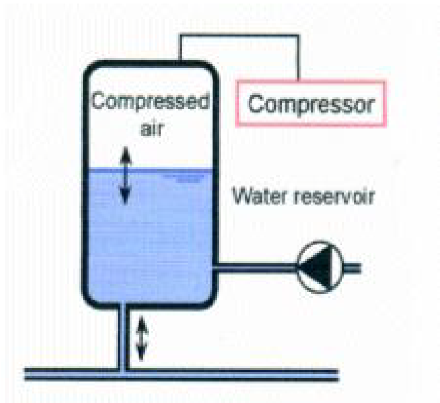


Figure 16.15 Air chamber

Air-Valve

Air valves, similar to Vented Air Chamber contain air which prevent very low minimum pressures in the pipeline and hence column separation. Air valves are modelled as small Vented Air Chamber equipped by dual-acting valves that allow air to be sucked into its chamber and to escape therefrom, while preventing the outflow of liquid. When the pressure inside the surrounding pipes drops below the atmospheric pressure, air-valve opens and the air flow into a system. The proper valve characteristics are required to set by a user. As soon as the liquid starts flowing back into the dual-acting valve, valve closes.

The amount of air that is entering the air valve during the sub-atmospheric conditions or leaving the air-valve during pressurized conditions is taken from the dual-acting valve characteristics. Next chart shows characteristics of Pont&Mousson, Ventex dual-acting valve, diameter of 150mm.

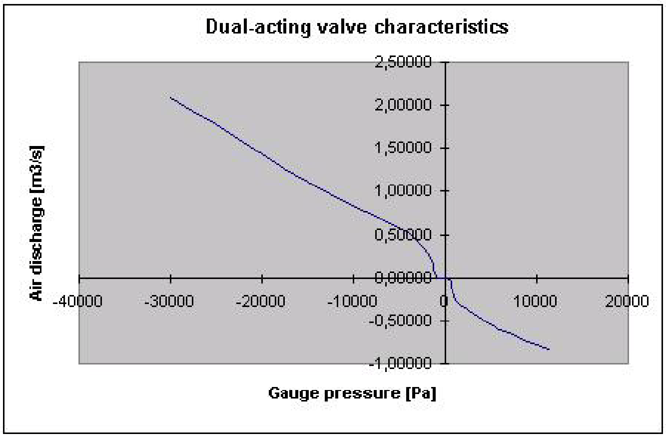


Figure 16.16 Dual-acting Valve Characteristics

When the pressure inside the air-valve drops below the atmospheric pressure, dual-acting valve opens and the air flows into a chamber. The proper valve characteristics must be defined by the user. As soon as the liquid starts flowing back out of the air-valve, the air-valve closes. The sizing of air-valve remains somewhat empirical, and it is recommended to contact the air-valve manufacturer as part of the design process.

The following fields need to be entered in addition to water distribution modeling:

- Valve diameter: diameter (mm or in)
- Polytropic expansion: the exponent in the polytropic gas equation (default value κ=1.2).
- Dual acting valve curve: air valve characteristics

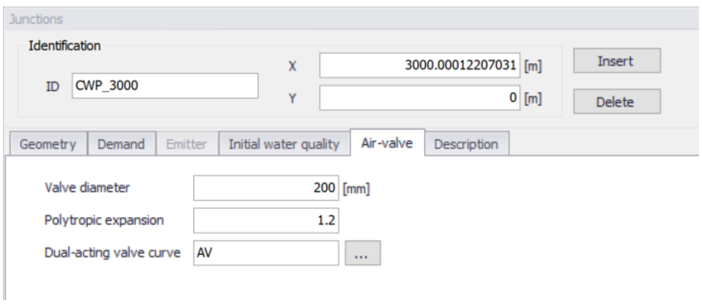


Figure 16.17 Air valve editor with water hammer settings

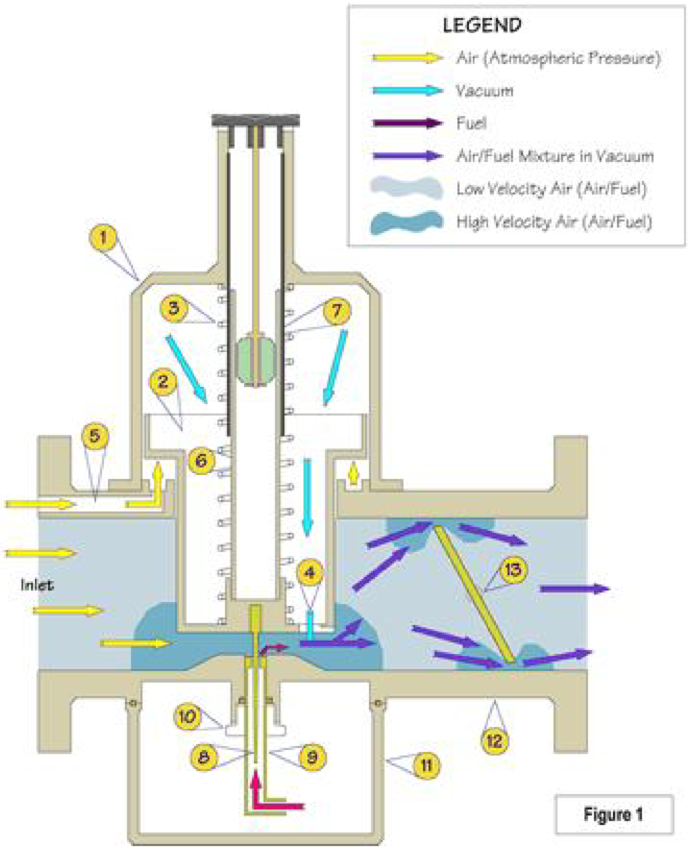


Figure 16.18 Air Valve

16.4.5 Tutorial

This section contains a brief summary describing how to set up Water Hammer when creating a new project based on a steady state model.

1. Add wave speed to every pipe.
2. Make sure every junction node has elevation defined.
3. Use hydraulic time step of 0.01 or 0.05 sec for networks in towns and 0.1sec for large transmission systems.
4. Use report time step 0.5 - 1 sec or 0.1 (as in above case)
5. Project options | Water hammer - set theta to 0.505 - 0.51 for better stability.
6. You do not need any water hammer boundaries for tanks/reservoirs (they are set automatically by the program)
7. You do not need any water hammer boundaries for junction node demands (they are set automatically by the program)

8. Set "user defined pipe length = 10m" for all pipes that have a shape length < 10m (for numerical purposes).
9. If you have pump stations with multiple pumps, close (=remove) all but 1 for the transient mode and use the equivalent pump characteristics.
10. Change all valves to TCV, e.g. PRV and PSV or FCV valves need to be replaced by a TCV with a setting (local loss) that will give the same pressures/flows.
11. You might need to add a TCV valve to an air chamber connecting pipe and close it initially and open with the pump fail.
12. If you have any inflows into the system, Q(t) boundary conditions must have positive flow values (negative flow values = outflow).

17 Optimization - Pump and Valve Scheduling

The purpose of this optimization-based tool is to support scheduling of pumps and operation of control valves. The optimization is based on optimization algorithms that can run with any extended period analysis model.

These two algorithms are available:

- SCE: The Shuffled Complex Evolution (SCE) method is a global optimization algorithm that combines various search strategies, including (i) competitive evolution, (ii) controlled random search, (iii) the simplex method, and (iv) complex shuffling.
- DDS: The Dynamically Dimensioned Search algorithm (DDS) is an optimization algorithm designed for large set of parameters. It automatically scales the search to find good solution within the maximum number of model calls. The algorithm starts to search globally, and then becomes more local as the number of models call approaches the maximum allowable number. Candidate solutions are created by perturbing the current values in the randomly selected dimensions only. The dimension of search is automatically adjusted.

The optimization setup includes "Controls" and "Targets". *Controls* are valves and pumps, and their operations and *Targets* are goals such as requested water levels or flows, pressures, etc.

Controls and their operations may be:

- Pump ON/OFF versus time (hourly basis)
- Pump ON/OFF defined as a repeating pattern (hourly basis)
- Pump speed versus time (hourly basis)
- Valve opening % versus time (hourly basis)
- Valve flow set-point versus time (hourly basis)

Targets are:

- Water level in a tank/reservoir (last value, minimum, maximum, average, span, time series)
- Water quality at a node (junction, tank)
- Total volume or a volume difference in a tank/reservoir (can be translated to the water level)
- Maximum or average or total flow across a valve
- Pump power
- Pump energy costs
- Source water balance (e.g. 60% of total water from a source 'A' and 40% of a total water flow from a source 'B')

The Optimization dialog box is reached by selecting 'Model type' from 'General Settings' from the Setup tree and then by selecting the 'Optimization' option. Once selected, the Optimization is added to the Special analyses group in the Setup tree.

17.1 Methods

A list of the Optimization dialog box data entries for Figure 17.1 and Figure 17.2 follows with a short description given for each entry. Note, that it is possible to insert multiple optimization analyses, each with its own settings, and then run the selected optimization by selecting it from the list. This is convenient when you need to investigate various optimization settings or different control strategies.

Insert

This button is used to insert a new Optimization into the list.

Delete

This button is used to delete an Optimization from the list.

Run

This button is used to run the simulation for the active Optimization in the list.

Stop

This button is used to stop (cancel) the optimization that is currently running.

Report

This button is used to report the optimization results.

17.1.1 DDS Optimization method data entries

This section describes settings for the DDS optimization method.

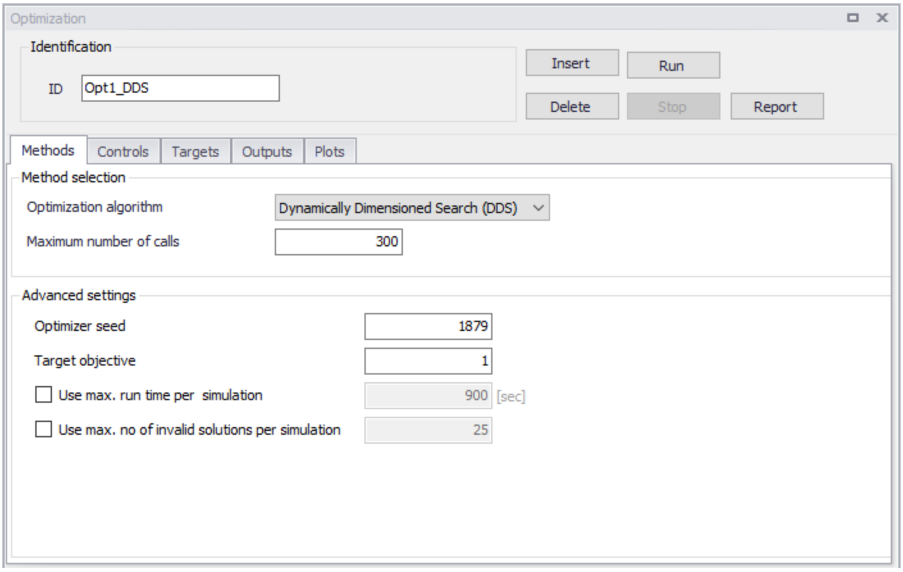


Figure 17.1 DDS Optimization settings

Maximum number of calls

This data entry is one of the stop criteria and it is the maximum number of model call during the optimization process. If the maximum number of model calls is reached, the optimization process will stop, and it will report the best solution that was found.

Optimizer seed

This data entry is used to initiate random number generator. There is no need to change the entry except in special cases.

Setpoint target objective

This data entry defines another stop criteria that compares the actual value of the objective function and it stops when its value is below this data entry.

Use maximum run time per simulation

This data entry can be used in special cases when the hydraulic simulation due to the “random” choice of control variables by the optimizer might take exceptionally long time to finish. In such case, if this criterion is used, the program will cancel the hydraulic simulation and it will use a high penalty for its settings for the optimizer.

Use maximum number of invalid solutions per simulation

This data entry can be used in special cases when the hydraulic simulation due to the “random” choice of control variables by the optimizer results in unbalanced or hydraulically unstable conditions during the extended period simulation run. In such case, if this criterion is used, the program will cancel the hydraulic simulation and it will use a high penalty for its settings for the optimizer.

17.1.2 SCE Optimization method data entries

This section describes settings for the SCE optimization method.

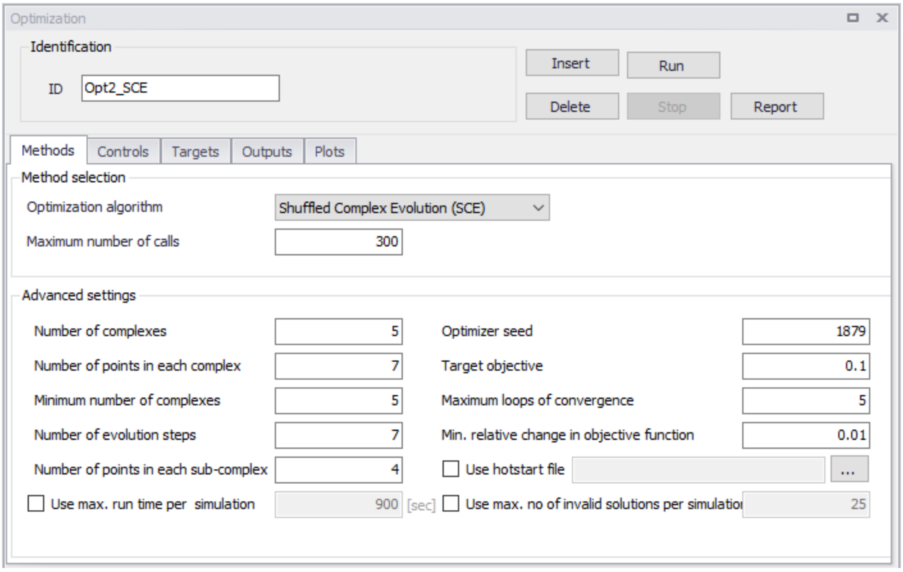


Figure 17.2 CSE Optimization settings

Maximum number of calls

This data entry is one of the stop criteria and it is the maximum number of model call during the optimization process. If the maximum number of model calls is reached, the optimization process will stop, and it will report the best solution that was found.

Number of complexes

This data entry defines the number of complexes (p) used in the optimization process. It is used to calculate the initial population (number of different model setups to run). The population size is s = p x m.

Number of points in each complex

This data entry defines the number of points in each complex (m) used in the optimization process. It is used to calculate the initial population (number of different model setups to run). The population size is s = p x m.

Minimum number of complexes

The minimum number of complexes is required when the number of complexes is reduced during the optimization.

Number of evolution steps

This data entry defines the number of evolution steps taken by each complex before complexes are shuffled.

Number of points in each sub-complex

This data entry defines the number of points in each sub-complex.

Optimizer seed

This data entry is used to initiate random number generator. There is no need to change the entry except in special cases.

Target objective

This data entry defines another stop criteria that compares the actual value of the objective function and it stops when its value is below this data entry.

Maximum loops of convergence

This data entry defines the maximum number of loops used in the optimization.

Minimum relative change in convergence function

This data entry is another stop criteria where solutions are compared in terms of a change in the objective function and the optimization stops when there is no improvement.

Use maximum run time per simulation

This data entry can be used in special cases when the hydraulic simulation due to the "random" choice of control variables by the optimizer might take exceptionally long time to finish. In such case, if this criterion is used, the program will cancel the hydraulic simulation and it will use a high penalty for its settings for the optimizer.

Use maximum number of invalid solutions per simulation

This data entry can be used in special cases when the hydraulic simulation due to the "random" choice of control variables by the optimizer results in unbalanced or hydraulically unstable conditions during the extended period simulation run. In such case, if this criterion is used, the program will cancel the hydraulic simulation and it will use a high penalty for its settings for the optimizer.

Use hotstart file

This data entry can be used to start the new optimization run from the results of the previous optimization run. Select the file with the extension .dat that contains the optimization results.

17.2 Controls

These data entries define *Controls* such as valves and pumps, and their operations. A list of the Control dialog box data entries for Figure 17.3 follows with a short description given for each entry. Multiple controls can be entered in this dialog box.

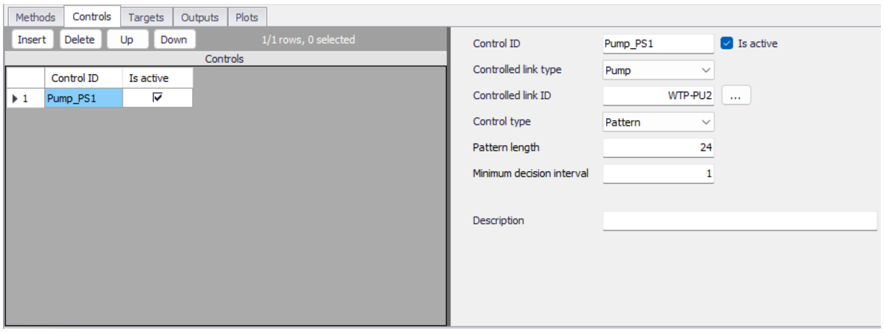


Figure 17.3 Optimization controls settings

Control ID

This data entry is used to identify the control.

Is active

This check box allows the user to toggle the Active status of the control on and off. The simulations will omit all controls that are not active.

Controlled link type

This data entry defines the type of the controlled link, the following options are available:

- Pump ON/OFF or relative speed
- Valve settings

Controlled link ID

This is the ID of the controlled link.

Control type

This data entry defines the how the link will be operated, the following options are available:

- Pattern (decisions about the link operation will be based on a repeating pattern)
- Time levels (decisions about the link operation will be based on predefined time levels)
- Clock time (decisions about the link operation will be based on predefined clock time levels)

Pattern length

This data entry is used when the Control type is 'Pattern'. It is the length (duration) of the repeating pattern that will be used for the link operation. A typical entry is 24 (hours).

Minimum decision interval

This data entry is when the Control type is 'Pattern'. It is the minimum time step of the repeating pattern that will be used for the link operation. A typical entry is 1 (hour).

Optimization set-point table

This data entry is when the Control type is 'Time levels' or 'Clock time'. It is the name of the time series curve that defines the time levels when the decision will be made for the link operation.

Description

This data entry is for the user defined description of the control.

17.3 Targets

These data entries define Targets that are goals such as requested water levels or flows, pressures, etc. A list of the Control dialog box data entries for Figure 4 follows with a short description given for each entry. Multiple targets can be entered in this dialog box.

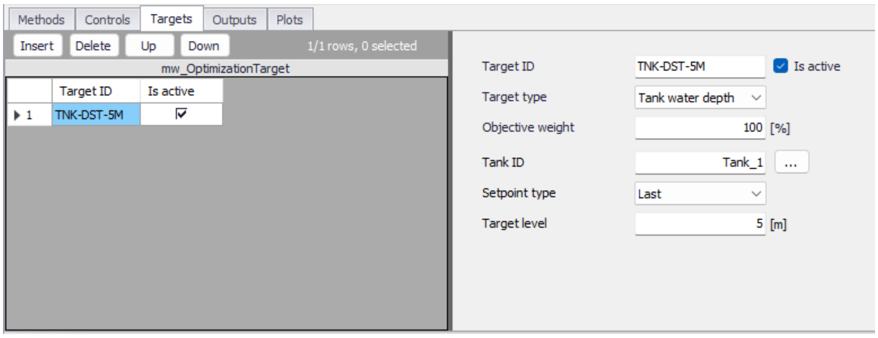


Figure 17.4 Optimization targets settings

Target ID

This data entry is used to identify the target.

Is active

This check box allows the user to toggle the Active status of the target on and off. The simulations will omit all targets that are not active.

Target type

This list defines the type of a target, the following options are available:

- Tank water level (depth of water in the tank)
- Water quality
- Pressure (junction pressure)

- Flow (link flow)
- Pump power (optimization pumps or all pumps)
- Pump energy costs (optimization pumps or all pumps)
- Source water balance (% of water supplied from a tank/reservoir)

Objective weight

This data entry is used to specify the weight of this target. Use different number if you want to prioritize one target over another).

Tank ID

This data entry is for entering the ID of the target tank (when the target type = tank water level)

Junction ID

This data entry is for entering the ID of the target junction (when the target type = pressure)

Link ID

This data entry is for entering the ID of the target link (when the target type = flow)

Water balance definition

This table is for entering the list of water sources and their water supply percentages (when the target type = source water balance). Note, that the data entry here is the outlet pipe from a storage tank and not the storage tank. That is in case that the storage tank have multiple outlets and some are feeding different zones or parts of the water supply system.

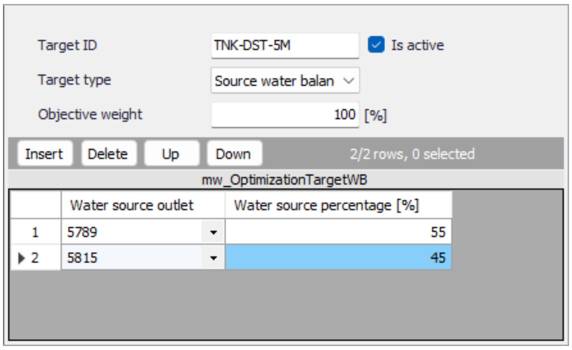


Figure 17.5 Inputs for Source water balance

Setpoint type

This list defines the set-point type. The following options are available:

- Last (this is the value at the end of the simulation)
- Minimum (this is the minimum value during the simulation)

- Maximum (this is the maximum value during the simulation)
- Average (this is the average value during the simulation)
- Span (this is the difference between the minimum and maximum value during the simulation).

Target level

This data entry is used to define the target set-point value.

17.4 Outputs

Outputs provides a list of computed controls and their optimized settings and a summary of targets with requested and computed values.

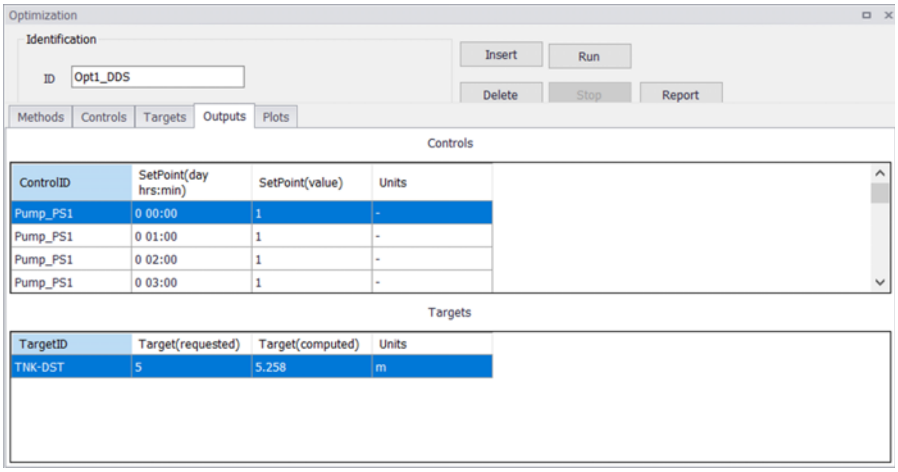


Figure 17.6 Optimization outputs

In addition to this summary, it is always possible to use the standard time series plots to select a target storage tank, for example, and plot the time series of the computed water level to see its full time history.

17.5 Plots

Plots provides time series graphs with computed controls and their optimized settings.

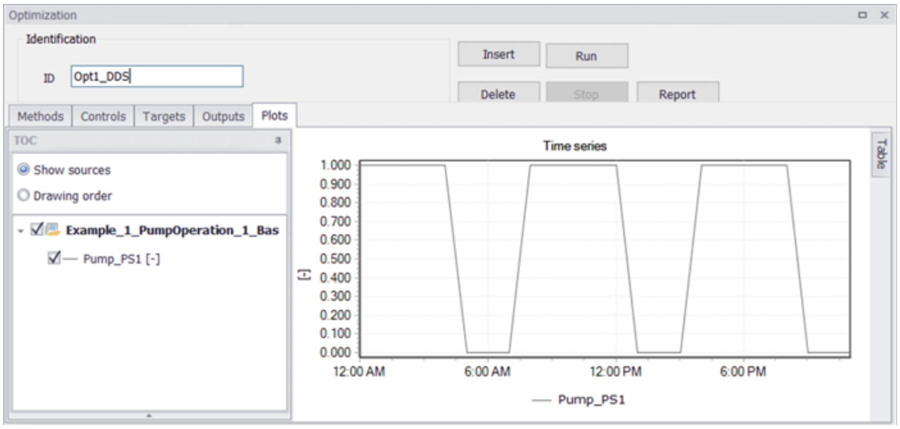


Figure 17.7 Optimization plots

17.6 Report

Report provides a HTML version of the outputs, i.e. a list of computed controls and their optimized settings and a summary of targets with requested and computed values.

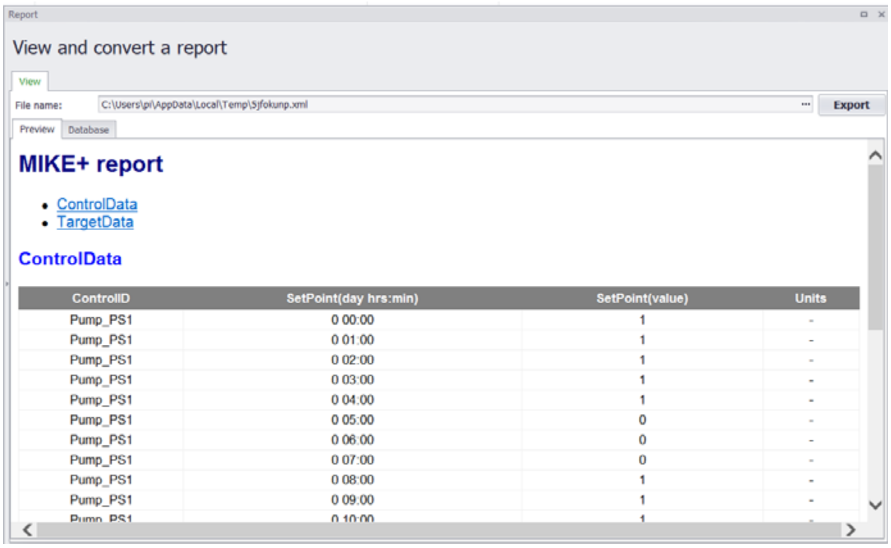


Figure 17.8 Optimization report

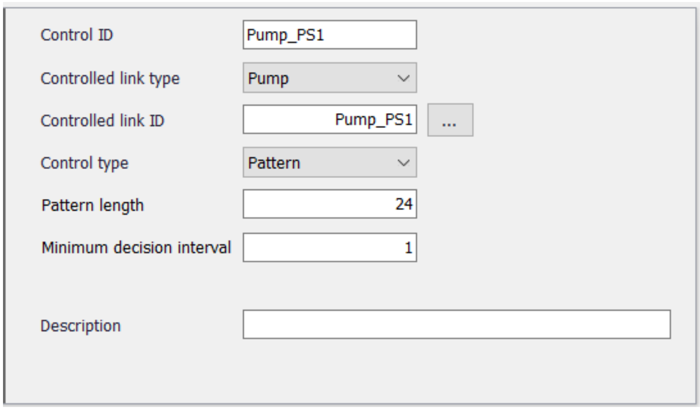
17.7 Examples

Several examples are provided here to illustrate how controls and targets can be defined.

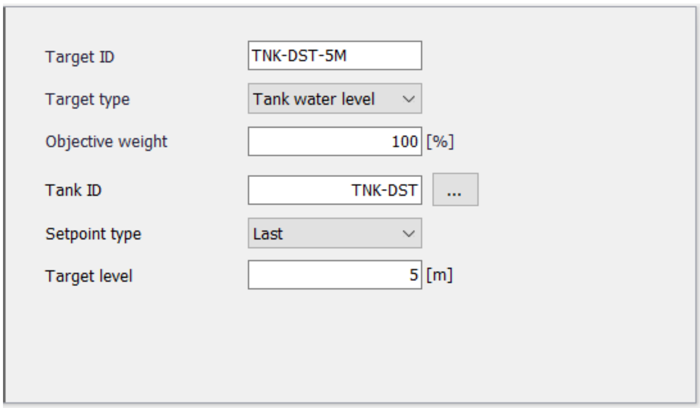
17.7.1 Example 1 - Pump control

The system contains a pump and a storage tank that is receiving the pumped water. The storage tank has also outlets that are supplied by gravity. The goal of the optimization is to find a pump operating strategy that will maintain the water level at the end of the simulation at 5m.

We will enter the pump into Controls:



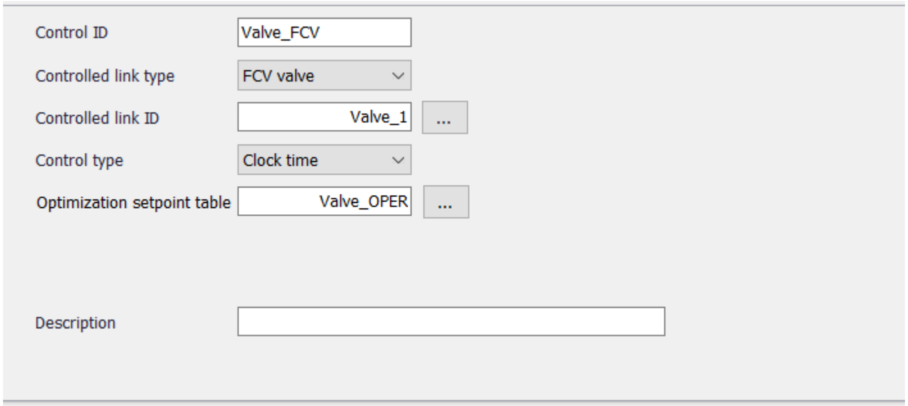
And we will enter the storage tank level into Targets:



17.7.2 Example 2 - Valve control

The system contains a gravity transmission pipeline and a downstream storage tank that is receiving the water. There is an inlet gate valve for the storage tank. The storage tank has also outlets that are supplied by gravity. The goal of the optimization is to find an operating strategy that will maintain the water level at the end of the simulation at 5m.

We will enter the flow control valve into Controls:

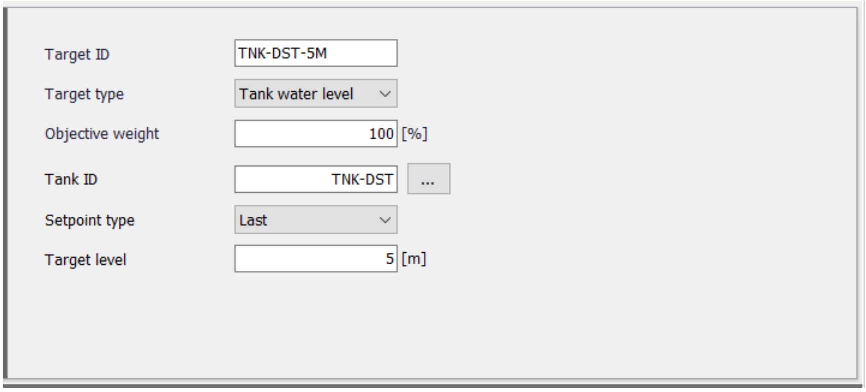


The valve set-point table for the decision levels could look something like this:

	Time of the day [h:mm tt]	Min. flow [l/s]	Initial. flow [l/s]	Max. flow [l/s]
1	12:00 AM	0	150	500
2	2:00 AM	0	150	500
3	4:00 AM	0	150	500
4	6:00 AM	0	150	500
5	8:00 AM	0	150	500
6	10:00 AM	0	150	500
7	12:00 PM	0	150	500
8	2:00 PM	0	150	500
9	4:00 PM	0	150	500
10	6:00 PM	0	150	500
11	8:00 PM	0	150	500
12	10:00 PM	0	150	500

These are the decision levels i.e. time levels when the valve settings can change. The minimum and maximum values are used to define the range within which the flow control valve can operate, and the initial value is the initial valve setting.

And we will enter the storage tank level into Targets:


Target ID
TNK-DST-5M
Target type
Tank water level
Objective weight
100
[%]
Tank ID
TNK-DST
...
Setpoint type
Last
Target level
5
[m]

18 Online Analysis

The purpose of the Online analysis is to support development of online versions of a hydraulic model, for which two other modules are also needed:

- Water Distribution Online to provide automatic updates of a hydraulic model based on SCADA data
- WaterNet Advisor to provide a user interface to the model results and SCADA data.

The Online analysis editor allows you define the online model configurations such as the sensor mapping (SCADA vs hydraulic model) and other entries. Here is an overview of the data input:

- Settings: general entries and configurations
- Sensors: mapping between the SCADA and hydraulic model
- Demand zones: definitions for demand zones and demand distribution
- Controls: definitions for pumps and control valves
- Comparisons: definitions for comparisons between the measured (SCADA) and simulated data (hydraulic model)
- Demand predictions: entries for demand prediction module.

The Online analysis editor is reached by selecting 'Model type' from General Settings from the 'Setup' tree and then by selecting the 'Online analysis' option. Once selected, the Online analysis is added to the Special analyses group in the 'Setup' tree.

18.1 Settings

A list of the Settings data entries follows with a short description given for each entry. This editor has a free-form i.e. it allows you to enter any number of entries, each consisting of a keyword and a section. The complete list and description of these entries is provided in the WD Online User's Guide. This editor contains a predefined set of commonly used configuration entries that can be used as a template or default settings.

Insert

This button is used to insert a new row into the list.

Delete

This button is used to delete the selected row from the list.

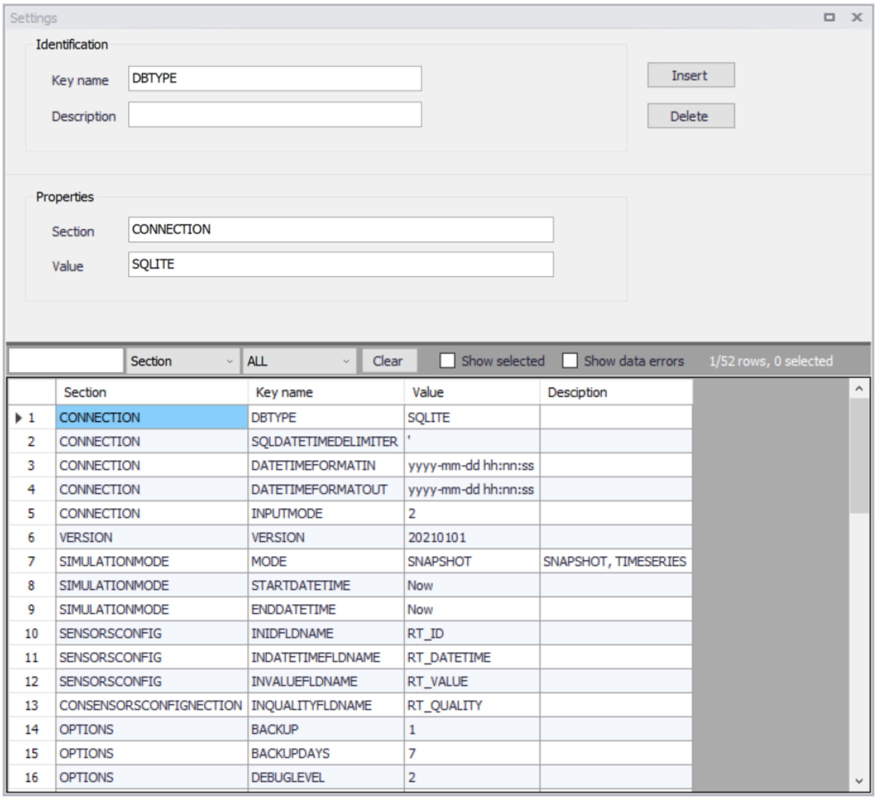


Figure 18.1 The Settings editor for the Online analysis

The editor entries are described below.

Keyname

Entry with the keyword name, see "Water Distribution Online User's Guide" for detailed descriptions.

Section

Entry with the section name, see "Water Distribution Online User's Guide" for detailed descriptions.

Value

Entry with the value, see "Water Distribution Online User's Guide" for detailed descriptions.

Description

User-defined description of the entry.

18.2 Sensors

A list of the Sensors editor's data entries follows with a short description given for each entry. Sensors allows you to define the mapping between the

SCADA tags and corresponding model elements (tanks, pumps, valves, etc.). Based on these entries, the hydraulic model can be automatically updated based on the SCADA (telemetry) data.

Insert
This button is used to insert a new row into the list.

Delete
This button is used to delete the selected row from the list.

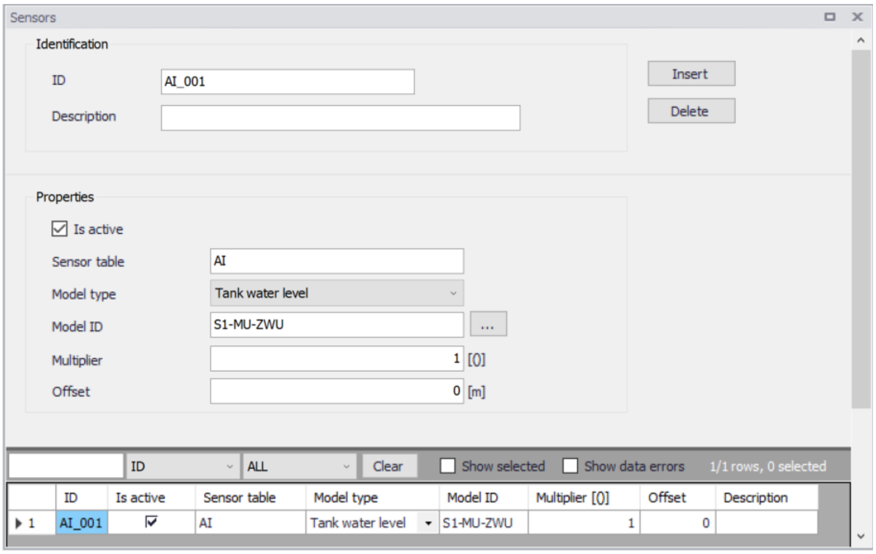


Figure 18.2 The Sensors editor for the Online analysis

The editor entries are described below.

(Sensor) ID
This is the SCADA tag name in the table with SCADA data.

Description
User-defined description of the entry.

Sensor table
This is the name of the table containing SCADA data.

Is active
This option allows you to enable or disable the sensor without adding or removing it from the list.

Model type
This list defines how the SCADA data will be used in the automatic model update. The following options are available:

- Node demand
- Reservoir level
- Tank water level
- PRV valve setting
- PSV valve setting
- PBV valve setting
- FCV valve setting
- TCV valve setting
- GPV valve setting
- Pump speed setting
- Pump open/closed
- Open/closed setting
- Valve or pump rule override
- Multiple node demand with pattern
- Node demand without pattern
- Node demand with pattern
- Special demand, not changed and with pattern based on SCADA data
- Average day demand multiple demand item, constant or with pattern
- Water quality baseline source strength
- Ambient temperature
- Global bulk reaction rate coefficient.

Model ID

This is the ID (MUID) of the hydraulic model element (node or link).

Multiplier

This entry is the multiplier "k" that will be used to multiply the SCADA value before using it in the model update. The model value = scada value * k + n.

Offset

This entry is the offset "n" that will be added to the SCADA value before using it in the model update. The model value = scada value * k + n.

18.2.1 Example

This entry will define a link / connection between the model storage tank "TNK-SWEETWATER" water depth and the corresponding SCADA sensor "AI_100_01" from a table "AI" and it will convert the measured water depth from "cm" into "m".

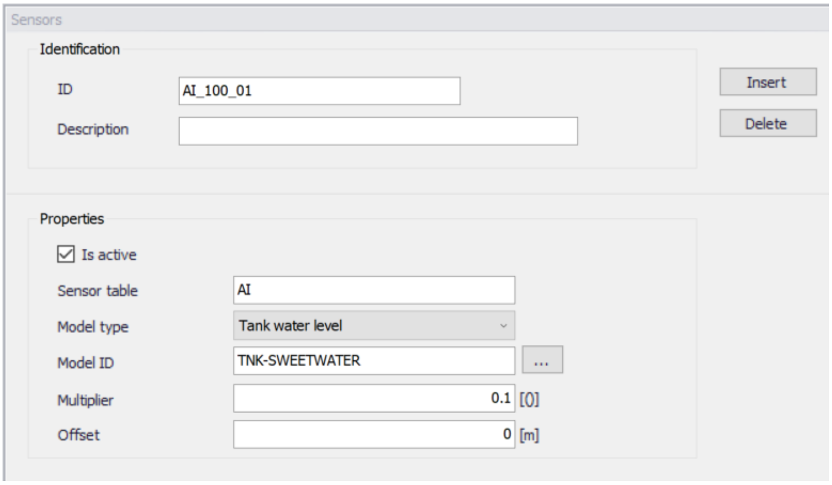


Figure 18.3 Example - Sensors data input

18.3 Controls

A list of the Controls editor's data entries follows with a short description given for each entry. Controls are used to define a link / connection between model pumps and control valves and their corresponding entries in the telemetry (SCADA) system. The purpose of this link is to be able to develop pump and valve operations based on the historical data (when the program runs in the hindcast mode) and to reproduce the behavior of the physical system in a hydraulic model. These entries are not needed for the real-time (snapshot) model or for the forecast mode.

Insert

This button is used to insert a new row into the list.

Delete

This button is used to delete the selected row from the list.

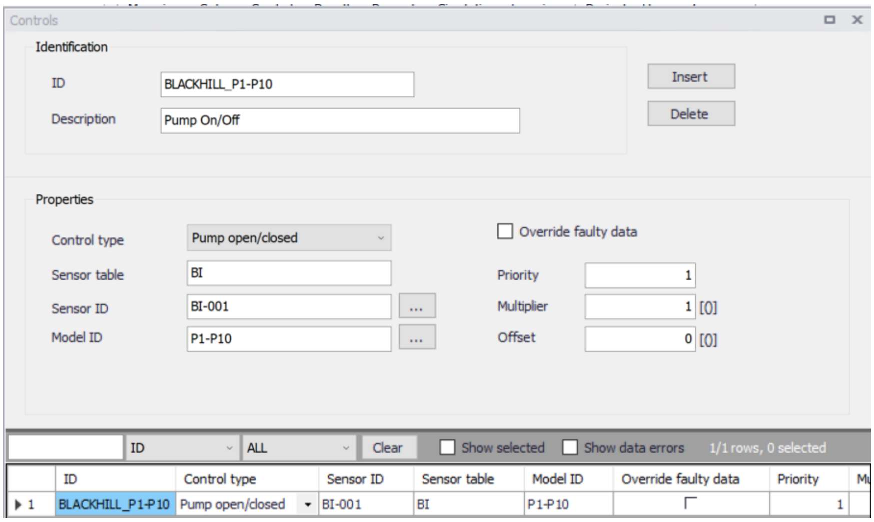


Figure 18.4 The Controls editor for the Online analysis

The editor entries are described below.

ID
This is the identification of the control.

Description
User-defined description of the entry.

Is active
This check box allows the user to toggle the Active status of the control on and off.

Sensor ID
This is the SCADA tag name in the table with SCADA data.

Sensor table
This is the name of the table containing SCADA data.

Control type
This list defines how the SCADA data will be used in the automatic model update. The following options are available:

- Tank HGL level
- Pump open/closed
- Open/closed
- PRV valve setting
- PSV valve setting
- TCV valve setting

- FCV valve setting
- Pump speed.

Model ID
This is the ID (MUID) of the hydraulic model element (node or link).

Priority
This data entry defines the rule priority in case multiple rules are used to control the same node or link. The priority is the number 1-5, the highest number has the highest priority.

Override faulty data
This option (when enabled) will use predefined model rules (if any) in case that the SCADA data are flagged as bad quality (Quality = 0).

Multiplier
This entry is the multiplier "k" that will be used to multiply the SCADA value before using it in the model update. The model value = scada value * k + n.

Offset
This entry is the offset "n" that will be added to the SCADA value before using it in the model update. The model value = scada value * k + n.

18.3.1 Example

This entry will define a link / connection between the model pump "PI-P10" and the corresponding SCADA tag "BI-001" that will be used in the hindcast simulation mode to replicate the pump operations (ON/OFF) based on the telemetry data.

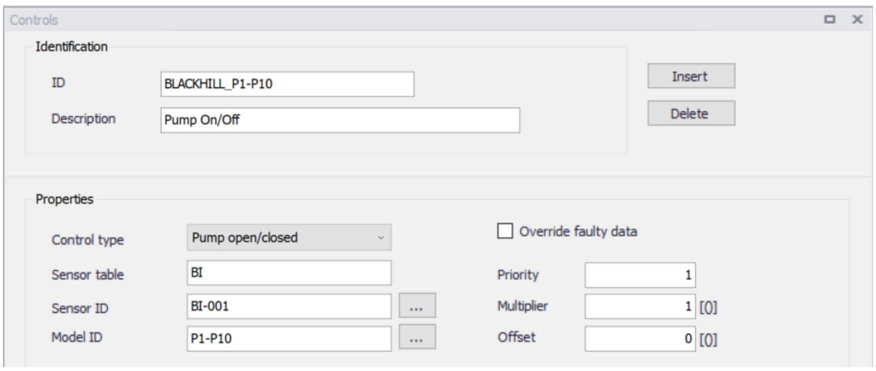


Figure 18.5 Example - Controls data input

18.4 Demand zones

A list of the Demand zones editor's data entries follows with a short description given for each entry. Demand zones allow you to define entries used by the program for updating water demands (water consumptions). The purpose of demand zones is to automatically scale water demands of nodes within a particular demand zone based on the total zone demand. That is because SCADA data are not available for each and every node demand (water connection, water meter) but there are real-time data available for the whole network or for zones (distribution zones, district meter areas, etc.).

Insert

This button is used to insert a new row into the list.

Delete

This button is used to delete the selected row from the list.

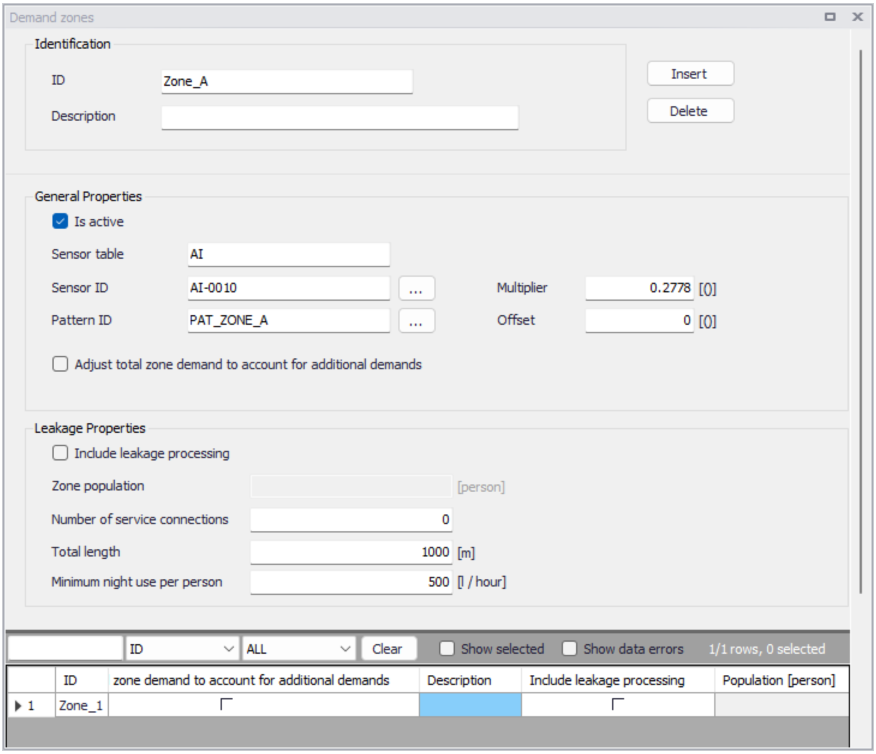


Figure 18.6 The Demand zones editor for the Online analysis

The editor entries are described below.

(Zone) ID

This is the name (identification) of the demand zone.

Description
User-defined description of the entry.

Is active
This check box allows the user to toggle the Active status of the demand on and off.

Sensor ID
This is the SCADA tag name in the table with SCADA data.

Sensor table
This is the name of the table containing SCADA data.

Pattern ID
This is the name of the pattern that will be used to scale demands within the zone. Using a pattern is optional. Note, that each zone must have its own pattern that is used for the demand scaling, i.e. if there are 5 zones, there must be 5 different patterns, one for each zone.

Multiplier
This entry is the multiplier "k" that will be used to multiply the SCADA value before using it in the model update. The model value = scada value * k + n.

Offset
This entry is the offset "n" that will be added to the SCADA value before using it in the model update. The model value = scada value * k + n.

Adjust total zone demand to account for additional demands
This option is used when the zone contains multiple demand types / patterns, and when the SCADA data only contains overall information about the entire zone. For example, if there are different demands/patterns within the zone residential and commercial, and for leakage, and if the SCADA sensor is used only for the residential and commercial pattern, then the model flow will be higher than measured because the model will have additional demands for leakage. When "Adjust total zone demand to account for additional demands" is selected, MIKE+ will make sure that the "demands/patterns" will be adjusted so that the SCADA zone demand matches this particular "demands/patterns" plus all other "demands" that are not separately measured.

Include leakage processing
This check box activates automatic processing of selected performance indicators as part of the WD Online analysis. The performance indicators include minimum night flow, current real losses (CARL), unavoidable real losses (UARL), and infrastructure leakage index (ILI).

Zone population
This is the population within the demand zone.

Number of service connections

This is the number of service connections (service pipes) within the zone.

Total length of service connections

This is the total length of service connections (service pipes) within the zone.

Minimum night use per person

This is the minimum night use of water per person within the demand zone.

18.4.1 Example

This entry will define a zone "Zone-A" with a pattern "PAT-ZoneAA" that will read the SCADA (total) flow from a SCADA tag "AI-0010", table "AI", and use it for scaling node demands within the same zone. Zone demand in m3/hour is converted into l/s.

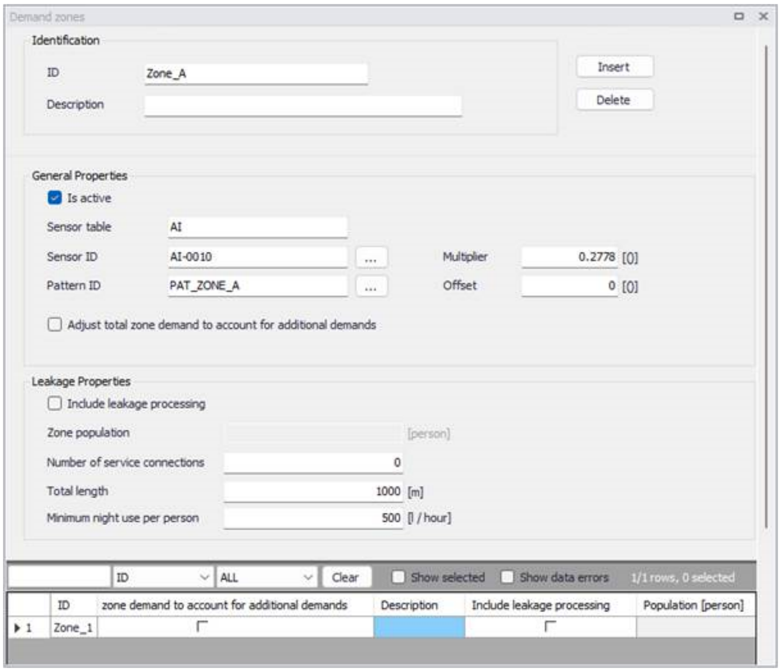


Figure 18.7 Example - Demand zones

Additionally, one can activate the automatic processing of the leakage data as part of the online analysis, by activating the 'Include leakage processing' check box.

With these settings, the WD Online engine will automatically process the following performance indicators for each zone where the 'Include leakage processing' check box is activated:

- Minimum night flow (l/s)
- CARL: current real losses (m^3/km.day)
- UARL: unavoidable real losses (m^3/km.day)
- ILI: infrastructure leakage index (-)

18.5 Demand predictions

A list of the Demand predictions editor's data entries follows with a short description given for each entry. Demand predictions allow you to use the demand prediction module that can predict future water consumptions (for an individual demand or for a zone) based on three methods:

- Statistical Winter-Holt method
- Machine learning
- 5-weeks method

Insert

This button is used to insert a new row into the list.

Delete

This button is used to delete the selected row from the list.

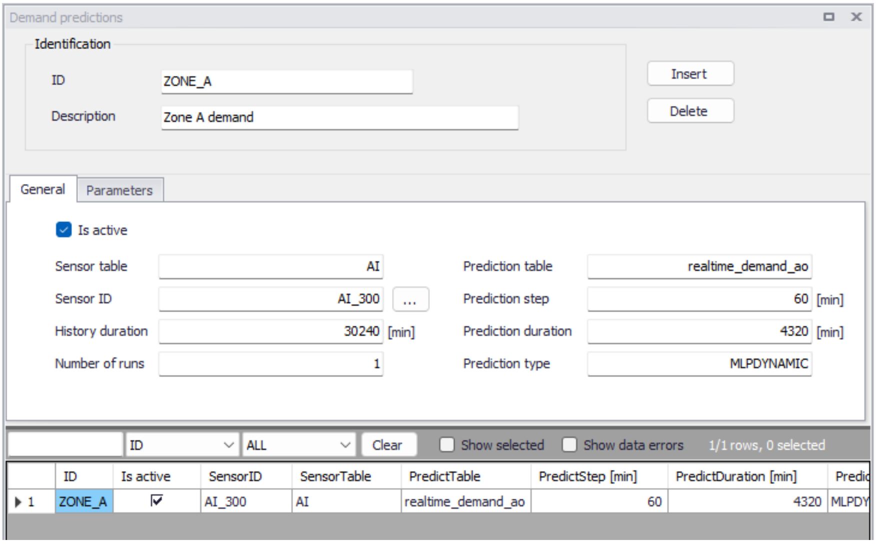


Figure 18.8 The Demand predictions editor for the Online analysis

The editor entries are described below.

Is active

This check box allows the user to toggle the Active status of the demand prediction on and off.

Sensor table

Name of the sensor table (input to demand prediction).

Sensor ID

Name of the sensor with demand data to be used for prediction.

Prediction table

Name of the output table for predicted demands.

Prediction step

Time step used for demand prediction time series.

Prediction duration

Duration of the demand prediction time series.

History duration

Duration of past data used for demand prediction.

Prediction type (method)

Demand prediction type: HOLTWINTERS or MLPDYNAMIC or 5WEEKS.

Number of runs

- HOLTWINTERS: always 1

- MLPDYNAMIC: number of repeated runs to obtain the margins for uncertainty, the demand prediction will contain average values.
- 5WEEKS: always 1

PARAM1

- HOLTWINTERS: "trend". Type of trend component: "add", "mul", "additive", "multiplicative".
- MLPDYNAMIC: "activation". Activation function for the hidden layer: 'identity', 'logistic', 'tanh', 'relu':
  - 'identity', no-op activation, useful to implement linear bottleneck, returns f(x) = x
  - 'logistic', the logistic sigmoid function, returns f(x) = 1 / (1 + exp(-x)).
  - 'tanh', the hyperbolic tan function, returns f(x) = tanh(x).
  - 'relu', the rectified linear unit function, returns f(x) = max(0, x)
- 5WEEKS: alpha value for the exponential smoothing, e.g. 0.12. The exponential smoothing is defined as follows: Q(t) = alpha Q(t) + (1-alpha)Q(t-1).

PARAM2

- HOLTWINTERS: "seasonal". Type of seasonal component: "add", "mul", "additive", "multiplicative".
- MLPDYNAMIC: "hidden_layer_sizes". The ith element represents the number of neurons in the ith hidden layer: tuple, length = n_layers - 2. Default = 100.
- 5WEEKS: not used (leave empty).

PARAM3

- HOLTWINTERS: "seasonal_periods". The number of periods in a complete seasonal cycle, e.g., 4 for quarterly data or 7 for daily data with a weekly cycle.
- MLPDYNAMIC: "solver". The solver for weight optimization: 'lbfgs', 'sgd', 'adam'. Note: The default solver 'adam' works pretty well on relatively large datasets (with thousands of training samples or more) in terms of both training time and validation score. For small datasets, however, 'lbfgs' can converge faster and perform better.
  - 'lbfgs' is an optimizer in the family of quasi-Newton methods.
  - 'sgd' refers to stochastic gradient descent.
  - 'adam' refers to a stochastic gradient-based optimizer proposed by Kingma, Diederik, and Jimmy Ba
- 5WEEKS: not used (leave empty).

PARAM4

- HOLTWINTERS: not used, leave empty

- MLPDYNAMIC: "learning_rate". Learning rate schedule for weight updates: 'constant', 'invscaling', 'adaptive'.
  - 'constant' is a constant learning rate given by 'learning_rate_init'.
  - ‘invscaling' gradually decreases the learning rate learning_rate_ at each time step 't' using an inverse scaling exponent of 'power_t'. effective_learning_rate = learning_rate_init / pow(t, power_t)
  - 'adaptive' keeps the learning rate constant to 'learning_rate_init' as long as training loss keeps decreasing. Each time two consecutive epochs fail to decrease training loss by at least tol, or fail to increase validation score by at least tol if 'early_stopping' is on, the current learning rate is divided by 5.
- 5WEEKS: not used (leave empty).

PARAM5

- HOLTWINTERS: not used, leave empty
- MLPDYNAMIC: "learning_rate_init". The initial learning rate used. It controls the step-size in updating the weights. Only used when solver='sgd' or 'adam'. default=0.001.
- 5WEEKS: not used (leave empty).

PARAM6

- HOLTWINTERS: not used, leave empty
- MLPDYNAMIC: "max_iter". Maximum number of iterations. The solver iterates until convergence (determined by 'tol') or this number of iterations. For stochastic solvers ('sgd', 'adam'), note that this determines the number of epochs (how many times each data point will be used), not the number of gradient steps. Default=200.
- 5WEEKS: not used (leave empty).

PARAM7

- HOLTWINTERS: not used, leave empty
- MLPDYNAMIC: "n_iter_no_change". Maximum number of epochs to not meet tol improvement. Only effective when solver='sgd' or 'adam'. default=10.
- 5WEEKS: not used (leave empty).

PARAM8

- HOLTWINTERS: not used, leave empty
- MLPDYNAMIC: "tol". Tolerance for the optimization. When the loss or score is not improving by at least tol for n_iter_no_change consecutive iterations, unless learning_rate is set to 'adaptive', convergence is considered to be reached and training stops. default=1e-4.
- 5WEEKS: not used (leave empty).

SensorMult
This is the multiplier "k" that will be used to multiply the SCADA value before using it in the model update. Model value = scada value * k + n.

SensorOffset
This is the offset "n" that will be added to the SCADA value before using it in the model update. Model value = scada value * k + n.

ZoneID
Name of the zone which corresponds to the Demand zones data.

18.5.1 Example

This entry will define a demand prediction for a zone "ZONE_A" demand (water consumption) that will be based on the machine learning principle with a history of 3 weeks (21 days = 30240 minutes) and the predicted demands will be for the next 3 days (4320 minutes) with a time step of 1 hour (60 minutes).

Other parameters are as follows: Param1= relu, Param2= (64,128,64), Param3 = adam, Param4 = adaptive; Param5 = 0.01; Param6 = 10000, Param7 = 1000, Param8 = 0.01.

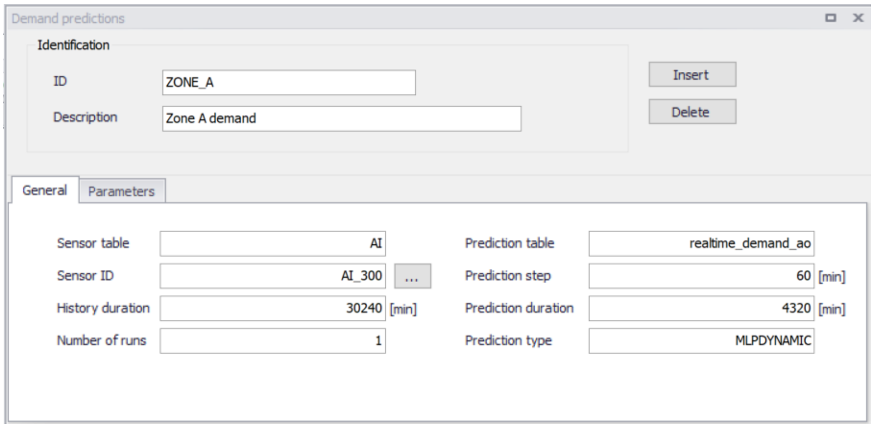


Figure 18.9 Example - Demand predictions data input

18.6 Comparisons

A list of the Comparisons editor's data entries follows with a short description given for each entry. Comparisons allow you to define pairs of matching entries: one for the measured data (SCADA) and another for the simulated data (hydraulic model). Comparisons can be the used for graphical visualization of differences in the Map or for alarming. Comparisons are also stored in the historical archive.

Insert

This button is used to insert a new row into the list.

Delete

This button is used to delete the selected row from the list.

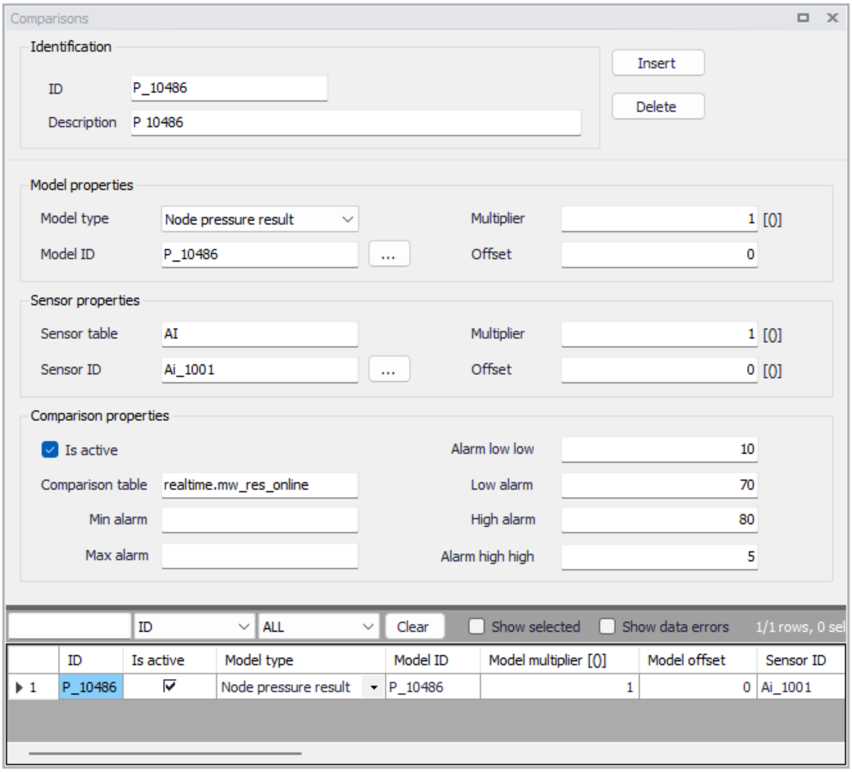


Figure 18.10 The Comparisons editor for the Online analysis

The editor entries are described below.

ID

This is the comparison ID.

Description

User-defined description of the entry.

Model type

The following outputs from the hydraulic model can be used for comparison with the telemetry (SCADA):

- Node pressure result
- Node Hydraulic Grade Line result
- Node demand result

- Node water quality result
- Calculated tank inflow/outflow result
- Link flow result
- Pipe velocity result
- Pipe headloss result
- Pipe water quality result
- Pipe status result (1 = temporarily closed, 2 = closed, 3 = open)
- Pipe setting result
- Pump status result (0 = closed (max. head exceeded), 1 = temporarily closed, 2 = closed, 3 = open, 5 = open (max. flow exceeded))
- Pump setting result
- Valve status result (1 = temporarily closed, 2 = closed, 3 = open, 4 = active (partially open), 7 = open (pressure setting not met))
- Valve setting result
- Calculated network zone demand
- Calculated tank flow for the network zone.

Model ID
This is the ID (MUID) of the hydraulic model element (node or link).

Model multiplier
This entry is the multiplier "k" that will be used to multiply the model value before using it in the comparison. The model value = mode value * k + n.

Model offset
This entry is the offset "n" that will be added to the model value before using it in the comparison. The model value = model value * k + n.

Sensor ID
Name of the sensor with demand data to be used for prediction.

Sensor table
This is the name of the table containing SCADA data.

Sensor multiplier
This entry is the multiplier "k" that will be used to multiply the SCADA value before using it in the comparison. The SCADA value = SCADA value * k + n.

Sensor offset
This entry is the offset "n" that will be added to the SCADA value before using it in the comparison. The SCADA value = SCADA value * k + n.

Is active

This check box allows the user to toggle the Active status of the comparison on and off.

Min alarm

This alarm value is used as follows: if the absolute value of a difference between simulated and observed is bigger than the "value" low alarm is triggered (alarm value is set to 1).

Max alarm

This alarm value is as follows: if the absolute value of a difference between simulated and observed is bigger than the "value" high alarm is triggered (alarm value is set to 2).

Low alarm

This alarm value is used as follows: if the simulated value is below the "value" low alarm is triggered (alarm value is set to 1)

Alarm low-low

This alarm value is used as follows: if the simulated value is below the "value" high alarm is triggered (alarm value is set to 2)

High alarm

This alarm value is used as follows: if the simulated value is above the "value" low alarm is triggered (alarm value is set to 1)

Alarm high-high

This alarm value is used as follows: if the simulated value is above the "value" high alarm is triggered (alarm value is set to 2)

18.6.1 Example

This entry will define a comparison between the simulated node pressure at the node "P_10" and the measured data with the telemetry (SCADA) tag "AI_10001" in the table "AI". The simulated pressure in "m" will be converted into "bar" for the comparison. Alarms will be generated as follows: low alarm pressure < 1 bar, low-low alarm pressure < 0.5 bar, high alarm pressure > 7 bar and high-high alarm pressure > 8 bar.

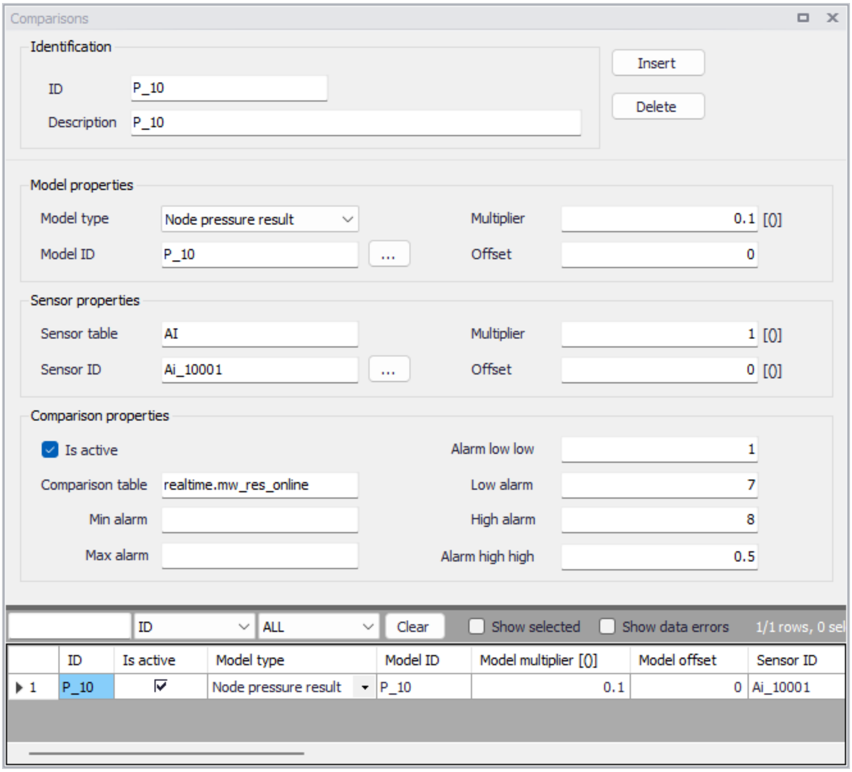


Figure 18.11 Example - Comparisons data input

18.7 Viewing locations of online analysis data on the Map

To visualise the location of selected sensors, controls, or comparison points on the Map, select them from their respective editors, and then use the special selection 'Network items associated to selected SCADA data' from the Map tab in the ribbon. This will select the model elements (nodes or links) where the selected sensors, controls, or comparison points are defined.

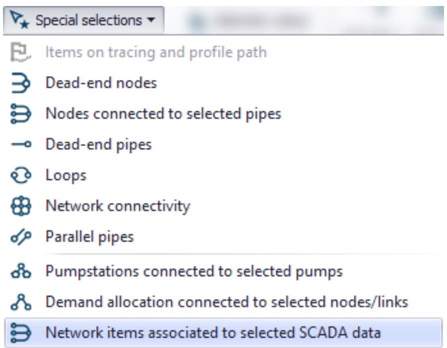


Figure 18.12 Selecting network items associated to selected SCADA data

19 Multi-Species Analysis

Standard EPANET-based water quality component is limited to tracking the transport and fate of just a single chemical species, such as fluoride used in a tracer study or free chlorine used in a disinfectant decay study. This multi-species analysis describes a water quality extension that allows it to model any system of multiple, interacting chemical species.

Many water quality problems in distribution systems can only be analyzed by using a multi-species approach. Consider the following descriptive examples:

- Free chlorine disinfectant is lost in bulk solution due mainly to oxidation-reduction reactions involving HOCl and OCl- and natural organic matter (NOM). The NOM itself is a heterogeneous mixture of organic compounds (e.g., humic and fulvic acids) of varying chemical characteristics. Current single-species models, however, must model free chlorine loss under the assumption that all other reactants are in excess and thus their concentrations can be considered constant. This limitation is responsible for the widespread observation that the water-specific decay rate constant of the common first-order model is not a constant at all, but rather varies significantly with chlorine dose (a clear indication of model structure error). The formation of regulated chlorination by-products, which results from free chlorine and NOM interactions, presents yet another set of reaction mechanisms involving multiple interacting species.
- Mono-, di-, and tri-chloramine result from interactions between free chlorine species and ammonia, and they are increasingly used as residual disinfectants. These chloramines also interact with NOM, though the reactions are slower than those for free chlorine. Thus, chloramine decay in distribution systems involves multiple interacting chemical species, which a single-species model is forced to simplify as a quasi-first order reaction. Furthermore, ammonia may be produced by auto-decomposition of chloramines, which is of significant practical importance for understanding nitrification episodes in distribution systems and storage tanks. Nitrification models may need to consider attached-growth nitrifying biofilms, suspended nitrifying biomass, and the electron donor (ammonia), electron acceptor (oxygen), and carbon source that supports microbial growth.
- For the relatively common situation where more than one water source supplies a distribution system, current models are not able to represent meaningful differences in source water quality, as they relate to water quality evolution in the distribution system. Modelers must try to compensate for this limitation by assigning bulk decay rate coefficients to specific pipes, according to which source supplies them. Such an approach has obvious deficiencies when attempting to model distribution system zones where sources blend together, and these zones are sometimes the focus of water quality issues.

None of these examples can be accurately modeled by using the single-species capabilities of the current EPANET program. This shortcoming provides the motivation to extend EPANET so that it can model reaction systems of any level of complexity.

For more information about the multi-species analysis visit:
https://www.epa.gov/water-research/epanet

The multi-species analysis in MIKE+ is first activated from the 'Model type' editor. This will add the Multi-species analysis into the model Setup tree.

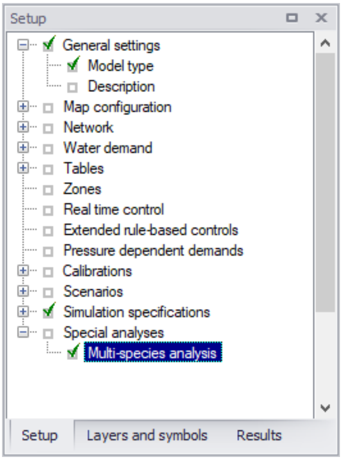


Figure 19.1 Accessing the multi-species analysis

19.1 Multi-species analysis editor

The editor allows to enter the input data by typing them into the text box. It also allows to load and export these data from/to a .msx file with the multi-species definitions.

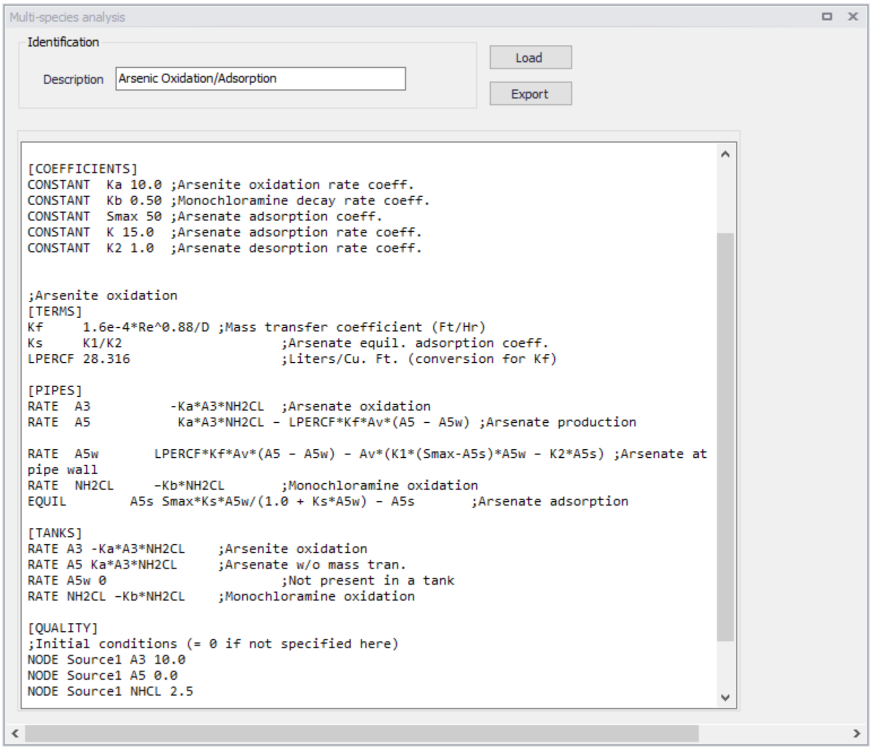


Figure 19.2 The Multi-species analysis editor

Description

An optional description of the species being described.

Text box

Input data for the muti-species simulation must be inserted in the wide text box. The format matches the format of the .msx file used by the EPANT MSX engine. This format is described in more details in the chapter 'Multi-species definition format'.

Load

Load/import the multi-species definition from a .msx file.

Export

Save the multi-species definition into a .msx file. This is optional, and not required prior to running the simulation.

19.2 Multi-species definition format

The text box in the 'Multi-species analysis' editor can contain the following sections:

Table 19.1 Sections in Multi-species analysis editor

[TITLE]	adds a descriptive title to the data set
[OPTIONS]	sets the values of computational options
[SPECIES]	names the chemical species being analyzed
[COEFFICIENTS]	names the parameters and constants used in chemical rate and equilibrium expressions
[TERMS]	defines intermediate / auxiliary terms used in chemical rate and equilibrium expressions
[PIPES]	supplies the rates and equilibrium expressions that govern species dynamics in pipes
[TANKS]	supplies the rates and equilibrium expressions that govern species dynamics in storage tanks
[SOURCES]	identifies input sources (i.e., boundary conditions) for selected species supplies
[QUALITY]	supplies initial conditions for selected species throughout the network
[PARAMETERS]	allows parameter values to be assigned on a pipe-by-pipe basis
[PATTERNS]	defines time patterns used with input sources
[REPORT]	specifies reporting options

Below is an example content. Reserved keywords are shown in bold and option choices are separated by slashes.

```
[TITLE]
<title line>

[OPTIONS]
AREA_UNITS FT2/M2/CM2
TIME_UNITS SEC/MIN/HR/DAY
SOLVER EUL/RK5/ROS2
COUPLING FULL/NONE
TIMESTEP <seconds>
ATOL <value>
RTOL <value>

[SPECIES]
BULK <specieID> <units> (<atol> <rtol>)
WALL <specieID> <units> (<atol> <rtol>)
```

```
[COEFFICIENTS]
PARAMETER <paramID> <value>
CONSTANT  <constID> <value>

[TERMS]
<termID> <expression>

[PIPES] or [TANKS]
EQUIL <specieID> <expression>
RATE <specieID> <expression>
FORMULA <specieID> <expression>

[SOURCES]
<type> <nodeID> <specieID> <strength> (<patternID>)

[QUALITY]
GLOBAL <specieID> <value>
NODE <nodeID> <bulkSpecieID> <value>
LINK <linkID> <bulkSpecieID> <value>

[PARAMETERS]
PIPE <pipeID> <paramID> <value>
TANK <tankID> <paramID> <value>

[REPORT]
NODES ALL
NODES <node1ID>  <node2ID> …
LINKS ALL
LINKS <link1ID>  <LINK2ID> …
SPECIES <species1ID>  YES/NO (<precision>)
FILE <filename>
PAGESIZE <lines>
```

Data sections are described below. For more information about the multi-species analysis visit: https://www.epa.gov/water-research/epanet.

19.2.1 [TITLE]

Purpose:

Provides a descriptive title to the problem being analyzed.

Format:

A single line of text.

Remarks:

The [TITLE] section is optional. In MIKE+, a default title made of the simulation ID and simulation description is used.

19.2.2 [OPTIONS]

Purpose:
Defines various simulation options.

Formats:

AREA_UNITS	FT2/M2/CM2
TIME_UNITS	SEC/MIN/HR/DAY
SOLVER	EUL/RK5/ROS2
COUPLING	FULL/NONE
COMPILER	NONE/VC/GC
TIMESTEP	seconds
ATOL	value
RTOL	value

Definitions:
AREA_UNITS sets the units used to express pipe wall surface area where:

FT2 = square feet
M2 = square meters
CM2 = square centimeters

The default is **FT2.**

TIME_UNITS is the units in which all reaction rate terms are expressed. The default units are hours (HR).

SOLVER is the choice of numerical integration method used to solve the reaction system where:

EUL = standard Euler integrator
RK5 = Runge-Kutta 5th order integrator
ROS2 = 2nd order Rosenbrock integrator.

The default solver is **EUL**.

COUPLING determines to what degree the solution of any algebraic equilibrium equations is coupled to the integration of the reaction rate equations. If coupling is **NONE**, then the solution to the algebraic equations is only updated at the end of each integration time step. With **FULL** coupling the updating is done whenever a new set of values for the rate-dependent variables in the reaction rate expressions is computed. This can occur at several intermediate times during the normal integration time step when using the **RK5** and **ROS2** integration methods. Thus, the **FULL** coupling option is more accurate, but can require significantly more computation time. The default is **NONE**.

COMPILER determines if the chemical reaction system being modeled should first be compiled before the simulation begins. This option is available on Windows systems that have either the Microsoft Visual C++ or the MinGW compiler installed or on Linux systems with the Gnu C++ compiler. The **VC** option is used for the Visual C++ compiler, the **GC** option is for the MinGW or Gnu compilers, while **NONE** is the default which means that no compilation is performed. Using this option can result in faster run times by a factor of 2 to 5.

TIMESTEP is the time step, in seconds, used to integrate the reaction system. The default time step is 300 seconds (5 minutes).

ATOL is the default absolute tolerance used to determine when two concentration levels of a species are the same. It applies to all species included in the model. Different values for individual species can be set in the **[SPECIES]** section of the input (see below). If no **ATOL** option is specified, then it defaults to 0.01 (regardless of species concentration units).

RTOL is a default relative accuracy level on a species' concentration used to adjust time steps in the **RK5** and **ROS2** integration methods. It applies to all species included in the model. Different values for individual species can be set in the **[SPECIES]** section of the input (see below). If no **RTOL** option is specified, then it defaults to 0.001.

19.2.3 [SPECIES]

Purpose:

Defines each chemical species being simulated.

Formats:

BULK	*name units* (Atol Rtol)
WALL	*name units* (Atol Rtol)

Definitions:

name species name

units species mass units

Atol optional absolute tolerance that overrides the global value set in the **[OPTIONS]** section

Rtol optional relative tolerance that overrides the global value set in the **[OPTIONS]** section

Remarks:

- The first format is used to define a bulk water (i.e. dissolved) species while the second is used for species attached (i.e. adsorbed) to the pipe wall.
- Bulk species are measured in concentration units of mass units per liter while wall species are measured in mass units per unit area.

- Any units can be used to represent species mass. The user is responsible for including any necessary unit conversion factors when specifying chemical reaction and equilibrium expressions that involve several species with different mass units.
- Values for both Atol and Rtol must be provided to override the default tolerances.

Examples:

```
[SPECIES]

;Bulk chlorine in mg/L with default tolerances

BULK CL2 MG

;Bulk biomass in ug/L with specific tolerances

BULK Xb UG 0.0001 0.01

;Attached biomass in ug/area with specific tolerances
WALL Xa UG 0.0001 0.01
```

19.2.4 [COEFFICIENTS]

Purpose:

Defines parameters and constants that are used in the reaction/equilibrium chemistry model.

Formats:

PARAMETER *name value*
CONSTANT *name value*

Definitions:

name coefficient's identifying name
value global value of the coefficient.

Remarks:

A **PARAMETER** is a coefficient whose value can be changed on a pipe-by-pipe (or tank-by-tank) basis (see the **[PARAMETERS]** section below) while a **CONSTANT** coefficient maintains the same value throughout the pipe network.

Examples:

```
[COEFFICIENTS]

;Kb can vary by pipe PARAMETER Kb 0.1
```

```
;Kw is fixed for all pipes CONSTANT Kw 1.5
```

19.2.5 [TERMS]

Purpose:
Defines mathematical expressions that are used as intermediate terms in the expressions for the chemical reaction/equilibrium model.

Formats:
termID expression

Definitions:
termID identifying name given to the term
expression any well-formed mathematical expression involving species, parameters, constants, hydraulic variables or other terms.

Remarks:
Terms can be used to simplify reaction rate or equilibrium expressions that would otherwise be unwieldy to write all on one line or have the same terms repeated in several different rate/equilibrium equations. The definition and use of TERMS, when those terms are common and appear in multiple rate or equilibrium expressions, may speed computation because the common term expression requires only one evaluation.

Hydraulic variables consist of the following reserved names:
D pipe diameter (feet or meters):
Q pipe flow rate (flow units)
U pipe flow velocity (ft/sec or m/sec)
Re flow Reynolds number
Us pipe shear velocity (ft/sec or m/sec)
Ff Darcy-Weisbach friction factor
Av Surface area per unit volume (area units/L)

Examples:

```
[TERMS]
;A mass transfer coefficient
Kf 1.2e-4*Re^0.88/D
;A reaction term
a1  k1*HOCL*NH3
```

19.2.6 [PIPES]

Purpose:
Supplies the rate and equilibrium expressions that govern species dynamics in pipes.

Formats:

EQUIL	*specieID expression*
RATE	*specieID expression*
FORMULA	*specieID expression*

Definitions:

specieID a species identifier

expression any well-formed mathematical expression involving species, parameters, constants, hydraulic variables or terms.

Remarks:

- There should be one expression supplied for each species defined in the model.
- The allowable hydraulic variables are defined above in the description of the **[TERMS]** section.
- The **EQUIL** format is used for equilibrium expressions where it is assumed that the expression supplied is being equated to zero. Thus, formally there is no need to supply the name of a species, but using one can help ensuring that all species are accounted for.
- The **RATE** format is used to supply the equation that expresses the rate of change of the given species with respect to time as a function of the other species in the model.
- The **FORMULA** format is used when the concentration of the named species is a simple function of the remaining species.

Examples:

```
[PIPES]
;Bulk chlorine decay
RATE CL2 -Kb*CL2

;Adsorption equilibrium between Cb in bulk and Cw on wall
EQUIL Cw Cmax*k*Cb / (1 + k*Cb) - Cw

;Conversion between biomass (X) and cell numbers (N)
FORMULA N log10(X*1.0e6)

;Bulk C formation plus non-equilibrium sorption between C
and Cs
;Using hydraulic variable Av [Area-Units/Liter]
RATE C K*C - Av*(K1*(Smax-Cs)*C - K2*Cs)
;Equivalent sorption model, using 1/hydraulic radius =
4/D ;Assumes area units are FT2 and diameter in FT
;CFPL is TERM equal to FT3/Liter, thus (4*CFPL/D) == Av
RATE C K*C - (4*CFPL/D)*(K1*(Smax-Cs)*C - K2*Cs)
```

19.2.7 [TANKS]

Purpose:
Supplies the rate and equilibrium expressions that govern species dynamics in storage tanks.

Formats:

EQUIL	*specieID expression*
RATE	*specieID expression*
FORMULA	*specieID expression*

Definitions:
specieID a species identifier
expression any well-formed mathematical expression involving species, parameters, constants, or terms.

Remarks:

- A **[TANKS]** section is always required when a model contains both bulk and wall species, even when there are no tanks in the pipe network. If the model contains only bulk species, then this section can be omitted if the reaction expressions within tanks are the same as within pipes.
- There should be one expression supplied for each bulk species defined in the model. By definition, wall species do not exist within tanks.
- Hydraulic variables are associated only with pipes and cannot appear in tank expressions.
- The **EQUIL** format is used for equilibrium expressions where it is assumed that the expression supplied is being equated to zero. Thus, formally there is no need to supply the name of a species but doing so helps ensuring that all species are accounted for.
- The **RATE** format is used to supply the equation that expresses the rate of change of the given species with respect to time as a function of the other species in the model.
- The **FORMULA** format is used when the concentration of the named species is a simple function of the remaining species.

Examples:
See the examples listed for the **[PIPES]** section.

19.2.8 [SOURCES]

Purpose:
Defines the locations where external sources of particular species enter the pipe network.

Formats:

sourceType nodeID specieID strength (patternID)

Definitions:

sourceType either **MASS**, **CONCEN**, **FLOWPACED**, or **SETPOINT**
nodeID the ID label of the network node where the source is located
specieID a bulk species identifier
strength the baseline mass inflow rate (mass/minute) for MASS sources or concentration (mass/L) for all other source types
patternID the name of an optional time pattern that is used to vary the source strength over time.

Remarks:

- Use one line for each species that has non-zero source strength.
- Only bulk species can enter the pipe network, not wall species.
- The definitions of the different source types conform to those used in the original EPANET program:
  - A **MASS** type source adds a specific mass of species per unit of time to the total flow entering the source node from all connecting pipes.
  - A **CONCEN** type source sets the concentration of the species in any external source inflow (i.e., a negative demand) entering the node. The external inflow must be established as part of the hydraulic specification of the network model.
  - A **FLOWPACED** type source adds a specific concentration to the concentration that results when all inflows to the source node from its connecting pipes are mixed together.
  - A **SETPOINT** type source fixes the concentration leaving the source node to a specific level as long as the mixture concentration of flows from all connecting pipes entering the node is less than the set point concentration.

If a time pattern is supplied for the source, it must be one defined in the **[PATTERNS]** section of the multi-species analysis definition, not a pattern from the associated EPANET simulation.

Examples:

```
[SOURCES]
;Inject 6.5 mg/minute of chemical X into Node N1
;over the period of time defined by pattern PAT1
MASS N1 X 6.5 PAT1
;Maintain a 1.0 mg/L level of chlorine at node N100
SETPOINT N100 CL2 1.0
```

19.2.9 [QUALITY]

Purpose:
Specifies the initial concentrations of species throughout the pipe network.

Formats:

GLOBAL		*specieID*	*concen*
NODE	*nodeID*	*specieID*	*concen*
LINK	*linkID*	*specieID*	*concen*

Definitions:
specieID a species identifier
nodeID a network node ID label
linkID a network link ID label
concen a species concentration

Remarks:
- Use as many lines as necessary to define a network's initial condition.
- Use the GLOBAL format to set the same initial concentration at all nodes (for bulk species) or within all pipes (for wall species).
- Use the NODE format to set an initial concentration of a bulk species at a particular node.
- Use the LINK format to set an initial concentration of a wall species within a particular pipe.
- The initial concentration of a bulk species within a pipe is assumed equal to the initial concentration at the downstream node of the pipe.
- All initial concentrations are assumed to be zero unless otherwise specified in this section.
- Models with equilibrium equations will require that reasonable initial conditions be set so that the equations are solvable. For example, if they contain a ratio of species concentrations then a divide by zero condition will occur if all initial concentrations are set to zero.

Examples:

```
[QUALITY]
;Set concentration of bulk species Cb to 1.0 at all nodes
GLOBAL Cb 1.0
;Override above condition for node N100
NODE N100 Cb 0.5
```

19.2.10 [PARAMETERS]

Purpose:

Defines values for specific reaction rate parameters on a pipe-by-pipe or tank-by-tank basis.

Formats:

PIPE	*pipeID paramID value*
TANK	*tankID paramID value*

Definitions:

pipeID the ID label of a pipe link in the network
tankID the ID label of a tank node in the network
paramID the name of one of the reaction rate parameters listed in the **[COEFFICIENTS]** section
value the parameter's value used for the specified pipe or tank.

Remarks:

Use one line for each pipe or tank whose parameter value is different than the global value.

19.2.11 [PATTERNS]

Purpose:

Defines time patterns used to vary external source strength over time.

Formats:

name multiplier multiplier ...

Definitions:

name an identifier assigned to the time pattern
multiplier a multiplier used to adjust a baseline value

Remarks:

- Use one or more lines for each time pattern included in the model.
- If extending the list of multipliers to another line remember to begin the line with the pattern name.
- All patterns share the same time period interval as defined in the [TIMES] section of the EPANET input file being used in conjunction with the EPANET-MSX input file.
- Each pattern can have a different number of time periods.
- When the simulation time exceeds the pattern length the pattern wraps around to its first period.

Examples:

```
[PATTERNS]
;A 3-hour injection pattern over a 24-hour period
;(assuming a 1-hour pattern time interval is in use)

P1 0.0 0.0 0.0 0.0 1.0 1.0
P1 1.0 0.0 0.0 0.0 0.0 0.0
P1 0.0 0.0 0.0 0.0 0.0 0.0
P1 0.0 0.0 0.0 0.0 0.0 0.0
```

19.2.12 [REPORT]

Purpose:

Describes the content of the output report produced from a simulation.

Formats:

NODES	**ALL**
NODES	*node1, node2, etc.*
LINKS	**ALL**
LINKS	*link1, link2, etc.*
SPECIES	*speciesID* **YES/NO** *(precision)*
FILE	*filename*
PAGESIZE	*lines*

Definitions:

node1,node2, etc. a list of nodes whose results are to be reported
link1,link2, etc. a list of links whose results are to be reported
specieID the name of a species to be reported on
precision number of decimal places used to report a species' concentration
filename the name of a file to which the report will be written
lines the number of lines per page to use in the report.

Remarks:

- Use as many NODES and LINKS lines as it takes to specify which locations get reported. The default is not to report results for any nodes or links.
- Use the SPECIES line to specify which species get reported and at what precision. The default is to report all species at two decimal places of precision.
- The FILE line is used to have the report written to a specific file. If not provided the report will be written to the same file used for reporting program errors and simulation status.

Examples:

```
[REPORT]
;Write results for all species at all nodes and links
;at all time periods to a specific file
NODES ALL
LINKS ALL
```

19.3 Running analysis

In order to run the multi-species simulation, open the Simulation setup editor and select the 'Multi-species water quality' module. When this module is selected, two simulation modes are available: you can either run both hydraulics and water quality simulations at the same time, or you can run only the water quality simulation using an "hydraulics" file resulting from an earlier hydraulics simulation. The latter helps reducing simulation times when the hydraulic simulation takes a long time and does not need to be repeated while running the water quality scenarios. The input hydraulics file is saved from the hydraulics simulation by selecting 'Save hydraulics file' in the 'Output' tab.

The simulation reports error and warning messages into the .sum file. Some of the simulation errors that result in the program termination are described below:

- Error 513: could not integrate reaction rate expressions:

  The differential equation solver employed by EPANET MSX could not successfully integrate the system's reaction rate equations over the current water quality time step. One could try re-running the analysis using a smaller time step or larger values for ATOL and RTOL (as specified in the [OPTIONS] or [SPECIES] sections of the multi-species analysis definition).

- Error 514: could not solve reaction equilibrium expressions:

  The non-linear equation solver employed by EPANET MSX could not successfully solve the system's set of equilibrium equations at the current simulation time. Users must ensure that the initial conditions set throughout the pipe network are sufficient and consistent so that a solution exists for the governing set of equilibrium equations.

In case that the multi-species simulation is unable to find the solution, adjustments to the model setup might be necessary such as:

- Use smaller time step or large values for tolerance parameters
- Change the solver type (numerical integration method)
- Change the coupling type

- Reduce the size and complexity of the model network by removing short dead-end pipes and merging pipes together in order to reduce number of pipes and to remove short pipe segments.

19.4 Simulation results

The simulation results are stored in the .msxr results file, and they can be processed in the same way as any other results.

19.5 Examples of multi-species analysis definition

Several examples of multi-species analysis definitions are provided in this section. They can be applied to any hydraulic model with only few or minor modifications, such as change of the tank ID's, for example. For more information about the multi-species analysis and sample MSX files, visit: https://www.epa.gov/water-research/epanet.

19.5.1 Two-source chlorine decay

Multi-source networks present problems when modelling a single species, such as free chlorine, when the decay rates observed in the source waters vary quite significantly. As the sources blend differently throughout the network it becomes difficult to assign a single decay coefficient that accurately reflects the decay rate observed in the blended water. One approach to reconciling the vastly different chlorine decay constants in this example, without introducing a more complex chlorine decay mechanism that attempts to represent the different reactivity of the total organics from the two sources, is to assume that at any time the chlorine decay constant within a pipe is given by a weighted average of the two source values, where the weights are the fraction of each source water present in the pipe. These fractions can be deduced by introducing a fictitious conservative tracer compound at Source 1, denoted as T1, whose concentration is fixed at a constant 1.0 mg/L. Then at any point in the network the fraction of water from Source 1 would be the concentration of T1 while the fraction from Source 2 would be 1.0 minus that value. The resulting chlorine decay model now consists of two-species, a tracer species T1 and a free chlorine species C.

```
[OPTIONS]
AREA_UNITS   FT2
RATE_UNITS   HR
SOLVER       RK5
TIMESTEP     300

[SPECIES]
BULK T1 MG ;Source 1 tracer
BULK CL2 MG ;Free chlorine
```

```
[COEFFICIENTS]
CONSTANT k1  1.3 ;Source 1 decay coeff.
CONSTANT  k2  17.7 ;Source 2 decay coeff.

[PIPES]
;T1 is conservative
RATE T1 0
;CL2 has first order decay
RATE CL2 -(k1*T1 + k2*(1-T1))*CL2

[QUALITY]
;Initial conditions (= 0 if not specified here)
[QUALITY]
;Initial conditions (= 0 if not specified here)
NODE Source1 T1 1.0
NODE Source1 CL2 1.2
NODE Source2 CL2 1.2

[REPORT]
NODES ALL
LINKS ALL
```

19.5.2 Mass transfer-limited arsenic oxidation/adsorption system

This example is an extension and more complete description of the arsenic oxidation/adsorption model. It models the oxidation of arsenite As+3 to arsenate As+5 by a monochloramine disinfectant residual NH2Cl in the bulk flow along with the subsequent adsorption of arsenate onto exposed iron on the pipe wall. We also include a mass transfer limitation to the rate at which arsenate can migrate to the pipe wall where it is adsorbed. It assumes that the network has a single source which is a reservoir node Source1,

```
[SPECIES]
BULK A3 UG ;Dissolved arsenite
BULK A5 UG ;Dissolved arsenate
BULK A5w UG ;Dissolved arsenate at wall
WALL A5s UG ;Adsorbed arsenate
BULK NH2CL MG ;Monochloramine

[COEFFICIENTS]
CONSTANT  Ka 10.0 ;Arsenite oxidation rate coeff.
CONSTANT  Kb 0.50 ;Monochloramine decay rate coeff.
CONSTANT  Smax 50 ;Arsenate adsorption coeff.
CONSTANT  K 15.0  ;Arsenate adsorption rate coeff.
CONSTANT  K2 1.0  ;Arsenate desorption rate coeff.

;Arsenite oxidation
[TERMS]
Kf     1.6e-4*Re^0.88/D;Mass transfer coefficient (Ft/Hr)
```

```
Ks      K1/K2;Arsenate equil. adsorption coeff.
LPERCF 28.316;Liters/Cu. Ft. (conversion for Kf)

[PIPES]
RATE   A3   -Ka*A3*NH2CL;Arsenate oxidation
RATE   A5    Ka*A3*NH2CL - LPERCF*Kf*Av*(A5 - A5w);Arseate
production

RATE   A5wLPERCF*Kf*Av*(A5 - A5w) - Av*(K1*(Smax-A5s)*A5w
- K2*A5s) ;Arsenate at pipe wall
RATE   NH2CL-Kb*NH2CL ;Monochloramine oxidation
EQUIL       A5s Smax*Ks*A5w/(1.0 + Ks*A5w) - A5s ;Arse-
nate adsorption

[TANKS]
RATE A3 -Ka*A3*NH2CL;Arsenite oxidation
RATE A5 Ka*A3*NH2CL;Arsenate w/o mass tran.
RATE A5w 0;Not present in a tank
RATE NH2CL -Kb*NH2CL ;Monochloramine oxidation

[QUALITY]
;Initial conditions (= 0 if not specified here)
NODE Source1 A3 10.0
NODE Source1 A5 0.0
NODE Source1 NHCL 2.5

[REPORT]
NODES ALL
LINKS ALL
```

19.5.3 Bacterial regrowth model with chlorine inhibition

This example models bacterial regrowth as affected by chlorine inhibition within a distribution system. There are six species defined for the model: bulk chlorine (CL2), bulk biodegradable dissolved organic carbon (S), bulk bacterial concentration (Xb), bulk bacterial cell count (Nb), attached bacterial concentration (Xa), and attached bacterial cell count (Na). CL2 and S are measured in milligrams. The bacterial concentrations are expressed in micrograms of equivalent carbon so that their numerical values scale more evenly. The bacterial cell counts are expressed as the logarithm of the number of cells. The model assumes that there is a single source node named Source1 that supplies all water to the system.

```
[OPTIONS]
AREA_UNITS   CM2
RATE_UNITS   HR
SOLVER       RK5
TIMESTEP     300
```

```
[SPECIES]
BULK CL2 MG ;chlorine
BULK S MG ;organic substrate
BULK Xb UG ; mass of free bacteria
WALL Xa UG ; mass of attached bacteria
BULK Nb log(N) ;number of free bacteria
WALL Na log(N) ;number of attached bacteria

[COEFFICIENTS]
CONSTANT  Kb 0.05 ;CL2 decay constant (1/hr)
CONSTANT  CL2C 0.20 ;characteristic CL2 (mg/L)
CONSTANT  CL2Tb 0.03 ;threshold CL2 for Xb (mg/L)
CONSTANT  CL2Ta 0.10  ;threshold CL2 for Xa (mg/L)
CONSTANT  MUMAXb 0.20 ;max. growth rate for Xb (1/hr)
CONSTANT  MUMAXa 0.20  ;max. growth rate for Xa (1/hr)
CONSTANT  Ks 0.40 ;half saturation constant (mg/L)
CONSTANT  Kdet 0.03 ;detachment rate constant
(1/hr/(ft/s))
CONSTANT  Kdep 0.08 ;deposition rate constant (1/hr)
CONSTANT  Kd 0.06 ;bacterial decay constant (1/hr)
CONSTANT  Yg 0.15 ;bacterial yield coefficient (mg/mg)

[TERMS]
Ib EXP(-STEP(CL2-CL2Tb)*(CL2-CL2Tb)/CL2C) ;Xb inhibition
coeff.
Ia EXP(-STEP(CL2-CL2Ta)*(CL2-CL2Ta)/CL2C) ;Xa inhibition-
coeff.
MUb MUMAXb*S/(S+Ks)*Ib ;Xb growth rate coeff.
MUa MUMAXa*S/(S+Ks)*Ia ;Xa growth rate coeff.

[PIPES]
RATE CL2 -Kb*CL2
RATE S -(MUa*Xa*Av + MUb*Xb)/Yg/1000
RATE Xb (MUb-Kd)*Xb + Kdet*Xa*U*Av - Kdep*Xb
RATE Xa (MUa-Kd)*Xa - Kdet*Xa*U + Kdep*Xb/Av
FORMULA Nb LOG10(1.0e6*Xb)
FORMULA Na LOG10(1.0e6*Xa)

[TANKS]
RATE CL2 -Kb*CL2
RATE S -MUb*Xb/Yg/1000
RATE Xb (MUb-Kd)*Xb
FORMULA Nb LOG10(1.0e6*Xb)

[SOURCES]
CONCEN Source1 CL2 1.2
CONCEN Source1 S 0.4
CONCEN Source1 Xb 0.01
```

```
[REPORT]
NODES ALL
LINKS ALL
```

20 Autocalibration

Hydraulic models need to be calibrated and verified based on the measured data, typically flows and pressures. Model calibration and verification is necessary in order to understand limitations of the model and ensure proper use of the model. The model can be calibrated in different ways including semi-automatic methods.

The purpose of this autocalibration-based tool is to support finding model settings, such as pipe friction, demands, isolating valves, that improve the match between the hydraulic model and observed data. The autocalibration is based on optimization algorithms that can run with any extended period analysis model.

These two algorithms are available:

- SCE: The Shuffled Complex Evolution (SCE) method is a global optimization algorithm that combines various search strategies, including (i) competitive evolution, (ii) controlled random search, (iii) the simplex method, and (iv) complex shuffling.
- DDS: The Dynamically Dimensioned Search algorithm (DDS) is an optimization algorithm designed for large set of parameters. It automatically scales the search to find good solution within the maximum number of model calls. The algorithm starts to search globally, and then becomes more local as the number of models call approaches the maximum allowable number. Candidate solutions are created by perturbing the current values in the randomly selected dimensions only. The dimension of search is automatically adjusted.

The autocalibration provides the following controls:

- Pipe friction autocalibration
- Water demand autocalibration (demand adjustment)
- Finding closed links
- Leakage adjustment.

These controls can be combined into the same autocalibration run even though it is unlikely doing different adjustments at the same time.

Each calibration process consists of several steps:

- Definition of controls, e.g., pipe roughness groups and pipe group assignment
- Definition of targeted pressure and/or flow values
- Automated calibration of control parameters e.g., pipe roughness coefficients by autocalibration methods

- Assignment of computed parameters to the model, e.g., calibrated pipe roughness values to the pipes.

20.1 Identification

A list of the Autocalibration parameters and buttons follows with a short description given for each of them. Note, that it is possible to insert multiple Autocalibration runs, each with its own settings, and then run the selected Autocalibration run by selecting it from the list. This is convenient when you e.g. need to calibrate different zones individually.

ID
The ID of the autocalibration run.

Simulation ID
This is the ID of the associated simulation (from the 'Simulation setup' editor), which is used as the base of the autocalibration.

Insert
This button is used to insert a new autocalibration into the list.

Delete
This button is used to delete an autocalibration from the list.

Run
This button is used to run the autocalibration currently active in the list.

Stop
This button is used to stop (cancel) the autocalibration that is currently running.

Report
This button is used to report the autocalibration results.

20.2 Methods

20.2.1 DDS Optimization parameters

This section describes settings for the DDS optimization method.

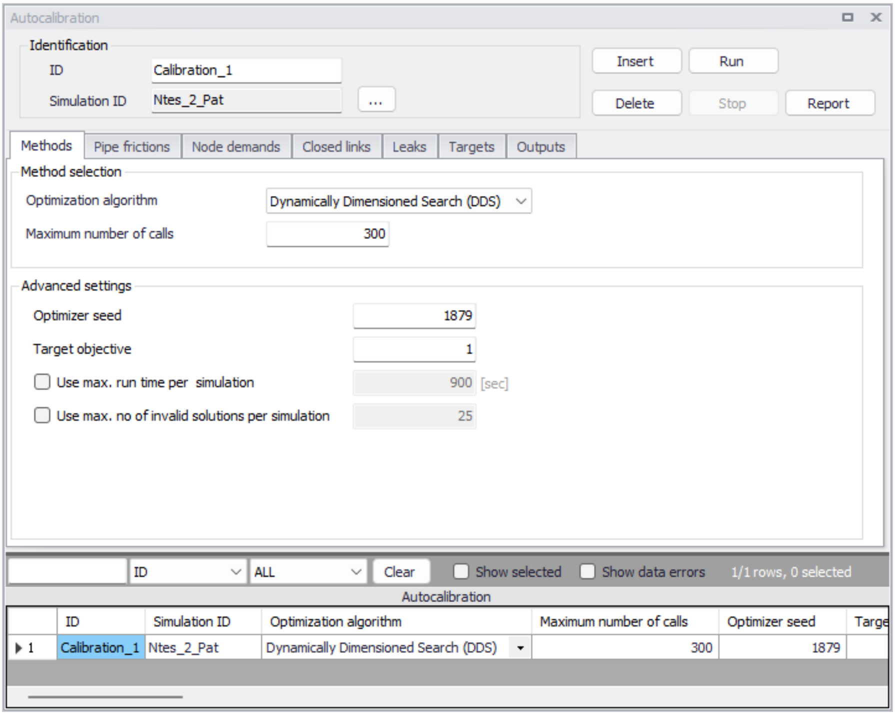


Figure 20.1 DDS Optimization settings

Maximum number of calls

This is one of the stop criteria and it is the maximum number of model call during the optimization process. If the maximum number of model calls is reached, the optimization process will stop, and it will report the best solution that was found.

Optimizer seed

This is used to initiate random number generator. There is no need to change this value except in special cases.

Target objective

This defines another stop criteria which compares the actual value of the objective function and it stops when its value is below the specified target objective.

Use maximum run time per simulation

This can be used in special cases when the hydraulic simulation due to the "random" choice of control variables by the optimizer might take exceptionally long time to finish. In such case, if this criterion is used, the program will cancel the hydraulic simulation and it will use a high penalty for its settings for the optimizer.

Use maximum number of invalid solutions per simulation

This can be used in special cases when the hydraulic simulation due to the "random" choice of control variables by the optimizer results in unbalanced or hydraulically unstable conditions during the extended period simulation run. In such case, if this criterion is used, the program will cancel the hydraulic simulation and it will use a high penalty for its settings for the optimizer.

20.2.2 SCE Optimization parameters

This section describes settings for the SCE optimization method.

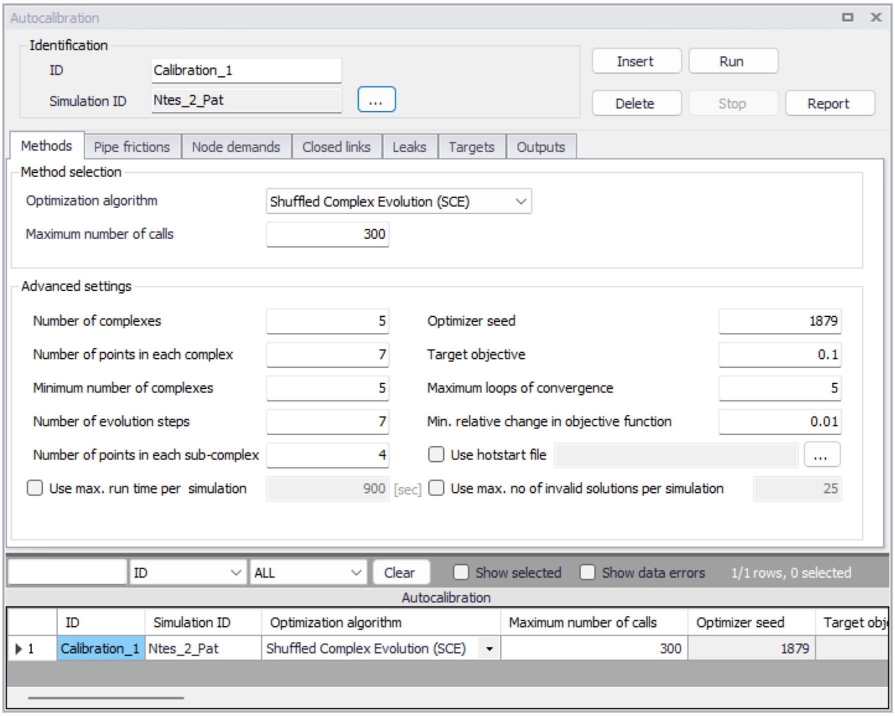


Figure 20.2 SCE Optimization settings

Maximum number of calls

This is one of the stop criteria and it is the maximum number of model call during the optimization process. If the maximum number of model calls is reached, the optimization process will stop, and it will report the best solution that was found.

Number of complexes

This defines the number of complexes (p) used in the optimization process. It is used to calculate the initial population (number of different model setups to run). The population size is s = p x m.

Number of points in each complex

This data entry defines the number of points in each complex (m) used in the optimization process. It is used to calculate the initial population (number of different model setups to run). The population size is s = p x m.

Minimum number of complexes

The minimum number of complexes is required when the number of complexes is reduced during the optimization.

Number of evolution steps

This defines the number of evolution steps taken by each complex before complexes are shuffled.

Number of points in each sub-complex

This defines the number of points in each sub-complex.

Optimizer seed

This is used to initiate random number generator. There is no need to change this value except in special cases.

Target objective

This defines another stop criteria which compares the actual value of the objective function and it stops when its value is below the specified target objective.

Maximum loops of convergence

This defines the maximum number of loops used in the optimization.

Minimum relative change in objective function

This is another stop criteria where solutions are compared in terms of a change in the objective function and the optimization stops when there is no improvement.

Use maximum run time per simulation

This can be used in special cases when the hydraulic simulation due to the "random" choice of control variables by the optimizer might take exceptionally long time to finish. In such case, if this criterion is used, the program will cancel the hydraulic simulation and it will use a high penalty for its settings for the optimizer.

Use hotstart file

This can be used to start the new autocalibration run from the results of a previous run. Select the file with the extension .dat that contains the previous results.

Use maximum number of invalid solutions per simulation

This can be used in special cases when the hydraulic simulation due to the "random" choice of control variables by the optimizer results in unbalanced or hydraulically unstable conditions during the extended period simulation run.

In such case, if this criterion is used, the program will cancel the hydraulic simulation and it will use a high penalty for its settings for the optimizer.

20.3 Pipe frictions

In this tab, it is possible to define settings for calibration of friction in pipes (parameter 'Roughness' in the pipes editor).

One of the common calibration parameters in the pipe network hydrodynamic model is the roughness coefficients. Pipe roughness values may be estimated in two ways: using values from literature or directly from field measurements. To obtain initial estimates of pipe roughness through field testing, it is a good practice to divide the water distribution system into composite zones that contain pipes of similar material and age. Additionally, several pipes of different diameters should be tested in each zone to obtain individual pipe roughness estimates. The process of calibration ideally requires simulation over an extended period, such as a time range for the maximum day - not just the maximum and minimum hour for the maximum day.

The pipe roughness can be autocalibrated no matter which head loss formulation is applied in the simulation: Hazen-Williams, Darcy-Weisbach or Manning.

Each record in the table corresponds to a geographical location to calibrate (defined by a list of pipes). Each list / location defined in this table will be calibrated with a uniform friction, therefore different locations should be defined in the table whenever different frictions are expected, depending on the known physical characteristics of the pipes such as material, age, and diameter.

After running the autocalibration, it is possible to review the calibrated value and adjust it, before applying the desired value to the network.

Pipe friction will be calibrated only if some pipes groups are defined in the table.

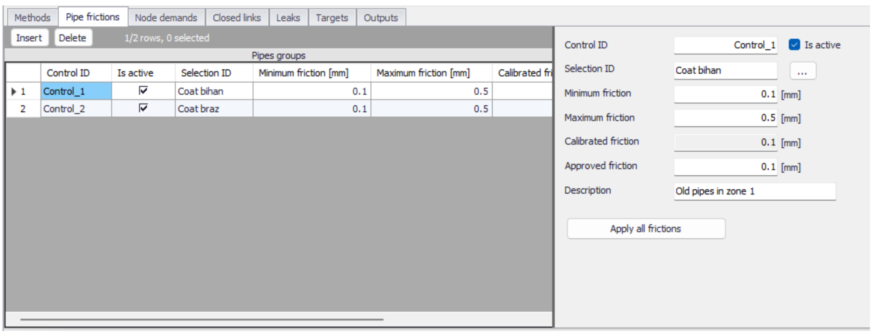


Figure 20.3 The pipe friction calibration settings

Calibration of pipe friction involves the settings described below.

Control ID
This is identification name for the active group of pipes to be calibrated with its uniform friction value.

Is active
The active pipes group will be calibrated only when its 'Is active' option is ticked.

Selection ID
This is the ID of the selection list, containing the group of pipes to be calibrated. This list can be reviewed and edited from the 'Selection manager'. The '…' button opens a list showing the valid list of selection IDs.

Minimum friction
This is the minimum value of the friction allowed during the autocalibration.

Maximum friction
This is the maximum value of the friction allowed during the autocalibration.

Calibrated friction
This is the calibrated friction value returned at the end of the autocalibration process. This value is computed during the simulation and cannot be edited manually.

Approved friction
This is the friction value approved, which will be applied to the pipes network.

At the end of the autocalibration process, this value matches by default the calibrated value. The modeler can then review the calibrated values and adjust the approved values, if necessary. The approved friction values will be actually applied to the pipes network only when pressing the 'Apply all frictions' button.

Description
This is an optional description of the pipes group.

Apply all frictions
This button updates the friction value for the various groups of pipes defined in the table, with their approved values. A group of pipes is updated only when its 'Is active' option is ticked.

20.4 Node demands

In this tab, it is possible to define settings for calibration of node demands' patterns.

Each record in the table corresponds to a specific pattern to calibrate. During the autocalibration, the pattern will be adjusted by a multiplier that will increase or decrease the water consumption for nodes that refer to the updated pattern.

For water distribution systems with multiple zones, such as pressures zones, distribution or DMA (district metered area) zones, it is recommended to develop individual patterns that are unique for the respective zone.

After running the autocalibration, it is possible to review the calibrated factor and modify it, before applying the desired value to the pattern.

Node demands will be calibrated only if some patterns to calibrate are added to the table.

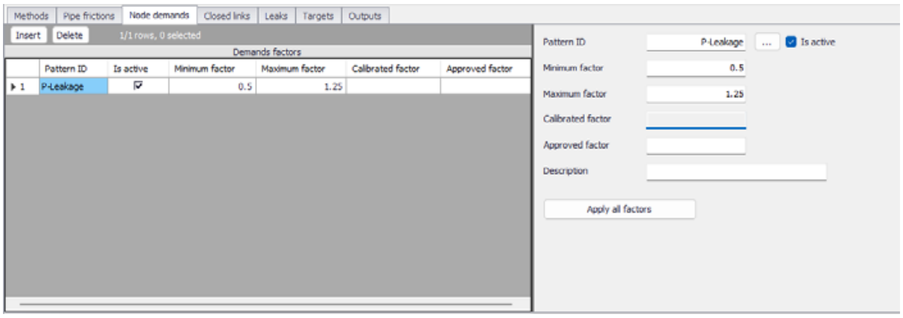


Figure 20.4 The node demands calibration settings

Calibration of node demands involves the settings described below.

Pattern ID

This is the ID of the selected pattern to be calibrated.

Is active

The pattern will be calibrated only when its 'Is active' option is ticked.

Minimum factor

This is the minimum factor for correction of the pattern.

Maximum factor

This is the maximum factor for correction of the pattern.

Calibrated factor

This is the calibrated factor returned at the end of the autocalibration process. This value is computed during the simulation and cannot be edited manually.

Approved factor

This is the approved factor, which will be applied to the pipes network.

At the end of the autocalibration process, this value matches by default the calibrated factor. The modeler can then review the calibrated factors and adjust the approved values, if necessary. The approved factors will be actually applied to the patterns only when pressing the 'Apply all factors' button.

Description

This is an optional description of the node demand pattern calibration.

Apply all factors

This button updates the patterns selected in the table, with their approved factor. A pattern is updated only when its 'Is active' option is ticked.

20.5 Closed links

In this tab, it is possible to optimize the Open / Closed status of selected links (pipes or TCV valves).

Unknown closed section or isolation valves cause often serious problems when calibrating the hydraulic model. This autocalibration option allows to pick multiple pipes or valves and test them for being potentially closed.

Each record in the table corresponds to a specific link. The autocalibration will return a proposed status, which can either be approved or rejected, before applying the desired value to the pattern.

Links statuses will be calibrated only if some links are added to the table.

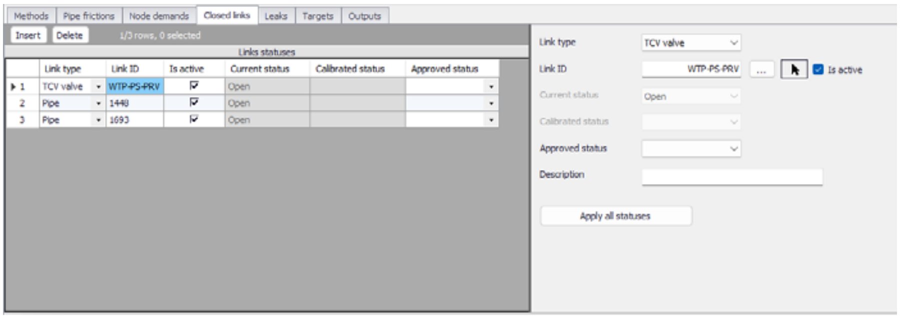


Figure 20.5 The closed links calibration settings

Calibration of links statuses involves the settings described below.

Link type

This controls the type of link to be selected and calibrated. It can either be a pipe or a TCV valve.

Link ID
This is the ID of the selected pipe or valve to be calibrated, depending on the selected link type. The '…' button opens a list showing the valid list of links. The arrow button can be used to select the link by clicking on the map.

Is active
The link will be calibrated only when its 'Is active' option is ticked.

Current status
This is the current status of the link, before starting the autocalibration. This value is shown for information only, and cannot be edited manually.

Calibrated status
This is the calibrated status returned at the end of the autocalibration process. This value is computed during the simulation and cannot be edited manually.

Approved status
This is the approved status, which will be applied to the link.

At the end of the autocalibration process, this value matches by default the calibrated status. The modeler can then review the calibrated statuses and adjust them, if necessary. The approved statuses will be actually applied to the links only when pressing the 'Apply all statuses' button.

Description
This is an optional description of the link status calibration.

Apply all statuses
This button updates the links selected in the table, with their approved status. A link is updated only when its 'Is active' option is ticked.

20.6 Leaks

In this tab, it is possible to optimize the node demands using the pressure dependent emitter coefficient of selected junctions ('Flow coefficient' parameter from the 'Junctions' editor).

The purpose of this control is not to simulate small background leakages that occur practically everywhere within the water distribution system but to identify major leaks that might be responsible for pressure loss that is measured but cannot be represented in the model using water usage data.

Each record in the table corresponds to a specific junction. After running the autocalibration, it is possible to review the calibrated emitter coefficient and adjust it, before applying the desired value to the network.

Emitter coefficients will be calibrated only if some junctions are added to the table.

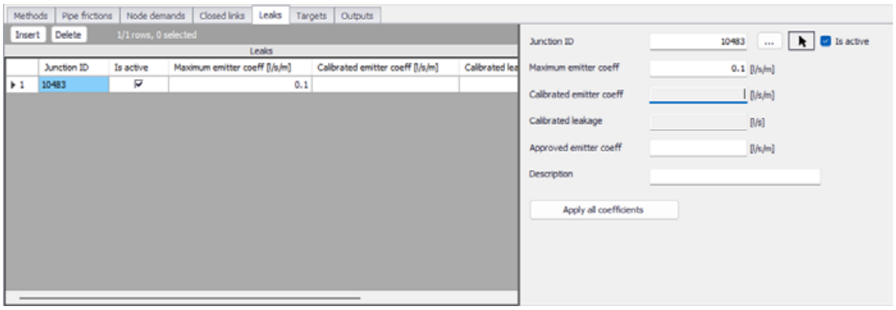


Figure 20.6 The leaks calibration settings

Calibration of emitter coefficients involves the settings described below.

Junction ID

This is the ID of the selected junction to be calibrated. The '…' button opens a list showing the valid list of junctions. The arrow button can be used to select the junction by clicking on the map.

Is active

The junction will be calibrated only when its 'Is active' option is ticked.

Maximum emitter coeff

This is the maximum value of the emitter coefficient allowed during the autocalibration.

Calibrated emitter coeff

This is the calibrated coefficient returned at the end of the autocalibration process. This value is computed during the simulation and cannot be edited manually.

Calibrated leakage

This is the calibrated total leakage at the junction, returned at the end of the autocalibration process. This value is computed during the simulation and cannot be edited manually.

Approved emitter coeff

This is the approved emitter coefficient, which will be applied to the junction.

At the end of the autocalibration process, this value matches by default the calibrated emitter coefficient. The modeler can then review the calibrated coefficients and adjust them, if necessary. The approved coefficients will be actually applied to the junctions only when pressing the 'Apply all coefficients' button.

Description

This is an optional description of the junction coefficient calibration.

Apply all coefficients

This button updates the junctions selected in the table, with their approved emitter coefficient / flow coefficient. All updated junctions are also changed to node type 'Emitter'. A junction is updated only when its 'Is active' option is ticked.

20.7 Targets

This tab contains the calibration objectives, which the simulation will aim to meet during the autocalibration process.

Each record in the table corresponds to a target, which is usually a location with measured data to match.

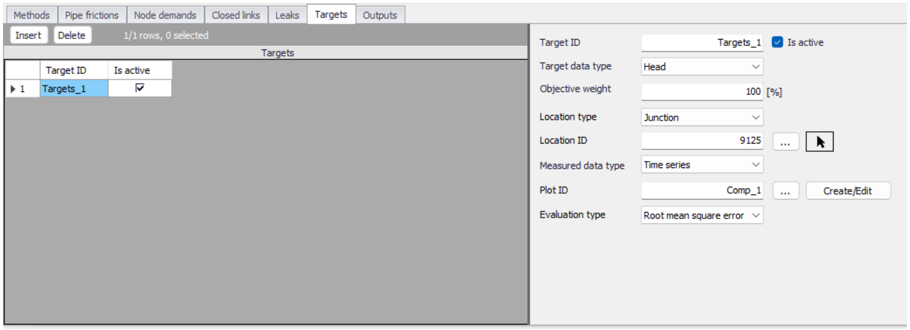


Figure 20.7 The calibration targets settings

Definition of targets involves the settings described below.

Target ID

This is the ID of the target.

Is active

The target will be used in the calibration only if its 'Is active' option is ticked.

Target data type

This list defines the type of the target. The following options are available:

- Tank water depth
- Pressure
- Flow
- Head.

Objective weight

This is used to specify the weight of this target. Use different number if you want to prioritize one target over another.

Location type
This is the type of network element where the target applies. Depending on the target data type, it can either be a junction, tank, pipe, pump or valve.

Location ID
This is the ID of the junction, tank, pipe, pump or valve where the target applies. The '…' button opens a list showing the valid list of elements. The arrow button can be used to select the element by clicking on the map.

Measured data type
The target measured data can either be defined as a time series or a constant value.

Plot ID
When the target measured data is a time series, this time series must be specified in a plot in the 'Plots and statistics' editor, and the plot ID must be selected here. The '…' button opens the list of existing plots to select from. The 'Create/Edit' button will open the 'Plots and statistics' editor to either edit the measured data time series, or to review the time series comparison after the autocalibration.

The selected time series will be compared to the results computed over the extended period simulation.

Evaluation type
When the target measured data is a time series, this controls how the goodness-of-fit between the measured and computed time series is evaluated:

- Root mean square error
- Standard deviation
- Pearson correlation
- Average.

Constant value
This is the constant target value, when the target measured data type is a constant value. It will be compared to the steady state results during the autocalibration.

20.8 Outputs

This tab provides a list of computed controls and their optimized settings and a summary of targets with requested and computed values.

20.9 Report

The 'Report' button provides a XML version of the outputs, i.e. a list of computed controls and their optimized settings and a summary of targets with requested and computed values.

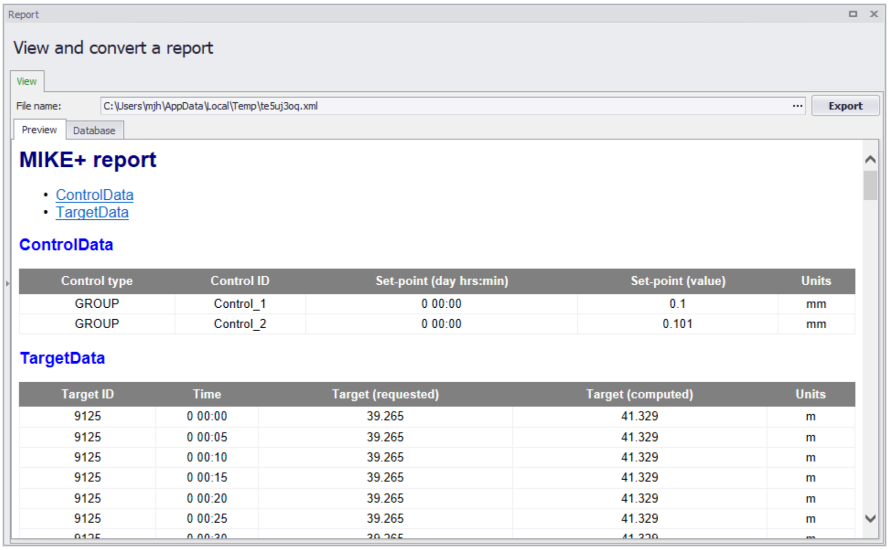


Figure 20.8 The autocalibration report

20.10 Examples

Several examples are provided here to illustrate how the autocalibration can be used.

20.10.1 Example 1 - Pipe friction steady state simulation

The treated water system contains a pump station, a long water supply pipeline consisting of multiple pipelines and a storage tank that is receiving the pumped water. The steady state model with assumed friction factor, entered as 0.1 mm (rugosity for Darcy-Weisbach friction formulation), gives the discharge of about 2500 l/s. Measured data show 2350 l/s.

To use the autocalibration for pipe friction, open the 'Autocalibration' editor and insert a new record, use the SCE method, for example, and accept all default settings but change the maximum number of calls to 5000.

Next, create a selection list called e.g. "CWP" (Clean Water Pipeline) and assign all pipes in the water supply pipeline from the treated water pump station pump station to the downstream reservoir into this selection.

Next, enter a new control into the 'Pipe frictions' tab:

- Selection ID: "CWP"
- Minimum friction: 0.1 mm (*)
- Maximum friction: 1.0 mm (*)

(*) We know that with 0.1 mm the discharge in the pipeline is 2500 l/s and so it is apparent that the friction factor (rugosity) must be bigger than 0.1mm in order to give smaller discharge.

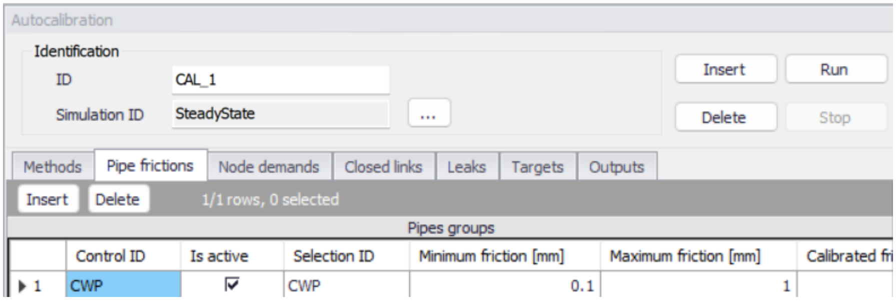


Figure 20.9 Example 1 - Pipe friction

Apply the following settings in the 'Targets' tab:

- Target data type: select 'Flow'
- Objective weight: 100%
- Link type: select 'Pipe'
- Link ID: enter a pipeline where the discharge of 2350 l/s is measured, it can be any pipeline along the water supply water main unless there are any turnouts (demands) in-between the pump station and the downstream storage tank
- Measured data type: select 'Constant value' for steady state discharge
- Constant value: enter the measured discharge of 2350 l/s.

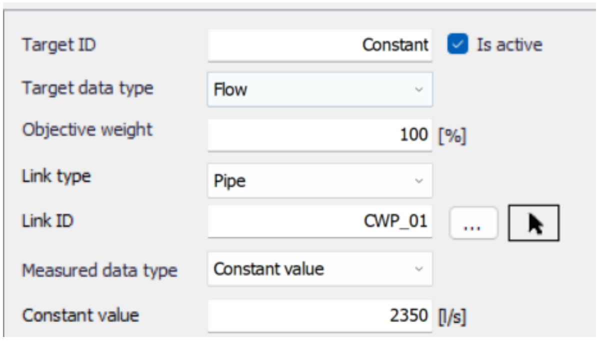


Figure 20.10 Example 1 - Targets

Next, click 'Run' and the program will start optimizing the friction factor.

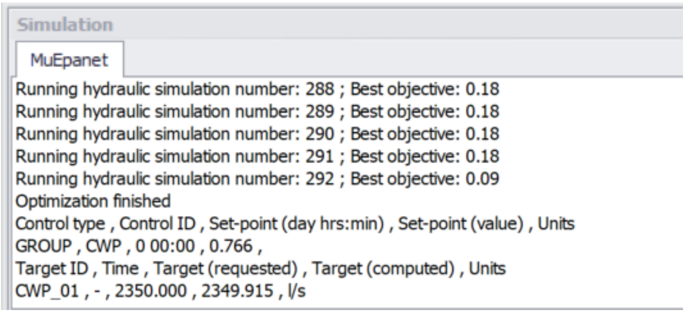


Figure 20.11 Example of simulation output in display

The report from the simulation shows that the simulation was completed and that the target flow was satisfied. Results are visible in the 'Outputs' tab.

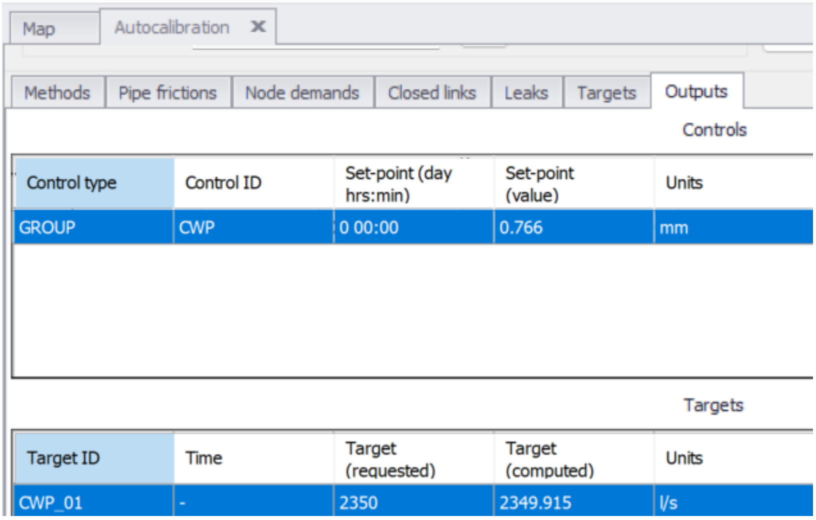


Figure 20.12 Example 1 - Outputs

The target discharge was 2350 l/s, and the computed discharge is 2349.91 l/s. The calibrated friction factor is 0.766mm.

Finally, go to the 'Pipe frictions' tab, and review the calibrated friction value. You can either accept or adjust the approved friction, and then click 'Apply all frictions' to apply the approved friction factor to all pipes within the "CWP" selection.

20.10.2 Example 2 - Pipe friction extended period simulation

The treated water system contains a pump station, a long water supply pipeline consisting of multiple pipelines and a storage tank that is receiving the pumped water. The extended period (24-hours) model was developed with assumed friction factor, entered as C values (Hazen-William's friction coefficient). Time series of measured pressure are available at 3 locations within the distribution system.

To use the autocalibration for pipe friction, open the 'Autocalibration' editor and insert a new record, use the SCE method, for example, and accept all default settings but change the maximum number of calls to 5000.

Next, create selection lists to group pipes (e.g. called Group_A, Group_B, Group_C) within the pipe network into groups based on similar material and installation date. We will assume 3 groups in this example.

Next, enter 3 new controls into the 'Pipe friction' tab: one for each pipe group. Define the minimum and maximum values of C-friction factor as illustrated below for each group.

- Selection ID: "Group_A"

- Minimum friction: 10
- Maximum friction: 140

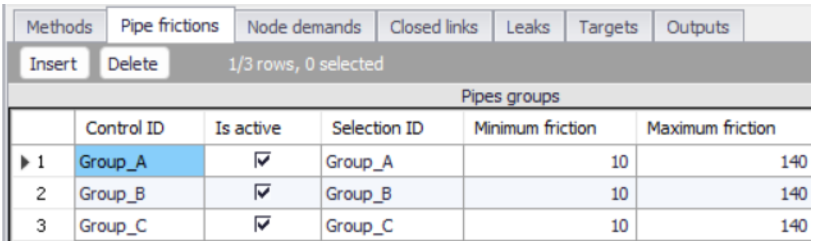


Figure 20.13 Example 2 - Pipe friction

Next, enter 3 new records in the 'Targets' tab (one for each of the 3 measurements):

- Target data type: select 'Pressure'
- Objective weight: 100%
- Junction ID: enter the junction node that corresponds to the pressure sensor location
- Measured data type: select "Time series" for extended period simulations with time series of measured data
- Plot ID: select the name of the Plot ID, defined in the 'Plots and statistics' editor, which contains the time series of measured data
- Evaluation type: select one of the available options.

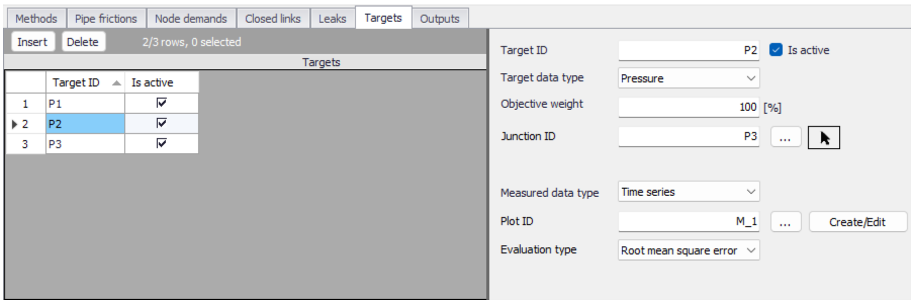


Figure 20.14 Example 2 - Targets

When 'Pipe frictions' and 'Targets' are defined, click 'Run' to run the autocalibration.

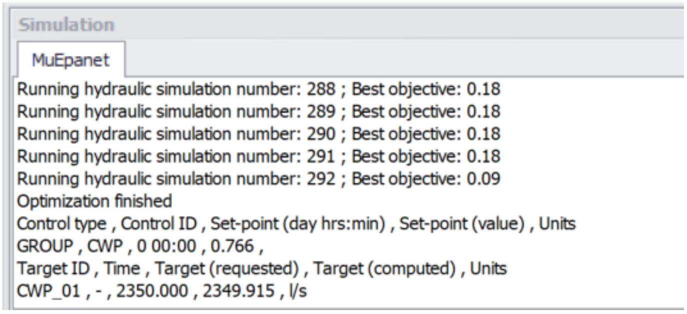
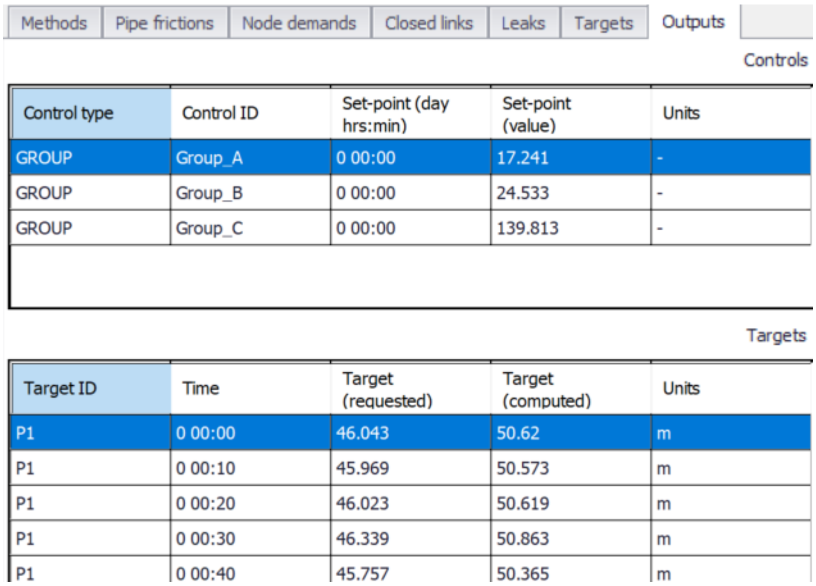


Figure 20.15 Example of simulation output in display

The report from the simulation shows that the simulation was completed and that the target flow was satisfied. Results are visible in the 'Outputs' tab.



Control type	Control ID	Set-point (day hrs:min)	Set-point (value)	Units
GROUP	Group_A	0 00:00	17.241	-
GROUP	Group_B	0 00:00	24.533	-
GROUP	Group_C	0 00:00	139.813	-

Targets

Target ID	Time	Target (requested)	Target (computed)	Units
P1	0 00:00	46.043	50.62	m
P1	0 00:10	45.969	50.573	m
P1	0 00:20	46.023	50.619	m
P1	0 00:30	46.339	50.863	m
P1	0 00:40	45.757	50.365	m

Figure 20.16 Example 2 - Outputs

The program also generates a XML report with autocalibration results, which is opened with the 'Report' button.

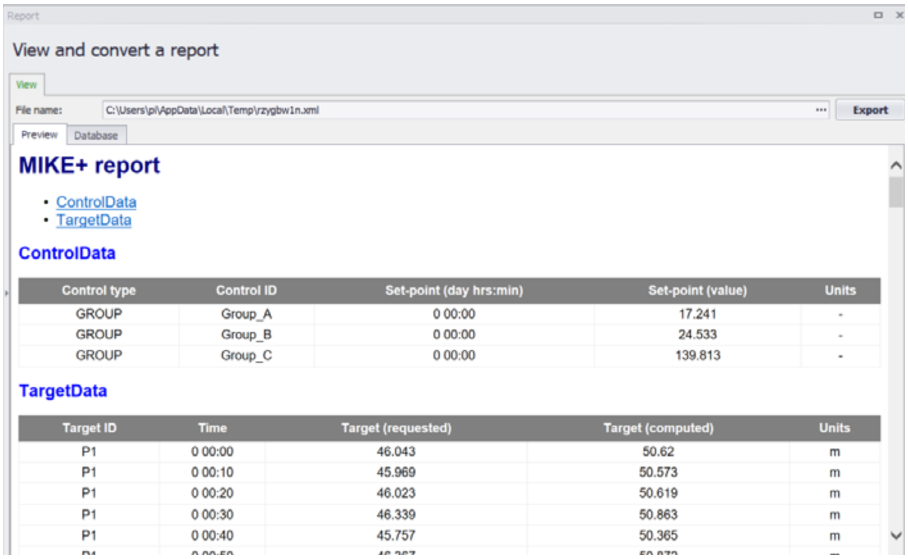


Figure 20.17 Example 2 - Reports

To review the match between computed and measured pressure time series, go to the 'Plots and statistics' editor, where the measured data have been specified. In this dialog, select the results file from autocalibration run and select 'Pressure' from the item list. Once defined, the time series plot will superimpose computed and measured data.

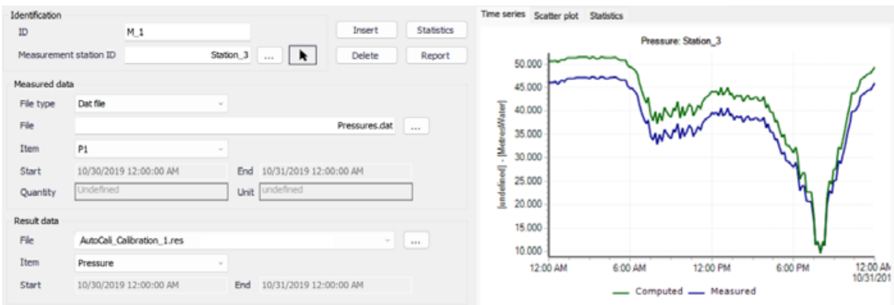


Figure 20.18 Example 2 - Plots and statistics with autocalibration results

Finally, go to the 'Pipe frictions' tab of the Autocalibration editor, and review the calibrated friction values. You can either accept or adjust the approved frictions, and then click 'Apply all frictions' to apply the approved friction factors to all pipes within the respective selections.

INDEX