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
- Pipe critically
- Cost analysis
- Shutdown planning
- Flushing analysis
- Water hammer analysis

These modules indicate which type of analysis will be modelled within the current project setup. For example if fire flow analysis its 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.

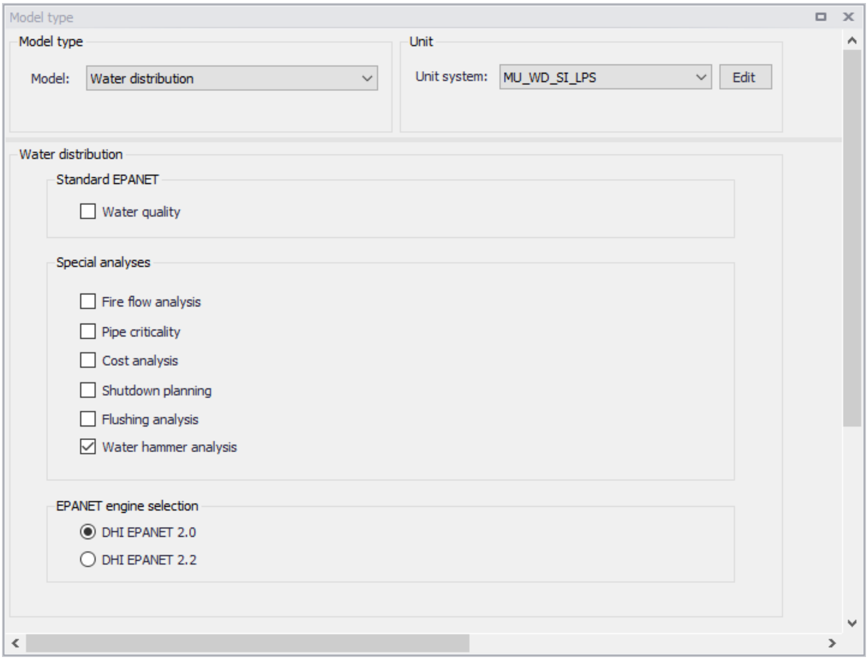


Figure 1.1 General Settings

The 'EPANET engine selection' option controls which version of the EPANET engine is to be used in the simulations, which can either be EPANET 2.0 or EPANET 2.2.

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.

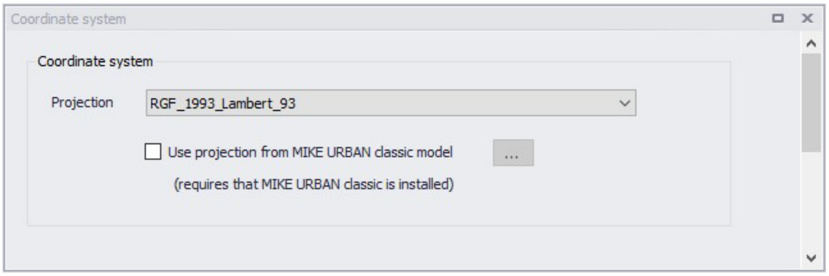


Figure 2.1 The Coordinate System dialog showing information on the projection system used in the project

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 bMIKE+ottom of the list: this will open a window listing the available projections, and where it is also possible to import new projections from a projection file (*.prj file).

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

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

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

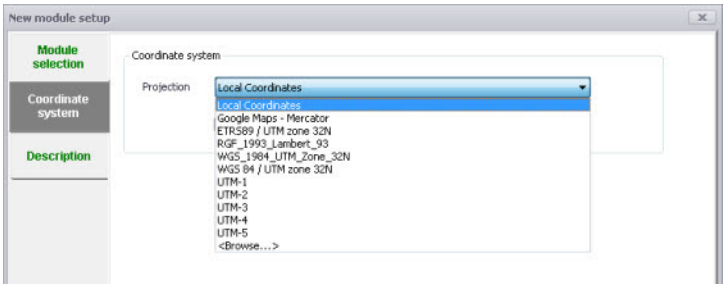


Figure 2.2 Specify the projection system for a new MIKE+ project on the New Module Setup window

2.2 Background Map

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

Activate a background map overlay by ticking the ‘Visible’ checkbox on the editor.

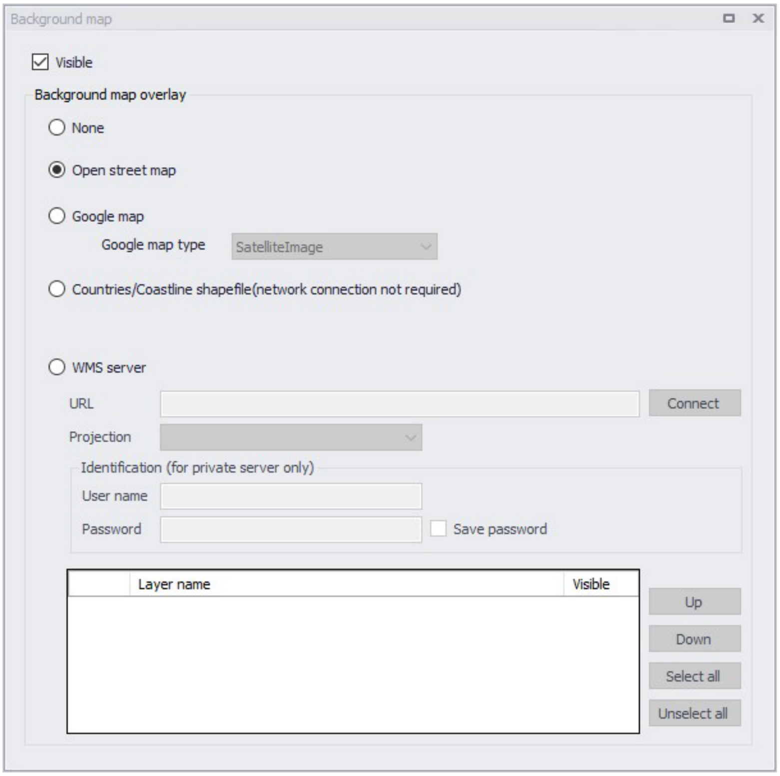


Figure 2.3 The Background Map Editor

The following background map overlay options are available:

- **None**
- **Open Street Map**
- **Google Map**. Select the Google map type to display (i.e. Street map, Satellite image, Terrain, or Hybrid).
- **Countries/Coastline Shapefile**. Polygon feature showing coastlines and demarcating oceans and inland areas.
- **WMS server**. Background maps obtained from a remote server. Enter the URL of the server and click 'Connect'. If the server is a private server, you will need to supply the user name and password for authentication, and you may optionally tick 'Save password' in order not to enter it again the next time you open MIKE+. When the connection is established, the table will provide the list of layers available on the server, and it is possible to select which layers to display in MIKE+ using the 'Visible' box. The list of projections will show the map projection(s) supported by the WMS server, and the one used for the model data in MIKE+ will be selected if possible. Note that displaying layers from a WMS server requires that the MIKE+ project uses the same map projection as the WMS layers: if the projection used in MIKE+ doesn't match any of the projections supported by the WMS server, you will be asked to update the map projection in MIKE+. Also note that it is only possible to connect to WMS servers using projected map projections (geographical coordinate systems not supported).An axis order also needs to be specified, defining the format of the coordinates on the WMS server: most of the servers provide coordinates in the XY order, but some servers provide coordinates in the opposite order and in this case the option must be changed to 'YX' otherwise the layers won't be displayed on the map.

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

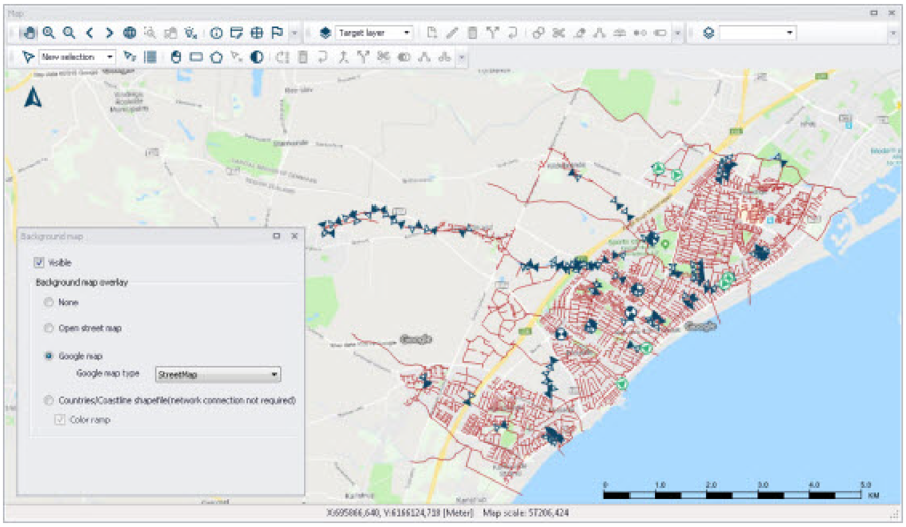


Figure 2.4 An example Google Map background on the Map View in MIKE+

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.

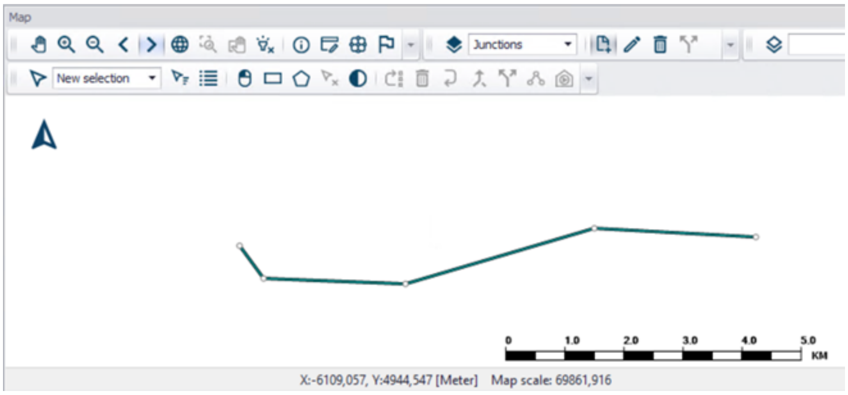


Figure 3.1 Draw Junctions

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

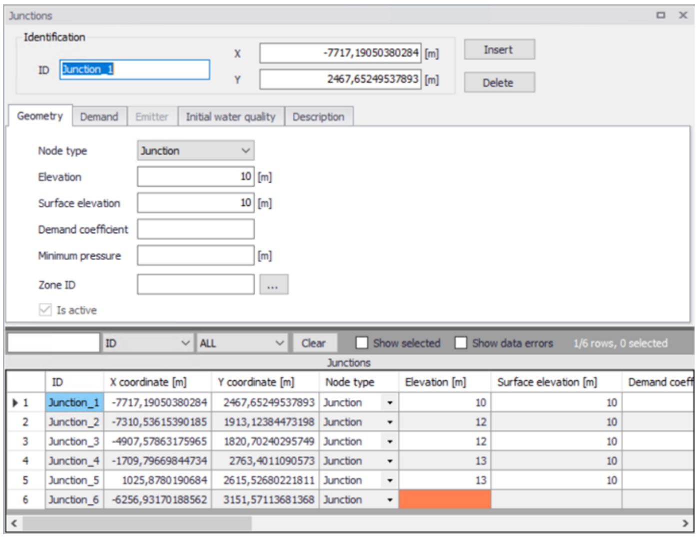


Figure 3.2 Junction Editor, Geometry tab

<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.

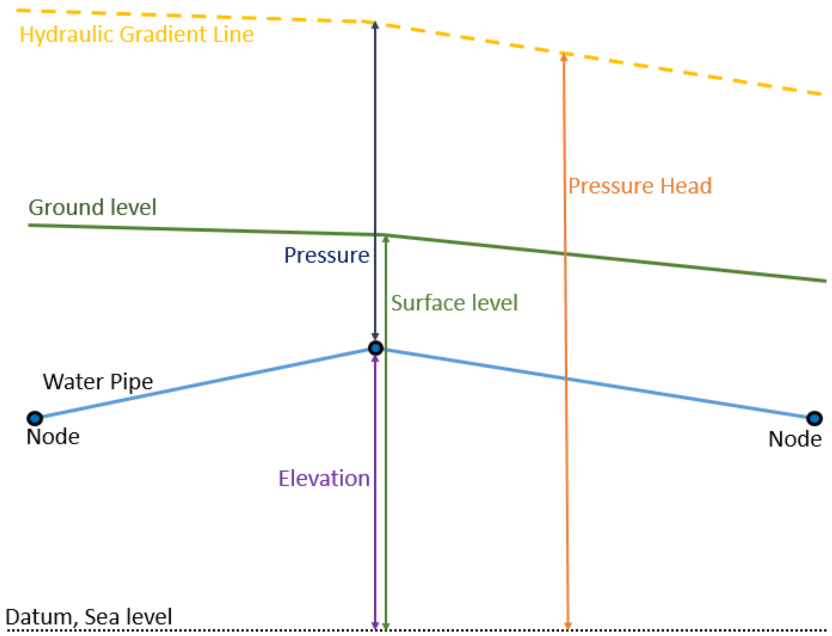


Figure 3.3 The difference in Elevation and Surface level

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

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

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.

Is active

This check box is always marked. The option to disable an object is not available for Junctions.

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.

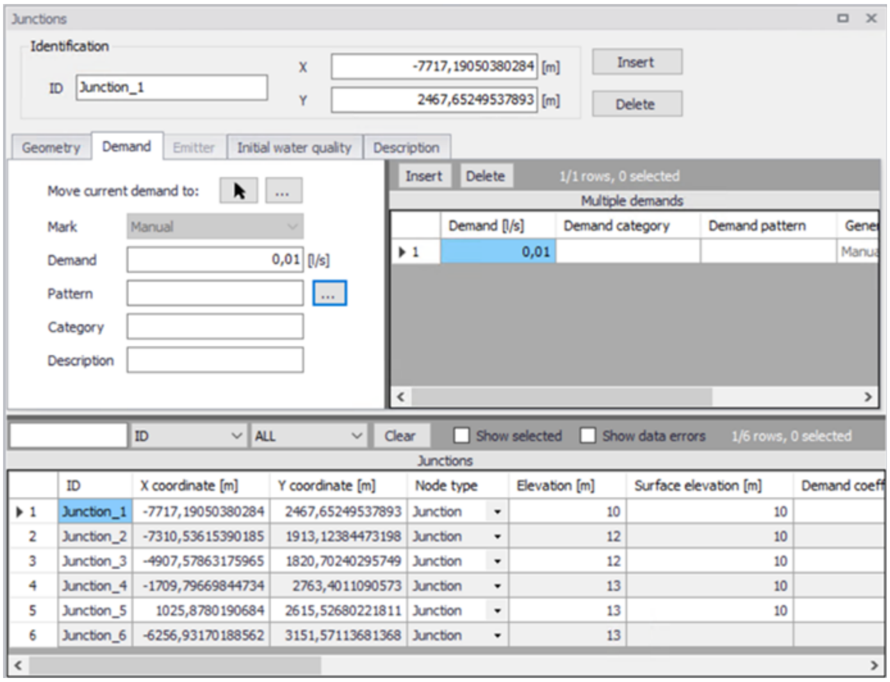


Figure 3.4 Junction editor, Demand tab

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.

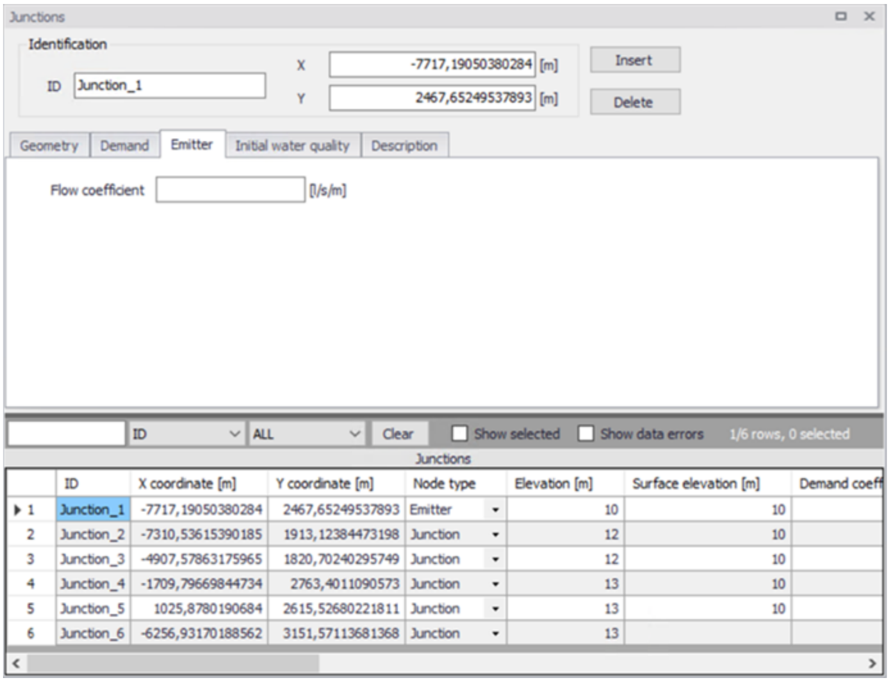


Figure 3.5 Emitter tab

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. The flow coefficient is defined in flow units per 1 psi or m pressure drop

Initial Water Quality

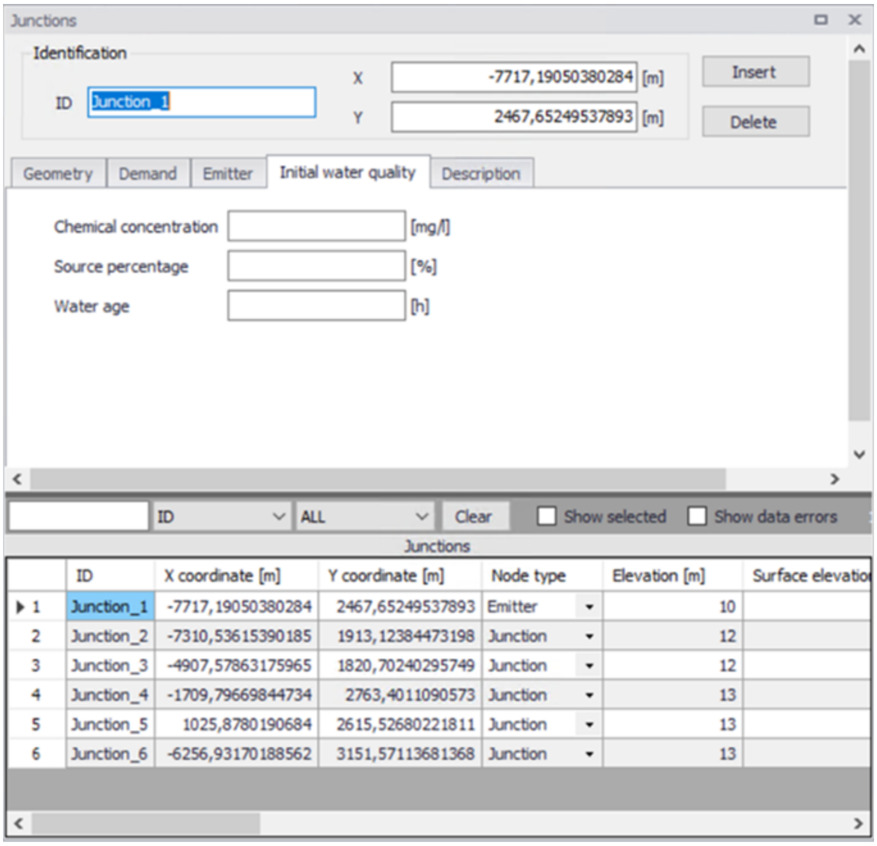


Figure 3.6 Junction editor, Initial water quality tab

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

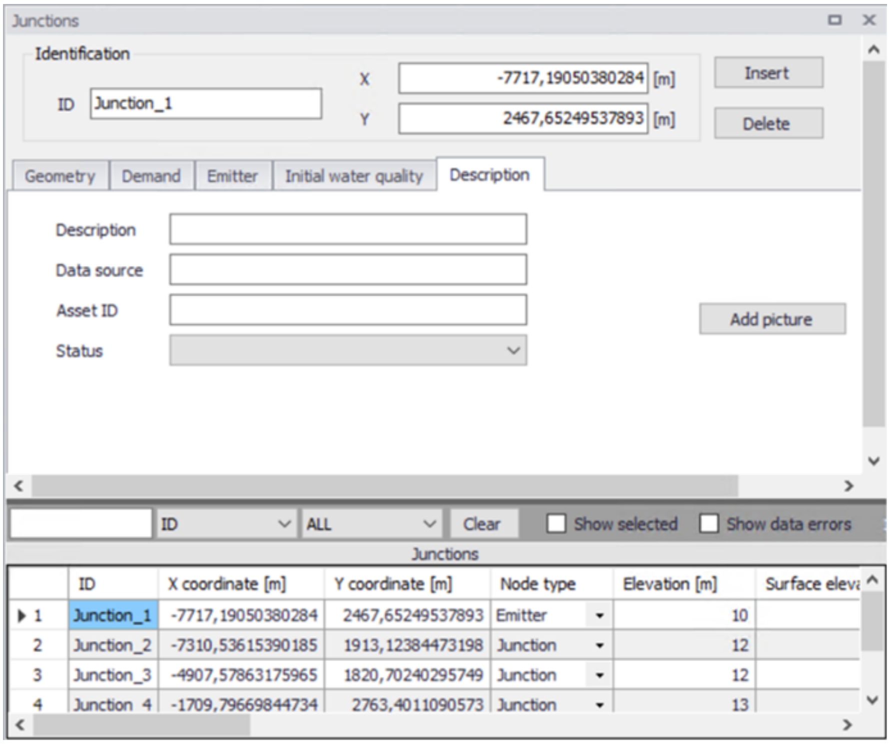


Figure 3.7 Junction editor, Description tab

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	

Table 3.1 Junction attributes

Field	Database name	Description	Mandatory?	Default value
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.

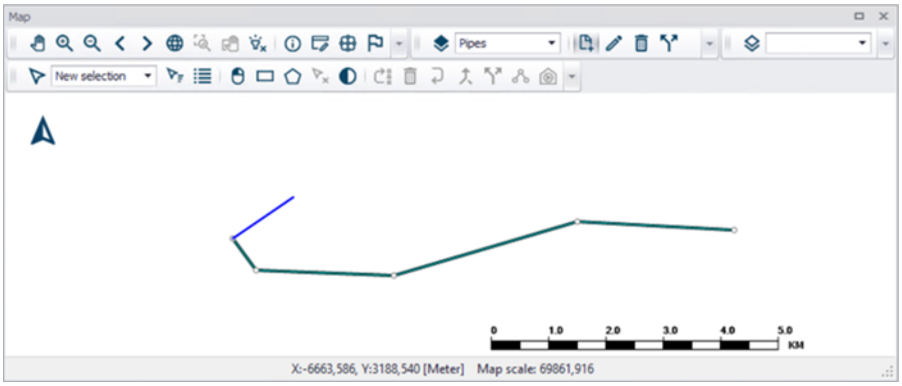


Figure 3.8 Pipes displayed in map

Geometry

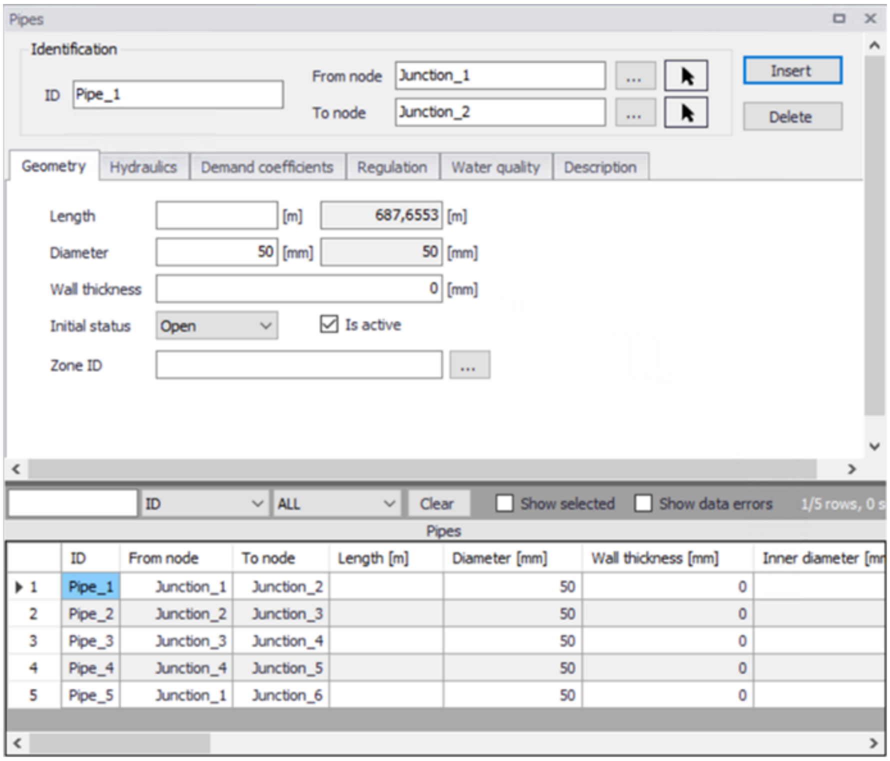


Figure 3.9 Pipe Geometry Editor

<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 data entry allows the user to toggle the Active status of the pipe on and off. The simulations will omit all pipes that are not active.

Hydraulics

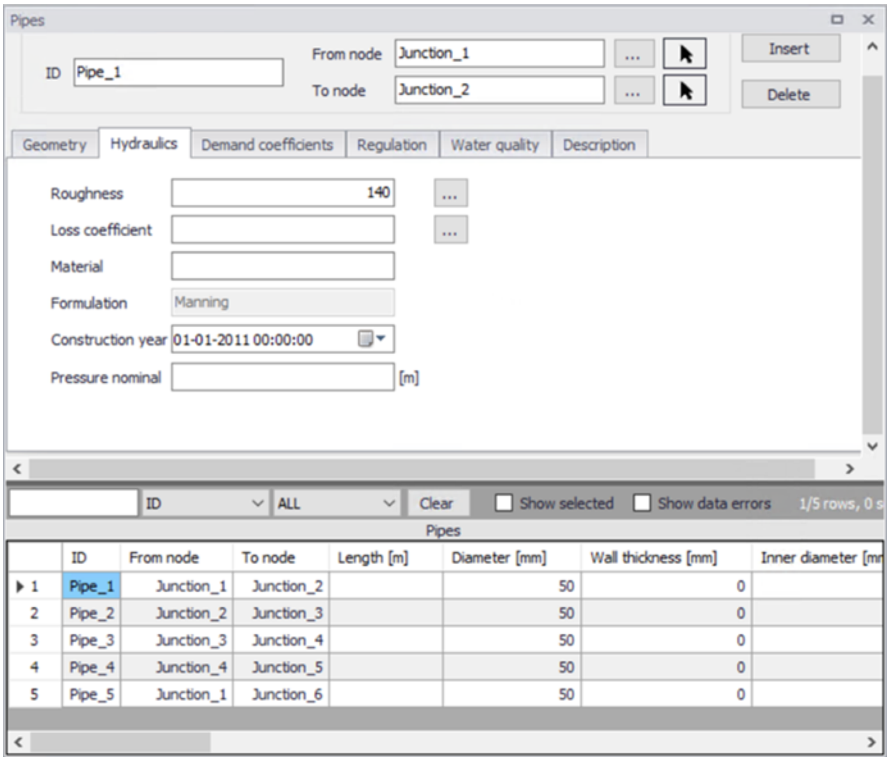


Figure 3.10 Pipe Hydraulics Editor

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.

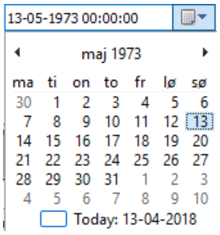


Figure 3.11 Calendar view

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.

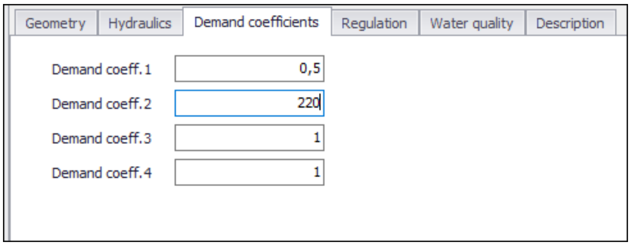


Figure 3.12 Pipe Demand Coefficients Editor

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.

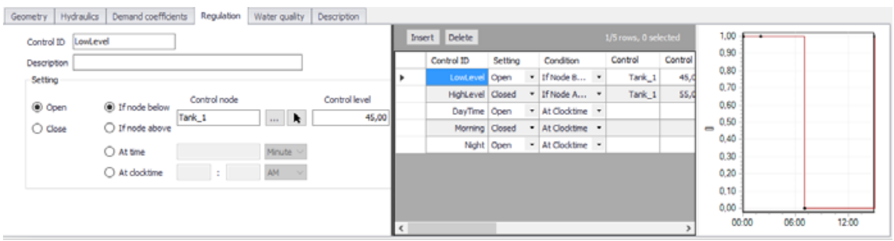


Figure 3.13 Regulation Tab

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.

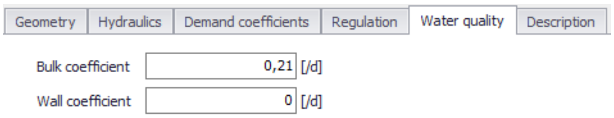


Figure 3.14 Water Quality Editor

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.

Description

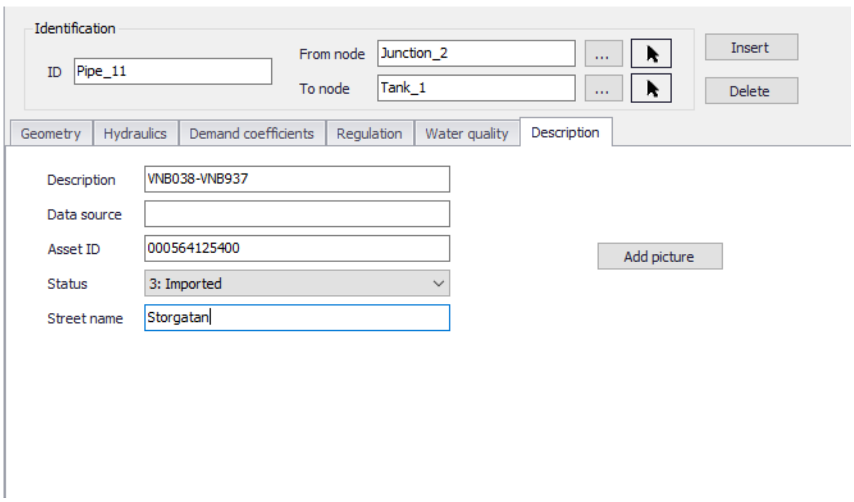


Figure 3.15 Pipe Description Editor

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	

Table 3.2 Pipe attributes

Field	Database name	Description	Mandatory?	Default value
Initial status	StatusNo	Sets the pipe to open, closed or check valve	Yes	Open
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

3.3.1 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

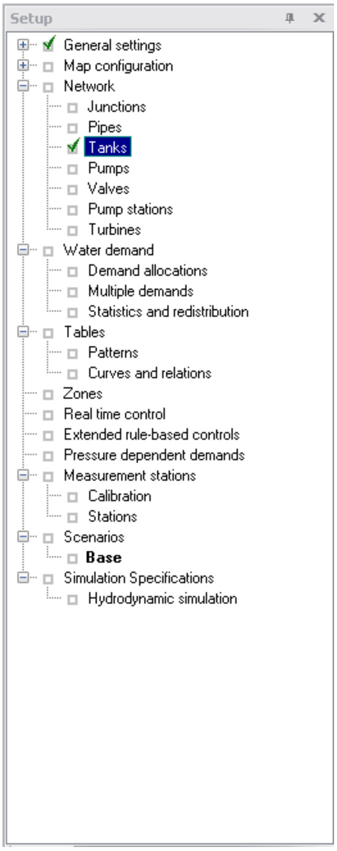


Figure 3.17 The Tank Editor dialog box is reached in Setup tree

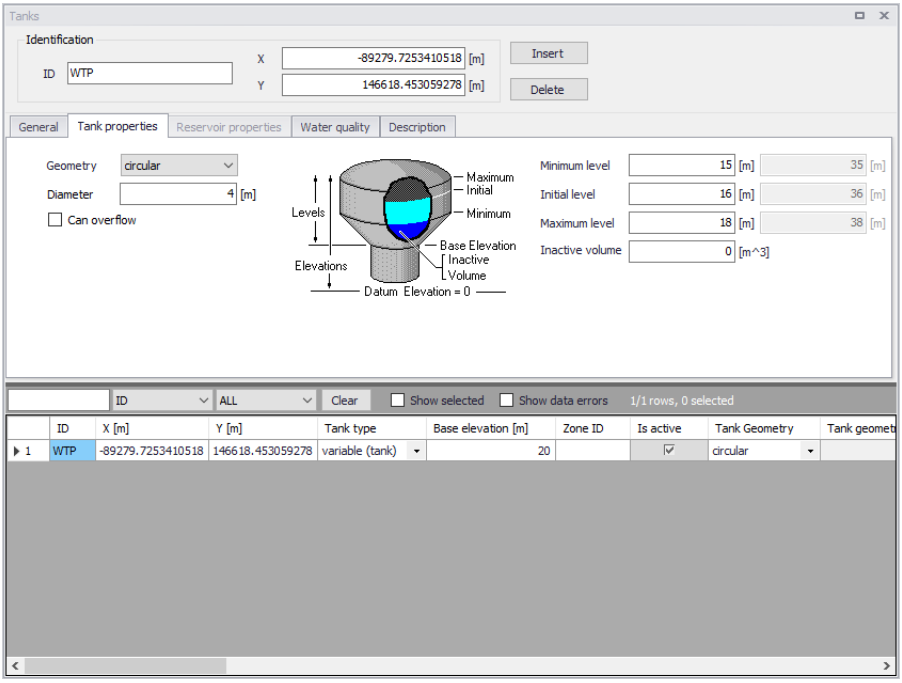


Figure 3.18 The Tank Editor allows the user to define the storage tank nodes that supply water to the distribution network

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

Identification

Tank ID (mandatory)

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. The check would be instant, when the user types an ID already used, there will be a warning message beside the field and the user would not be able to type anything else (see).

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.

When importing (or merging) multiple water distribution network models into a single network model, MIKE+++ will check for collisions between identical node IDs and will automatically assign a new node ID value for any node being imported that contains the same node ID value as what already exists in the network model.

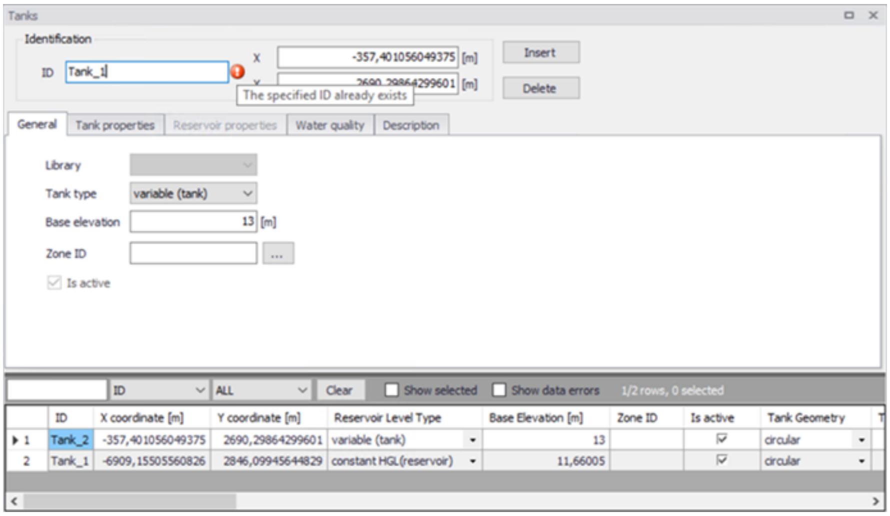


Figure 3.19 Warning message displayed when a tank has a repetitive ID

X and Y COORDINATE (optional)

The X and Y data entries are used to define the physical (map) location of the tank node, in units of ft. or m. This location definition is optional. In some cases, the actual location of the tank node is not known—especially in older, legacy networks. However, if the location is defined, then the tank will be displayed in the Map window. When defining the tank nodes graphically on the Map window using the Add Tank tool, the X, Y location is automatically entered.

Grid Attribute

The grid at the bottom of Tank Editor shows the attribute table of tanks. When selecting a tank from the table, that tank will be located on the network map with selected state. On the top of the grid, users can search and select tanks according to their attributes. It also gives options for clearing selection, showing selected junction attributes and showing data errors.

Right click on the header of column, showing a menu with following items:

- Statistic
- Field calculator
- Select by expression

These functions should be all applied on the selection.

Right click on the grid origin, showing a menu of Hide columns: the fields of feature can be selected to be seen or hidden by users.

Right click on anywhere else origin, there is a pop up menu showing with following items:

- Clear selection
- Copy to clipboard
- Copy to clipboard with header text
- Paste from clipboard
- Reset layout
- Show columns in active tab
- Add user defined column

Tank Properties

It contains input fields for geometry, Tank Properties, Reservoir Properties and Description. Detailed information of each section is shown below.

General

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

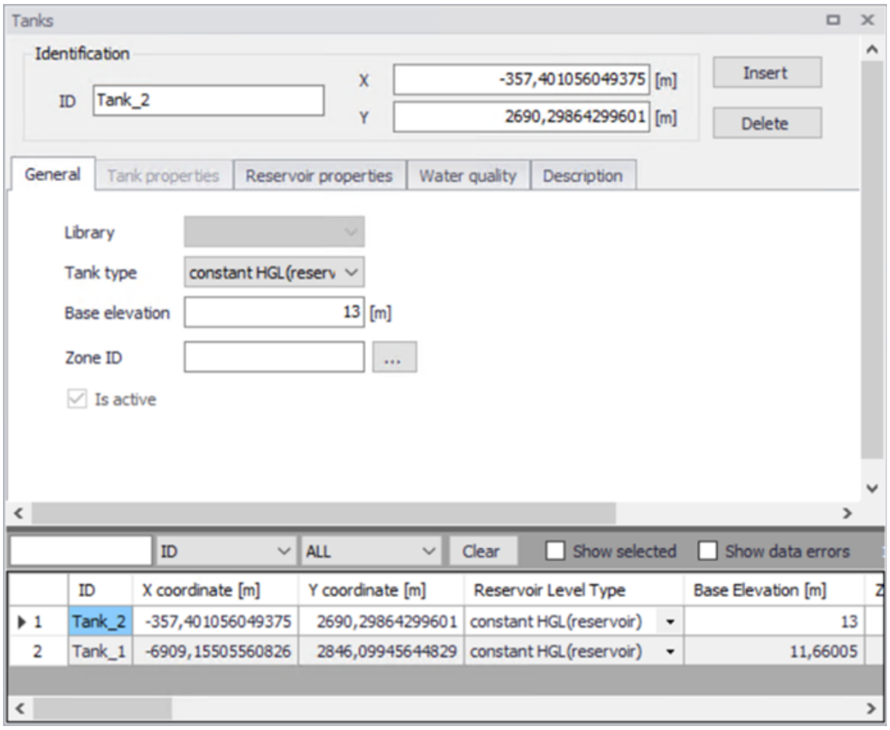


Figure 3.20 The general information of Tank

Tank Type (mandatory)

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.

Is Active (mandatory)

It defines whether the tank is active or not. If the tank is active, it would be included in the model, otherwise it would be omitted.

TANK PROPERTIES

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

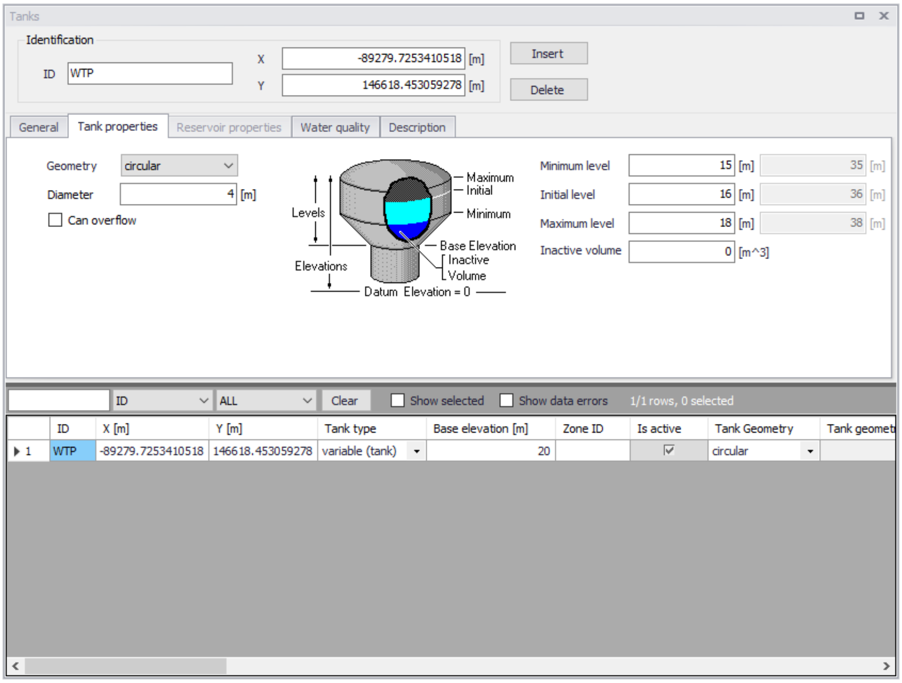


Figure 3.21 The Tank Properties

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)

A geometry ID 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.

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 ft3 or m3, of the storage tank, as shown in Figure 3.22.

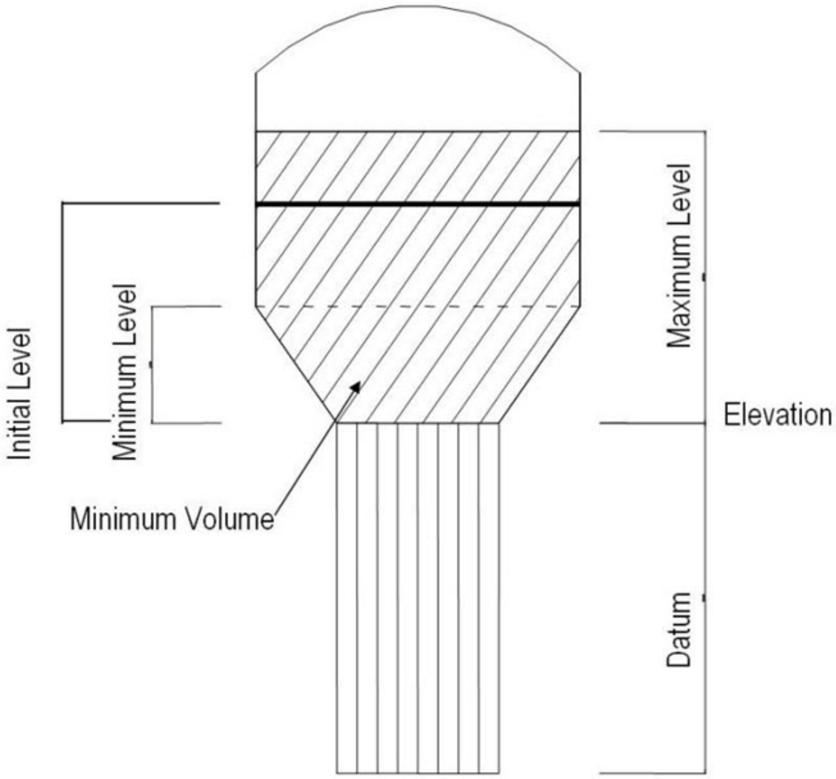


Figure 3.22 Definition of storage tank levels

RESERVOIR PROPERTIES

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

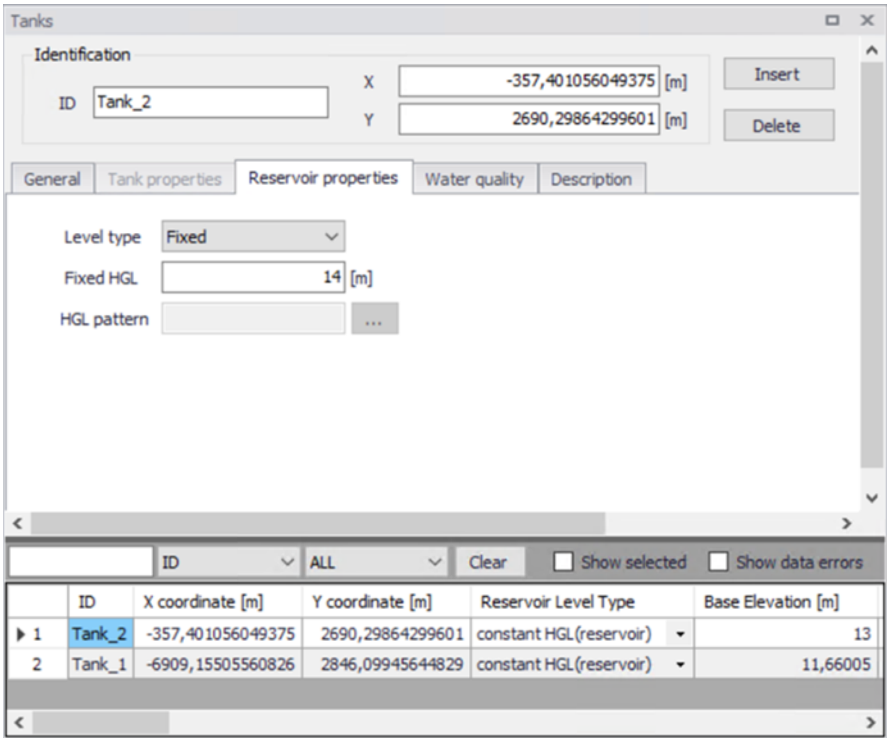


Figure 3.23 The Reservoir Properties

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.

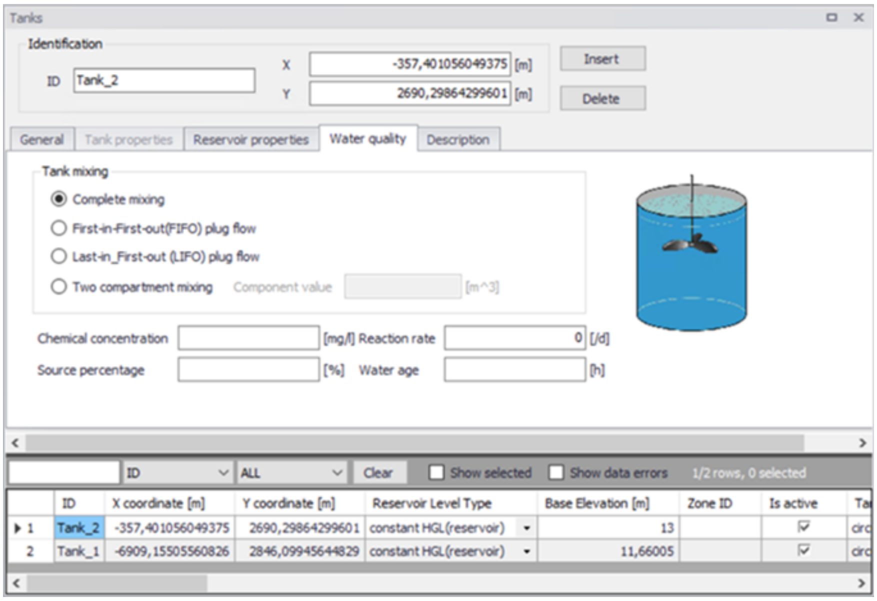


Figure 3.24 The water quality parameters of tanks

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.

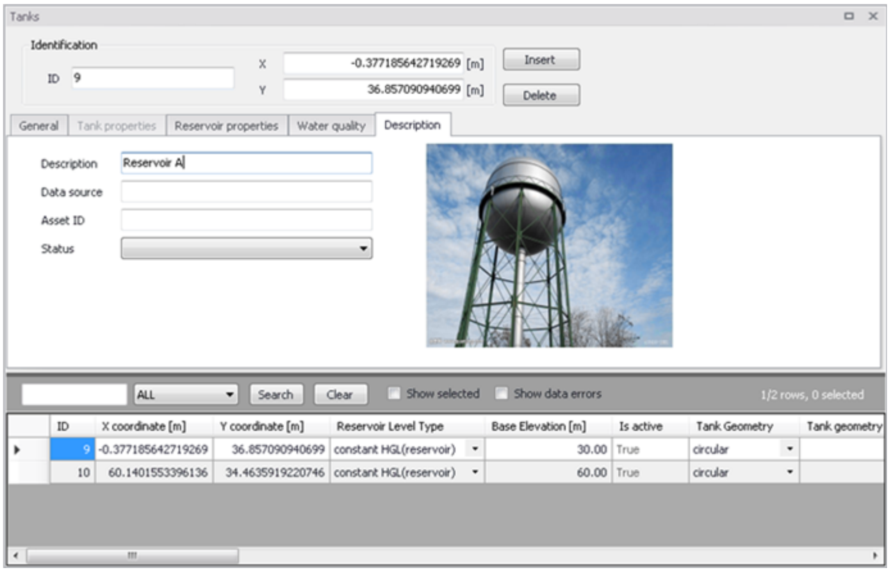


Figure 3.25 Tank Editor Picture

3.4 Pump Editor

Pumps are used to raise the hydraulic head of water. Pumps are represented as short links of negligible length. *MIKE+ WATER* analysis 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 box in Figure 3.27.

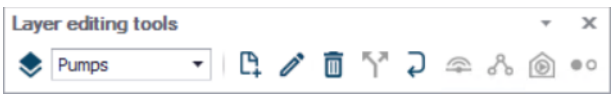


Figure 3.26 Pumps Drawing and Editing Tool

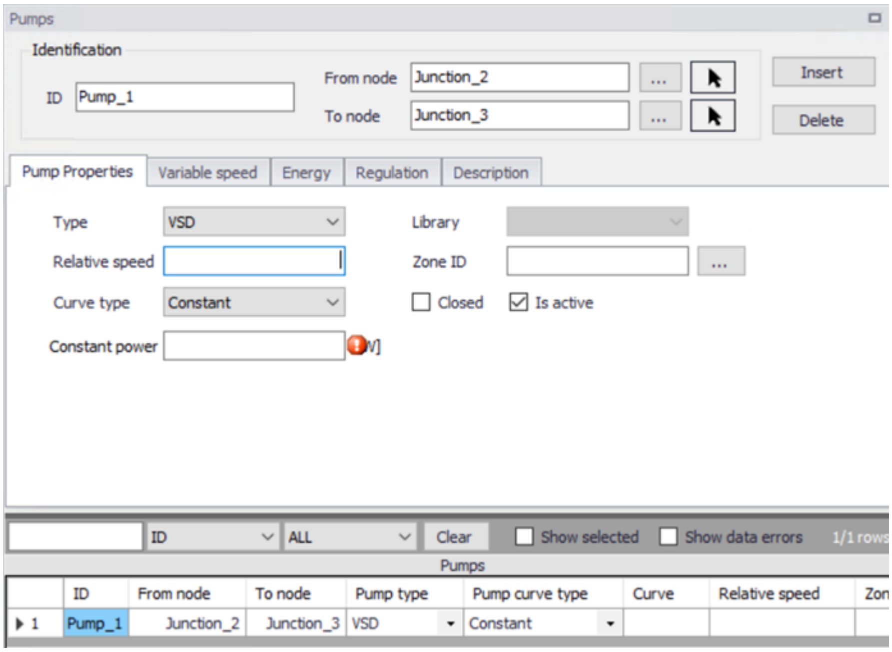


Figure 3.27   Pump Configuration Window

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

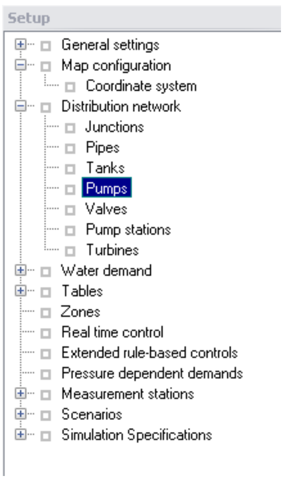


Figure 3.28   Layout of Setup Tree

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. See Figure 3.29

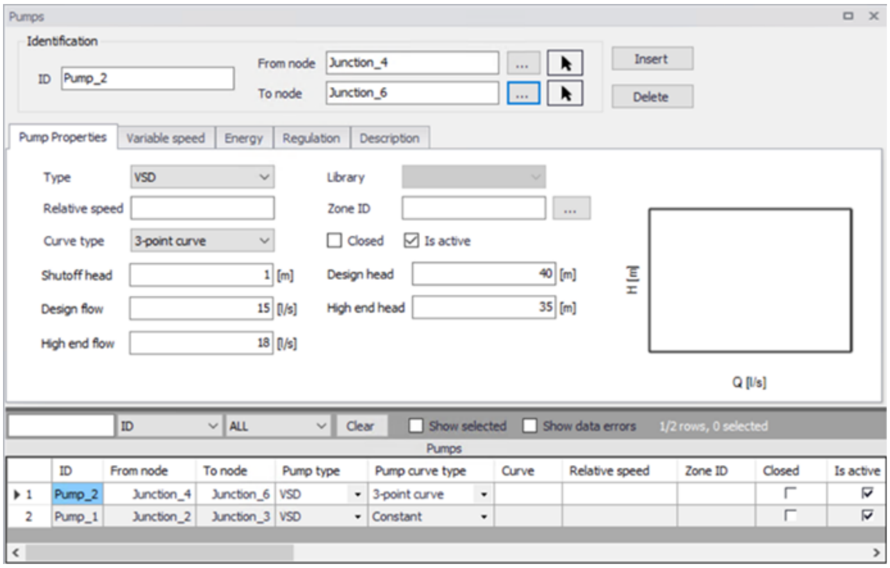


Figure 3.29 Identification in Pump Editor

From node, To node

These data entries define the ID of the pump's starting and ending nodes. Clicking , ID selection window pops up and the Node Type pull-down selection list allows to specify what type of node is connected to the pump. The type selection is either Junction or Tank in Figure 3.30. 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.

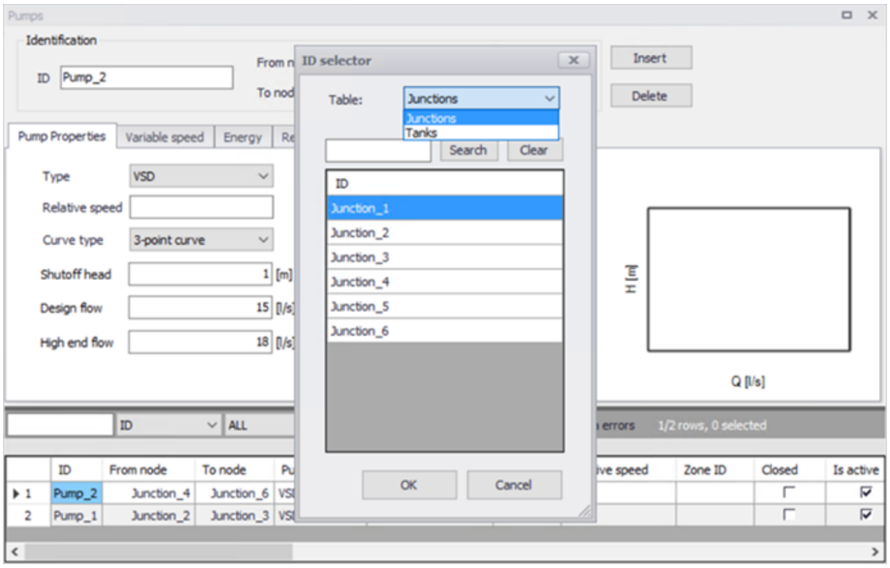


Figure 3.30 ID Selection Window

Grid Attribute

The grid at the bottom of Pump Editor shows the attribute tables of pumps. When selecting a pump from the table, the pump link will be located in yellow on the network map. There is a filter on top of the grid. The blank box allows users to type the Pump ID to be filtered whilst users can also search the specific pump by pump types (VSD or CSP). It also let users clear selection criterion by clicking <Clear> button.

There are two check box to either show selected pumps or filter the data entry errors.

<Show Selected> check box allows users to select the pumps selected in the grid (i.e. select pumps on the map and it'll be popped up in the grid). <Show Data Errors> check box allows to filter out errors in settings or data in the grid where it is highlighted in red.

Clicking the top left corner of the grid, it pops up a list of items to be displayed in the grid. By default all boxed are checked but users can uncheck the items that are not to be displayed. See Figure 3.31

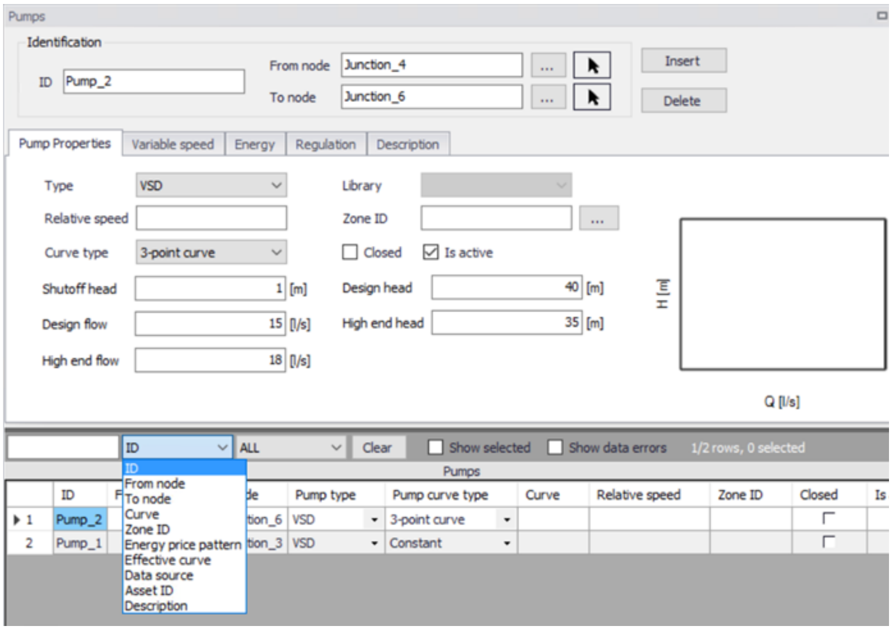


Figure 3.31 Selection Lists of Items Displayed

By right clicking the header of each column, a list of following options can be selected from (Figure 3.32):

- Select column
- Field calculator
- Select by attribute

By right clicking each cell in the grid, a list of following options pops up:

- Clear selection
- Copy to clipboard
- Copy to clipboard with header text
- Paste from clipboard
- Reset layout
- Show columns in active tab
- Add user defined columns

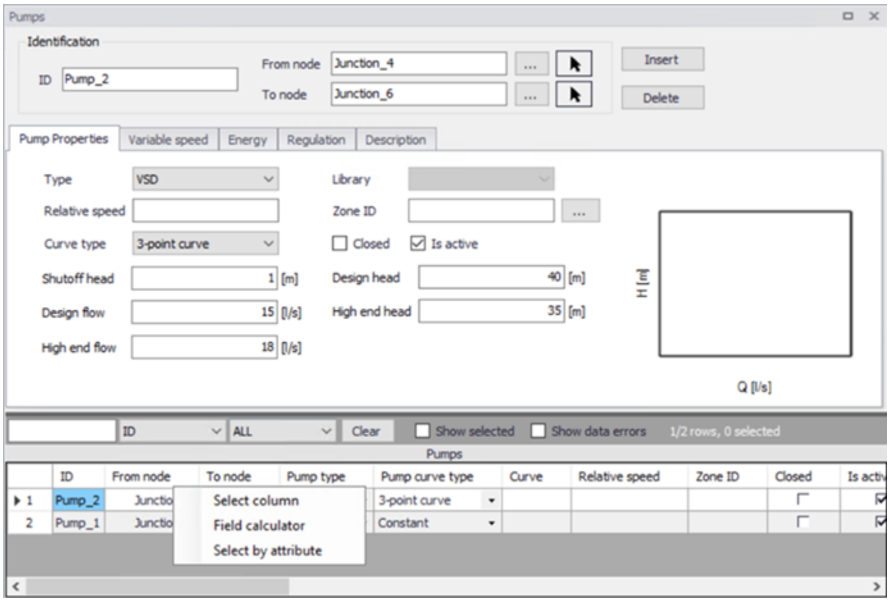


Figure 3.32 List of Selection Options

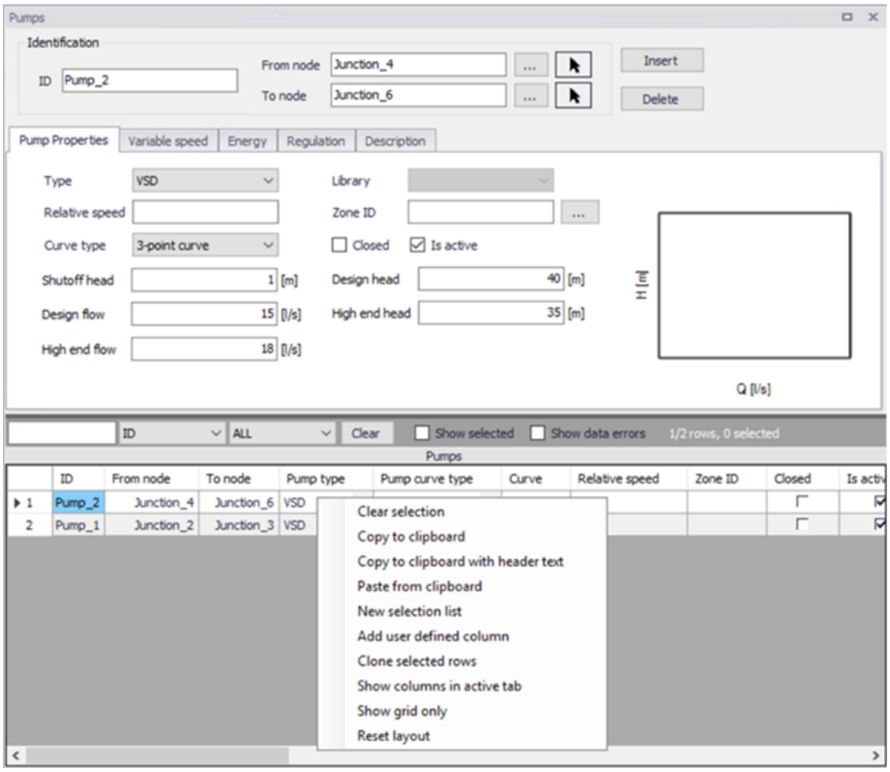


Figure 3.33 List of Selection Options

Pump Properties

This option button data entry defines the pump operating characteristics.

Pump 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 control at node. 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. See

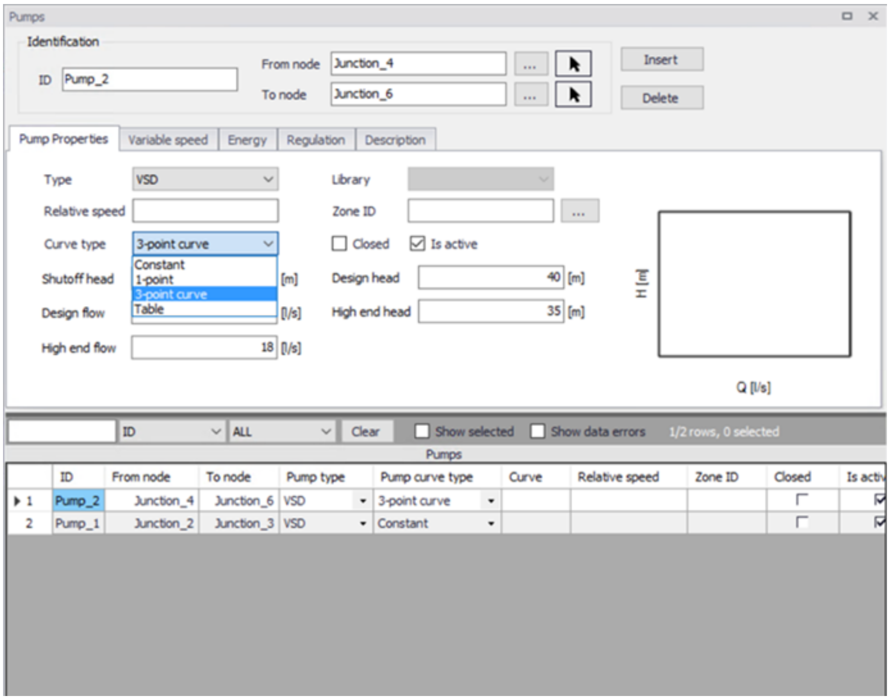


Figure 3.34 Pump Curve Type

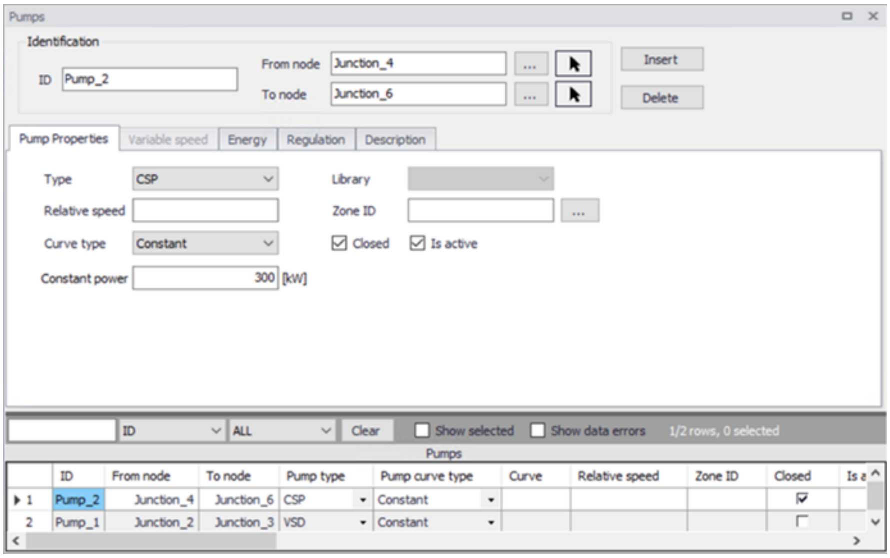


Figure 3.35 Pump Curve Type: Constant Power Setting

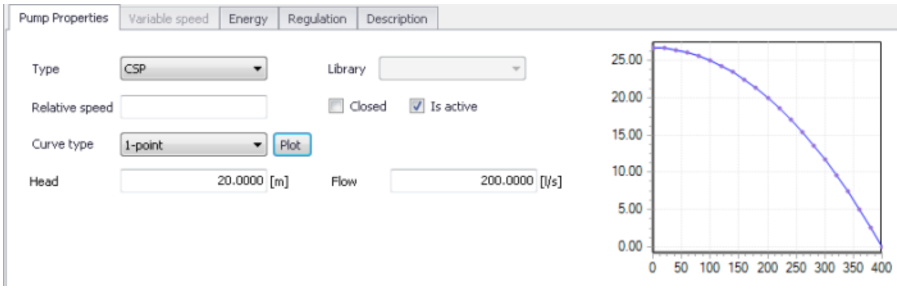


Figure 3.36 1-Point Pump Curve

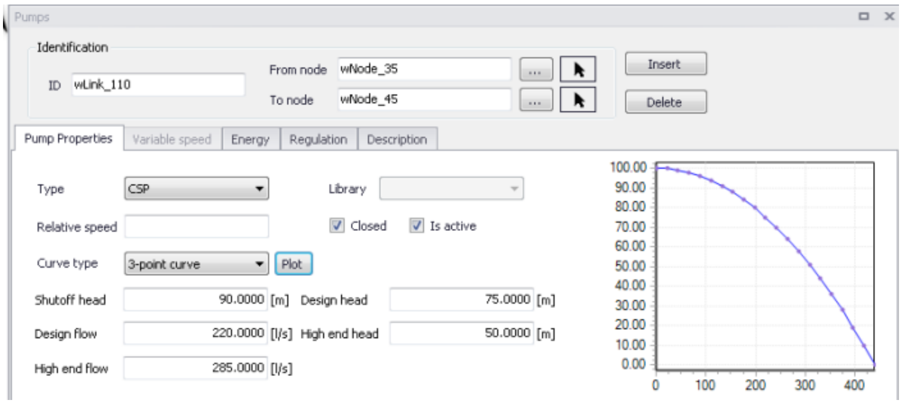


Figure 3.37 3 Point Pump Curve

Constant Power is used 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. See Figure 3.35

1-Point Pump Curve pump, as shown in Figure 3.36, 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 Pump Curve, as shown in Figure 2.2, 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. See Figure 3.37

The Q-H Pump Curve can be 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 can be created in the Curve Relation Editor found in the <Tables> menu in Figure 3.38 and Figure 3.39 (see Tables section for more details). Once the Q-H pump curve is created, it can then be assigned to a pump in the Pump Editor under option 4 <Table>. See Figure 3.40.

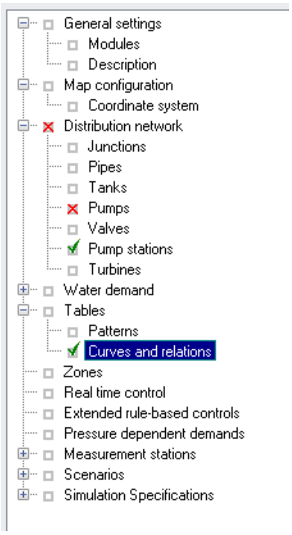


Figure 3.38 Curve and Relations in Setup Tree

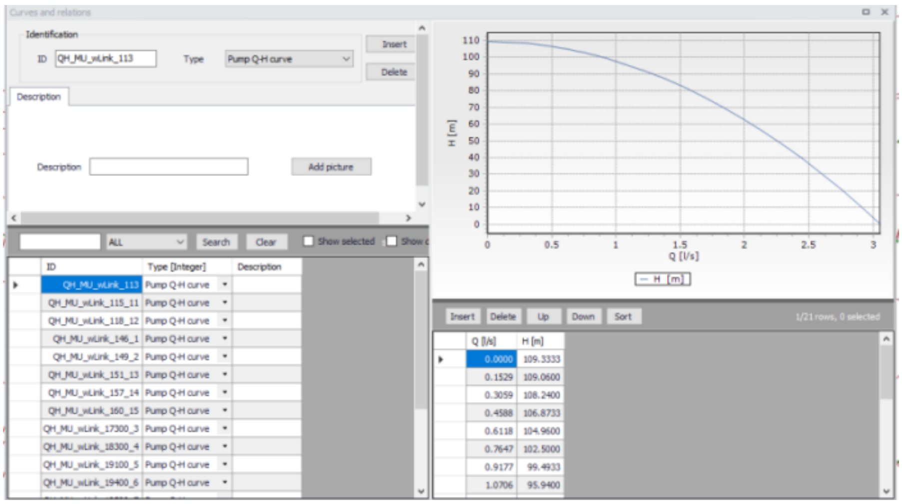


Figure 3.39 Curves and Relations Settings

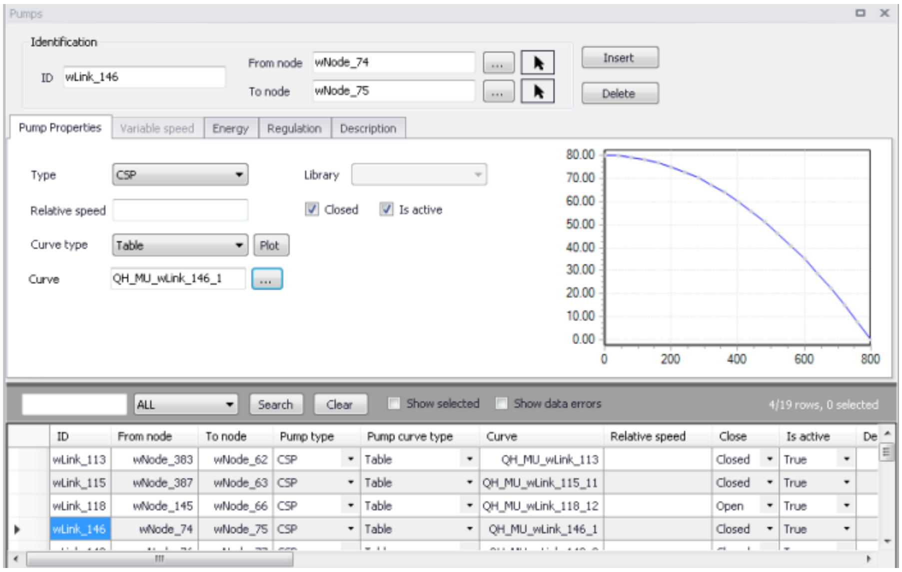


Figure 3.40 Q-H Pump Curve in Table

Variable Speed

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

Control type

Two 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.

Control node

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

Control pressure

Users can type the pressure value in meter 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'.

Curve

The selected 'Pump pressure 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.

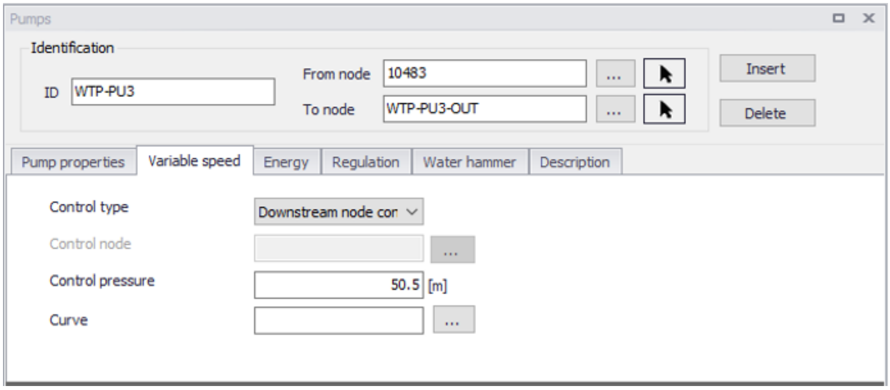


Figure 3.41 Variable Speed Setting

Energy

MIKE+ Water is capable of modelling the cost of operating pumps. Within the Pump Energy Editor, the user can define a method for cost calculation. See Figure 3.42

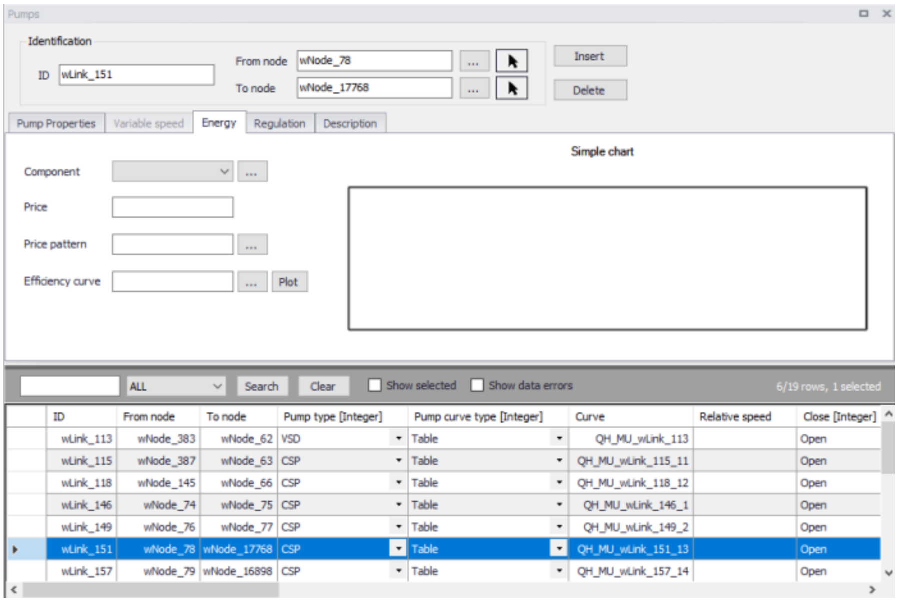


Figure 3.42 Energy Settings

Component=not in current release, will need to be completed when development finish.

Price

The user defines an energy price ($/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

Control ID

When clicking <Insert> in the control section located in the middle of REGULATION settings (see Figure 3.43), an ID will be automatically generated and displayed in Control ID field.

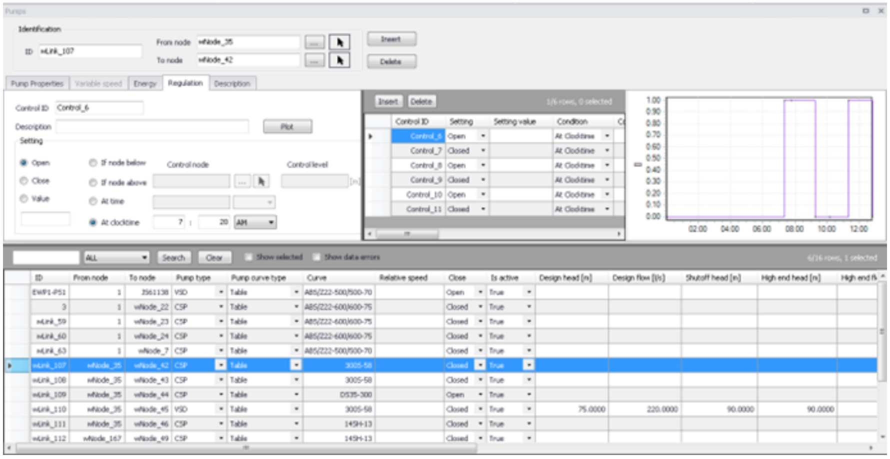


Figure 3.43 Regulation Setting Tab

Description

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

Settings

This field is enabled only when a control is inserted or already existing in the middle section in Figure 3.44

When clicking <Insert>, a Control ID will be automatically created and the controls can be edited in either Setting field or from the drop-down list in Control field (i.e. OPEN or CLOSED at what time or under what conditions). The controls for an individual pump can also be plotted in graph in the right field by clicking <Plot>.

Settings in REGULATION suits CSP (Constant Speed Pumps). VSD pumps can be controlled in RTC (Real-time Control) settings. Please refer to the specific settings.

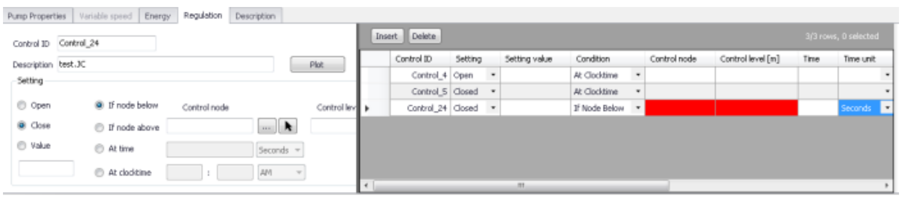


Figure 3.44 Control Settings in Regulation

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 3.45

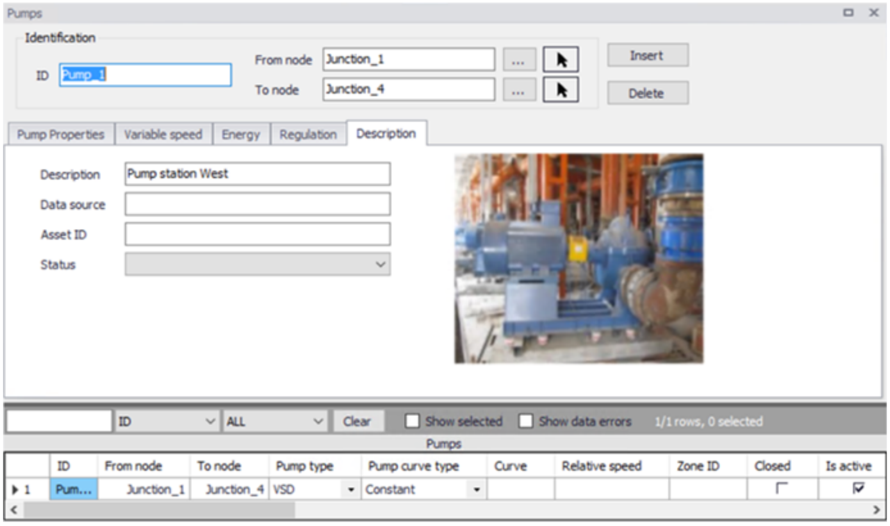


Figure 3.45 Layout of Description Settings

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 3.46

Figure 3.46 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

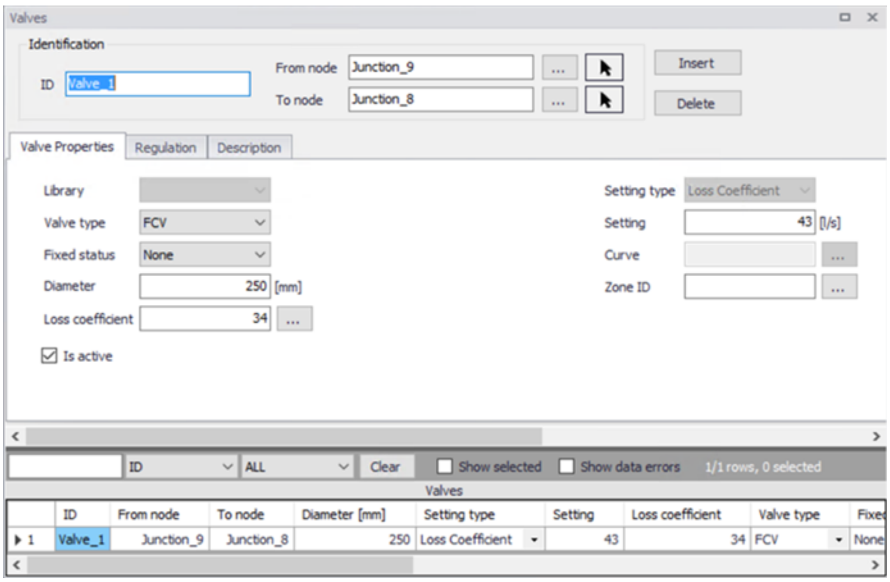


Figure 3.47 Valve properties Editor

<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 datebase. 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.

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 characteristics 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 a extended period simulation, and thereby get another head loss coefficient from the curve.

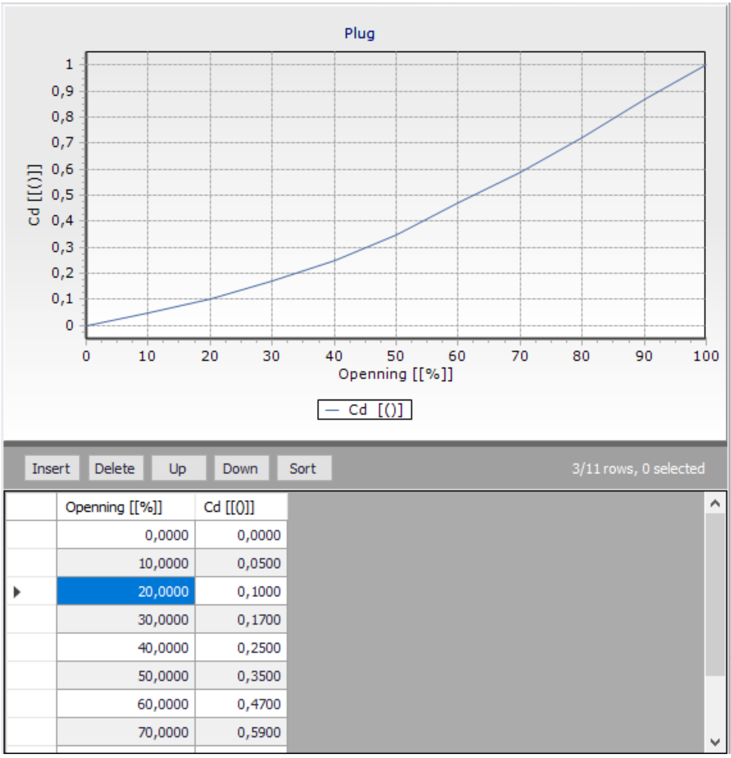


Figure 3.48

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.

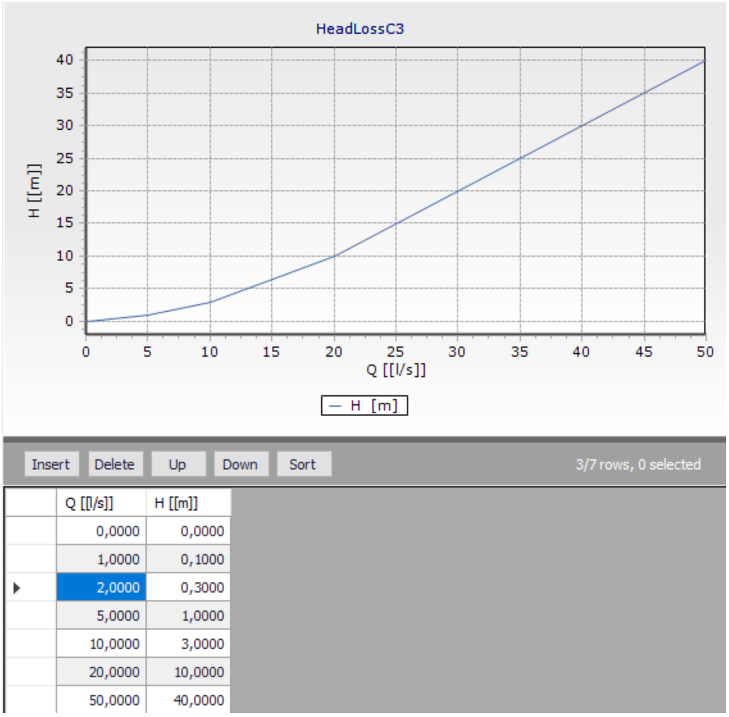


Figure 3.49

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 Valve. Two options for the valve setting are available:

- 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.

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 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 specifdied for a GPV valve. Both curve types are generated from Tables > Curves and relations.

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.

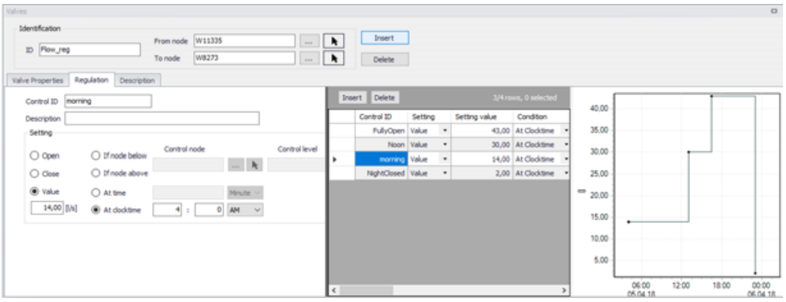


Figure 3.50 Regulation tab

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.

Description

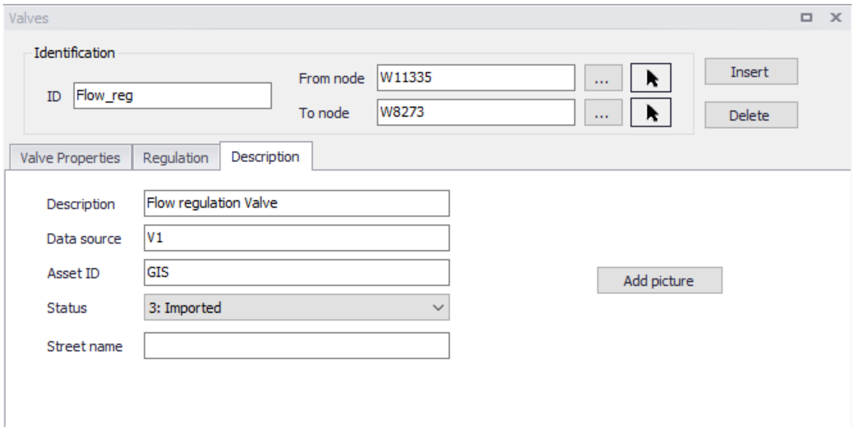


Figure 3.51

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	
Description	Description	Descriptive text	No	

Table 3.3 Valve attributes

Field	Database name	Description	Mandatory?	Default value
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 Editor

Pump stations (or pumps) are represented by “points“ in the MIKE+ WATER network model. Pumps are either defined interactively on the Map window using the <Drawing> tool, or by manual data entry using the Pumps Editor in Distribution Network. The Pump Stations Editor allows you to define the pump’s ID, description, and other attributes. The Pump Stations Editor is reached by double clicking <Pump Stations> in Distribution Network in

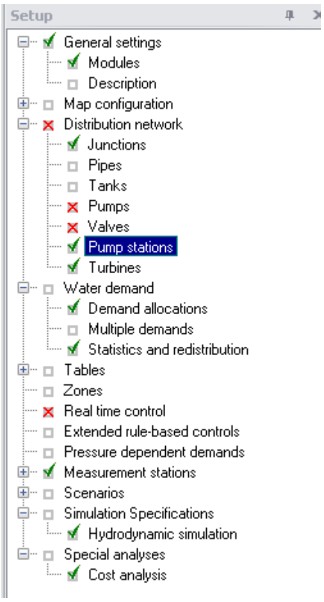


Figure 3.52 Layout of Setup Tree

Identification

This data entry is used to specify a corresponding asset pump ID, which uniquely identifies the pump in the asset management system (such as GIS, for example). See Figure 3.53

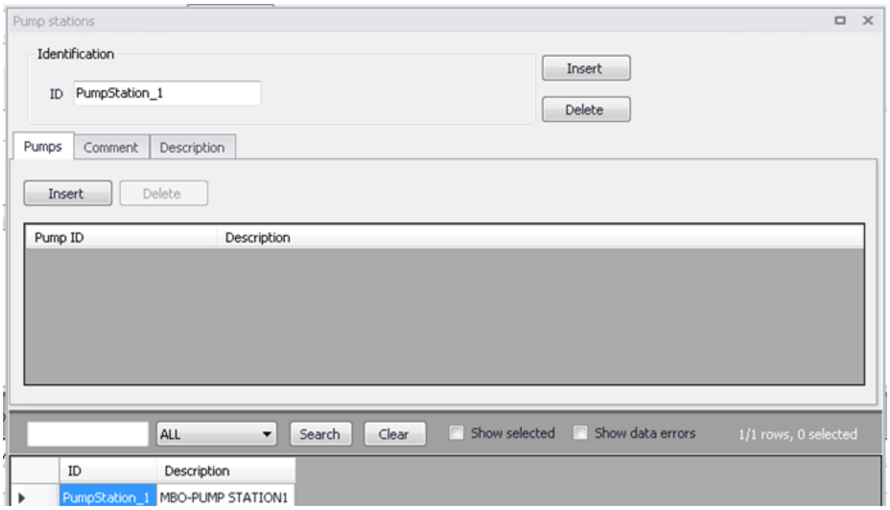


Figure 3.53 Layout of Pump Station Editor

<Insert> button allows users to create a new pump station and an asset ID will be automatically generated by default but editable.

<Delete> button allows users to delete already existing pump stations.

Grid Attribute

The grid at the bottom of Pump Station Editor shows the attribute tables of pump stations. There is a filter on top of the grid.

<Search> button allows users to filter pump station by typing pump station ID in the blank box. <Clear> button let users clear selection criterion.

<Show selected> check box allows to filter the selected rows in the grid.

<Show data error> check box allows to filter the rows where an error occurs.

By right clicking the header of each column, a list of following options can be selected from (Figure 1.3):

1. Select column
2. Field calculator
3. Select by attribute

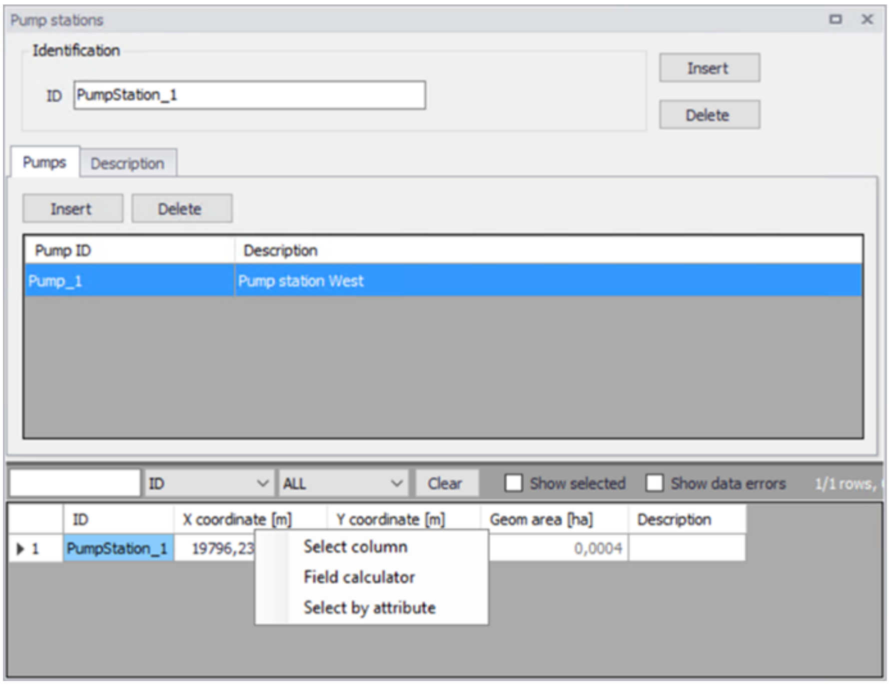


Figure 3.54 The Layout of Pump Stations Attribute

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 3.55), or by manual data entry using the Turbine Editor dialog box as shown in Figure 3.57

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 3.56).

Figure 3.55 The Turbine Editing Tool

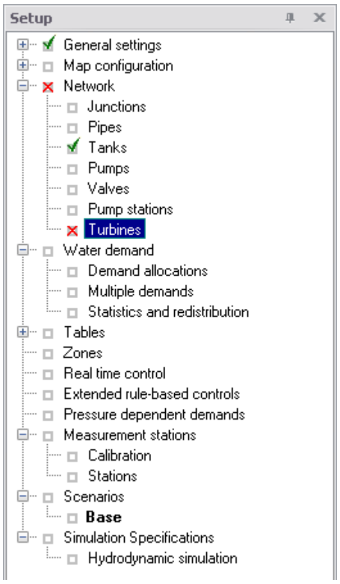


Figure 3.56 The Turbine editor in Setup Tree

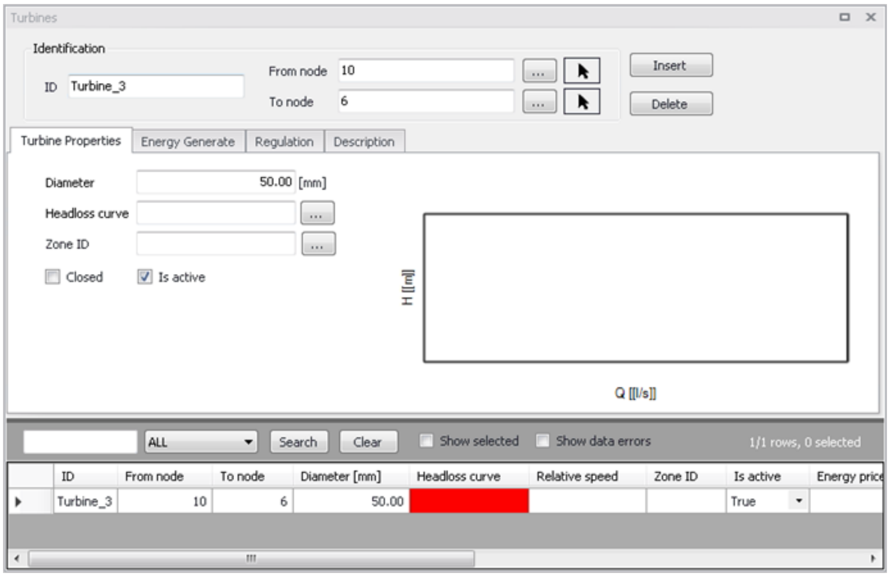


Figure 3.57 The Turbine editor allows the user to define the storage tank node that supply water to the water distribution network

A list of the Turbine editor data entries for Figure 3.56 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.

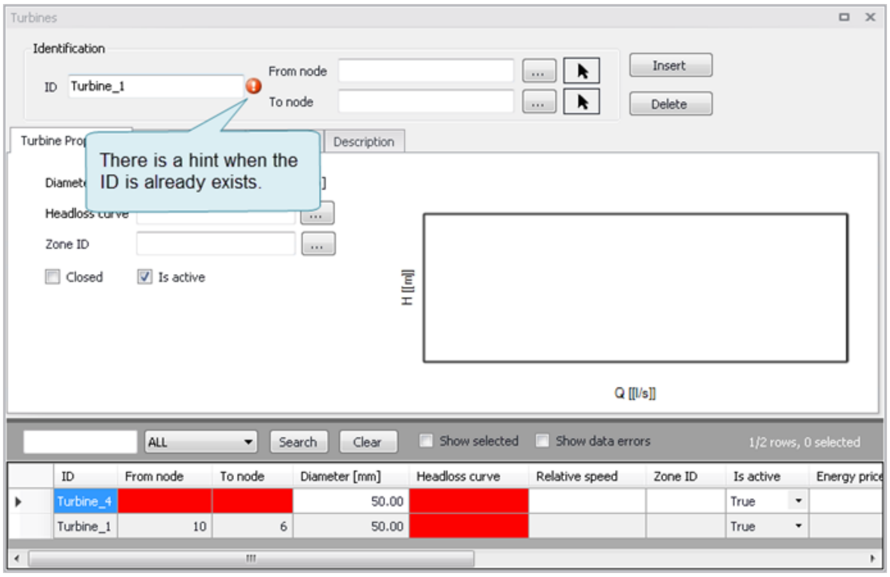


Figure 3.58 Hint when a turbine ID is repeated.

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.

Grid Attribute

The grid at the bottom of Turbine editor shows the attribute table of turbines. When selecting a turbine from the table, that turbine will be located on the network map with selected state. On the top of the grid, users can search and select turbines according to their attributes. It also gives options for clearing selection, showing selected turbine attributes and showing data errors.

Right click on the header of column, showing a menu with following items:

- Statistic
- Field calculator
- Select by expression

These functions should be all applied on the selection.

Right click on the grid origin, showing a menu of Hide columns: the fields of feature can be selected to be seen or hidden by users.

Right click on anywhere else origin, there is a pop up menu showing with following items:

- Clear selection
- Copy to clipboard
- Copy to clipboard with header text
- Paste from clipboard
- Reset layout
- Show columns in active tab
- Add user defined column

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 1.5.

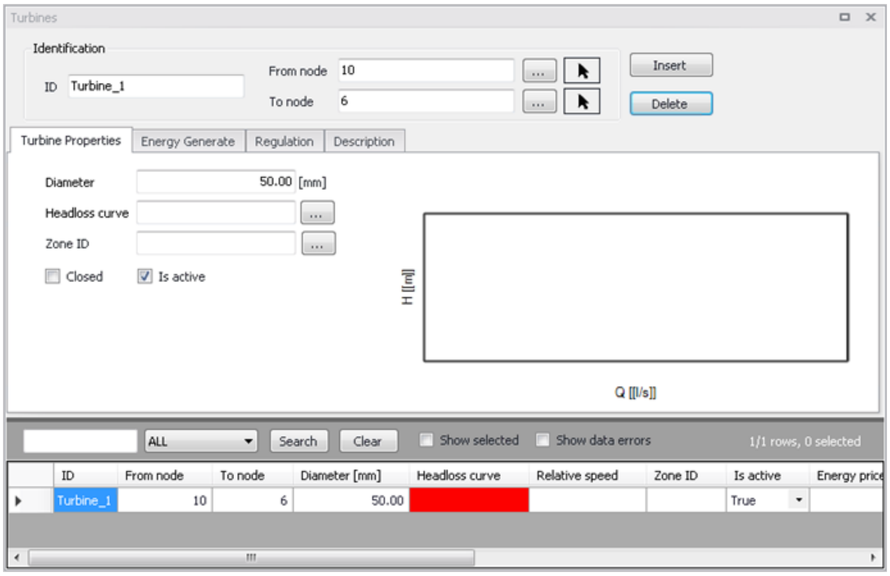


Figure 3.59 The General Information Turbine

Turbine Type

This drop down selection list data entry allows you to define the turbine types. There are two options available:

- Impulse Turbines
- Reaction Turbines

Diameter
This data entry allows you to define the Turbine size (ft or m).

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
It defines the zone ID of turbines.

IS Active (mandatory)
It defines whether the turbine is active or not. If the turbine is active, it would be included in the model, otherwise it would 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 3.60

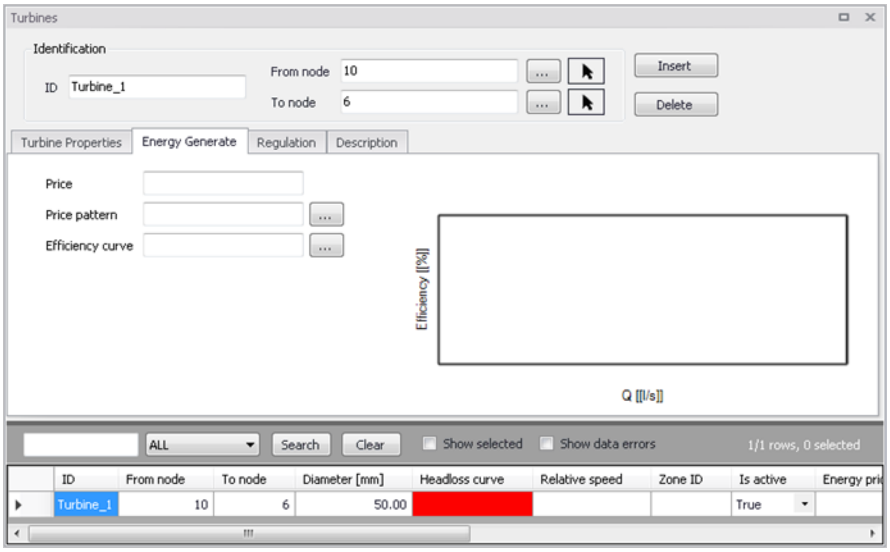


Figure 3.60 The Energy Generated of Turbines

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 3.61).

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.

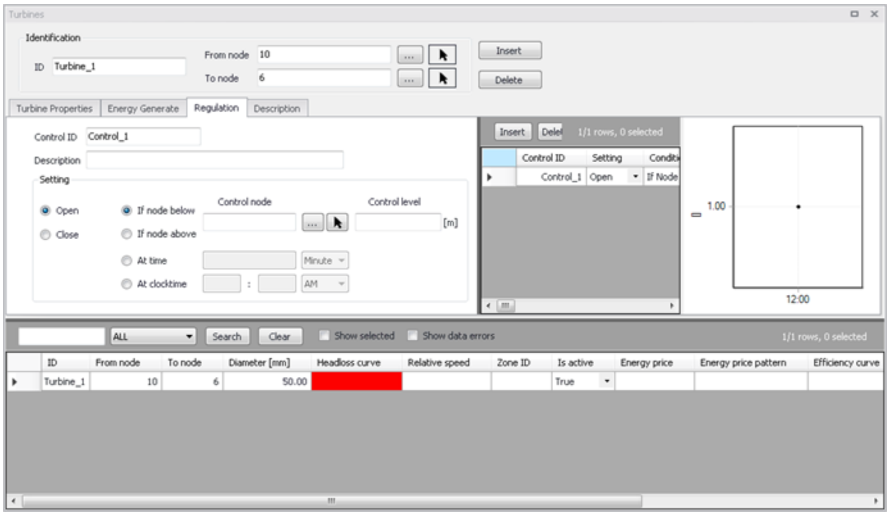


Figure 3.61 The Regulation of Turbines

A list of the data entries for Regulations follows:

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 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 1.8.

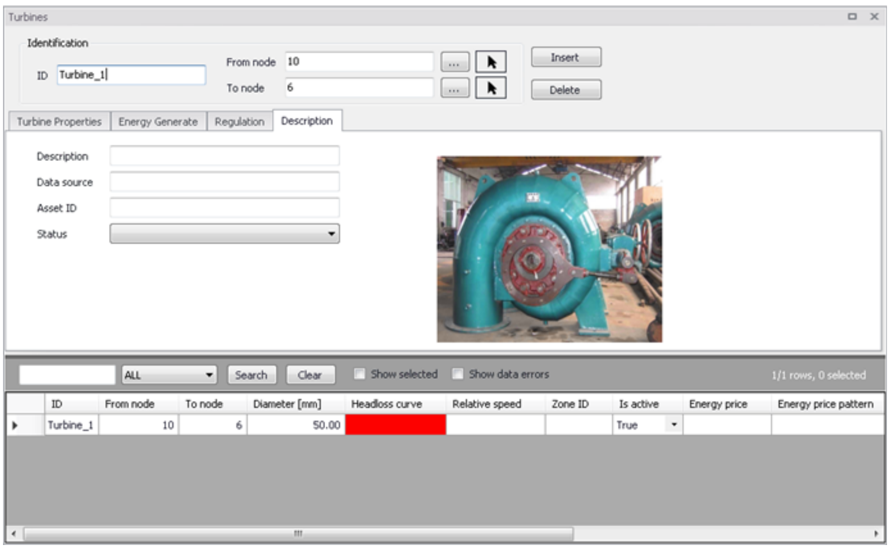


Figure 3.62 The Description of Turbines

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

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.

The Zones Tab in MIKE+, contains two groups of data is accessed through MIKE+ TOC. One for identification and another one to set the zone specifics.

In the Identification group it is possible to insert new zones and delete different zones. These zones can be defined as different zone types; DMA zone, Demand zone, Region zone, Ward and Other.

Identification ID

This data entry is used to define a unique positive integer ID value that specifies the network pressure zone. Pressure zones are defined at junction nodes, storage tanks and reservoirs. Note that, by default, all nodes are defined as belonging to pressure zone 1. Therefore, pressure zone 1 is always defined and not listed within the Pressure Zone Editor

Zone type

Zone type is used to identify the distributing demand by the kind of zone established. There are five types predefined in MIKE+: DMA zone (“0”), pressure zone(“1”), demand zone (“2”), region zone (“3”), and other (“4”). Per default the DMA zone is the initial status of this entry.

Range 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.

Description (Optional)

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

Generate

These buttons helps automating the creation of new zones. Two options are available:

- GIS layer: This method uses a polygon shape file to read the zones' extents from.
- Zone IDs from pipes' properties: With this option, the zones are created by grouping the pipes according to their 'Zone ID', as specified in the Pipes editor.

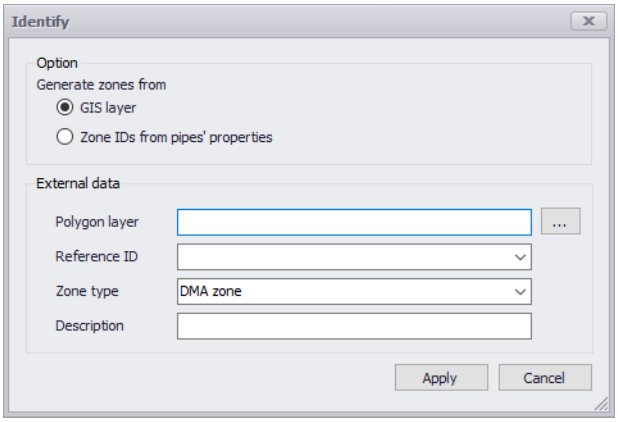


Figure 4.1 Creating new zones from a GIS layer or from pipes' properties

4.1.2 Demand Allocation

An alternative way to generate junction node demands can be developed based on geocoding 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 geocoded it based on X, Y geographical coordinates.

The Demand Allocation Editor box is reached directly from the TOC under the Water demand layer group.

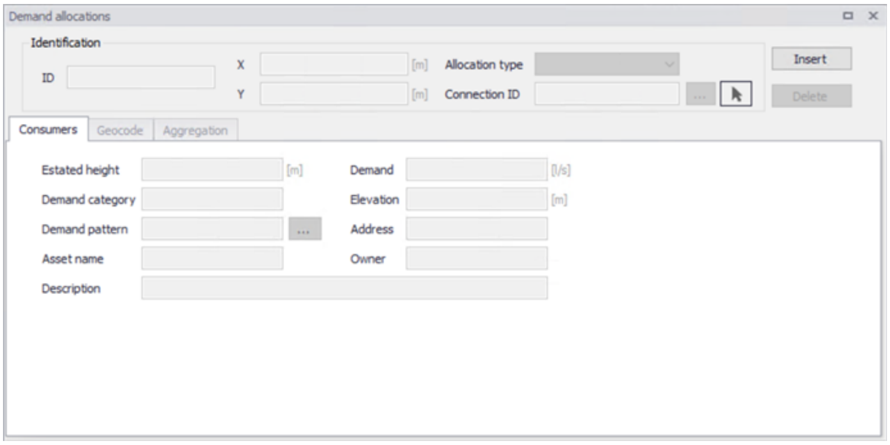


Figure 4.2 Demand Allocation Editor

(Fig 4.1 Demand Allocation Editor dialog box is used to store and edit the consumption data defined as consumption points and link it to the appropriate network junctions or pipes)

A list of the Demand Allocation Editor data entries for Figure 4.2 follows, with a short description of each entry.

Identification 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.

Allocation type

In this data entry the user can specify if the connected demand is loaded to either a junction or a pipe.

Consumers tab

In this data group the user can specify all the information relevant to the demand allocation point.

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.

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.

Metered Demand

This data entry stores data imported by the user. The consumption database often stores aggregated data over several months. The user can import this into “Metered consumption” and then obtain pipe demand coefficient for demand distribution.

Geocode tab

Demand geocoding allows you to connect customers to model nodes. This connection will then be used to develop node demands (demand aggregation). Customers (allocation points) can be connected to nodes by several different methods:

- Finding the nearest node
- Finding the nearest pipe and then the nearest node on that pipe
- By a pipe ID defined in the customer point and then to the nearest node on that pipe.

The selection of the method depends on the nature of how demand allocation points (consumers) were developed and what they represent.

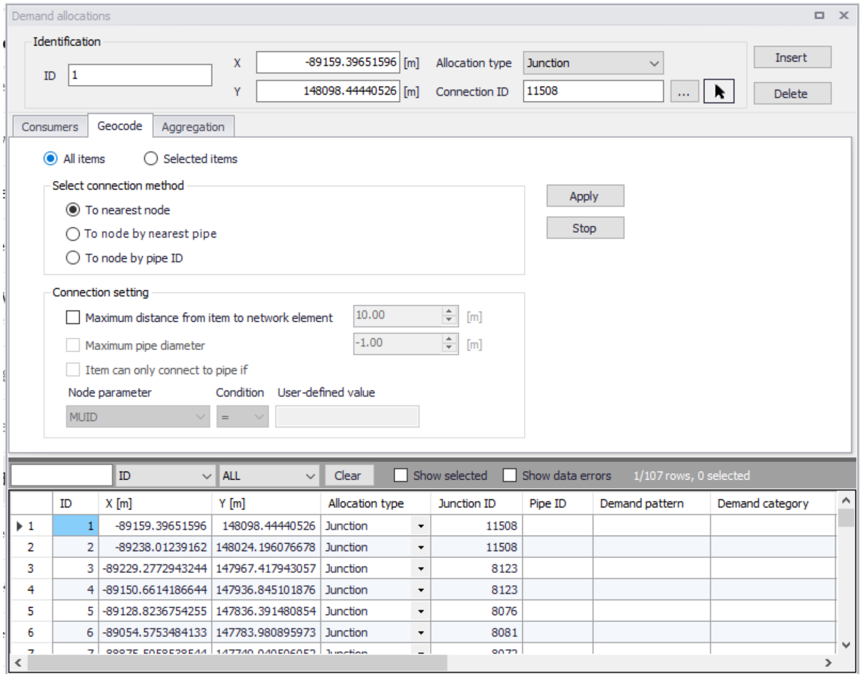


Figure 4.3 Demand geocoding

It is possible to use all records (all consumers) or only selected ones (e.g. new consumers) using the radio buttons at the top:

- All items
- Selected items

Select connection method:

- To nearest node: the program will find the nearest node to the customer point
- To nearest node by nearest pipe: the program will first find the nearest pipe and then it will use the nearest node to the customer point on that pipe
- To node by pipe ID: the program will find the nearest node on a pipe that has the same ID as defined for the customer point. This method is suitable when customers are already "geocoded" to pipes in the source data.

Connection settings can be used in order to restrict pipes that will be used for the customer allocation:

- Maximum distance from item to network element: this is the maximum distance between the customer point and the node (to nearest node method) or the nearest pipe (to nearest node by nearest pipe method)

- Maximum pipe diameter: this is the maximum pipe diameter of the nearest pipe. If the nearest pipe diameter is too big the program will skip the pipe and will keep searching for another pipe. This is suitable in order not to allocate demand points to main or transmission pipes.
- Item can only connect to node if the node meets certain criteria, e.g. "node description = xxx"
  - Node parameter: select the node attribute
  - Condition: select the condition (<, =, >, LIKE)
  - User-defined value: specify the attribute value
- Item can only connect to pipe if the pipe meets certain criteria, e.g. "pipe description = xxx"
  - Pipe parameter: select the pipe attribute
  - Condition: select the condition (<, =, >, LIKE)
  - User-defined value: specify the attribute value

Apply: starts the demand geocoding process.

Stop: cancels the running demand geocoding process.

Aggregation tab

Demand aggregation allows you to use customer connections (geocoding) to model nodes 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

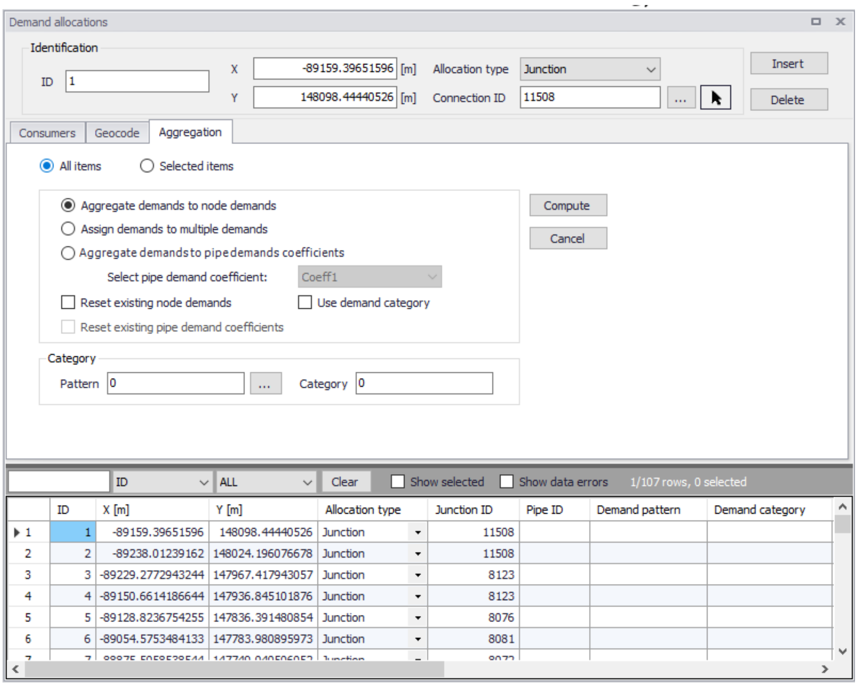


Figure 4.4 Demand aggregation

It is possible to use all records (all consumers) or only selected ones (e.g. new consumers), using radio buttons at the top:

- All items
- Selected items

Select aggregation method:

- 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

Reset existing node demands: the program will remove all existing multiple node demands.

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. 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).

Pattern: if this field is not empty, the program will use the pattern name for multiple demands that will be created in this process.

Category: if this field is not empty, the program will use the category name for multiple demands that will be created in this process.

Compute: starts the demand aggregation/assignment process.

Stop: cancels the running demand aggregation/assignment process.

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 box is reached by MIKE+ | Multiple Demands.

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.

4.1.4 Statistics and redistribution

MIKE+ can generate statistical information for junction node demands. Demand statistics is generated for each pressure zone as well as for the complete network. Additionally, demand statistics dialog box allows the user to redistribute node demands by changing the calculated statistical results.

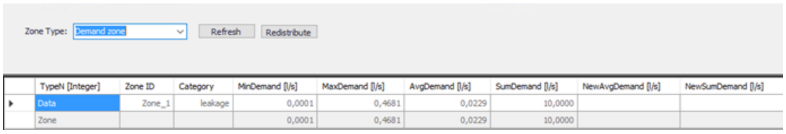


Figure 4.5 Statistics and Distribution

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

Type

This data entry 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.

Category Type

This option entry allows the user to specify the category type such as *fixed* or *scaled*, which will be used in the demand redistribute process. Junction node demand with fixed demand category will not be scaled during the demand redistribution.

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 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 MIKE+ Tool.

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 IFigure 4.8 is used to automatically assign the demands at the appropriate junction nodes. The Distributed Demands tool box is reached thorough the WD toolbox in the Tools Section.

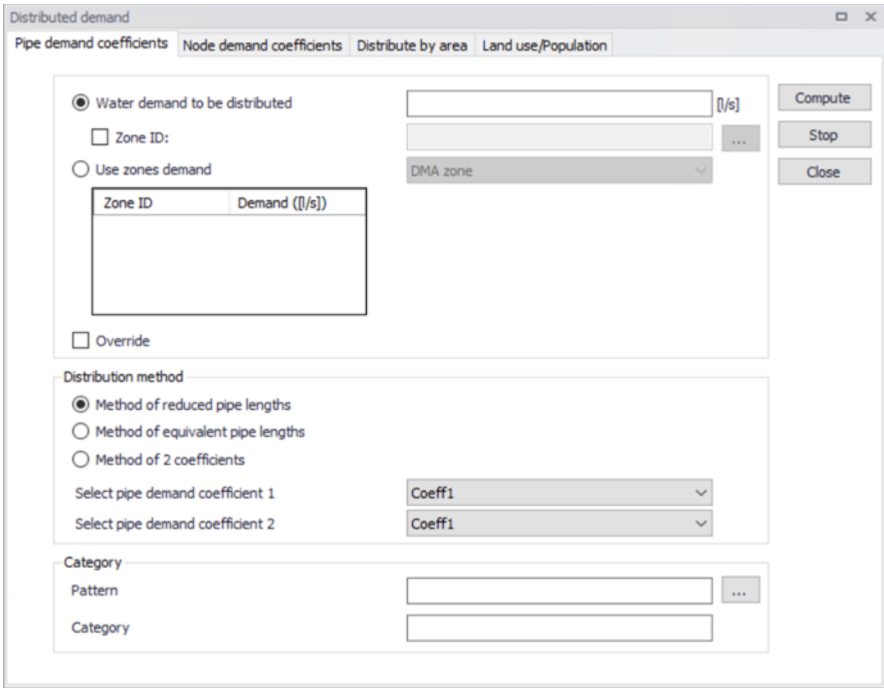


Figure 4.6 Distributed Demands Pipe Demands Coefficients

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

Water Demand to be distributed (mandatory)

This data entry 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.

Zone ID

There are two methods for distributing the water demands, distribute by zone ID (zone by zone) or using the zone table (one-time process).

The Zone ID check box allows the user to select whether the total network demand corresponds to the entire network or a single zone. Checking this box applies the specified water demand to a single zone, leaving the box unchecked will apply the specified water demand to the entire water distribution network.

The zone must be specified in the provided data entry field. Selecting <<...>> displays the Zone ID selection in the dialog box, where the appropriate Zone ID can be selected.

The “Use zones demand” will enable the use of the zone table by this means the user can define the total demand of each zone type.

The Override check box when checked will clear the demand in the junctions first before distributing. By default this check box is marked.

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:

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

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:

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

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:

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

where:

qpi = Total water demand applied to the pipe, split between the two end nodes.

Q = total network water

li = Pipe length

k1i, K2i = pipe demand coefficient

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.

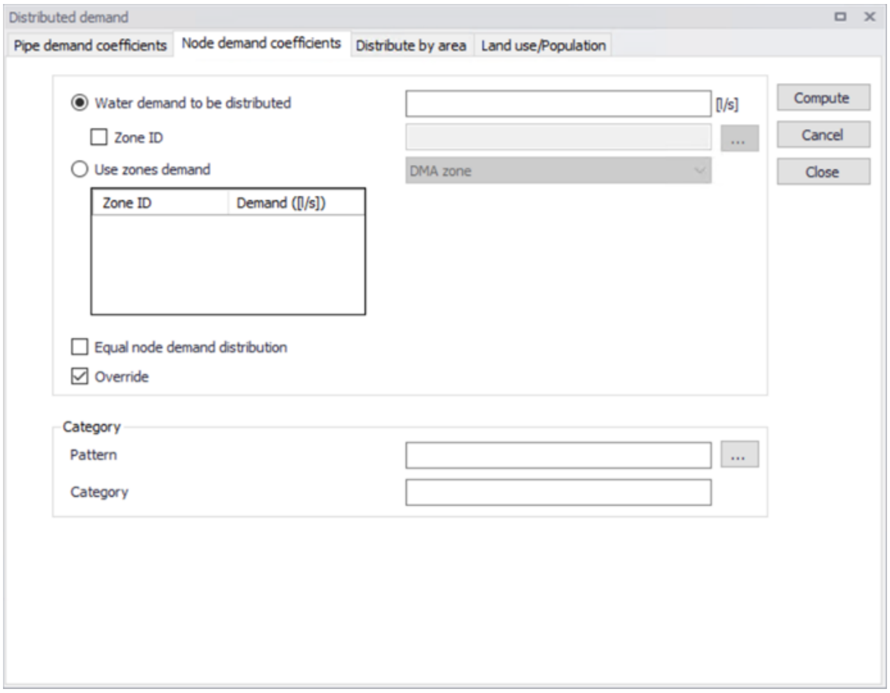


Figure 4.7 Node Demands Coefficients

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. Similarly to Pipe Demand Coefficients there are two methods of distributing the demand: zone by zone or distribute by the zone table.

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

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

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.

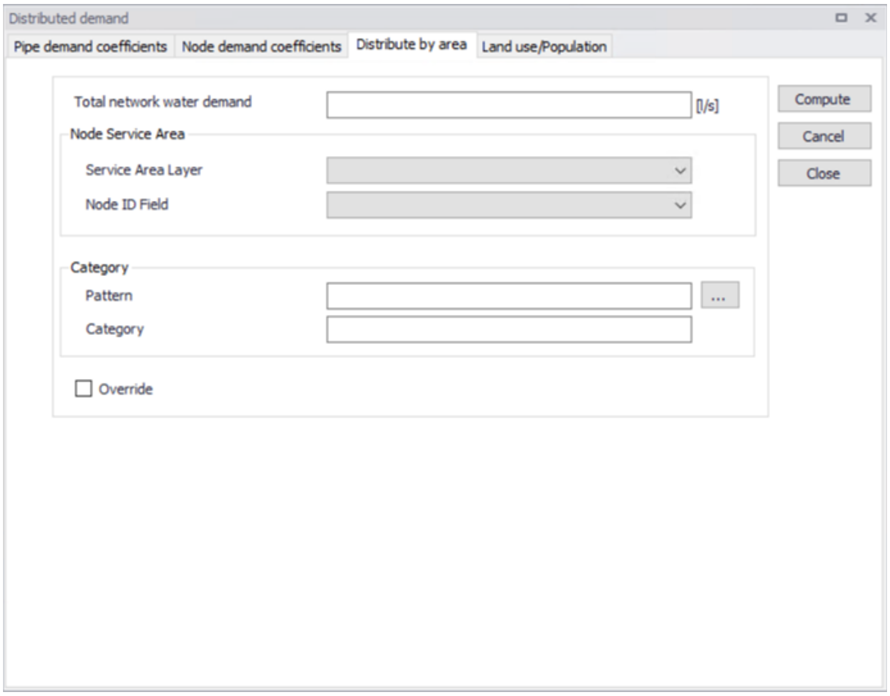


Figure 4.8 Distributed Demands Area

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.

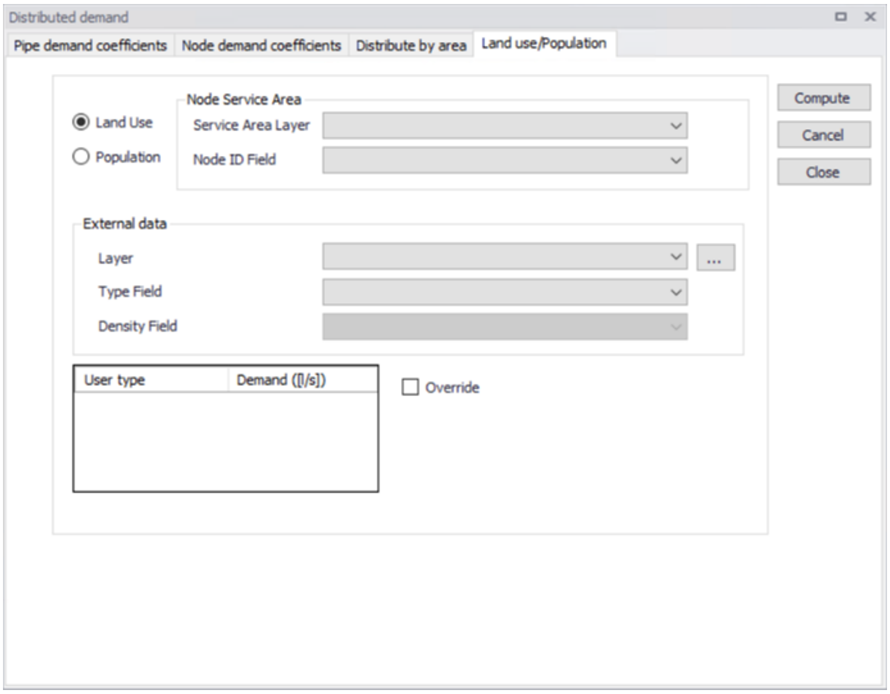


Figure 4.9 Distributed Demands Land use

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.

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.

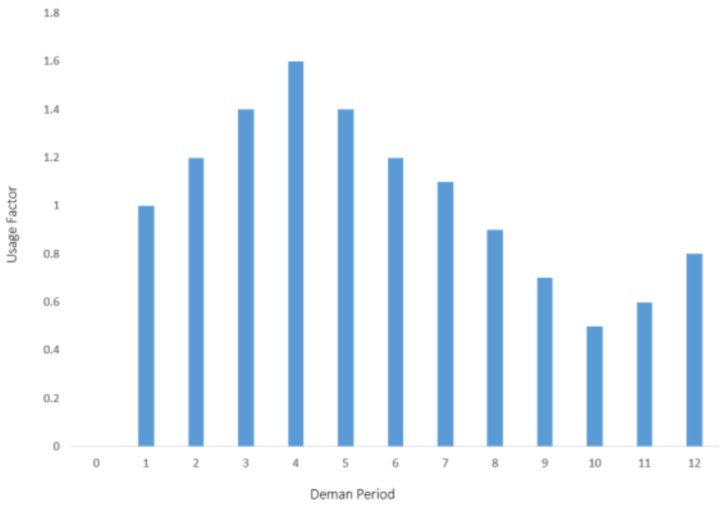


Figure 5.1 Time pattern for water usage

The definition of repetitive profiles consists of two main steps; the definition of diurnal profiles and the definition of cyclic profiles, which are combining one or more profiles together.

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 duration of such diurnal profile curve is one day (24 hours). The diurnal profiles settings can be accessed from the Patterns tab.

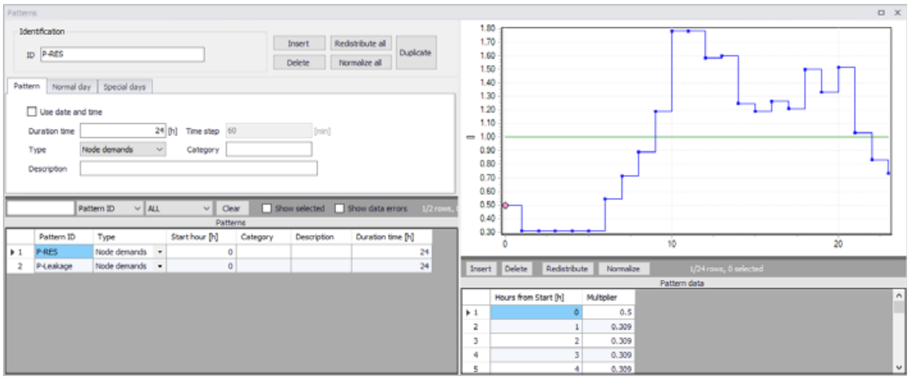


Figure 5.2 Diurnal Pattern

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.

Time Step

The pattern time 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 hydrodynamic simulation settings.

Figure 5.3 Pattern time step in Hydrodynamic simulation settings

Duration time

The duration is required to specify the extension of the diurnal pattern. The duration time and time step, decides the number of records in the pattern data table.

Use data and time sets whether the absolute date and time will be used for each and every multiplier. If it is set as true then the column “Date and time” will be enabled and must be defined in the subsequent field.

Type

The user can define the type of diurnal pattern, the options are *Node demands, Water quality, Tank water, Energy Price* or *Undefined.*

Category

This data enty allows you to enter a description identifying the demand pattern being defined. This description can optionally excluded of the reports.

Description

This data enty 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. This description can optionally be included in the reports generated.

Redistribute

Select redistribute in case that the pattern time step was changed in the simulation settings and you want to adjust the pattern time steps (number of patterns) to match the pattern time step. Note, that this will not the pattern values (multipliers) but it will change their count.

Normalize

Select normalize in case that you want to adjust the pattern multipliers to the average value of “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 time (only for Water Hammer Analysis)
- Valve Characteristics Kv. Valve flow coefficient Kv versus time (only for Water Hammer Analysis)
- 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).

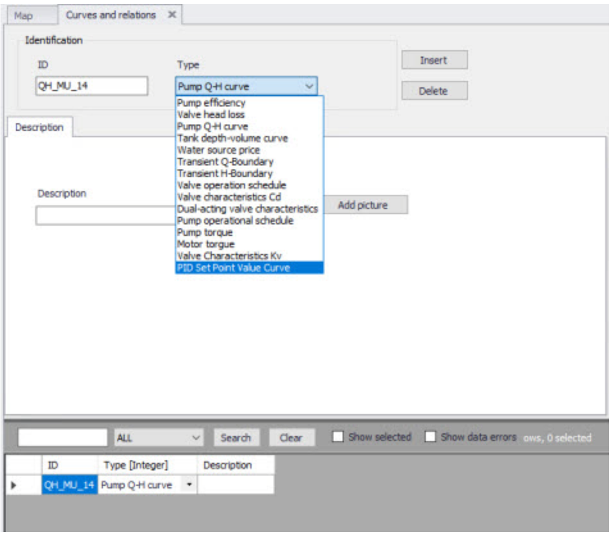
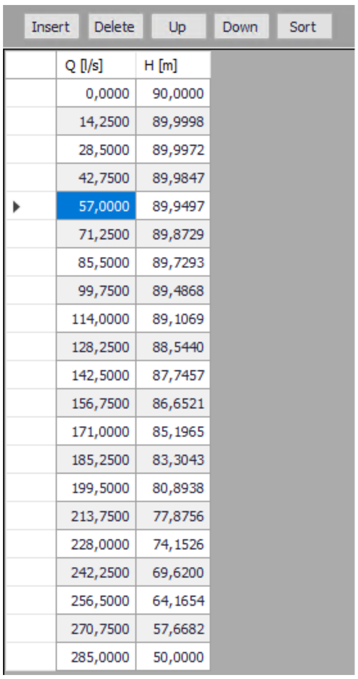


Figure 5.4 Curve and Relations, Identification

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

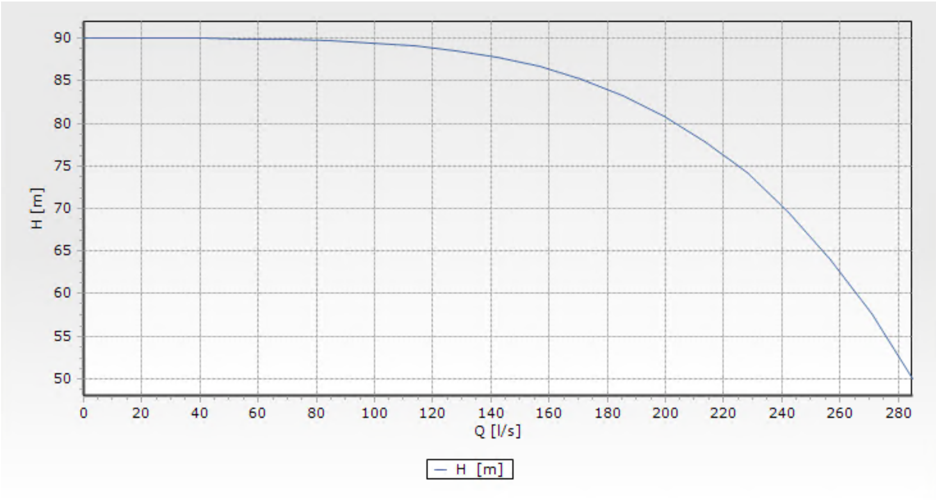


Figure 5.6 Preview of a defined curve in Curve Editor

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 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 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.

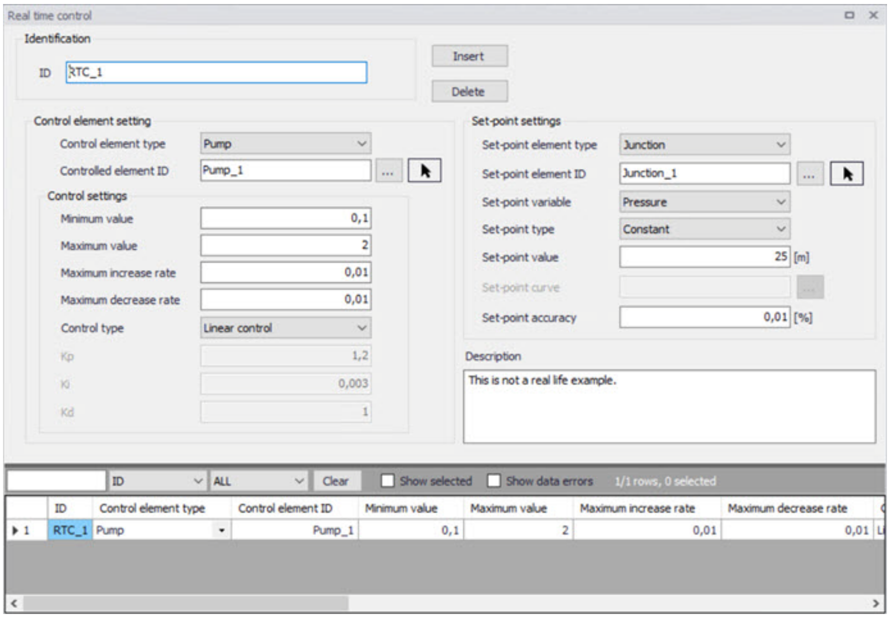


Figure 6.1 The Real Time Control dialog box is used to the analysis parameters

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.

Valve type

In case of a TCV valve control element, this data entry allows you to define the type of a valve, such as butterfly valve, globe valve, plug valve, ball valve, fast opening valve, or equal percentage valve. The program will apply a built-in valve characteristics curve based on the valve type.

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%

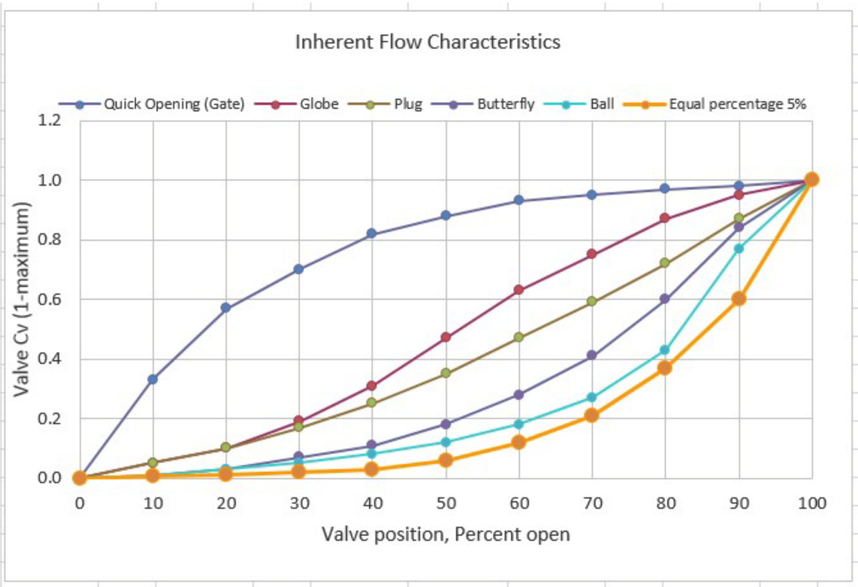


Figure 6.2 Default valve characteristics

7 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.

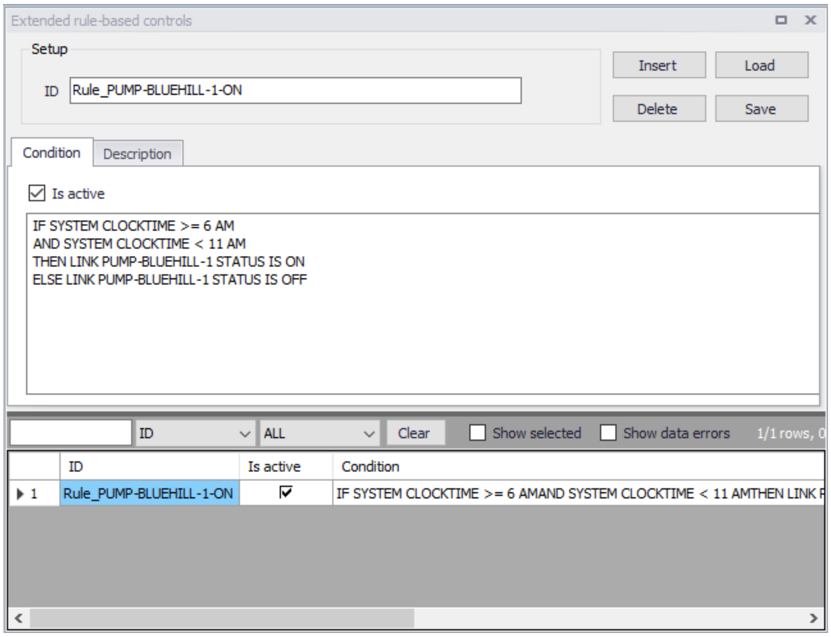


Figure 7.1 The Extended rule-based control editor

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.

7.1 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.

7.1.1 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
<=> =

7.1.2 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.

7.2 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:

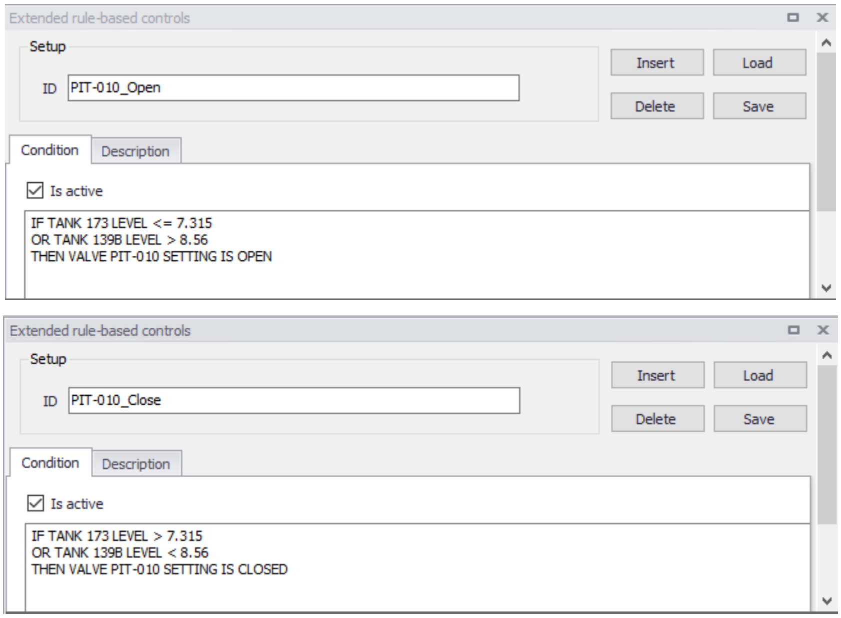
IF PUMP 3 SETTING LIKE PUMP 4
THEN …

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

7.3 Controls Examples

7.3.1 Control of a valve

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



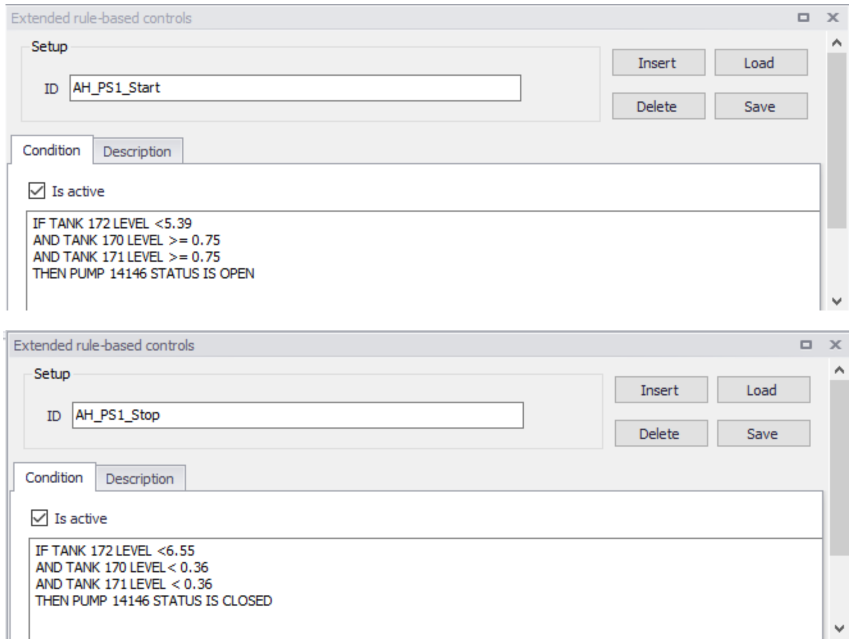

```
RULE   PIT-010_Open;    BUNKER RD NORTH CV OPEN
IF TANK 173 LEVEL <= 7.315
OR TANK 139B LEVEL > 8.56
THEN VALVE PIT-010 SETTING IS OPEN

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 7.2 Rules in EPANET *.inp file

7.3.2 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 7.3 Rules in EPANET *.INP file

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

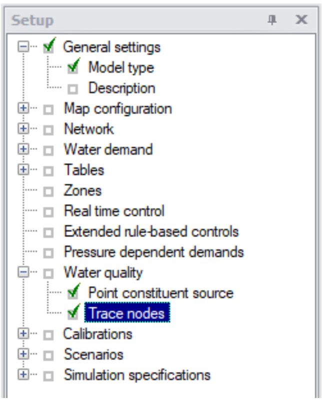


Figure 8.1 Water Quality setup tree layout

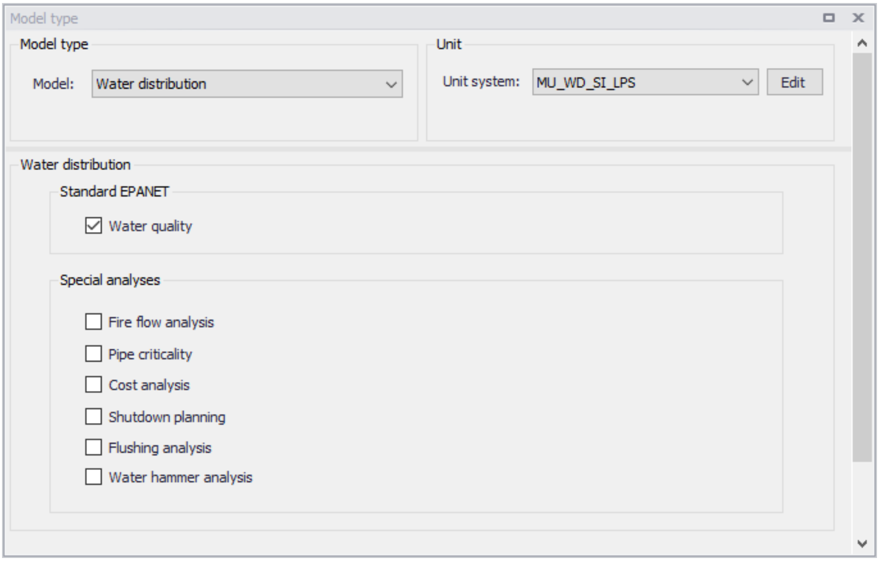


Figure 8.2 Layout of Models

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 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.

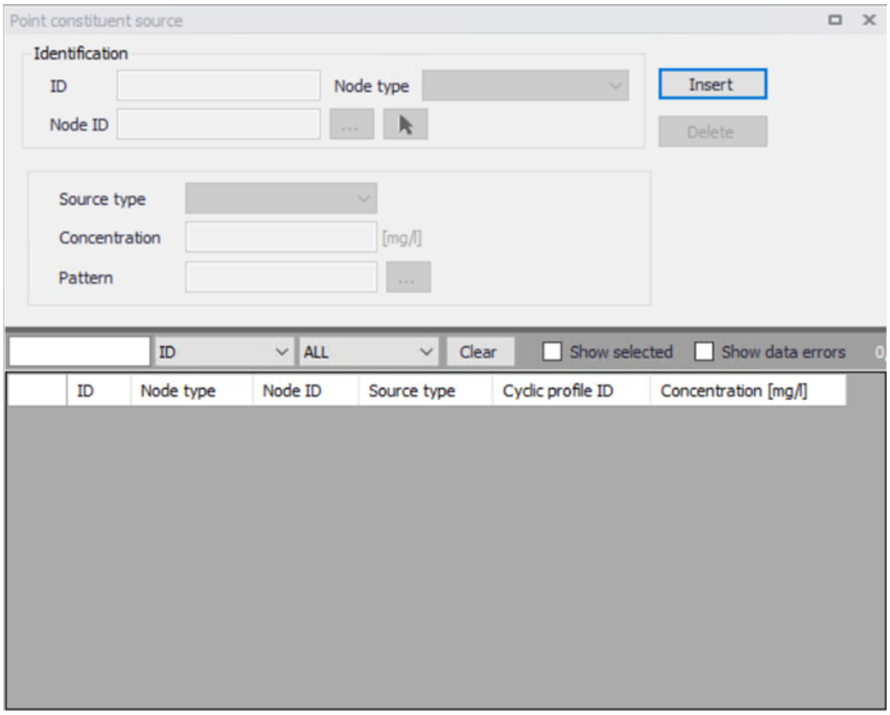


Figure 8.3 Layout of Point Constituent Source

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

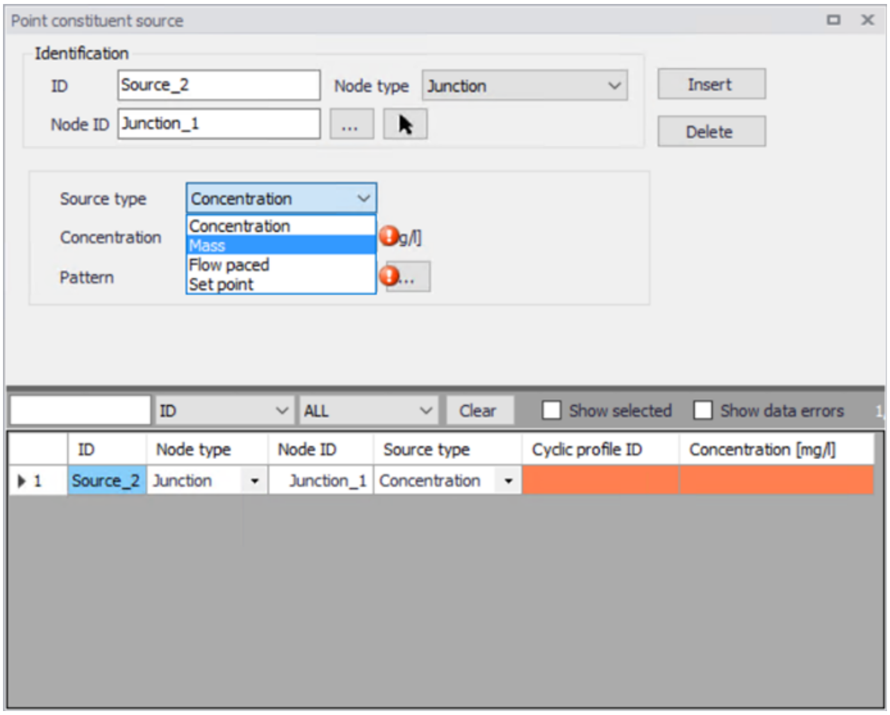


Figure 8.4 Source Type Options

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.

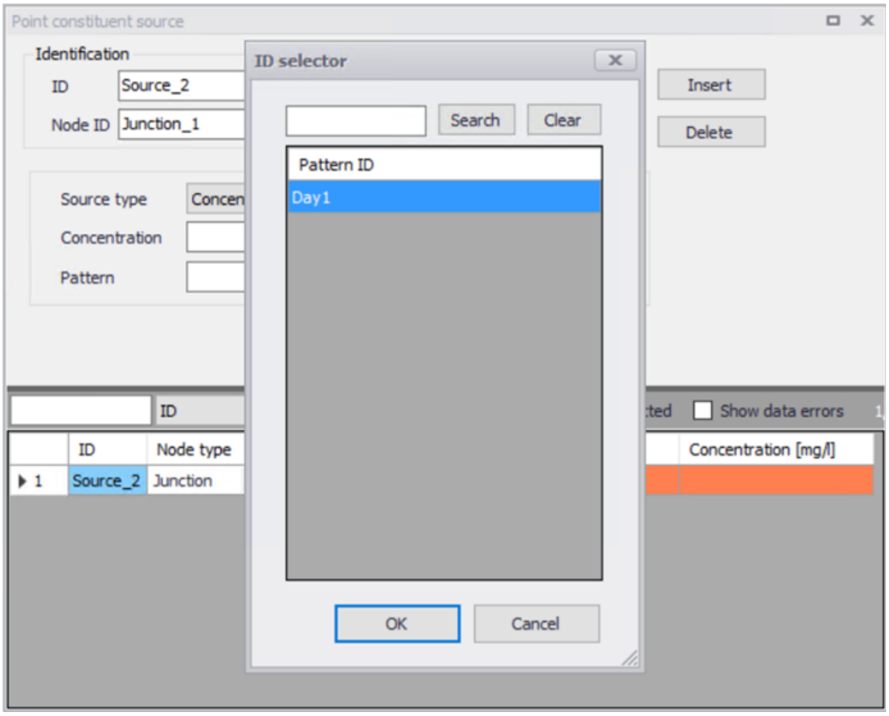


Figure 8.5 ID Selector Window

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.

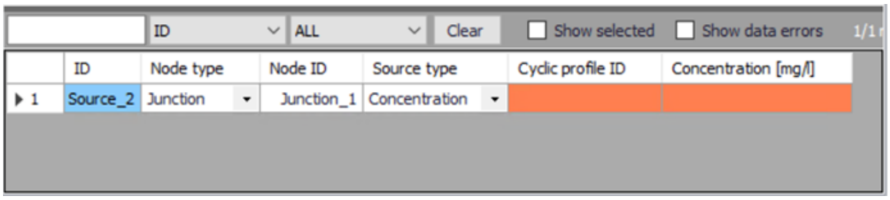


Figure 8.6 Detail Table of Point Constituent Source Settings

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.

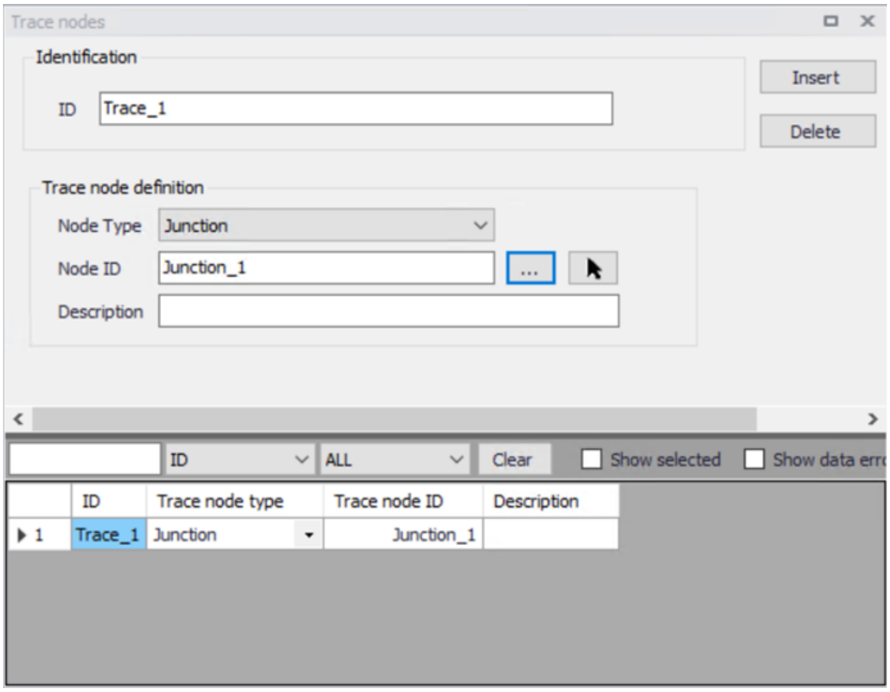


Figure 8.7 Point constituent source editor

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:

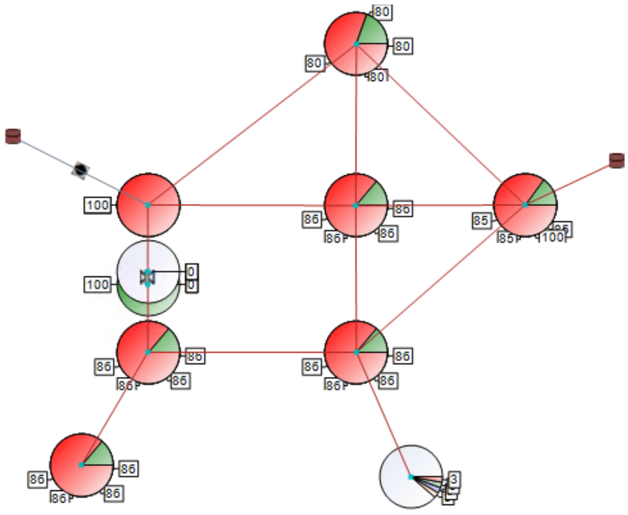


Figure 8.8 Multiple source blending (detail from the supply zone, colour shows water from different water sources)

9 Scenarios

The water distribution and wastewater collection data models are commonly used for the system performance analysis and in the planning process. The complexity of the involved systems, the various uncertainties about the future conditions and usually huge costs associated with maintenance, rehabilitation and development necessitate a thorough investigation of alternative system configurations in a search for the technically feasible, environmentally sound and economically efficient solution. These alternative configurations - scenarios - may differ by the system's physical layout, loading conditions, operational strategies, etc. Various projects, such as development of a Sewerage Master Plan, Wastewater Transportation Strategy, an Overflow Abatement Strategy, etc. would typically result in a large number of scenarios, either representing alternative system configurations at a given time and/or representing the system at various development stages. Test of each scenario against the prescribed legislation or the standards of service that the authorities provide requires a numerical model on its own.

These scenarios are always related to each other through the common origin ('Base') and the differences typically epitomize a smaller part of the total data. Moreover, scenarios representing a development of the system through time are subject to the dependencies propagating along with the timeline. Analysis of the scenarios as separate projects creates major inconveniences, such as:

- Large number of models, even when differences between them are mino
- Missing the efficient overview over the entire set of solutions
- Inability to maintain the existing dependencies between the individual scenarios automatically. Thus, the updating of the models with additional information requires editing of multiple files to change the same element, e.g. if a pipe diameter is found to have been incorrectly registered in the GIS database, it will have to be updated multiple times in each of the scenario project files.

9.1 What is Scenarios

The MIKE+ Scenarios is a user interface for a set of MIKE+ features, enabling the definition, organisation management and reporting of alternative scenarios, such as:

- Augmentation of existing trunk sewer mains;
- Increased wastewater loading from increased population;
- Increased water demands from increased population;
- Alternative design loads, e.g. rainfall-runoff of different return period;
- Alternative alignment of sewer and storm mains;

- Building of a new sewer trunk and water supply mains in order to cater for a new development area.

within the same MIKE+ project.

9.2 Design of the MIKE+ Scenario Manager

The MIKE+ Scenario group is based on the concept of Data Groups, Alternatives and Scenarios. In this context, a Data Group is a set of database tables which form a meaningful set. E.g. all database tables containing collection system network data belong to the data group "Network Data". Every database table relevant for the scenario manager is included in one of the Data Groups.

Each Data Group can appear in the MIKE+ project in any number of Alternatives. The initial alternative is named with a default name 'Base'. Any further alternative is created upon user request and gets a user-specified name. The Alternatives for a certain data group are organised in a tree-like structure, where dependencies propagate along the branches -from the "parent" to all the "heirs" i.e. "children".

A scenario represents a complete set of consistent data, featuring the system configuration for a given situation. In other words, a scenario contains one alternative of each Data Group. Actually, individual alternatives are used as building blocks for constructing scenarios. A moderate number of data groups (eight for collection system and nine for water distribution) allows for a manageable structure of scenarios, while ensuring the high level of flexibility.

The initial scenario is named with a default name 'Base', and consists of the 'Base' alternative of each data group. Any further scenario is created upon user request and gets a user-specified name. The scenarios are organised in a tree-like structure of "parents" and "children".

Alternatives

The alternatives represent the components of the scenarios. The various alternatives contain the actual data belonging to a certain data group. Actually, each subsequent alternative only contains the information on the differences relative to its immediate "parent", while the rest of data is inherited from the "parent" through the principles of inheritance.

Grouping of various alternatives belonging to different data groups into scenarios is sometimes subject to limitations, because the data groups have not been formed on the basis of data independency, but rather following the logical data grouping. E.g. the alternative of the "Catchment connections" CS data group, which specifies a catchment connection to node 'A', cannot be used with the alternative of the "Network data" data group where node 'A' has been renamed or deleted. Obviously, the catchment would remain disconnected.

Collection System Alternatives

For Collection Systems, the scenarios are composed of the following data groups:

- Network
- Loads and Boundary
- Catchment & Hydrological
- WQ
- Operational RTC
- LTS
- Profiles and Curves

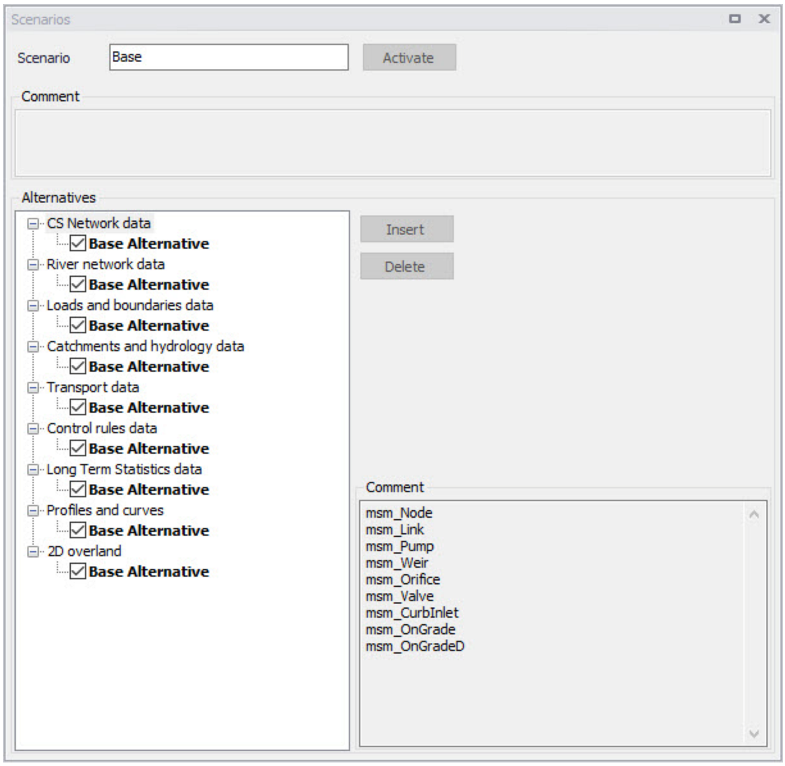


Figure 9.1 Scenarios Alternatives for Collection System

Water Distribution Alternatives

For Water Distribution Systems input data can be grouped the following way, corresponding to the different types of available alternatives:

- Networkt

- Water Demands
- Control
- Water Quality
- Pattern and Curves

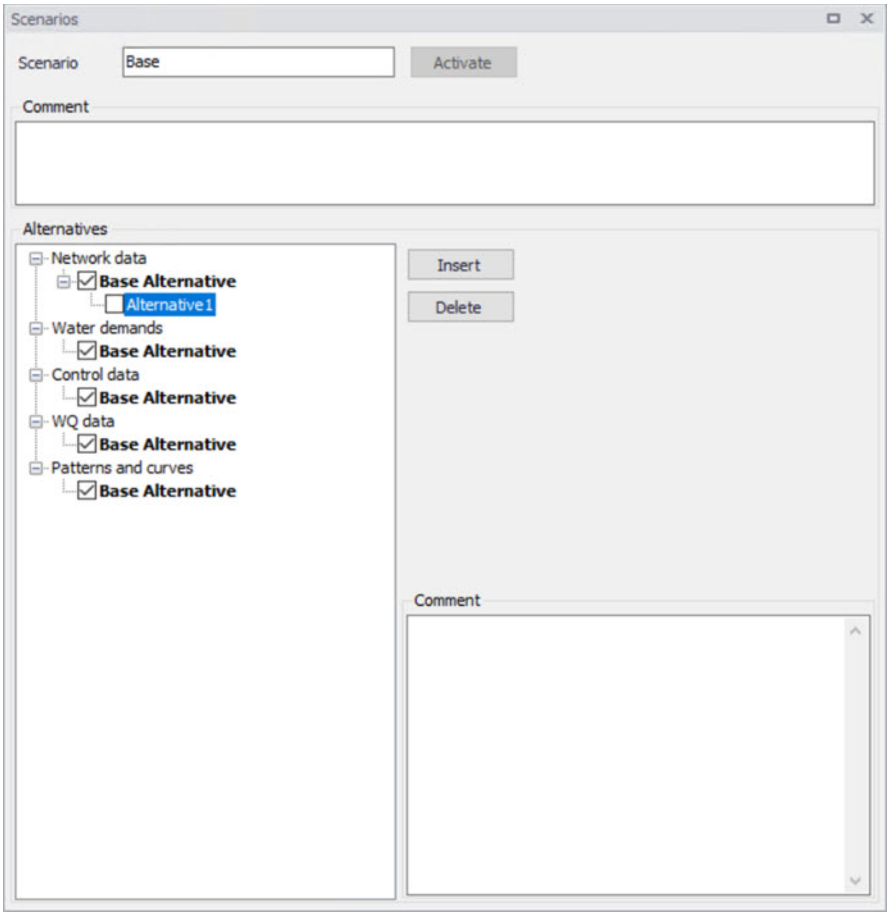


Figure 9.2 Scenarios Alternatives for Water Distribution

Inheritance principles

With the inheritance from 'parent' alternatives to 'child' alternatives, some specific items must be kept in mind.

- Making a change to an alternative will affect all descendent ('child') alternatives of that alternative. This means that it will impact all the scenarios where either the alternative or the children of that alternative are applied. This also ensures that if one value needs updating it will be updated in all the scenarios where the alternative is applied (e.g. if a pipe diameter is found to have been incorrectly registered in the GIS data during the course of a project then the pipe diameter can be changed one place only, regardless of the number of scenarios and alternatives that reference to this alternative).
- The chain of inheritance for a certain data record stops where any change (or delete) of that element has occurred in earlier work. E.g. if a bottom level of a node 'A' has been edited in some child alternative, some later update of the bottom level in 'Base' will only propagate through the alternative tree until the alternative containing the old change.
- Adding an element (e.g. a node) in the 'parent' with an ID that already exists in one or more of its descendants ('children') will overwrite the content of the 'child' element
- If adding an element (e.g. pump/link) in the parent that cannot be added to all the children (because some parts may have been deleted/changed there), the element is added where possible and omitted elsewhere.

9.3 Managing Scenarios and Alternatives

The Scenario Manager has two parts:

- The Scenario part
- The Alternatives part

The scenario part is for creating, editing and managing scenarios, while the alternatives part is for creating, editing and managing alternatives.

Under Scenarios in the tree view one can “right-click” to get the option of creating a new child scenario.

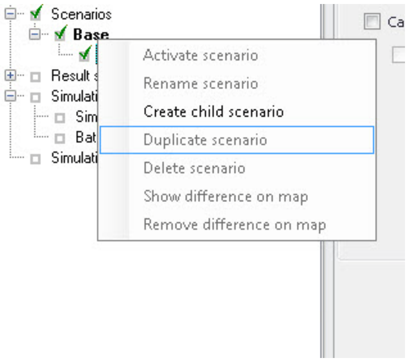


Figure 9.3 Create a new child scenario

Right-clicking on a child scenario enables the options to activate, rename, duplicate, delete, create a new child scenario and visualize the differences in the map.

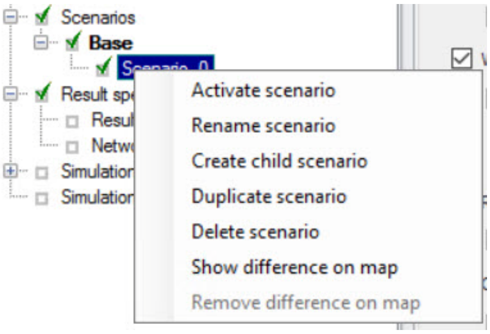


Figure 9.4 Management of child scenarios

Scenarios

The scenario part is used for creating, editing, and managing scenarios. Per default there will one built-in scenario, i.e. the Base scenario. The Base scenario cannot be edited or deleted. An unlimited number of additional scenarios can then be added to cover the various 'What if' scenarios.

Create a Child Scenario,

This option adds a scenario that is a child of the highlighted (not to be confused with the active/current scenario), i.e. to begin with the alternatives of a new scenario will be that of the highlighted scenario. A name for the new scenario is suggested by default. The name can be changed through the option of Rename scenario or by editing directly in the ID field in the Scenario editor.

Remove,

The remove option will delete the highlighted scenario. The Base scenario cannot be deleted. Note that deleting a scenario will not delete any data as the alternatives hold the data (the scenarios just refer to alternatives). The comments for the scenario being deleted, however, will also be deleted.

Rename,
The rename option will make the scenario name active so it can be easily renamed.

Activate,
The activate button will load the scenario, i.e. the project data is manipulated so that all editors contain the appropriate data. Depending on the size of the project this may take some time.

Alternatives
Alternatives can be edited only if the appropriate scenario is made active. Alternatives can, however, be added regardless of the active scenario. When a scenario is loaded, the project data is manipulated so that all editors contain the appropriate data.

Alternatives can be activated, inserted and deleted in the editor through the visual buttons.

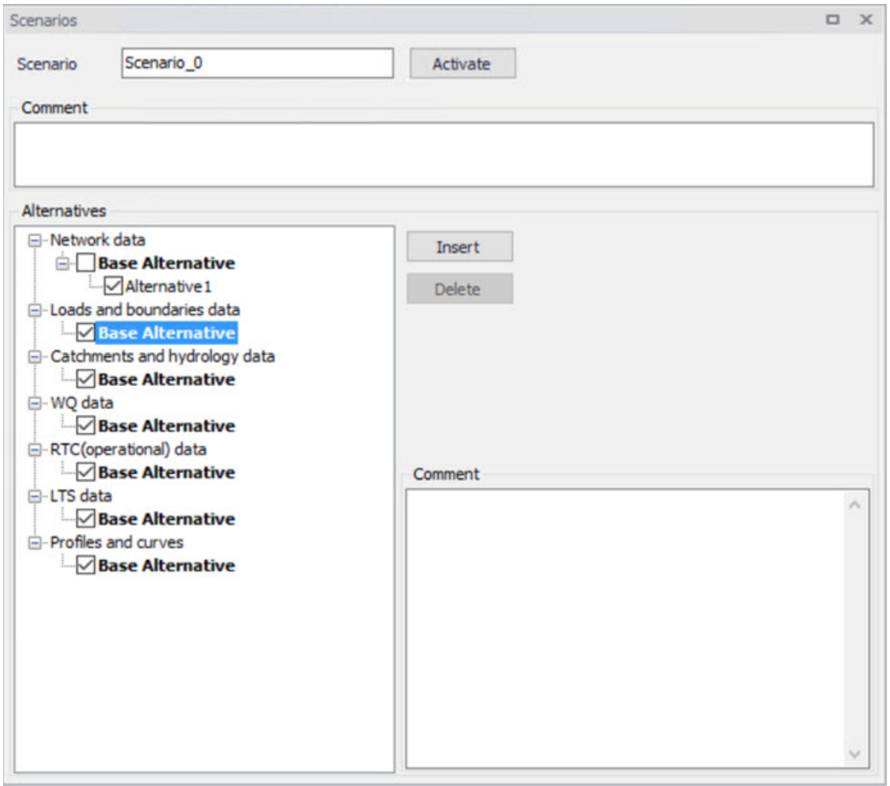


Figure 9.5 Alternative options

9.4 How to Start Working with Scenarios

Creating alternatives and scenarios

- Right click on Scenario in the Tree view
- Create a child scenario
- Select the alternative group that you wish to add an alternative to and

press the ‘Insert’ button

- You can now rename it and/or continue to make alternatives
- Select the alternative and click on Activate

The activated scenario is displayed in bold font. Equally, all the alternatives belonging to the active scenario, are displayed bolded in the 'Alternatives' page.

Both the data tables, the graphics and any MIKE+ tools work ONLY with the data belonging to the currently active scenario. Access to the data belonging to other scenarios is not possible through MIKE+ interface.

10 Simulation Specifications

10.1 Hydrodynamic Simulation

There are six primary tabs in Hydrodynamic Simulation 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 and in ID section, a new session name can be revised (the default name is Project _X). Alternatively, “Delete” button allows to delete the existing simulation projects. This feature is highlighted in Figure 10.1.

The table under tabs shows the project simulation details including ID, hydraulic parameters, etc. The parameters can be revised by double clicking the editable cells. Once revised, these settings are automatically saved to individual projects. To exit editing mode, press Enter or click somewhere else.

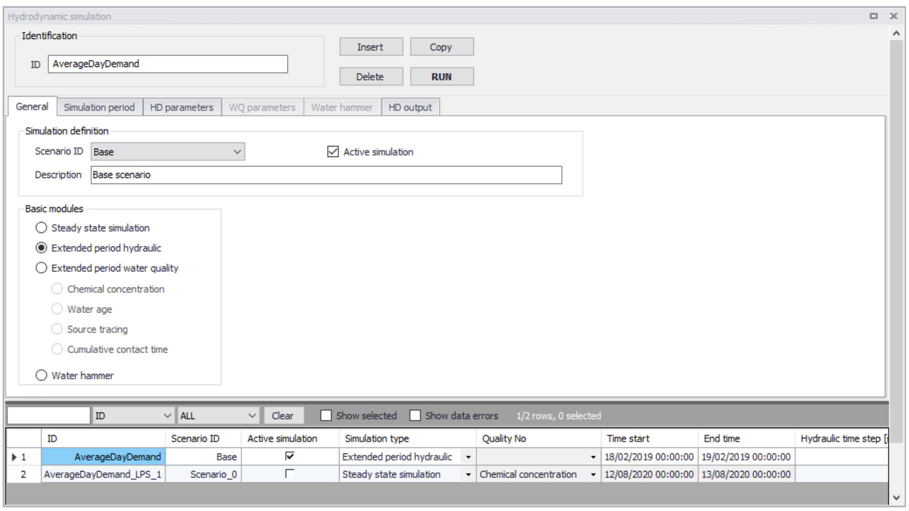


Figure 10.1 Layout of Hydrodynamic Simulation Editor

General

Under the General setting tab:

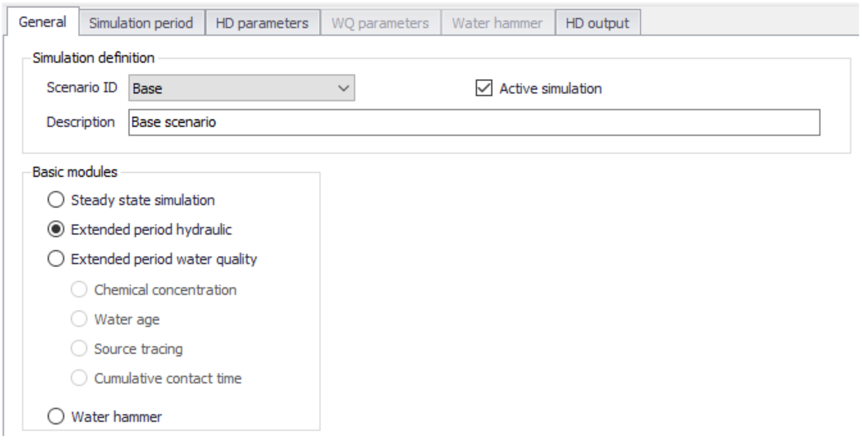


Figure 10.2 The General setting tab.

Simulation definition

Simulation ID: it is Base by default.

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

Active simulation: a mandatory check field, necessary to indicate which simulation will be used when running the simulation accessed from the “Run” ribbon.

Basic modules

There are 4 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).
- Water hammer

Water quality model is normally disabled but users can enable it by double clicking ‘Model type’ under General settings and selecting “Water quality” in Distribution System. Figure 10.3 and Figure 10.4 illustrate the layout of Setup tree and Module options.

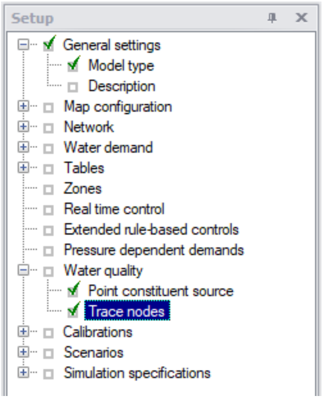


Figure 10.3 Layout of Setup Tree

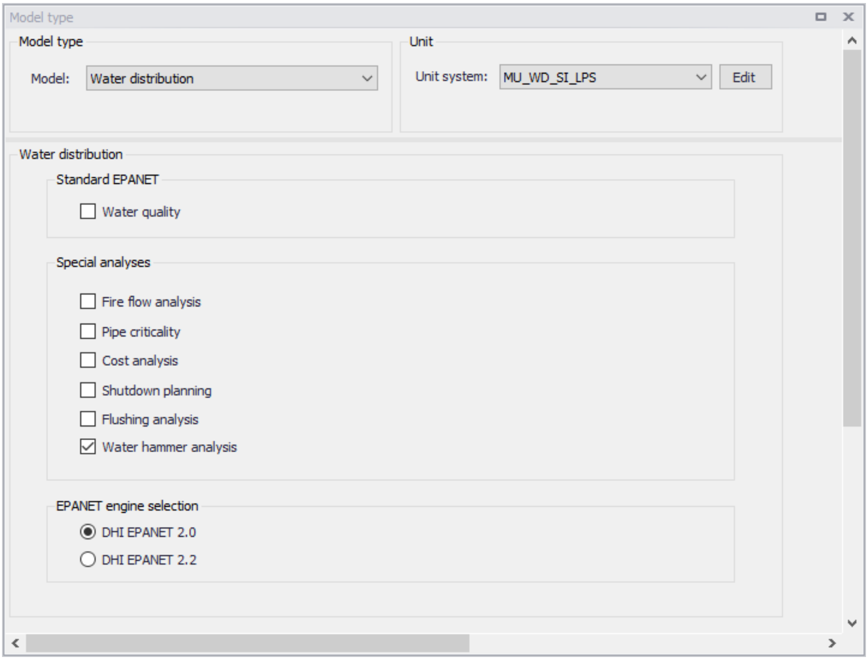


Figure 10.4 Layout of Model type editor

The following buttons are also located at the top of the editor with the Identification group:

'Copy' button

Duplicates an existing (currently active) simulation setup record.

'RUN' button

Triggers export of the current simulation job and execution of the simulation.

10.1.1 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.

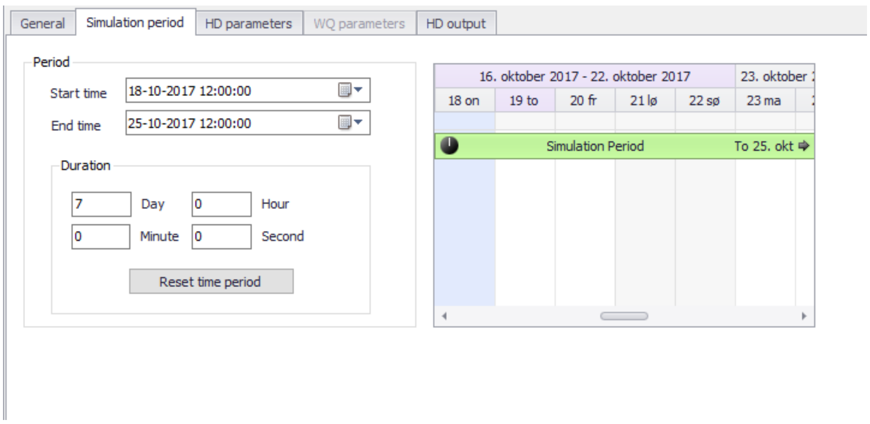


Figure 10.5 Layout of Simulation Period Settings Tab

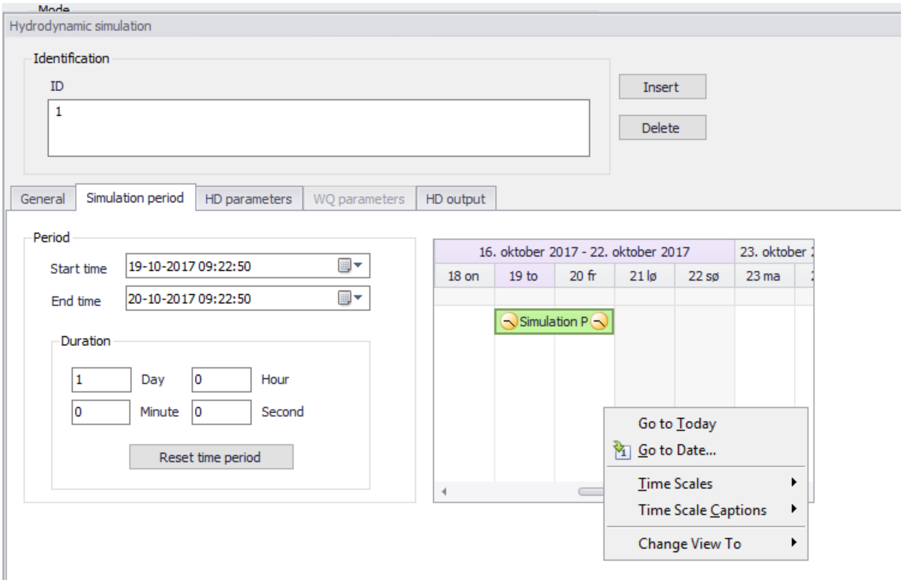


Figure 10.6 Layout of Gantt Chart View Option in Simulation Period tab

10.1.2 HD parameters

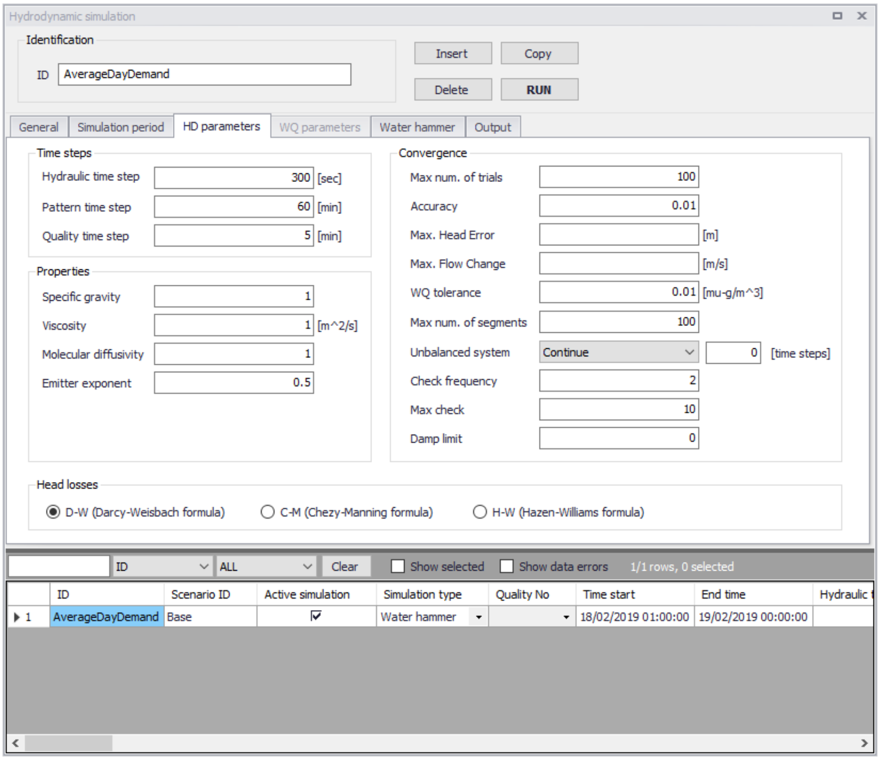


Figure 10.7 Layout of HD Parameters Tab

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.

Head Losses

This section specifies 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.

It has three choices:

D-W: Darcy-Weisbach formula

C-H: Chezy-Manning formula

H-M: Hazen-Williams formula

Each formula has its corresponding roughness coefficient.

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.

10.1.3 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 Setup Tree.

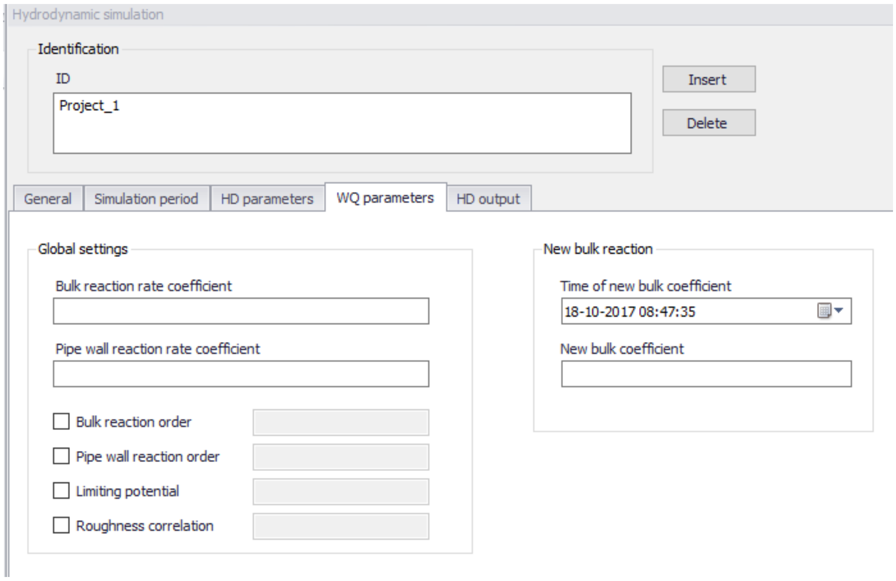


Figure 10.8 Layout of WQ Parameters Tab

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.

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.

10.1.4 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.

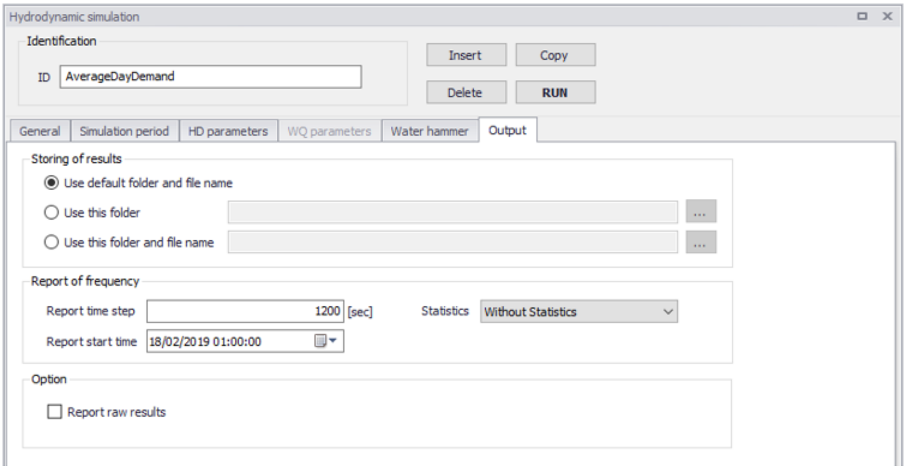


Figure 10.9 Layout of Output Setup

10.2 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.

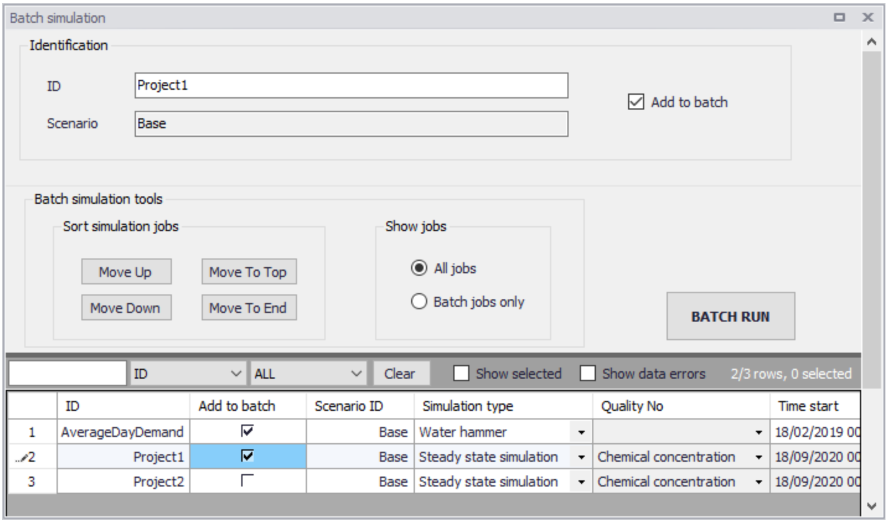


Figure 10.10 The Batch Simulation Editor

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 10.1 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
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.

11 Advanced Analysis

11.1 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 two 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.

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. 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.

11.1.1 Setup

A list of the Fire Flow Analysis dialog box data entries for Figure 11.1 follows, with a short description given for each entry.

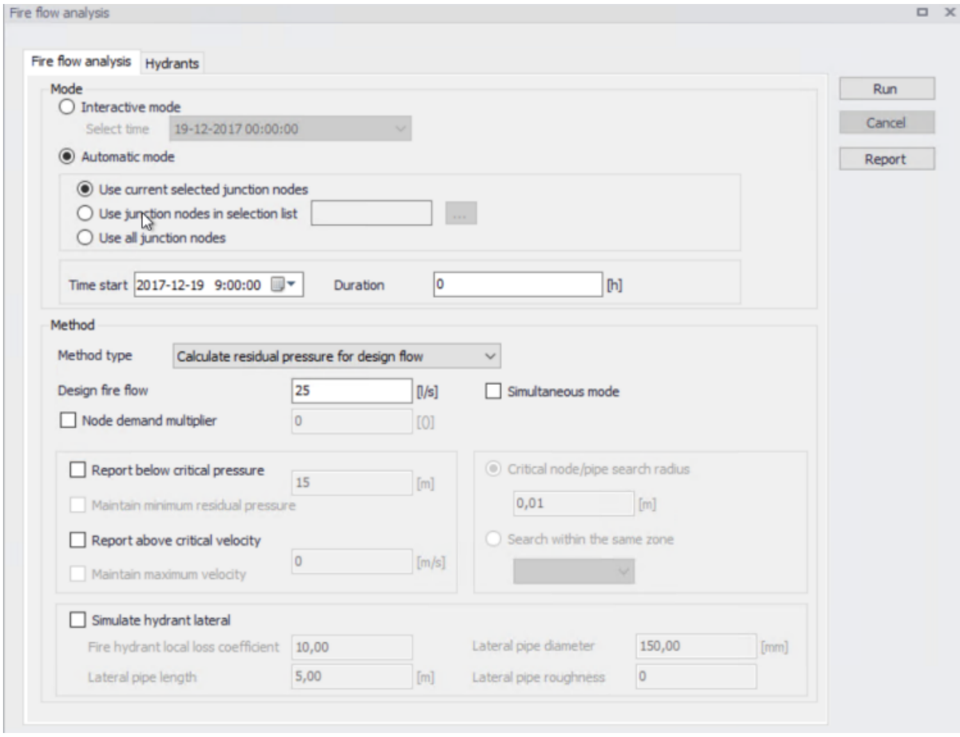


Figure 11.1 The Fire Flow dialog box is used to specify fire flow analysis parameters

Interactive mode

This radio button selection allows you to select the fire flow analysis mode. It is possible to run the analysis in automatic mode for number of selected nodes or it is possible to run the fire flow simulation interactively for a specific node.

Select time

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.

A list of the Interactive fire flow analysis dialog box data entries for Figure 11.2 follows, with a short description given for each entry.

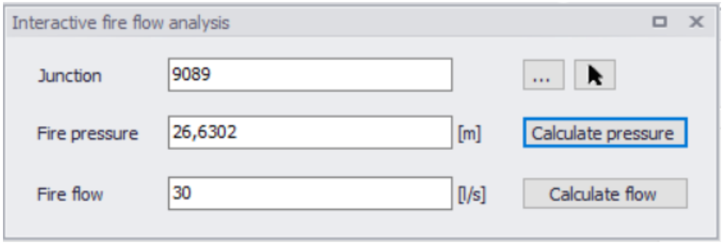


Figure 11.2 The Fire Flow dialog box used in Interactive simulation

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 selecting "↑" button.

Fire pressure

This data entry allows you to define the required design pressure that will be used in the calculation when you select "Calculate flow" or this field will display the computed residual pressure when you click "Calculate pressure".

Fire flow

This data entry allows you to define the required design flow that will be used in the calculation when you select "Calculate pressure" or this field will display the computed fire flow when you click "Calculate flow".

Calculate pressure

This button will allow you to compute the residual pressure for a design fire flow.

Calculate flow

This button will allow you to compute the available fire flow for a design pressure.

Automatic mode

This radio button selection allows you to run the analysis in automatic mode for number of selected nodes.

Use only selected junction nodes, Use junction nodes in selection list, Use all junction nodes

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" from the main program ribbon menu.

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.

Duration

This data entry Is used to the duration of the fire flow event and it is used by the program to compute remaining volumes of water in storage tanks.

Calculate residual pressure for design flow, Calculate available flow for design pressure, Calculate fire hydrant Q-H curve for selected node

This radio button selection allows you to select the fire flow analysis type. 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 flow

This data entry is used to define the design (required) fire flow for which the available (residual) fire pressure will be calculated.

Node 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.

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

Simultaneous mode

This data entry allows you to run the fire flow simulation (*Calculate residual 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.

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.

Fire hydrant 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 pipe diameter

This data entry allows you to define a diameter of the fire hydrant lateral pipe.

Lateral pipe length

This data entry allows you to define a length of the fire hydrant lateral pipe.

Lateral pipe roughness

This data entry allows you to define a roughness coefficient of the fire hydrant lateral pipe.

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

This data entry allows you to define that the search for the nodes and pipes should happen within the same pressure zone. 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.

Run fire flow simulation

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+ program 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.

11.2 Hydrants

A list of the Hydrant dialog box data entries for Figure 11.3 follows, with a short description given for each entry.

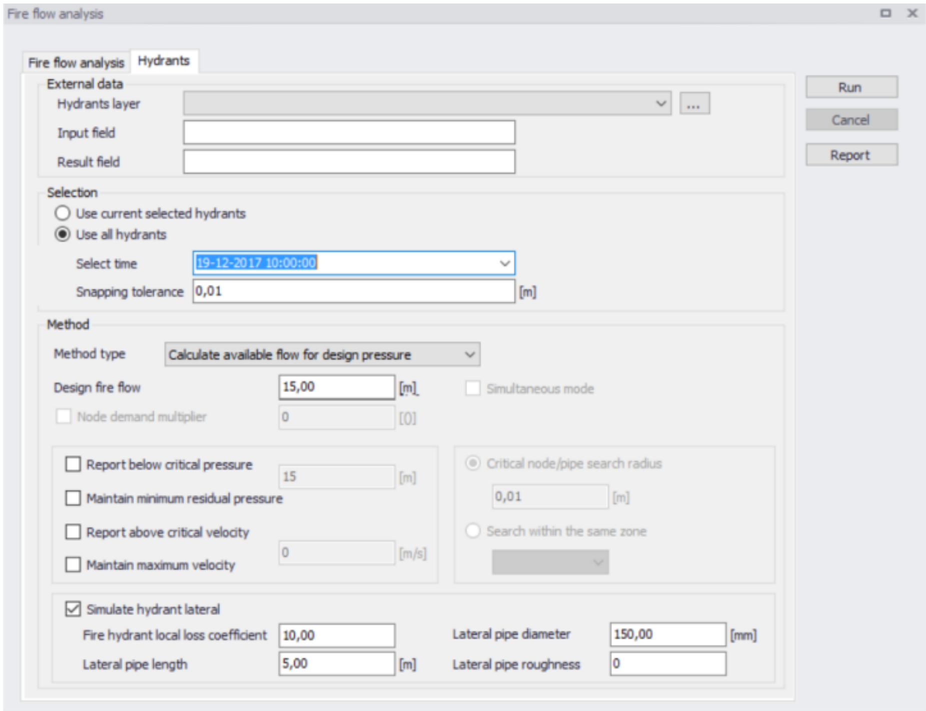


Figure 11.3 The Fire Flow dialog box is used to specify fire flow analysis parameters

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).

Use selected hydrants

This data entry allows you to run the fire flow simulation for selected hydrants (from the layer with reference shapefile).

Use all hydrants

This data entry allows you to run the fire flow simulation for all hydrants in the reference shapefile.

Snapping tolerance

This data entry allows you to define the spatial tolerance that will be used to find the node in the hydraulic model that is the nearest junction node to the hydrant.

The description of the remaining fields is the same as described for “Fire flow analysis”.

11.3 Running Simulations

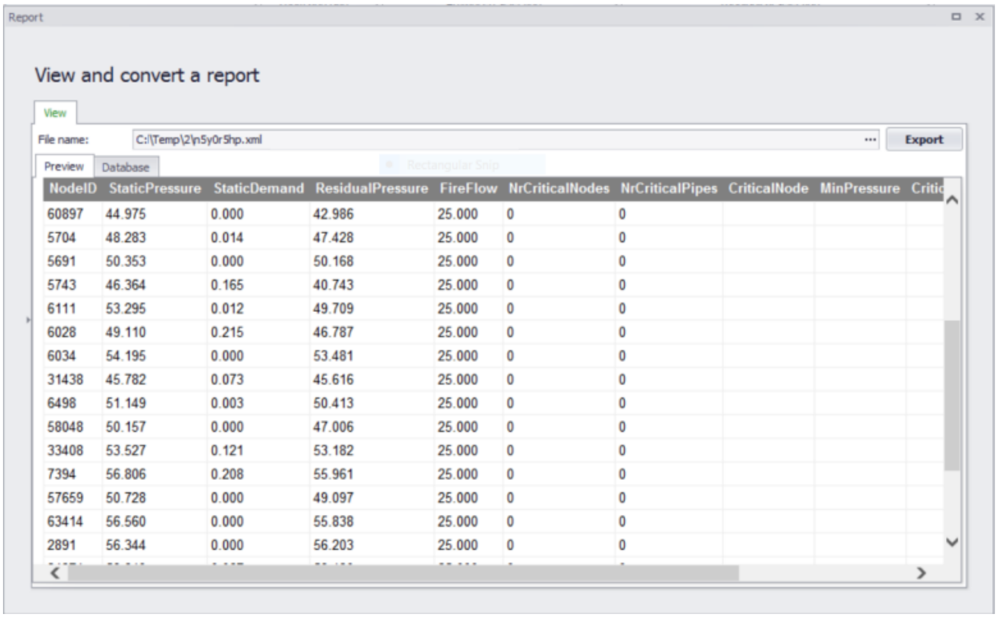
Select “Run” from within the Fire Flow dialog box in order to run the simulation. The simulation progress will be displayed in the application status window. The simulation can be interrupted (cancelled) by pressing “Esc”.

11.4 Browsing Results

Results of the fire flow simulations can be displayed in different ways.

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 suitable for importing into Microsoft Excel, for example. The tabular results can also be displayed directly from within the Fire Flow Analysis dialog box by selecting “Report”.



NodeID	StaticPressure	StaticDemand	ResidualPressure	FireFlow	NrCriticalNodes	NrCriticalPipes	CriticalNode	MinPressure	Critic
60897	44.975	0.000	42.986	25.000	0	0			
5704	48.283	0.014	47.428	25.000	0	0			
5691	50.353	0.000	50.168	25.000	0	0			
5743	46.364	0.165	40.743	25.000	0	0			
6111	53.295	0.012	49.709	25.000	0	0			
6028	49.110	0.215	46.787	25.000	0	0			
6034	54.195	0.000	53.481	25.000	0	0			
31438	45.782	0.073	45.616	25.000	0	0			
6498	51.149	0.003	50.413	25.000	0	0			
58048	50.157	0.000	47.006	25.000	0	0			
33408	53.527	0.121	53.182	25.000	0	0			
7394	56.806	0.208	55.961	25.000	0	0			
57659	50.728	0.000	49.097	25.000	0	0			
63414	56.560	0.000	55.838	25.000	0	0			
2891	56.344	0.000	56.203	25.000	0	0			

Figure 11.4 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:

- Static pressure: steady state pressure at the fire flow node
- Static demand: steady state demand at the fire flow node
- Residual pressure: simulated or given residual pressure during the fire flow simulation at the fire flow 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

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 Parameters

The user must define the hydraulic results file at first, then define the energy price, price pattern, efficiency and Demand charge of Pumps/Turbines.

- Global Price: Average cost per kW/hour.
- Global Price Pattern: ID label of time pattern describing how energy price varies with time.
- Pump/Turbine Efficiency: Either a single perfect efficiency for global setting or the ID label of an efficiency curve for a specific pump/Turbine.
- Demand Charge: Added cost per maximum kW usage during the simulation period.
- Currency: currency name
- Carbon emission factor: this optional entry is used for computing carbon emissions related to pump energy in kg/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.

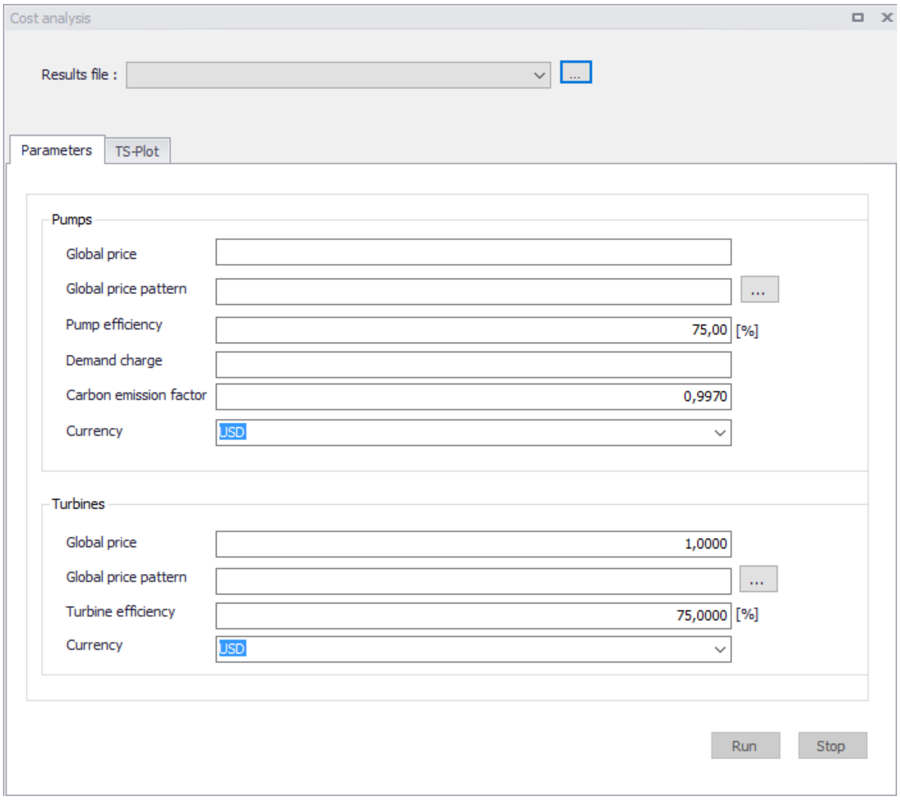


Figure 12.1 The Cost Analysis Parameters dialog box

12.2 Time Series Plot

The Time Series Plot (TS 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 TS-Plot would displaythe graph varied with time, and the result data table on the right side of the panel.

- Efficiency: efficiency of pump/ turbine with time (%)
- kW-hr/volume: Power consumption/production (kW per hour) per millions gallonsn (or cubic meters).
- Energy Consumption/Generated: Energy consumption/production of the production of the pump/turbine operation with time (negative to represent generated energy)
- Energy Cost: The cost of energy consumption or generated of the pump/turbine operation with time (negative to represent generate energy)

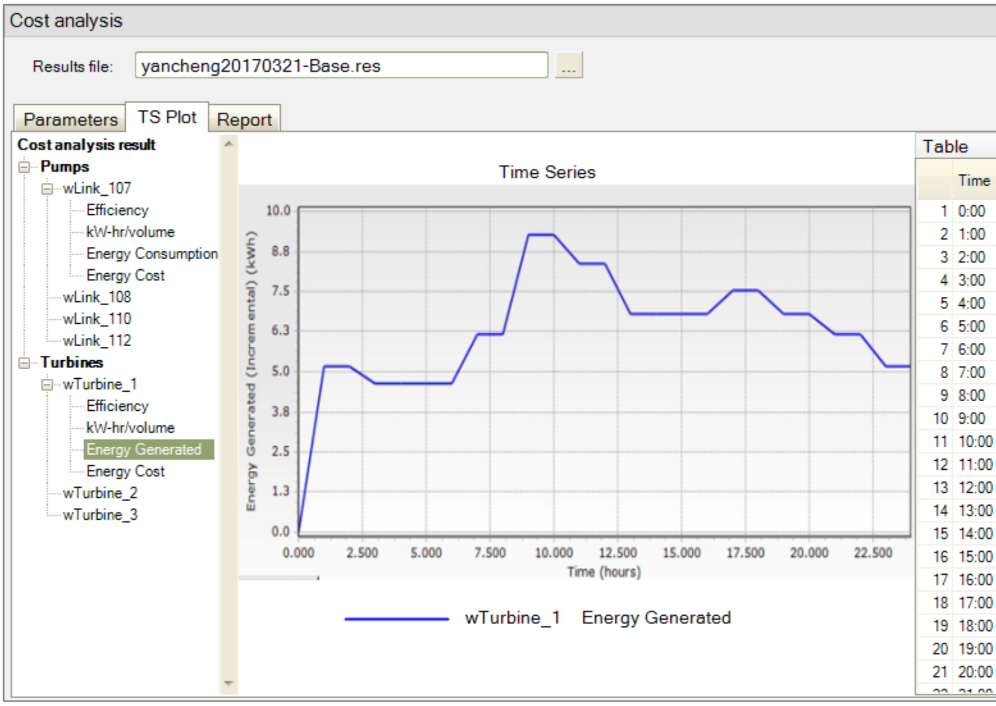
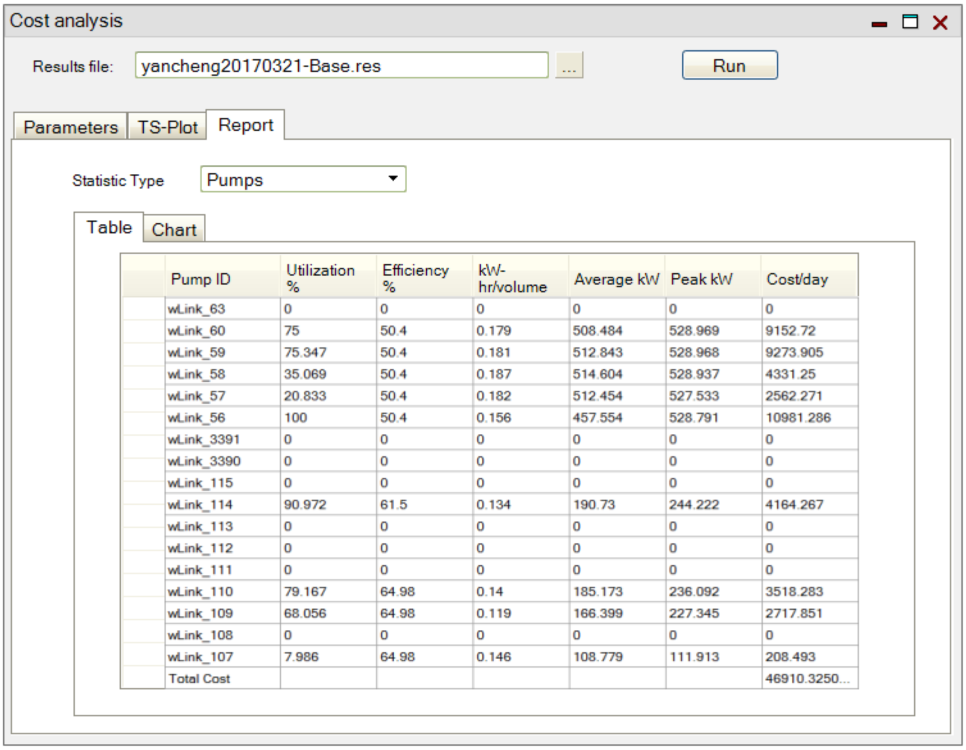


Figure 12.2 Cost Analysis Time Series plot

12.3 Report

12.3.1 Pump energy table

The pump energy table displays the statistical data of pump utilization, average efficiency, average power consumption, peak power consumption, and cost per day.



Pump ID	Utilization %	Efficiency %	kW-hr/volume	Average kW	Peak kW	Cost/day
wLink_63	0	0	0	0	0	0
wLink_60	75	50.4	0.179	508.484	528.969	9152.72
wLink_59	75.347	50.4	0.181	512.843	528.968	9273.905
wLink_58	35.069	50.4	0.187	514.604	528.937	4331.25
wLink_57	20.833	50.4	0.182	512.454	527.533	2562.271
wLink_56	100	50.4	0.156	457.554	528.791	10981.286
wLink_3391	0	0	0	0	0	0
wLink_3390	0	0	0	0	0	0
wLink_115	0	0	0	0	0	0
wLink_114	90.972	61.5	0.134	190.73	244.222	4164.267
wLink_113	0	0	0	0	0	0
wLink_112	0	0	0	0	0	0
wLink_111	0	0	0	0	0	0
wLink_110	79.167	64.98	0.14	185.173	236.092	3518.283
wLink_109	68.056	64.98	0.119	166.399	227.345	2717.851
wLink_108	0	0	0	0	0	0
wLink_107	7.986	64.98	0.146	108.779	111.913	208.493
Total Cost						46910.3250...

Figure 12.3 Cost Analysis Pump Energy Table

The following fields are calculated and reported:

- Utilization: percent utilization i.e. percent of the time that the pump was operating (%)
- Efficiency: Average efficiency of the pump (%)
- kW-hr/m3 or kW-hr/gallon: Average power consumption (kW per hour) per million gallons (or cubic meters) pumped
- Average kW: Average rate of energy usage if the pump (kW)
- Peak kW: Peak rate of energy usage of the pump operation (kW)
- Cost/day: total cost of the pump operation per day (monetary units)

12.3.2 Pump energy graphs

The pump energy including pump utilization, power consumption, and cost can be displayed for every pump in the model.

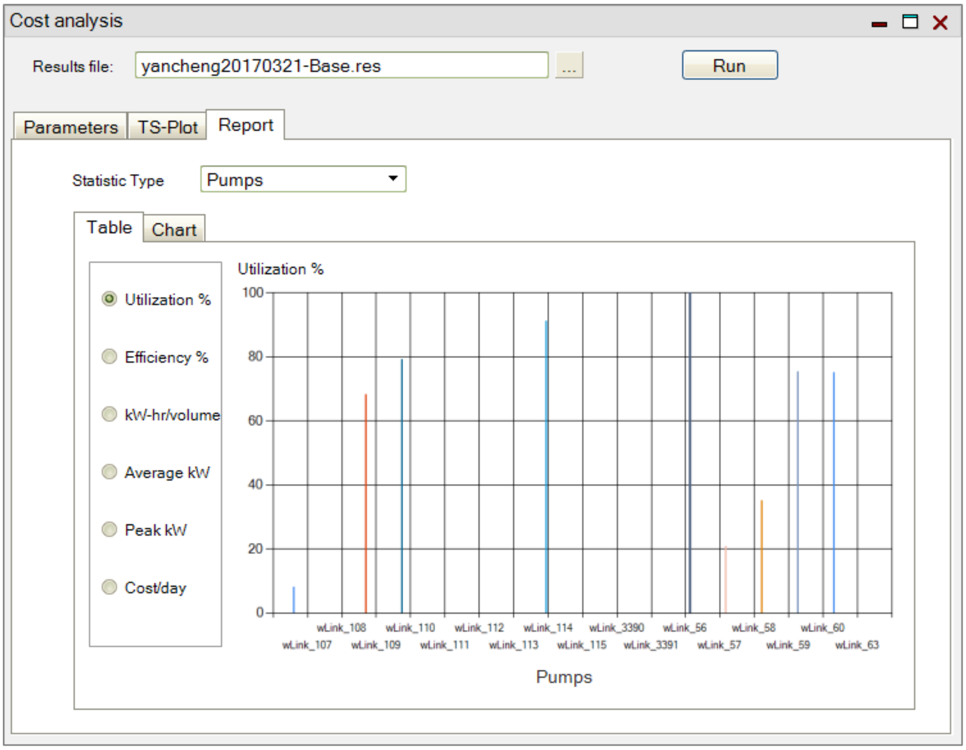
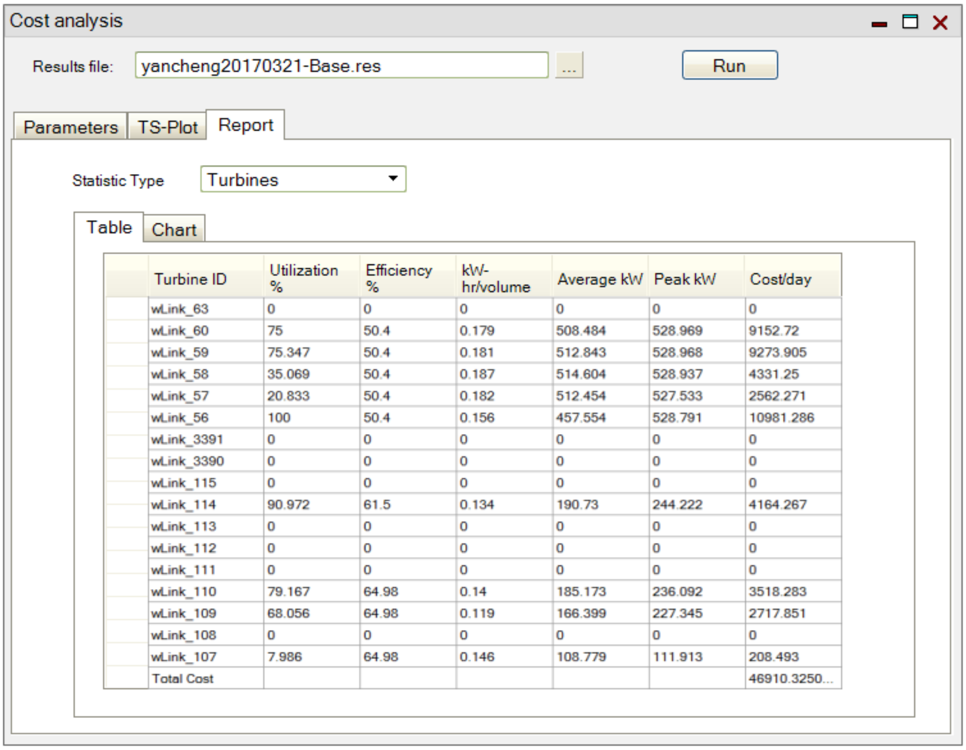


Figure 12.4 Pump Energy Graph

12.3.3 Turbines

Turbine energy table

The turbine energy table reports turbines utilization, average efficiency, average power production, peak power production and energy saving per day.



Turbine ID	Utilization %	Efficiency %	kW-hr/volume	Average kW	Peak kW	Cost/day
wLink_63	0	0	0	0	0	0
wLink_60	75	50.4	0.179	508.484	528.969	9152.72
wLink_59	75.347	50.4	0.181	512.843	528.968	9273.905
wLink_58	35.069	50.4	0.187	514.604	528.937	4331.25
wLink_57	20.833	50.4	0.182	512.454	527.533	2562.271
wLink_56	100	50.4	0.156	457.554	528.791	10981.286
wLink_3391	0	0	0	0	0	0
wLink_3390	0	0	0	0	0	0
wLink_115	0	0	0	0	0	0
wLink_114	90.972	61.5	0.134	190.73	244.222	4164.267
wLink_113	0	0	0	0	0	0
wLink_112	0	0	0	0	0	0
wLink_111	0	0	0	0	0	0
wLink_110	79.167	64.98	0.14	185.173	236.092	3518.283
wLink_109	68.056	64.98	0.119	166.399	227.345	2717.851
wLink_108	0	0	0	0	0	0
wLink_107	7.986	64.98	0.146	108.779	111.913	208.493
Total Cost						46910.3250...

Figure 12.5 Turbine Energy Table

The following fields are calculated and reported:

- Utilization: Percent utilization i.e. percent of the time that the turbine was operating (%)
- Efficiency: Average efficiency of the turbine (%)
- kW-hr/m3 or kW-hr/gallon: Average power production (kW per hour) per million cubic metters (or gallons) pumped.
- Average kW: Average rate of energy production of the turbine (kW)
- Peak kW: Peak rate of energy production of the turbine operation per day, negative to represent generated energy.
- Cost/day: total cost of the turbine operation per day, negative to represent generated energy (in monetary units).

Turbine energy graphs

Turbine energy data is plotted including turbine utilization, power production, and energy saving can be displayed for every turbine in the model. These graphs enhance the analysis of turbine energy and evaluation and comparison of different turbines.

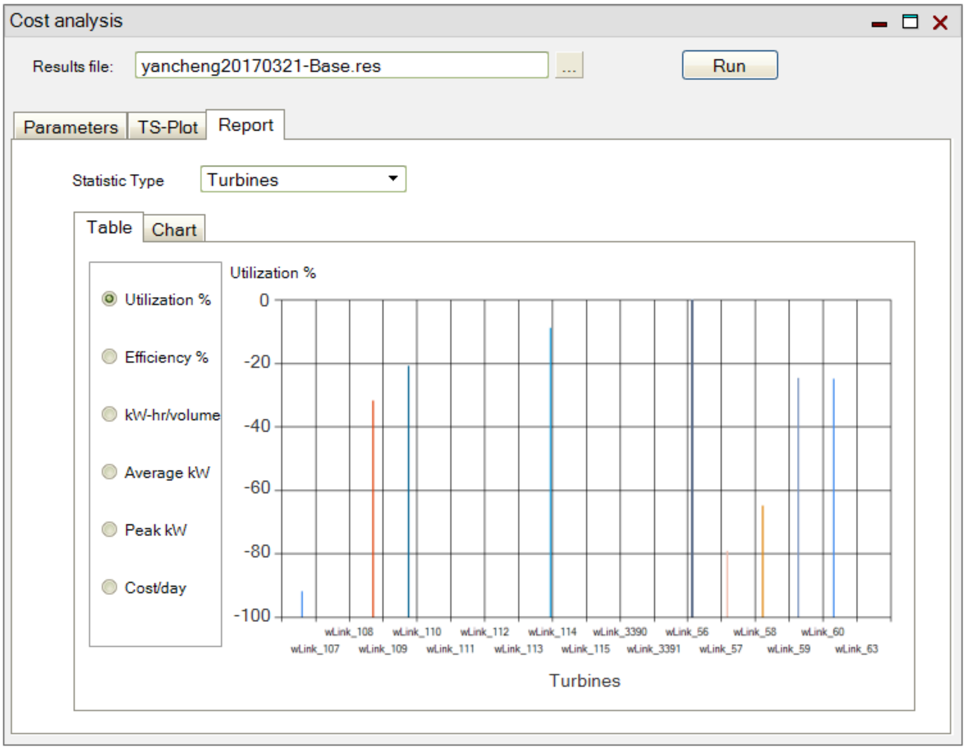
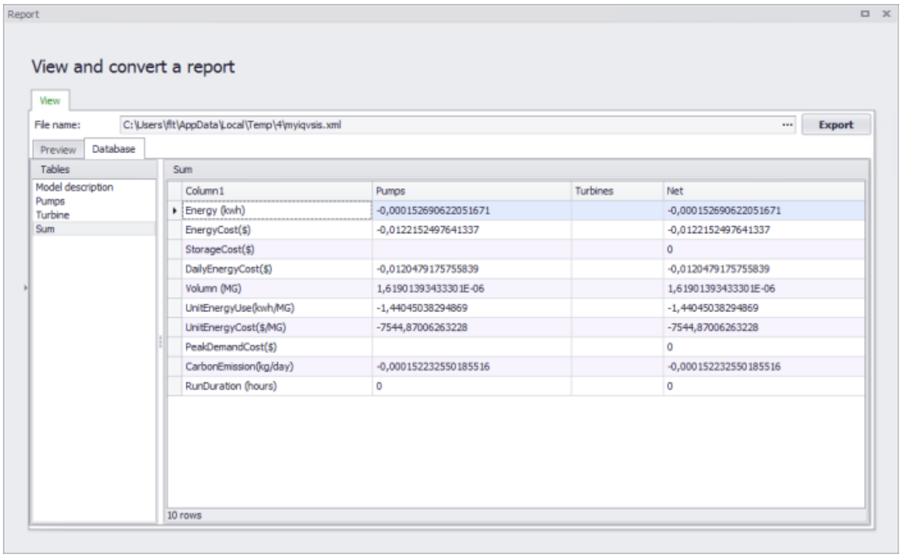


Figure 12.6 Cost Analysis Turbine Energy graph

Sum Report

This type of report presents all the statistical energy data of all pumps and turbines. Including net energy and energy cost, storage cost, daily/ unit energy cost, peak demand cost and carbon emission per day.



Column1	Pumps	Turbines	Net
Energy (kwh)	-0,000152690622051671		-0,000152690622051671
EnergyCost($)	-0,0122152497641337		-0,0122152497641337
StorageCost($)			0
DailyEnergyCost($)	-0,0120479175755839		-0,0120479175755839
Volumn (MG)	1,61901393433301E-06		1,61901393433301E-06
UnitEnergyUse(kwh/MG)	-1,44045038294869		-1,44045038294869
UnitEnergyCost($/MG)	-7544,87006263228		-7544,87006263228
PeakDemandCost($)			0
CarbonEmission(kg/day)	-0,000152232550185516		-0,000152232550185516
RunDuration (hours)	0		0

Figure 12.7 The Sum Energy Report table

13 Pipe criticality

Pipe criticality modelling is required to predict the water distribution system response to pipe breaks situations, planned reconstructions, and other scenarios of limited water supply. Pipe criticality 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 pipe criticality is determined based on evaluation of several (four) 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 $(P1+P2+P3+P4)_{(pipe\ i)}$

- 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 water demand that cannot be delivered due to insufficient service pressure, the value of 1 corresponds to the total water demand consumption. P3 is computed as total demand that cannot be delivered in nodes where the pressure is below the required service level.
- Pipe length criteria (P4) is computed as a total length of pipes affected by the respective pipe. 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. P4 is computed as a total length of pipes where the pressure is below the required service level
- 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.

Performance indicators P are computed as follows:

- P1 is computed as water (in flow units) that cannot be delivered through the pipe
- P2 is computed as number of nodes, where the pressure is below the required service level
- P3 is computed as total demand that cannot be delivered in nodes where the pressure is below the required service level.
- P4 is computed as a total length of pipes where the pressure is below the required service level
- P5 is computed as a total (sum of) “criteria” of pipes where the pressure is below the required service level

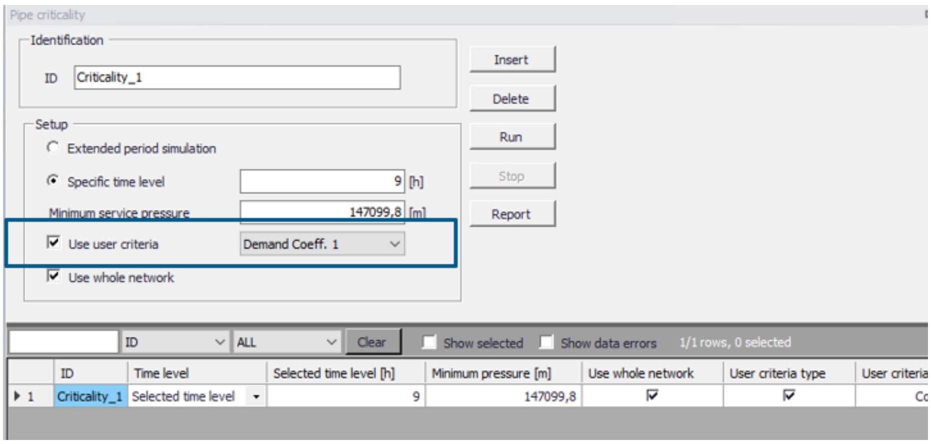


Figure 13.1 Pipe criticality user defined criteria

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”.

The Pipe Criticality Analysis dialog box is reached by selecting ‘Model type’ from General Settings from the Table of Contents and then by selecting the

‘Pipe criticality’ option. Note that to run the pipe criticality analysis, you need to select *Run* from within the Pipe Criticality Analysis dialog box.

13.1 Setup

A list of the Pipe Criticality Analysis dialog box data entries for Figure 13.2 follows, with a short description given for each entry.

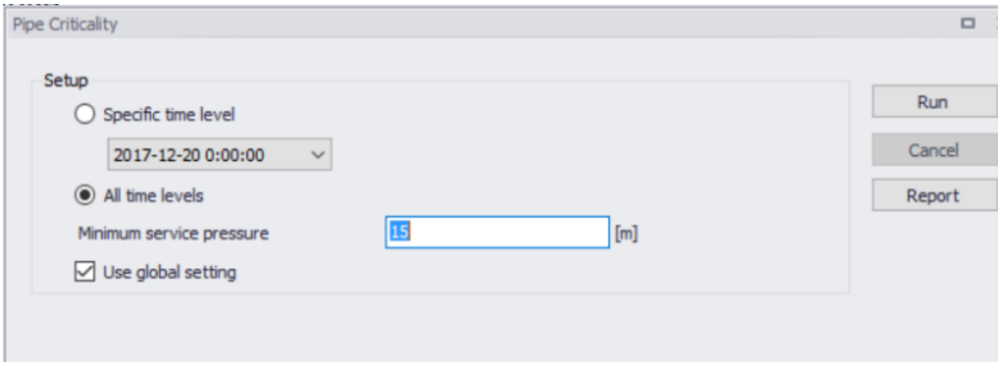


Figure 13.2 Pipe Criticality dialog box is used to the analysis parameters

Specific time level

This data entry allows you to select the time level from the extended period simulation that will be used to compute the pipe criticality.

All time levels

This data entry allows you to compute the pipe criticality 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 global settings

This data entry allows you to run the pipe criticality for all pipes within the model or for pipes within selected pressure zone(s).

13.2 Running Simulations

Select “Run” from within the Pipe Criticality 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 Browsing Results

Results of the pipe criticality simulations can be displayed in different ways.

Tabular Results

The simulated pipe criticality results are written into the output CSV file. The CSV file is a comma separated text file in a format that is suitable for importing into Microsoft Excel, for example. The tabular results can also be displayed directly from within the Pipe Criticality Analysis dialog box by selecting “Report”.

Figure 4 Pipe criticality results report

Thematic Maps

The simulated pipe criticality results can be displayed using the colour coded maps. Select Layers and Add Layer and select one of the pipe criticality result items to create a colour coded map with the fire flow results.

List of available pipe criticality 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)
- PI-1: performance indicator PI-1 (-)
- SumNodes: number of nodes where the service pressure is insufficient
- PI-2: performance indicator PI-2 (-)
- SumDemand: demand or total water volume in case of extended period simulation for all time levels)
- PI-3: performance indicator PI-3 (-)
- SumLength: total pipe length where the service pressure is insufficient
- PI-4: performance indicator PI-4 (-)
- C: performance indicator C (-)
- PI-5: Performance Indicator PI-5

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.

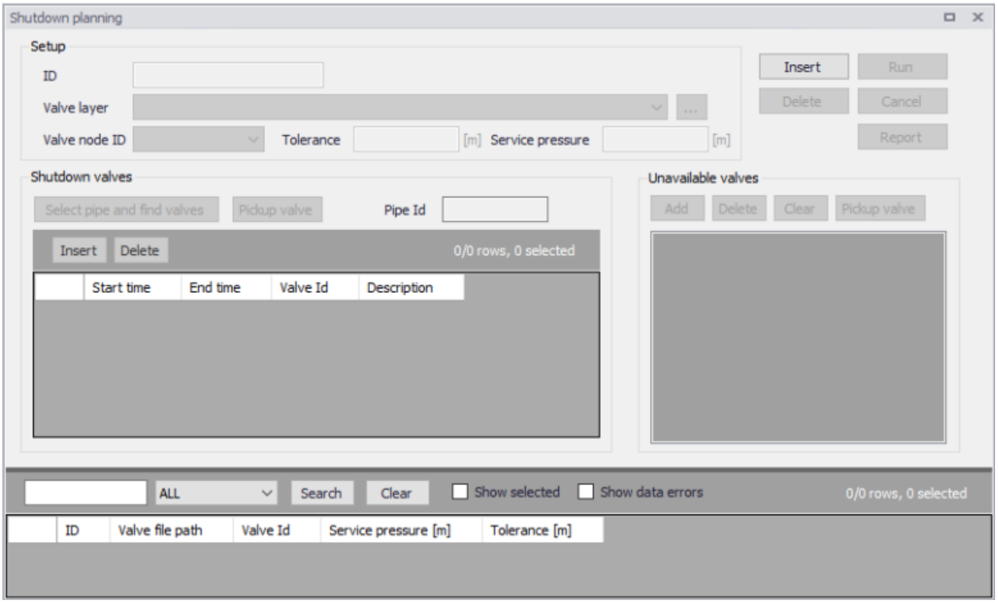


Figure 14.1 The Shutdown Planning dialog box is used to define the analysis parameters

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. The layer with simulated pressures is automatically added into the Map layers at the end of the shutdown planning analysis.

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.

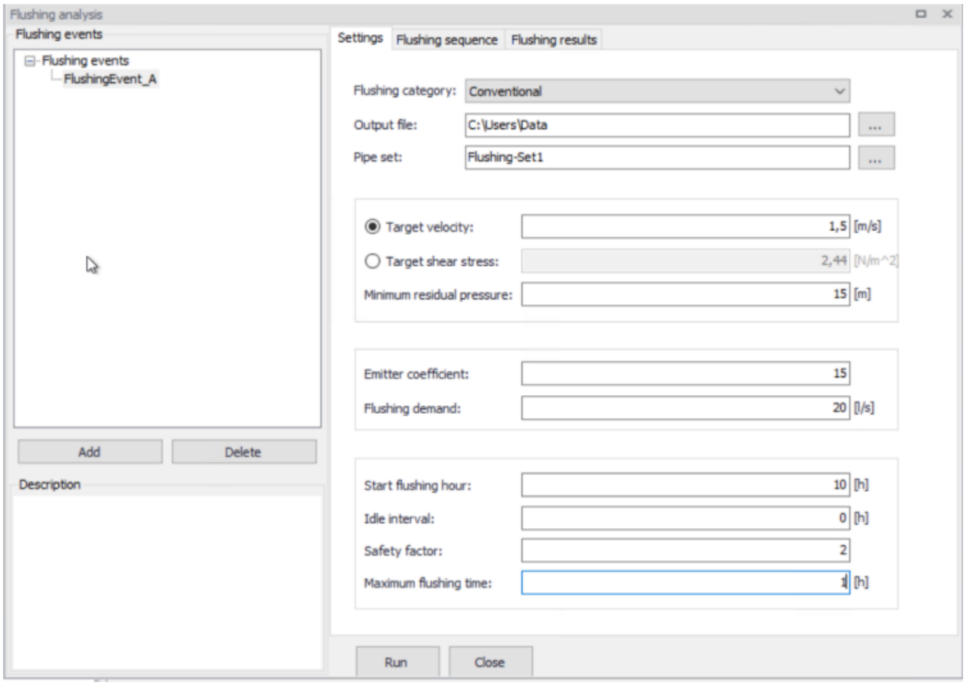


Figure 15.1 The Flushing settings box is used to the analysis parameters

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.

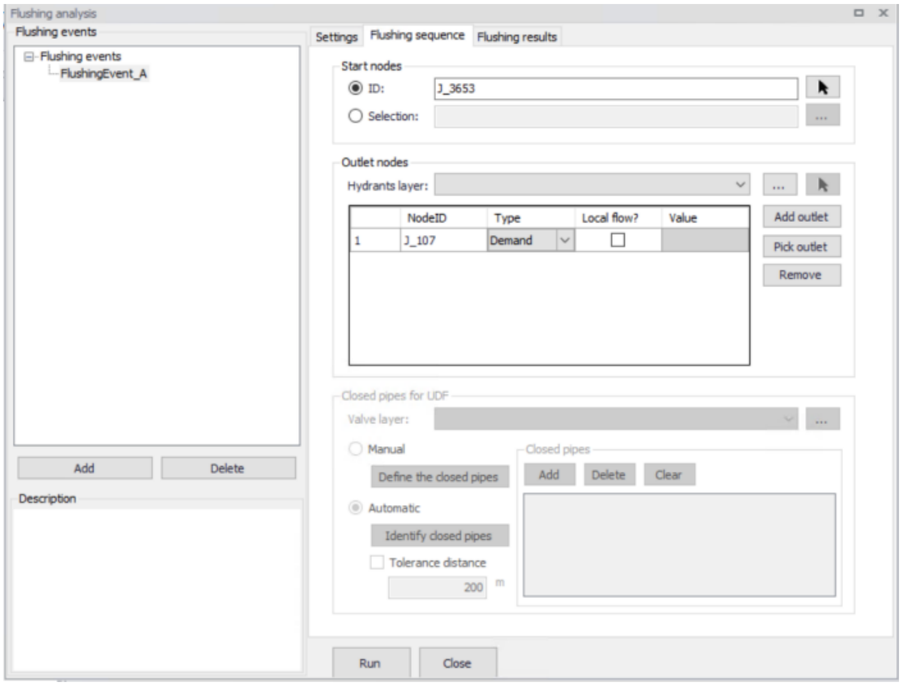


Figure 15.2 The Flushing sequence box is used to the analysis parameters

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.

Tabular Results

The simulated flushing results are written into the output CSV file. The CSV file is a comma separated text file in a format that is suitable for importing into Microsoft Excel, for example. The tabular results can also be displayed directly from within the Flushing Analysis dialog box by selecting “Flushing results”.

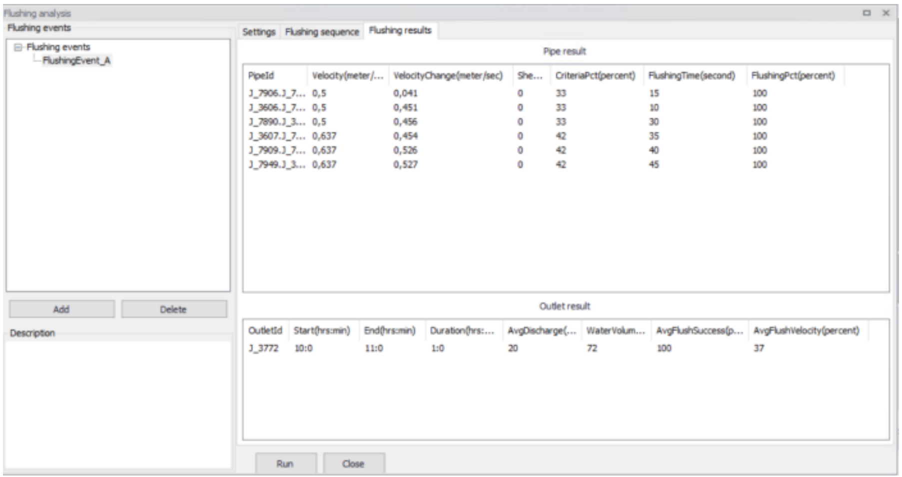


Figure 15.3 Flushing results report

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 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.

16.1 Settings

A list of the Pressure dependent demands data entries for Figure 16.1 follows, with a short description given for each entry.

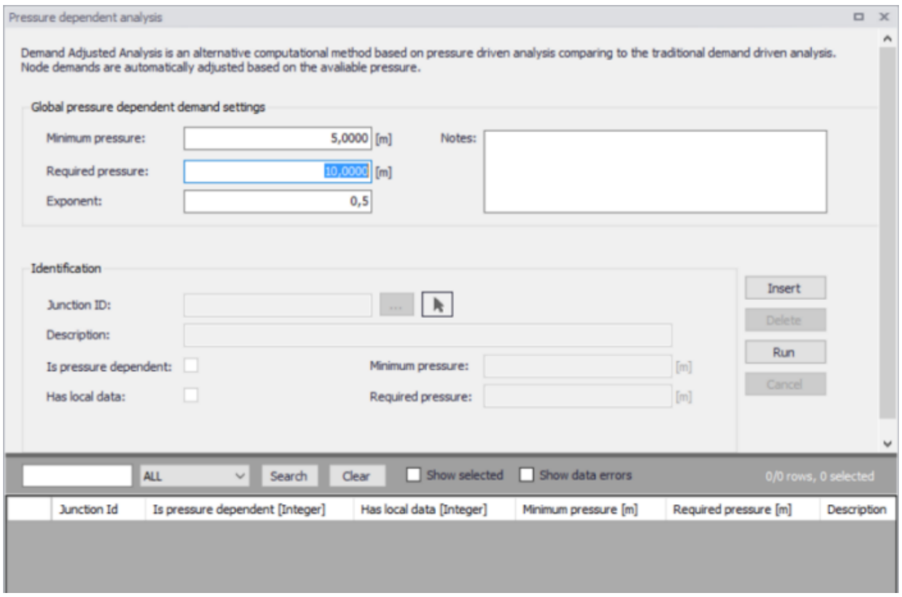


Figure 16.1 The Pressure dependent demands dialog box is used to the analysis parameters

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.

16.2 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”.

16.3 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)

17 Valve Criticality

Valve criticality allows you to select any valve from your GIS valve layer and find what other valves need to be closed in order to replace the selected valve, such as if the valve would be malfunctioning. In order to use this tool, select Valve Criticality from Tools menu and define the layer containing your pipe network and valves. Please note that you can select any layer including shapefiles and that you can combine e.g. pipe network from your hydraulic model with a shapefile containing GIS valves.

Valve criticality can operate in two modes:

- Interactive mode: allows you to inspect valve one by one by pointing and clicking the Valve
- Automatic mode: allows you to run the valve criticality for selected valves in the automatic manner and store the results in the database.

Valve criticality tool helps you to understand the important of isolation valves and assists you in the valve maintenance and replacement program.

The Valve Criticality dialog box is reached by selecting Tools from the program main menu and the by selecting Valve Criticality. Note that to run the valve criticality analysis, you need to select *Run* from within the Valve Criticality dialog box.

17.1 Settings

A list of the Valve Criticality data entries for Figure 17.1 follows, with a short description given for each entry.

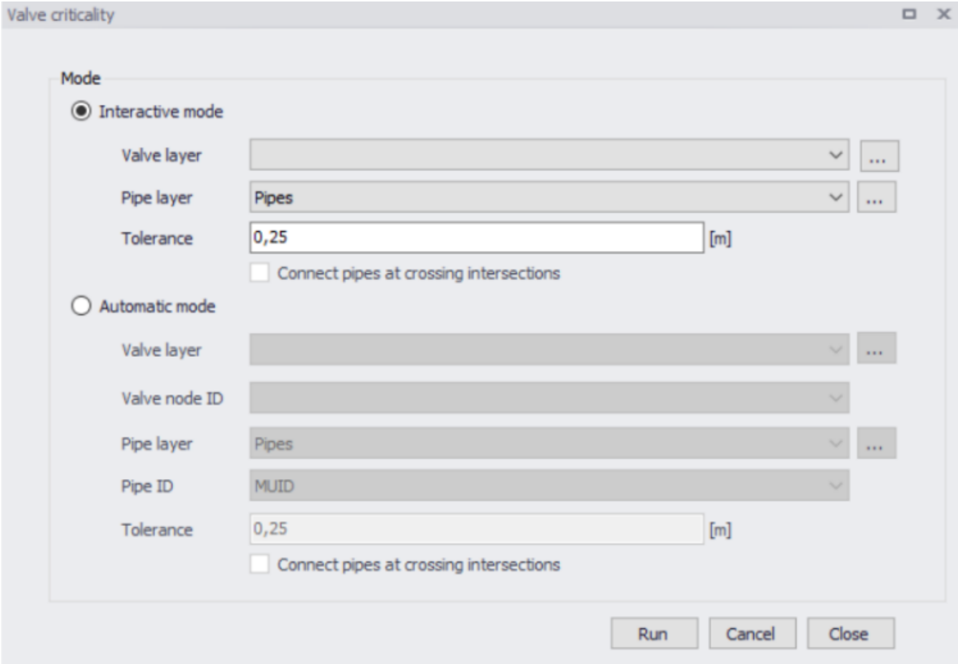


Figure 17.1 The Valve Criticality dialog box is used to define the analysis parameters

Interactive mode

This data entry 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 data entry 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

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 pipe layer in this data entry, the pipe layer needs to be already added to the Map layers.

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

This data entry 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.

17.2 Running analysis

Select “Run” from within the Valve Criticality dialog box in order to run the simulation. In case of “interactive” simulation the program will allow you select the valve from the Map window. In case of “automatic” simulation, the program will start analysing all valves and the simulation progress will be displayed in the application status window. The simulation can be interrupted (cancelled) by pressing “Esc”.

17.3 Valve criticality results

Results of the automatic valve criticality can be displayed in the Map where each valve can be colour coded by the number that represents the number of substitute valves.

18 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

18.1 Water Hammer Calculation

MIKE+ Water Hammer computes hydraulic transients in pipe networks. The computations are based on the continuity equation:

$$\frac{\partial Q}{\partial x} + \frac{gA}{a^2}\frac{\partial H}{\partial t} = 0 \tag{18.1}$$

and the equation of motion:

$$\frac{\partial Q}{\partial t} + gA\frac{\partial H}{\partial x} + \frac{f}{2DA}Q\left|Q\right| = 0 \tag{18.2}$$

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

$$a = \sqrt{\frac{K}{\rho\left[1 + (K/E\ )\psi\ \right]}} \tag{18.3}$$

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.

18.2 Theoretical Background

The following section describes the MIKE+ water hammer numerical engine.

18.2.1 Description of Water Hammer Model

The water hammer computation is based on the Continuity equation

$$\frac{\partial Q}{\partial x} + \frac{g\,A}{a^2}\frac{\partial H}{\partial t} = 0 \tag{18.4}$$

and the Momentum equation

$$\frac{\partial Q}{\partial t} + g\,A\frac{\partial H}{\partial x} + \frac{f}{2DA}Q\left|Q\right| = 0 \tag{18.5}$$

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

$$a = \sqrt{\frac{K}{\rho[1 + (K/E)\psi]}} \tag{18.6}$$

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

$$\psi = 0 \tag{18.7}$$

Thick-Walled Elastic Conduit (D/e<=10)

- anchoring at both ends = full restraint

$$\psi = 2(1 + \nu)\left(\frac{R_o^2 + R_i^2}{R_o^2 - R_i^2} - \frac{2\nu R_i^2}{R_o^2 - R_i^2}\right) \tag{18.8}$$

in which ν is the Poison's ratio, R_o is an external diameter, R_i is an internal diameter.

- upstream anchoring = upper restraint

$$\psi = 2\left(\frac{R_o^2 + 1.5R_i^2}{R_o^2 - R_i^2} + \frac{\nu(R_o^2 - 3R_i^2)}{R_o^2 - R_i^2}\right) \tag{18.9}$$

- frequent expansion joints = expansion joints

$$\psi = 2\left(\frac{R_o^2 + R_i^2}{R_o^2 - R_i^2} + \nu\right) \tag{18.10}$$

Thin-Walled Elastic Conduit (D/e>10)

- anchoring at both ends = full restraint

$$\psi = \frac{D}{e}\left(1 - \nu^2\right) \tag{18.11}$$

in which D is the conduit diameter and e is the wall thickness

- upstream anchoring = upper restraint

$$\psi = \frac{D}{e}\left(1 - 0.5\,\nu\right) \tag{18.12}$$

- frequent expansion joints = expansion joints

$$\psi = \frac{D}{e} \tag{18.13}$$

Tunnels Through Solid Rock, Parmakian 1963

- Unlined tunnel

$$\psi = 1 \quad E = G \tag{18.14}$$

where G is the modulus of rigidity of the rock.

- Steel - lined tunnel

$$\psi = \frac{DE}{GD + Ee} \tag{18.15}$$

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.

$$e_e = E_r \ e_c + \frac{A_s}{L_s} \tag{18.16}$$

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.

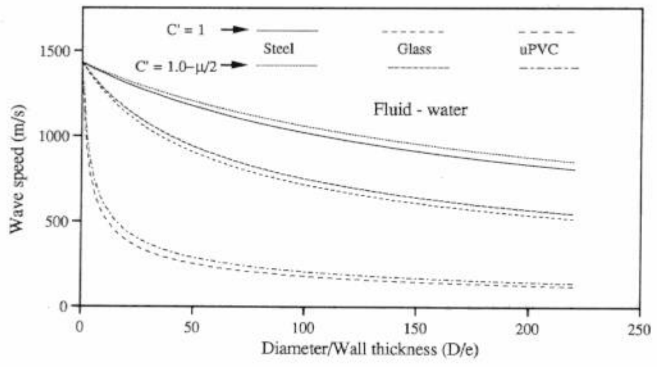


Figure 18.1 Fluid water

Values of Young's Modulus of Elasticity and Poisson's Ratio for a range of common materials are available in the following table.

Table 18.1 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 18.1 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.

18.3 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

$$\frac{f}{2DA}\left|Q\right|Q \approx \frac{1}{2}\frac{f}{2DA}\left(\left|Q_{j-1}^{n}\right|Q_{j-1}^{n+1} + \left|Q_{j}^{n}\right|Q_{j}^{n+1}\right) \tag{18.17}$$

The coefficients for the water hammer model have been derived and have the following form:

18.3.1 Coefficients for the numerical scheme

$$\alpha = \frac{gA}{a^2} \tag{18.18}$$

$$fric = \frac{\lambda}{2AD} \tag{18.19}$$

$$Cr = \frac{a\ \Delta t}{\Delta x} \tag{18.20}$$

$$\alpha_c = \left(6\psi^2 - 6\psi + 1\right) + Cr^2\left(6\theta - 6\theta^2 - 1\right)\frac{\Delta x^2}{3a^2\Delta t^2} \tag{18.21}$$

$$\alpha_m = \left(6\psi^2 - 6\psi + 1\right) + Cr^2\left(6\theta - 6\theta^2 - 1\right)\frac{gA\,\Delta x^2}{3a^2\Delta t^2} \tag{18.22}$$

$$A1 = \alpha\left(1 - \psi\right)\Delta x \tag{18.23}$$

$$B1 = -\theta\ \Delta t + \alpha_c \tag{18.24}$$

$$C1 = \alpha\ \psi\ \Delta x \tag{18.25}$$

$$D1 = \theta\ \Delta t - \alpha_c \tag{18.26}$$

$$E1 = -(1-\theta)\Delta t(Q_{j-1}^{n} + Q_{j-1}^{n-1} - Q_{j}^{n} - Q_{j}^{n-1}) + \theta\Delta t(Q_{j-1}^{n} - Q_{j}^{n}) +$$
$$+\alpha_c(Q_{j}^{n-1} - 2Q_{j}^{n} - Q_{j-1}^{n-1} + 2Q_{j-1}^{n}) + \alpha_c(1-\psi)\Delta x H_{j-1}^{n-1} + \alpha_c\psi\Delta x H_{j}^{n-1} \tag{18.27}$$

$$A2 = -g\ a\ \theta\ \Delta t + \alpha_m \tag{18.28}$$

$$B2 = (1-\psi)\Delta x + fric\,\Delta t \Delta x \left| Q^{n}_{j-1} \right| \tag{18.29}$$

$$C2 = g\ a\ \theta\ \Delta t - \alpha_{m} \tag{18.30}$$

$$D2 = \psi \Delta x + fric \Delta t \Delta x \left| Q^{n}_{j} \right| \tag{18.31}$$

$$E2 = (1-\psi)\Delta x Q^{n-1}_{j-1} + \psi \Delta x Q^{n-1}_{j} + gA(1-\theta)\Delta t\left(H^{n}_{j-1} - H^{n}_{j} + H^{n-1}_{j-1} - H^{n-1}_{j}\right) + \\ + gA\theta\Delta t\left(H^{n}_{j-1} - H^{n}_{j}\right) + \alpha_{m}\left(H^{n-1}_{j} - 2H^{n}_{j} - H^{n-1}_{j-1} + 2H^{n}_{j-1}\right) \tag{18.32}$$

18.3.2 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

$$A1_{j} H^{n+1}_{j-1} + B1_{j} Q^{n+1}_{j-1} + C1_{j} H^{n+1}_{j} + D1_{j} Q^{n+1}_{j} = E1_{j} \tag{18.33}$$

$$A2_{j} H^{n+1}_{j-1} + B2_{j} Q^{n+1}_{j-1} + C2_{j} H^{n+1}_{j} + D2_{j} Q^{n+1}_{j} = E2_{j} \tag{18.34}$$

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 18.2.

For any computational grid point, equations (18.35), (18.36) may be written as

$$H_i = L1_i H_I + M1_i H_{jj} + N1_i \tag{18.35}$$

$$Q_i = L2_i H_I + M2_i H_{jj} + N2_i \tag{18.36}$$

where L,M,N are functions of coefficients A, B, C, D, E, found through a double sweep elimination

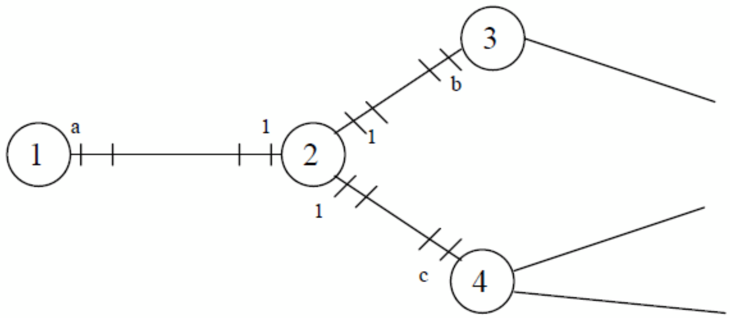


Figure 18.2 Part of a looped pipe network

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.

$$\sum_{k=1}^{m} Q_{1k}^{n+1} = 0 \tag{18.37}$$

$$h_{11}^{n+1} = h_{12}^{n+1} = \ldots = h_{1k}^{n+1} = \ldots = h_{1m}^{n+1} \tag{18.38}$$

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.

$$[S]\,\{h\} = \{T_L\} \tag{18.39}$$

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 (18.37) and (18.38).

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.

18.3.3 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.

18.4 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.

18.4.1 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

$$Cr = \frac{a\Delta t}{\Delta x} \tag{18.40}$$

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).

18.4.2 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.

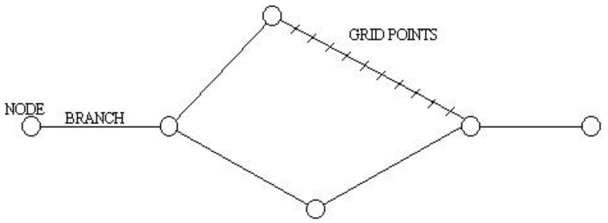


Figure 18.3 Definition Network Layout

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 18.2 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)

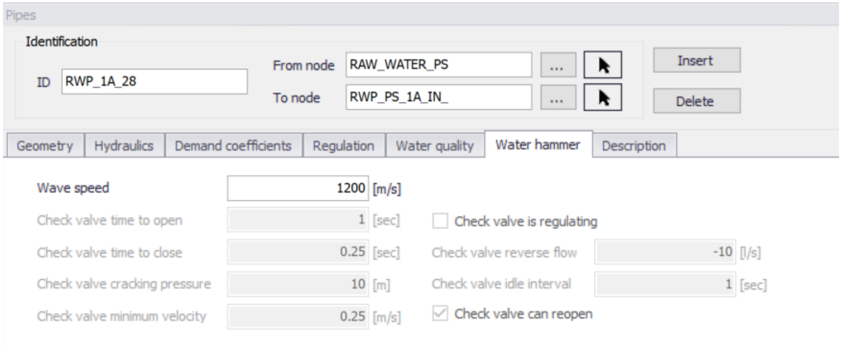


Figure 18.4 Pipe editor with water hammer settings

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:

$$tdH = H = \left(\frac{V_d^2}{2g} + \frac{p_d}{\rho g} + z_d \right) - \left(\frac{V_s^2}{2g} + \frac{p_s}{\rho g} + z_s \right) \tag{18.41}$$

where the subscripts, d and s denote the discharge and suction flanges, respectively. Power input P (kW) is defined as:

$$P = \frac{\rho g Q H}{\eta} = T \omega = \frac{T 2 \pi N}{60} \tag{18.42}$$

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 18.5. 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.

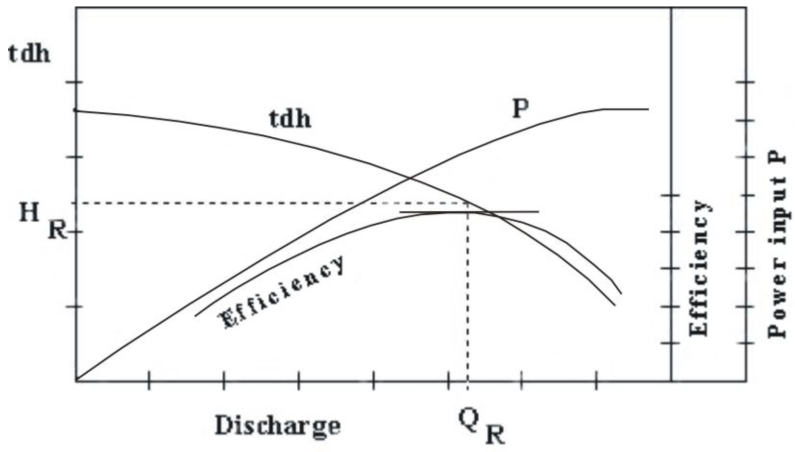


Figure 18.5 Q - H Curve

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

$$\frac{Q_1}{Q_2} = \frac{N_1}{N_2} \quad \frac{H_1}{H_2} = \left(\frac{N_1}{N_2}\right)^2 \quad \frac{P_1}{P_2} = \left(\frac{N_1}{N_2}\right)^3 \tag{18.43}$$

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

$$P = \frac{\rho g Q H}{\eta} = T\omega = \frac{T 2\pi N}{60} \tag{18.44}$$

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

$$\Delta N = \frac{(\ Tm - T\ )\Delta t 30}{I_{\varpi} \pi} \tag{18.45}$$

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:

$$tdH_t = \frac{tdH_{100\%}}{N_{100\%}^2} N_t^2 \tag{18.46}$$

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:

$$I\omega \sum_{i=1}^{n} mr_j \tag{18.47}$$

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.

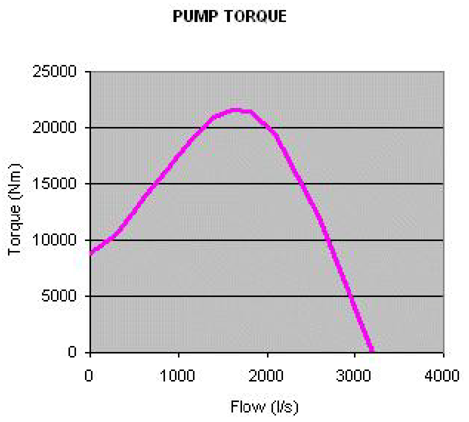


Figure 18.6 Pump torque curve

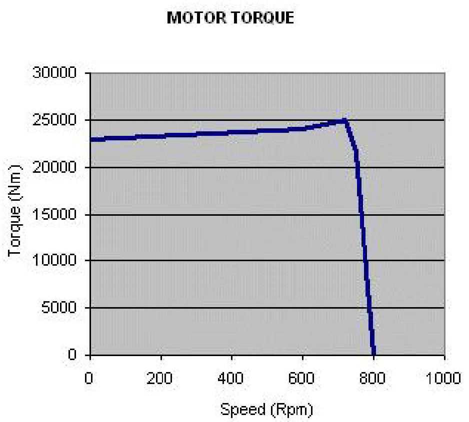


Figure 18.7 Motor torque curve

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)

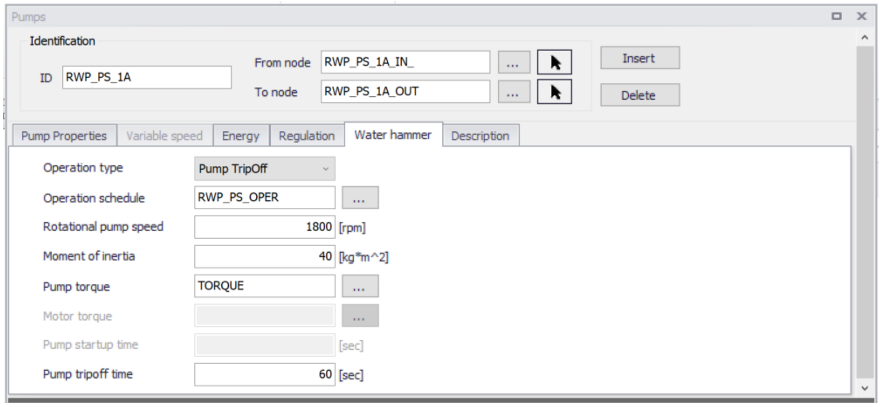


Figure 18.8 Pump editor with water hammer settings

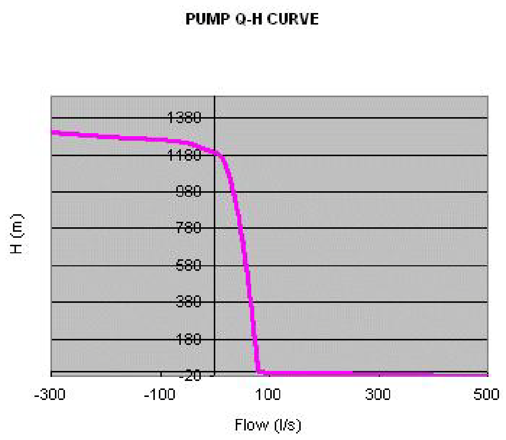


Figure 18.9 Pump Q-H curve defined for 100% rated speed

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 DH is expressed using a discharge coefficient Cd for:

In-line valve

$$Q = Cd\ Av\sqrt{2g\Delta H}\ Cd = \frac{1}{\sqrt{\xi}} \tag{18.48}$$

where Av is the valve area and x is the valve minor loss coefficient.

Free-discharge valve

$$Q = Cd\ Av\sqrt{2g\Delta H}\ Cd = \frac{1}{\sqrt{\xi + 1}} \tag{18.49}$$

where xf 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

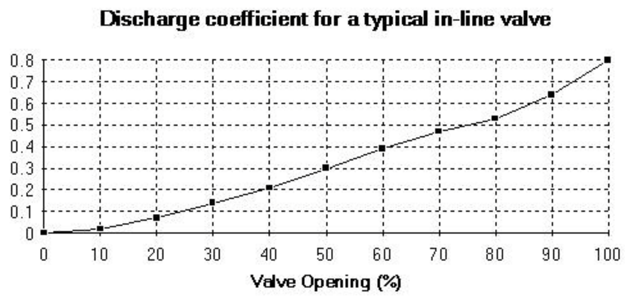


Figure 18.10 Discharge coefficient for a typical in-line valve

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

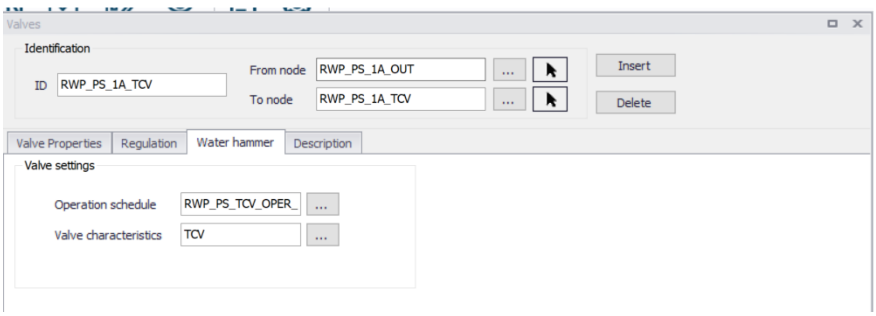


Figure 18.11 Valve editor with water hammer settings

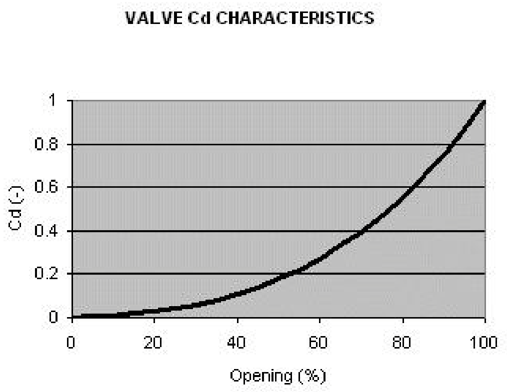


Figure 18.12 Valve Cd Characteristics

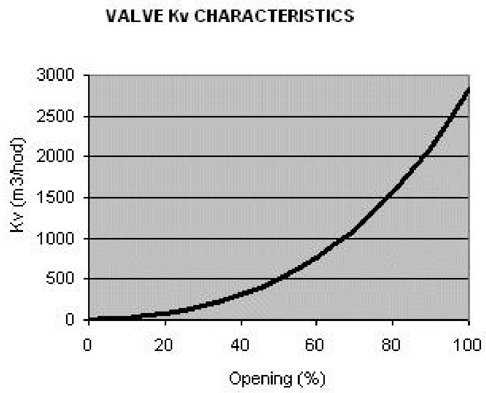


Figure 18.13 Valve Kv Characteristics (Example)

The relation between Cd and Kv valve coefficients is given by the following equation:

$$Cd = \frac{Kv}{3600A\sqrt{2g}} \tag{18.50}$$

The relation between Cd valve coefficient and x minor loss coefficient is given by the following equation:

$$Cd = \frac{1}{\sqrt{\xi - 1}} \tag{18.51}$$

or, for an in-line valve:

$$Cd = \frac{1}{\sqrt{\xi}} \tag{18.52}$$

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.

$$Cr = 1 = a\,\frac{\Delta t}{\Delta tx} \tag{18.53}$$

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 18.3 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 18.3 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

18.4.3 List of components

List of supported components

The following components are supported by the Water Hammer simulations.

Table 18.4 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 18.5 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 18.6 List of new components

Component	Remark
Air Chamber	Supported
Vented air-chamber	Supported
Air Valve	Supported

18.4.4 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 18.15 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

$$C = H_{air} * \forall^{\kappa}_{air} \tag{18.54}$$

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).

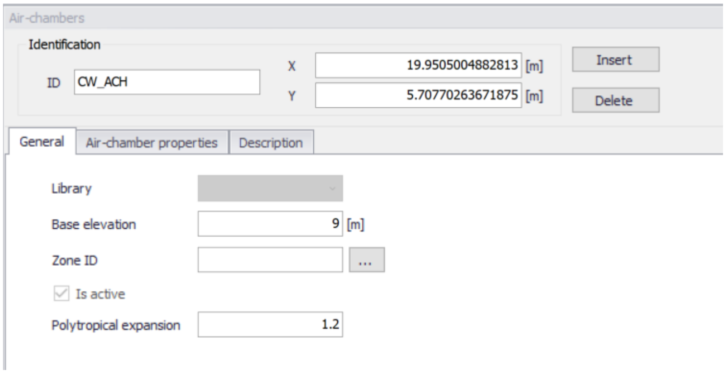


Figure 18.14 Air chamber editor with water hammer settings

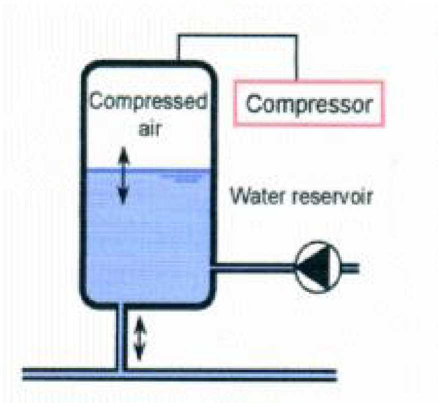


Figure 18.15 Air chamber

Vented air chambers

Vented Air Chambers contain air which prevent very low minimum pressures in the pipeline and hence column separation. Vented Air Chambers are equipped by dual-acting valves that allow air to be sucked into its chamber and to escape there from, while preventing the outflow of liquid. The outflow of air is usually restricted and the escaping air is compressed by the liquid in the chamber. They are frequently used behind the pumps in water supply pipelines. Mostly they are cylindrical with a vertical and/or horizontal axis.

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 izotermic relation for a perfect gas

$$C = H_{air} * \forall^{\kappa}_{air} \tag{18.55}$$

in which Hair and "air are the absolute pressure head and the volume of the enclosed air, k is the exponent in the polytropic gas equation (k = 1.0 for an isothermal expansion, k = 1.4 for adiabatic expansion). m is the entrapped air volume, pair is the air pressure inside a chamber for which the Qair (air discharge) is taken from the dual-acting valve characteristics. Next chart shows characteristics of Pont&Mousson, Ventex dual-acting valve, diameter of 150mm.

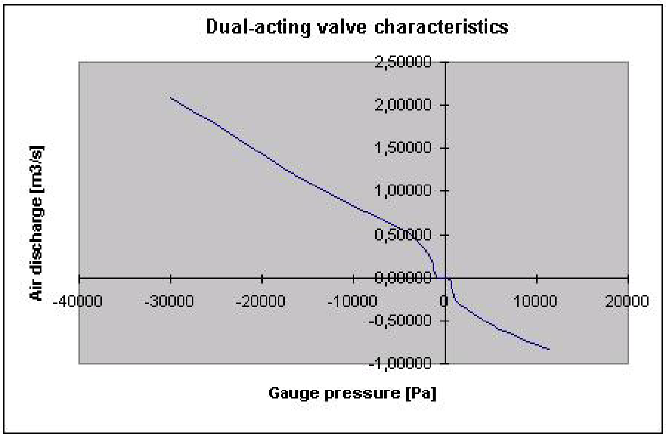


Figure 18.16 Dual-acting Valve Characteristics

When the pressure inside the Vented Air Chamber drops below the atmospheric pressure, dual-acting valve opens and the air flow into a chamber. 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.

Parameters:

- Node ID. Vented-air chamber can be located only in the "dead-end" node, i.e. node with only one adjacent pipe. Vented air-chambers are defined as junction nodes with an air-valve.
- Dual-acting Valve characteristics: Relation between gauge pressure and the air flow through the dual-acting valve.

In order to define the venter air chamber, use the Junction Editor and set the node to a "air valve" and use the valve equivalent diameter to represent the vented air chamber volume.

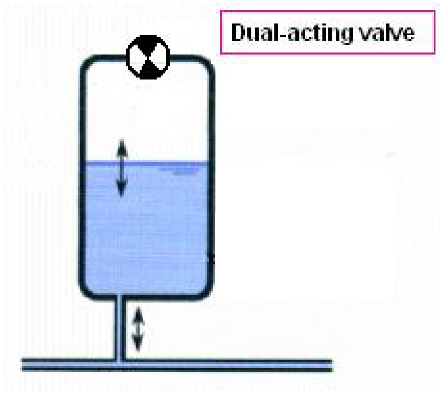


Figure 18.17 Vented Air-Chamber

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 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

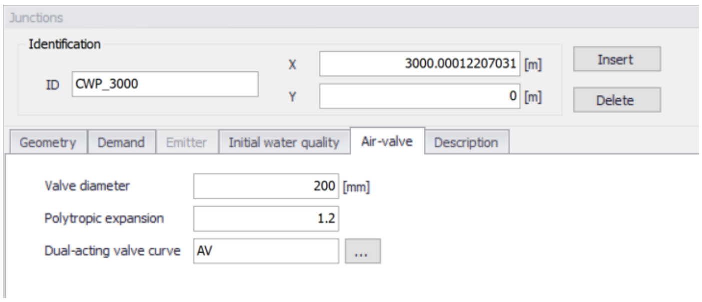


Figure 18.18 Air valve editor with water hammer settings

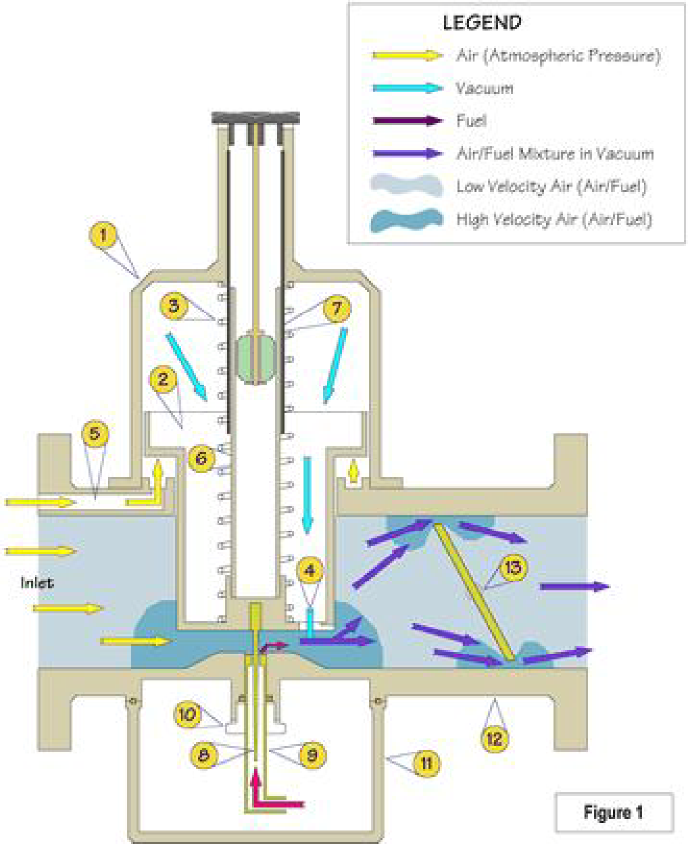


Figure 18.19 Air Valve

18.4.5 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).

INDEX