

MIKE URBAN

Water Distribution





PLEASE NOTE

COPYRIGHT

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

LIMITED LIABILITY

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

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

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

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

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





1	EPANET Input Description	11
1.1	Network Component Editors	11
1.1.1	Junction Editor	12
	Emitter	16
	Air-Valve	16
	Graphical Placement and Editing of Junction Nodes	17
1.1.2	Pipe Editor	18
	Warning	23
	Flow Direction	23
	Graphical Placement and Editing of Pipes	23
	Snap Tolerance	24
	Defining Curved Pipes	24
1.1.3	Pump Editor	25
	Pump Energy Settings	29
	Pump Water Hammer Settings (optional)	30
	Pump Variable Speed Settings (optional)	31
	Variable Pumps	32
	Flow Direction	33
	Graphical Placement and Editing of Pumps	33
1.1.4	Valve Editor	34
	Valve Water Hammer Settings (optional)	37
	Variable Valves	38
	Check Valves	38
	Flow Direction	38
	Graphical Placement and Editing of Valves	38
1.1.5	Tank Editor	39
	Graphical Placement and Editing of Tanks	44
1.1.6	Air-Chamber Editor	44
	Graphical Placement and Editing of Air-Chambers	47
1.2	Other Editors	48
1.2.1	Project Options Editor	48
1.2.2	Pressure Zone Editor	52
1.2.3	Energy Editor	53
1.2.4	Report Editor	53
1.2.5	Curves and Relations Editor	56
1.2.6	Boundary Conditions Editor	58
1.3	Network Demand	59
1.3.1	Distributed Demands	59
1.3.2	Demand Allocation Editor	64
	Geocoding	66
	Aggregation	68
1.3.3	Multiple Demand Editor	69
1.3.4	Demand Statistics	71
1.4	Extended Period Simulations	73
1.4.1	Simple Control Editor	74
	Multiple Controls	76
	Pipes with Check Valves	77
	Available Operational Controls	77



	Time Based Controls77
	Storage Tank Water Level Based Controls77
	Junction Node Pressure Based Controls77
1.4.2	Rule Based Controls	78
	Multiple Pumps, Valves (WD-Tools)81
1.4.3	Repetitive Profiles (Pattern) Editor	83
	Diurnal Patterns83
	Cyclic Profiles86
1.4.4	Time Settings Editor	87
	Time Units88
	Diurnal Curves and Patterns88
1.5	Water Quality Simulations	89
1.5.1	Water Quality Analysis Selection	89
1.5.2	Water Quality Analysis Parameters	90
1.5.3	Initial Water Quality Editor	91
1.5.4	Point Constituent Source Editor	93
	Point Constituent Source Time Patterns94
1.5.5	Reaction Rate Editor	95
	Global Settings95
1.5.6	Local Settings	96
	Tank Reaction Rate Assignments96
	Pipe Reaction Rate Assignments96
	Note97
1.5.7	Source Tracing	97
1.6	Tools	98
1.6.1	Send Data	98
1.6.2	Engineering Tables	98
1.6.3	Convert Asset Valves/Pumps to Model Valves/Pumps	100
1.6.4	Duplicate Pipe Attributes	101
MIKE URBAN WD User Guide		105
2	Using the Model	107
2.1	Overview	108
2.1.1	EPANET	108
	Pipe Flow Module	108
	Water Quality Module	108
2.1.2	WD-Tools	109
	Fire Flow Analysis	109
	Pipe Roughness Calibration	109
	PID Control	109
	Variable Speed Drive	110
	Extended Rule-Based Controls	110
	Demand Adjusted Analysis	110
	Pressure Zone Mapping	110
	Water Hammer	110
2.2	Application Basics	111
2.2.1	Defining the Model	111



	Constructing Complex Networks	113
	Roughness Coefficient Data	113
2.2.2	Input Data Requirements	113
	Project Data	113
	Network Data	113
	Extended Period Data	115
	Water Quality Data	115
	Fire Flow Data	116
	Water Hammer Data	116
2.3	Entering Data	116
2.3.1	Interactive Data Entry	116
2.3.2	Graphical Input	117
2.3.3	Importing Graphical Data	118
	Hiding and Showing Graphical Data	119
2.4	Demand Processing	119
2.4.1	Distributed Demands	119
	Developing Pipe Demand Coefficients	122
	Demand Allocation	124
	Demand Editing and Demand Scenarios	124
2.5	Performing an Analysis	125
2.5.1	Executing the Analysis	125
2.5.2	Batch simulations	126
2.6	Viewing Analysis Results	127
2.6.1	Comparing Alternative Solutions	128
	Different Demands	129
	Different Pipe Roughness	129
	Different Pumping Scenarios	129
	Effect of Pressure Reducing Valves	129
	Water Quality Parameters	129
2.6.2	Viewing the EPANET Analysis Results	129
2.6.3	Result Browser Window	130
2.6.4	Results Table	130
2.6.5	Thematic Mapping	130
2.6.6	Profile Plots	131
2.6.7	Time Series Plots	131
2.6.8	Link Q-H Plots	132
2.6.9	Fire Hydrant Q-H Plots	132
2.6.10	Current Operating Pump Q-H Point	132
2.6.11	Check for Pipes and Nodes	132
2.6.12	Animation	132
2.6.13	EPANET Error and Warning Messages	132
2.6.14	Store Results to Database	134
3	Water Distribution Tools (WD-Tools)	135
3.1	Fire Flow Analysis	135
3.1.1	Browsing Results	138
3.1.2	Tabular Results	138
3.1.3	Thematic Maps	138



3.2	Pipe Roughness Calibration	139
3.2.1	Evolutionary Algorithms - Theoretical Background	139
3.2.2	Introduction	141
3.2.3	Pipe Roughness Calibration	141
3.3	PID Control	145
3.3.1	Architecture of RTC Systems	145
3.3.2	MIKE URBAN vs. Real Life	147
3.3.3	The PID Algorithm	149
	Control Settings	151
	Set Point Settings	152
	PID Settings	152
3.4	Demand Adjusted Analysis	152
3.4.1	Low Pressure Systems	155
3.4.2	Intermittent Water Supply Systems or Modelling System Shutdown and Maintenance	157
3.5	Variable Speed Drive Pump	157
3.6	Extended Rule-Based Controls	157
3.7	Pressure Zone Mapping	157
3.8	Advisor	159
3.8.1	Pipe Criticality	159
3.8.2	Sustainability Analysis	162
3.8.3	Cost Analysis	165
	Pump Energy Table	165
	Pump Energy Graphs	166
3.8.4	Alarms and Violations	167
4	MIKE URBAN Water Hammer	171
4.1	Water Hammer Calculation	171
4.2	Theoretical Background	172
4.2.1	Description of Water Hammer Model	172
	Wave Speed	172
	Rigid Conduit	173
	Thick-Walled Elastic Conduit ($D/e \leq 10$)	173
	Thin-Walled Elastic Conduit ($D/e > 10$)	173
	Tunnels Through Solid Rock, Parmakian 1963	174
	Reinforced Concrete Pipe	174
	Diagrams	175
4.3	Numerical Scheme and Algorithm	176
4.3.1	Coefficients for the Numerical Scheme	177
4.3.2	Looped Network Solution Algorithm	178
4.3.3	Hydraulic Structures	180
4.4	Water Hammer Calculations	181
	Water Hammer Data Preparation	181
	Water Hammer Result Presentation	181
4.4.1	Running Water Hammer Simulations	182
	Initial Conditions	182
	Boundary Conditions	182
	Computational Parameters	182



4.4.2	Definition of Network Layout	183
	Branches	184
	Nodes	184
	Grid Points	185
	Computational Grid and Hydraulics Structures	185
	Specific Pipe Data	185
	Junction Node Demands	185
	Control Rules	185
	Specific Pump Data	185
	Specific Valve Data	191
	List of Supported Components	196
	List of Unsupported Component	196
4.4.3	List of Additional Components	196
4.4.4	Components Located in Nodes	197
	Tanks	197
	Air-Chambers	197
	Vented Air Chambers	199
	Air-Valve	201
4.4.5	Tutorial	202
5	EPANET	205
5.1	History	206
5.1.1	Analysis Methods	206
	Steady State Hydraulics	206
	Extended Period Hydraulics	207
	Water Quality Analysis	207
5.1.2	Applications of MIKE URBAN WATER	207
	Planning	208
	Design	208
	Operation	208
5.1.3	Skeletonization	208
5.2	The Water Distribution Network	210
5.2.1	Pipes	211
	Modeling Parallel Pipes	213
5.2.2	Pumps	214
	Pumping Rate Limits	216
	Controlling Pumps	217
	Modeling Pumps in Parallel and Series	217
5.2.3	Valves	220
	Pressure Reducing Valves (PRV)	220
	Pressure Sustaining Valves (PSV)	221
	Pressure Breaker Valves (PBV)	221
	Flow Control Valves (FCV)	221
	Throttle Control Valves (TCV)	221
	General Purpose Valves (GPV)	221
5.2.4	Minor Losses	221
5.2.5	Nodes	222
	Modeling Pumped Groundwater Wells	223
	Modeling Hydraulically Adjacent Storage Tanks	225



5.3	Time Patterns	225
5.4	Hydraulic Simulation Model	226
5.4.1	Water Age and Source Tracing	233
5.5	Water Quality Simulation Model	234
5.5.1	Advective Transport in Pipes	234
5.5.2	Mixing at Pipe Junctions	234
5.5.3	Mixing in Storage Facilities	235
5.5.4	Bulk Flow Reactions	235
5.5.5	Pipe Wall Reactions	237
5.5.6	Lagrangian Transport Algorithm	238
5.5.7	Tank Mixing Models	240
5.6	Model Calibration	243
5.6.1	Accuracy Concerns	243
	Pressure Differential Method	244
	Percent Pressure Differential Method	244
	Head Loss Differential Method	244
5.6.2	Reasons for Calibrating a Model	244
	Model Credibility	245
	Benchmark	245
	Predict Potential Problems	245
	Understand System Operations	245
	Uncover Errors	245
5.6.3	Calibration Model Data Requirements	246
	Simulation Type Considerations	246
	Data Acquisition	247
	Operational Review	247
	Recalibration Frequency	247
5.6.4	Calibration Simulations	248
	Initial Simulation	248
	Comparing Model Output	248
5.6.5	Model Adjustments	250
	Operational Data Adjustments	251
	Consumption Data Adjustments	251
	Network Data Adjustments	252
	Pipe Roughness Adjustments	252
	Number of Simulation Iterations	252



1 EPANET Input Description

This chapter provides an overview of the MIKE URBAN WATER data input dialog boxes. For a detailed overview on using this program, see Chapter 2.

The Model Manager User Guide will provide information on how to work with the various editing tools, loading of results etc.

1.1 Network Component Editors

The following sections describe the various network component editors (i.e., junctions, pipes, pumps, reservoirs, tanks, and valves). These network component editors allow you to define each component individually using a spearheads like data entry dialog box. This enables you to view and edit the attributes of all of the components that have been defined. These component editors also allow you to define a network model without a graphical layout, if that is necessary.

The network component editors are all available from the **EPANET** Menu, as shown in Figure 1.1.

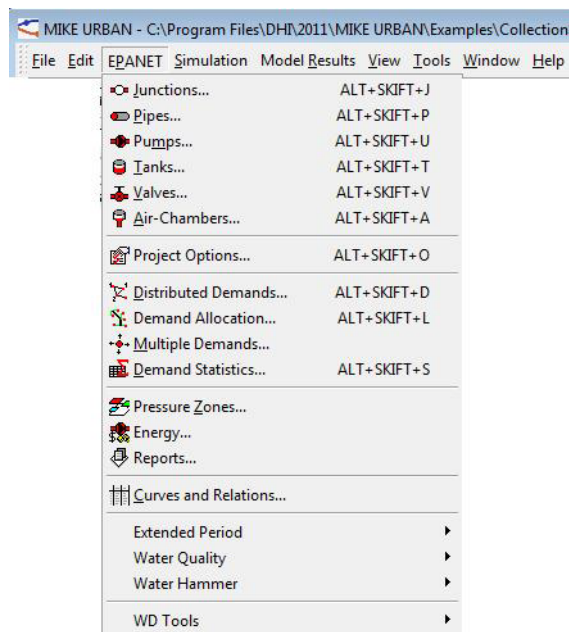


Figure 1.1 The EPANET Menu allow access to the network component editors

1.1.1 Junction Editor

The first step in defining a water distribution network is to define the junction nodes that define the interconnection between the pipes, pumps, valves, tanks, and reservoirs 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 points where pipe attributes (e.g., diameter, roughness, etc.) change.

Junction nodes are either defined interactively on the graphical Map window using the Junction tool, or by manual data entry using the Junction Editor dialog box as shown in Figure 1.2. The Junction Editor allows you to define the junction's ID, location, any external demand, and a description. The Junction Editor dialog box is reached by selecting **EPANET | Junctions**

Demand	Description	Category	Profile
0,04	<Null>	<Null>	P-1

Junction ID	Asset ID	Data source	Description	Elevation	Surface Ele	Pressure z	State	Den
10337	10337	<Null>		29,70	<Null>	<Null>	unmarked	<Nul
10480	10480	<Null>		14,27	<Null>	<Null>	unmarked	<Nul
10483	10483	<Null>		18,52	<Null>	<Null>	unmarked	<Nul

Figure 1.2 The Junction Editor allows you to define the junction nodes that define the inter connectivity between the water distribution network components

A list of the Junction Editor data entries for Figure 1.2 follows, with a short description given for each entry.

ASSET ID (optional)

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



JUNCTION ID (mandatory)

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 URBAN whenever a new junction node is placed into the list by pressing «Insert». When defining the junction nodes graphically on the Map window using the Add Junction tool, the junction ID is automatically defined.

X and Y COORDINATE (optional)

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

DESCRIPTION (optional)

This data entry allows you to enter a description identifying the junction 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 junction node location (such as database table or a database file name) in the asset management system.

STATUS (optional)

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

PRESSURE ZONE ID (optional)

This data entry allows you to define the ID of the pressure zone that the junction node lies within. The pressure zone ID can be used when distributing the zone demand as well as to graphically display different zones on the Map.

Selecting «...» allows you to display the Pressure Zone selection dialog box, where the appropriate pressure zone ID can be selected.



TYPE (mandatory)

This drop down selection list data entry allows you to define whether the junction node is a simple *junction*, or is an *emitter*, or an *air-valve*. By default, any junction nodes added are *junctions*.

DEMAND COEFFICIENT

The new field was added into the Junction editor. 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.

Example: the network has 3 nodes, where the demand coefficient is defined; these values are 10, 10, and 30. For each node, the weighted coefficient is calculated and based on it; the total network demand is distributed. The node, where the demand coefficient is not defined will get no demand from the total network demand: Q

$$q_i = \frac{Q_T}{\sum_{i=0,n} C_i} C_i \quad (1.1)$$

where:

q_i = node demand

Q_T = total network demand

c_i = demand coefficient

MINIMAL PRESSURE (optional)

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.

STATE (optional)

This drop down selection list data entry allows you to define whether the junction node is *unmarked* (i.e. an existing node already contained in the water distribution network), or is *marked* (i.e. one that is being considered for future expansion, etc.). By default, any junction nodes added are unmarked.

ELEVATION (mandatory)

This data entry defines the elevation above a common datum for the junction node, in units of ft. or m. This value is used to determine the pressure head at the node during a simulation. The default elevation is zero.

Junction nodes should have their elevation specified above zero datum (i.e., sea level) so that pressure computations can be carried out.

SURFACE ELEVATION (optional)

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



MULTIPLE DEMAND (optional)

Junctions may have more than one demand assigned to them. This function is particularly useful if the demand patterns of multiple water users are known for a given junction. It is also possible to assign separate patterns to the demands assigned to a given junction. The demands and demand patterns can be assigned from the Junction Editor, by selecting the Multiple option for base demand.

If no water is entering or leaving the network system at this junction node location, this entry should be specified as zero. 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 (e.g., from a groundwater extraction well), then a negative value should be specified.

This demand value will be adjusted during an extended period simulation according to the assigned demand pattern ID. Caution should be exercised if a groundwater extraction well is defined using this entry, since the groundwater extraction rate will be adjusted according to the assigned demand pattern ID.

Note that the baseline demand at this junction node can be (optionally) computed by globally defining the demand for the entire network and then having MIKE URBAN distribute this demand to each of the network nodes using the Distributed Demand dialog box. See the section titled “Distributed Demands” on page 59 for more information on computing distributed demands.

For extended period simulations, the baseline demand can be adjusted during the simulation due to a predefined time pattern factor. See the section titled “Repetitive Profiles (Pattern) Editor” on page 83 for more information.

DEMAND DESCRIPTION (optional)

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

DEMAND CATEGORY (optional)

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.

DEMAND PATTERN ID (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. See the section titled “Repetitive Profiles (Pattern) Editor” on page 83 for more information on demand patterns.

Emitter

The following data entries defines parameters of an emitter located at the junction node. 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).

A list of the Emitter data entries for Figure 1.3 follows, with a short description given for each entry.

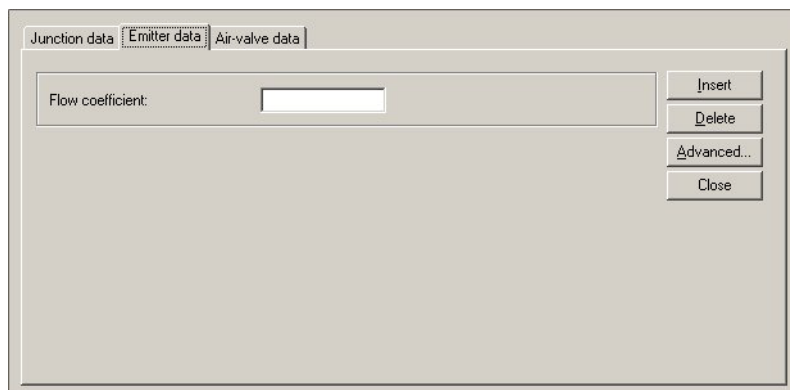


Figure 1.3 Emitter data entries

FLOW COEFFICIENT (mandatory)

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 at 1 psi (1 meter) pressure drop

Air-Valve

The following data entries defines parameters of an air-valve located at the junction node and are used only in case of a water hammer analysis. A list of the Air-Valve data entries for Figure 1.3 follows, with a short description given for each entry. See the section titled “Water Hammer Calculation” on page 171 for more information on demand patterns.

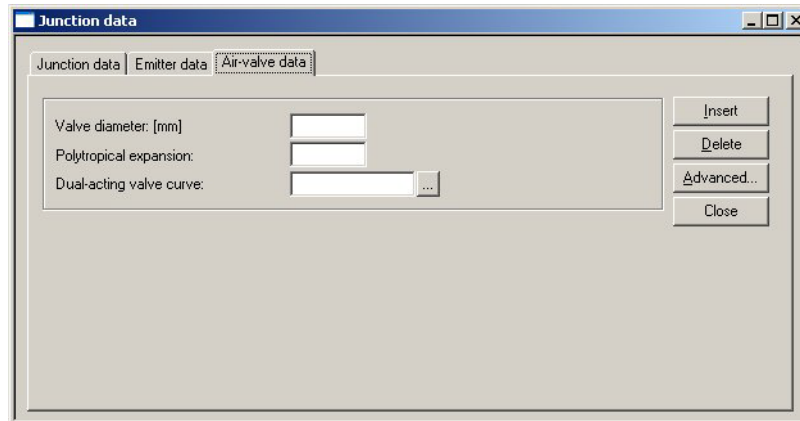


Figure 1.4 Air-Valve data entries

VALVE DIAMETER (mandatory)

This data entry allows you to define the diameter of the air-valve. See the section titled “Specific Valve Data” on page 191 for more information on demand patterns.

POLYTROPIC EXPANSION FACTOR (mandatory)

This data entry allows you to define the polytropic exponent in the polytropic gas equation.

By default this data entry is $k=1.2$. See the section titled “Specific Valve Data” on page 191.

DUAL-ACTING VALVE CHARACTERISTICS CURVE (mandatory)

This data entry allows you to define the ID of the dual-acting valve characteristics curve. This is the relation between gauge pressure and the air flow through the dual-acting valve. See the section titled “Specific Valve Data” on page 191.

Graphical Placement and Editing of Junction Nodes

From the Map window, the Add Junction tool can be used to add a junction node. As nodes are added graphically to the Map window, their X, Y locations are stored into the MIKE URBAN database. While placing a new node, if the user clicks on an existing node MIKE URBAN will display an error message informing the user that an invalid location had been selected.

If desired, from the Map window, the user can move an existing junction node using the ‘select’ tool. By selecting the node and holding down the left mouse button, the node can be dragged to a new location. As the node is dragged, its connecting links (i.e., pipes, pumps, and valves) rubber band along with it.

To delete an existing junction node graphically, select the node using the ‘Select’ tool and then press «Delete». The selected node and all its connect-

ing links will then be deleted. To edit a junction node, double click on the node using the Select tool. The Junction Editor will then be displayed allowing you to edit the attributes of the selected node.

1.1.2 Pipe Editor

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 'Add pipe' tool, or by manual data entry using the Pipe Editor dialog box as shown in Figure 1.5

Pipe ID	Description	CDate	Constructi	Asset ID	Data sourc	Demand co	Demand
1447	89193	01-01-2005	<Null>	<Null>	<Null>	1,000	1,0
1448	89291	01-01-2005	<Null>	<Null>	<Null>	1,000	1,0
1449	89292	01-01-2005	<Null>	<Null>	<Null>	1,000	1,0

Figure 1.5 The Pipe Editor allows you to define the pipes that make up the water distribution network

The Pipe Editor allows you to define the pipe's ID, diameter, loss coefficients, nodal connectivity, description, and other attributes. The Pipe Editor dialog box is reached by selecting **EPANET | Pipes**.

A list of the Pipe Editor data entries for Figure 1.5 follows, with a short description given for each entry.



ASSET ID (optional)

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

PIPE ID (mandatory)

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

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

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

FROM NODE, TO NODE (mandatory)

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.

The Node Type pull-down selection list allows the user to specify what type of node is connected to the end of the pipe. Then, choosing «...» will display the Select Node selection dialog box from which the user can select the appropriate node. Or, choosing «Pick» allows the user to graphically select the node from the Map window. To reverse the order of the nodes, choose «Advanced - Swap Nodes».

If 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.

DESCRIPTION (optional)

This data entry allows you to enter a description identifying the pipe 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 pipe location (such as database table or a database file name) in the asset management system.



STATUS (optional)

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.

PRESSURE ZONE ID (optional)

This data entry allows you to define the ID of the pressure zone that the pipe lies within. The pressure zone ID can be used when distributing the zone demand as well as to graphically display different zones on the Map.

Selecting «...» allows you to display the Pressure Zone selection dialog box, where the appropriate pressure zone ID can be selected.

LENGTH (mandatory)

This data entry defines the pipe length, in the unit of your choice. It is possible to define a specific pipe length—independent of the pipe network layout. Otherwise, MIKE URBAN recomputes the pipe length based upon the pipe layout.

A global recomputing of pipe lengths can be performed by using the functionality under the Advanced button on the Pipe Editor.

DIAMETER (mandatory)

This data entry defines the internal diameter of the pipe, in inches or mm. 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.

MATERIAL (optional)

This option allows the user to define the material of pipe construction. The Pipe Material is defined by a “string” and the pipe roughness can be globally assigned based upon the pipe material and pipe construction year, for example.

CONSTRUCTION YEAR (optional)

This option allows the user to define the age of the pipe. Pipe age is defined by a float number, meaning that alpha characters are not allowed to be entered in this field. There are two common ways the user can define pipe age, either by year of construction year, or number of years the pipe has been in service. The choice is left to the user.

WALL THICKNESS (optional)

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.

WAVE SPEED (water hammer mandatory)

This field is used to define the celerity of the pressure wave. This field is used only in case of water hammer calculations.



FORMULATION (read-only)

This field is used to display the current friction loss formulation type, such as Darcy-Weisbach, Hazen-Williams, or Manning. To change the friction formulation, use the Project Options dialog box.

ROUGHNESS (mandatory)

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. The roughness type is specified by the user within the Project Options dialog box. Choosing «...» will display the Select Pipe Roughness Coefficient selection dialog box, allowing the user to select the appropriate roughness value to use.

LOSS COEFFICIENT (optional)

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 than one minor loss component exists along the pipe, then the sum of the corresponding minor loss coefficients should be entered.

PRESSURE NORMAL (optional)

This field is used to define the normal operating pressure for a pipe. This is an optional field and can be used when the calculated pressures are distributed to pipes.

BULK COEFF. and WALL COEFF. (optional)

Locally defined reaction rates. Please refer to section on reaction rates for further.

DEMAND COEFFICIENT 1,2 (optional)

MIKE URBAN allows the user to recalculate the nodal water demands based upon the total network demand using two methods: the Method of Pipe Lengths and the Method of Two Coefficients. 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 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 pre-defined demand coefficients.

These two data entries define the demand coefficients, k_{1i} and k_{2i} which are used in these two computational methods. Each method proportions the total network demand along the pipe by splitting the computed demand between the two nodes at each end of the pipe.



The Method of Pipe Lengths computes the total water demand assigned to the current pipe (which is then split between the starting and ending nodes) as:

$$q_{pi} = \frac{(Q)l_i k_{li}}{\Sigma(k_{li}l_i)} \quad (1.2)$$

The Method of Two Coefficients computes the total water demand assigned to the current pipe (which is then split between the starting and ending nodes) as:

$$q_{pi} = \frac{(Q)k_{li}k_{2i}}{\Sigma(k_{li}k_{2i})} \quad (1.3)$$

where:

q_{pi} =total water demand applied to the pipe, split between the two end nodes.

Q = total network water demand. Note that the total network water demand is defined in the Distributed Demands dialog box, available from the Edit Menu.

l_i = pipe length

See the section titled “Distributed Demands” on page 59 for more information on computing distributed demands.

Note that there is also Demand Coefficient 3, 4 which is available only in the table grid.

STREET NAME (optional)

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.

STATE (optional)

This drop down list box data entry allows you to define whether the pipe is *unmarked* (i.e. a pipe already contained in the water distribution network), or is *marked* (i.e. one that is being considered for future expansion, etc.). By default, any pipes added are unmarked.

CHECK VALVE (optional)

This check box data entry defines the presence of a check valve 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. By default, no check valve is present in a pipe.



CLOSED (optional)

This check box data entry allows the user to toggle the OPEN and CLOSED status of the pipe. Choosing CLOSED effectively removes the pipe from the network system.

Warning

Note that you cannot set the pipe status of a pipe containing a check valve. 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).

Flow Direction

Note that the flow direction for pipes is assumed to be from the starting (upstream) node to the ending (downstream) node. The order in which these nodes are specified for each pipe is critical for a proper network representation. To reverse the order of the nodes, choose «Swap Nodes».

If 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.

Graphical Placement and Editing of Pipes

From the Map window, the Add Pipe tool can be used in different ways to place a pipe. If the user clicks on or near an existing node, MIKE URBAN will snap to the node and treat this as a *starting* node. A rubber-banding line will then be drawn from this node, representing the pipe, while the user selects the *ending* node. Then, if the user clicks on or near an existing node, it will snap to that node and treat the selected node as the *ending* node and then draw the pipe. If the user clicks anywhere else, it will place an *ending* node at the selected location.

While selecting the *starting* node, if the user clicks with the 'Add pipe' tool in the Map window somewhere else other than a node or a pipe, MIKE URBAN will place a *starting* node at the clicked location.

If the user selects an existing pipe while selecting either the *starting* or *ending* nodes, MIKE URBAN will display a query dialog, as shown in Figure 1.6, asking the user whether to split the pipe and insert a node at the selected location. If the user selects to insert a node into the pipe, the pipe will be broken into two segments at the point of selection with a new node added. The length of the two pipe segments will be adjusted accordingly so that the total length of the two segments equals that of the original pipe.

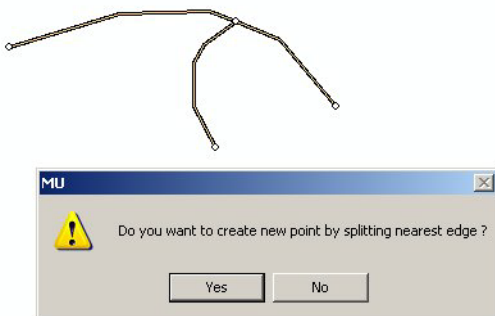


Figure 1.6 When clicking on an existing pipe with the Add Pipe tool, MIKE URBAN will ask whether to insert a junction node into the existing pipe for connecting the new pipe

To delete an existing pipe graphically, select the pipe using the 'Select' tool and then select «Delete Selected Features». The selected pipe will then be deleted. To edit a pipe, double click on the pipe using the 'Select' tool. The Pipe Editor will then be displayed allowing you to edit the attributes of the selected pipe.

Snap Tolerance

As you graphically layout pipes in the Map window by clicking on existing nodes, MIKE URBAN will snap to the nearest node if it is within a specified snap tolerance. However, if the cursor is not placed directly on top of the node or within the specified snap tolerance, MIKE URBAN may not snap to the node but instead will add a new junction node.

A snap tolerance is specified in the Configuration dialog box, available from the Tools Menu. See the section titled "Snap Tolerance" on page 24. This snap tolerance is specified in screen pixels, and defines how close the cursor must be to the node in order to snap to the node. Anywhere outside of this snap tolerance and a new junction node is created.

Defining Curved Pipes

MIKE URBAN uses multi-segmented polylines to represent curved pipes. Intermediate vertices are used to define the curvature of the pipe. A curved pipe is defined graphically in the Map window. To define a curved pipe:

1. Select the 'Add Pipe' tool from the Components toolbar and click on the starting point in the Map window.
2. After selecting the starting point, construct the curve of the pipe by clicking in the Map window at the points the curvature changes to create intermediate vertices so that an arc or a curved line is formed. Double-click on the last position to end the curved pipe. If the last position is an



existing node, the end point of the curved pipe will snap to the existing node.

To calculate the pipe length based upon the curved layout, press the Advanced button on the Pipe Editor and select either 'Recompute selected pipe length' or 'Recompute all pipes length'. MIKE URBAN will then recompute the pipe length for either the selected pipes or all the pipe based upon the pipe layout.

1.1.3 Pump Editor

Pumps are used to raise the hydraulic head of water. Pumps are represented as short links of negligible length. MIKE URBAN 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 Add Pump tool, or by manual data entry using the Pump Editor dialog box as shown in Figure 1.7.

The Pump Editor allows you to define the pump's ID, pump power curve, status, nodal connectivity, description, and other attributes. The Pump Editor dialog box is reached by selecting **EPANET | Pumps**.

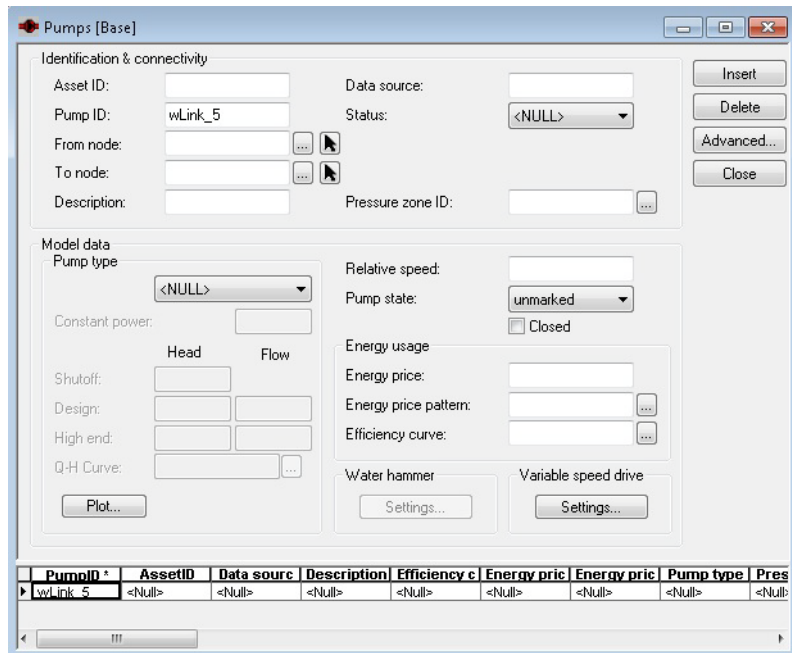


Figure 1.7 The Pump Editor allows you to define the pumps used in the water distribution network



A list of the Pump Editor data entries for Figure 1.7 follows, with a short description given for each entry.

ASSET ID (optional)

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).

PUMP ID (mandatory)

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

A new pump ID is automatically suggested by MIKE URBAN whenever a new pipe is placed into the list by pressing «Insert». When defining the pipes graphically on the Map window using the Add Pump tool, the pump ID is automatically defined.

FROM NODE, TO NODE (mandatory)

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

The Node Type pull-down selection list allows the user to specify what type of node is connected to the end of the pump. Then, choosing «...» will display the Select Node selection dialog box from which the user can select the appropriate node. Or, choosing «Pick» allows the user to graphically select the node from the Map window.

Pumped flow is always assumed to move from the starting (upstream) node to the ending (downstream) node. To reverse the order of the nodes, choose «Swap Nodes».

DESCRIPTION (optional)

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.

DATA SOURCE (optional)

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 (optional)

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.



PRESSURE ZONE ID (optional)

This data entry allows you to define the ID of the pressure zone that the pump lies within. The pressure zone ID can be used when distributing the zone demand as well as to graphically display different zones on the Map.

Selecting «...» allows you to display the Pressure Zone selection dialog box, where the appropriate pressure zone ID can be selected.

PUMP TYPE (mandatory)

This option button data entry defines the pump operating characteristics. There are four options available to define the pump specifications. By default, a constant power pump is selected.

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

1xQ-H Point Pump Curve pump, as shown in Figure 1.8, 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.

The *Design Head* is in units ft. or m, and are by default zero. The *Design Flow* is in the user-specified units, and by default zero.

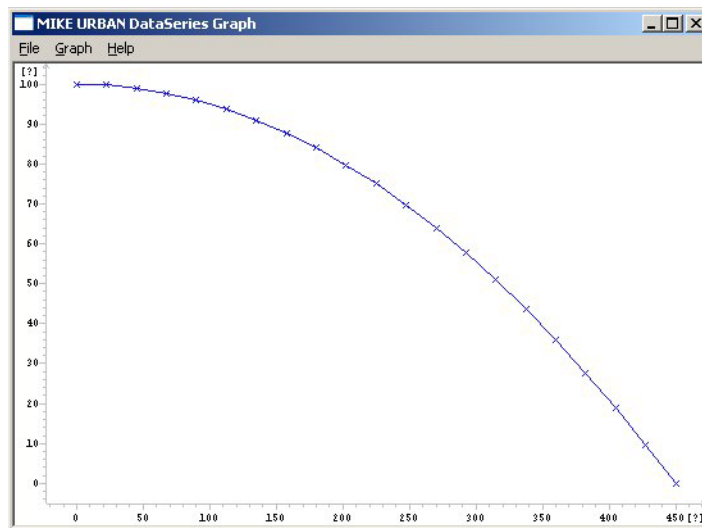


Figure 1.8 A 1-point pump curve contains no extended flow range

3xQ-H Point Pump Curve, as shown in Figure 1.10 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. The *Design Flow* is corresponding flow rate. 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.

Q-H Pump Curve can alternatively. The *Q-H Pump Curve* is created by providing either a pair of head-flow points, or four or more such points. MIKE URBAN creates the pump curve by connecting the points with straight line segments. The Q-H pump curve should be created in the Curve Editor found in the Edit menu. Once the Q-H pump curve is created, it can then be assigned to a pump in the Pump Editor.

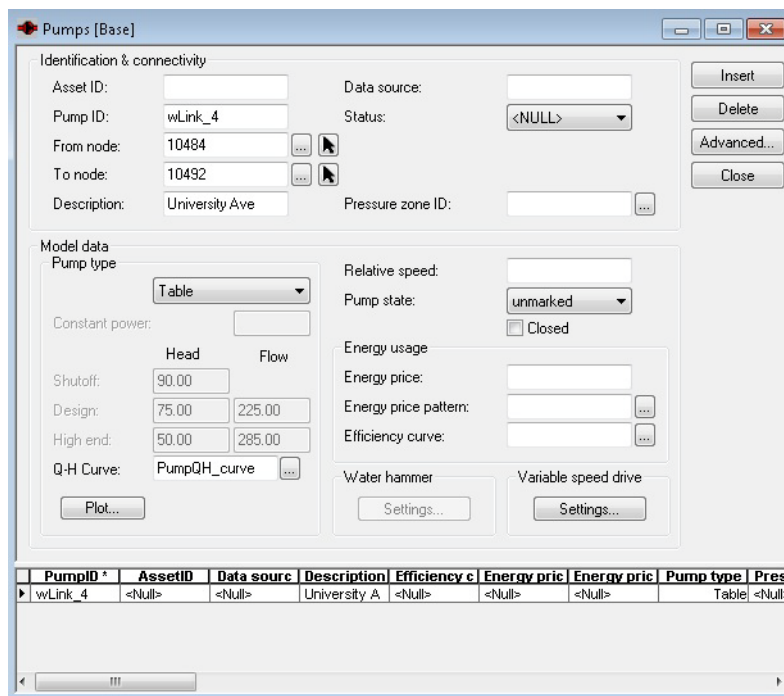


Figure 1.9 A Q-H Table used to define the multi-point pump curve



PumpID *	AssetID	Data sourc	Description	Efficiency c	Energy pric	Energy pric	Shutoff	Desi
wLink_4	<Null>	<Null>	University A	<Null>	<Null>	<Null>	90,00	

Figure 1.10 A 3-point pump curve with no extended flow range.

RELATIVE SPEED (optional)

Note that the 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.

PUMP STATE (optional)

This drop down list box data entry allows you to define whether the pump is *unmarked* (i.e. a pump already contained in the water distribution network), or is *marked* (i.e. one that is being considered for future expansion, etc.). By default, any pumps added are unmarked.

CLOSED (optional)

This check box data entry allows the user to toggle the OPEN and CLOSED status of the pump. Choosing CLOSED effectively removes the pump from the network system.

Pump Energy Settings

MIKE URBAN is capable of modelling the cost of operating pumps. Within the Pump Energy Editor, the user can define a method for cost calculation.



ENERGY PRICE (optional)

The user defines an energy price (\$/kw-hour) to be used. In this method, MIKE URBAN 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 project's Energy Options will be used

ENERGY PRICE PATTERN (optional)

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 (optional)

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.

Pump Water Hammer Settings (optional)

MIKE URBAN is capable of modelling pumps under the water hammer flow conditions. Within the Pump Water Hammer Editor, the user can define the required pump parameters as shown in Figure 1.11.

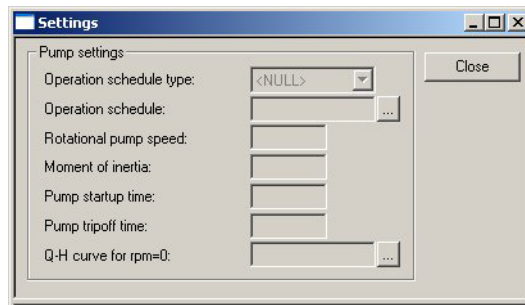


Figure 1.11 A pump water hammer settings

OPERATIONAL SCHEDULE TYPE (mandatory)

This list box data entry defines the pump operating schedule. There are three options available to define the pump schedule (rotational speed of pump N versus time).

- a. The pump is controlled by a pump operation schedule (N-time) curve.
- b. The 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.



- c C) The pump is primarily stopped (N equals zero) until time of pump start-up is reached, then pump start-up equation is applied.

OPERATIONAL SCHEDULE (mandatory)

This data entry is used to select the pump operation schedule ID. To define the pump schedule, use the Curves and Relations dialog box.

ROTATIONAL PUMP SPEED (mandatory)

This data entry is used to specify the initial rotational speed of a pump in rpm (rotation per minute).

MOMENT OF INERTIA (optional)

The moment of inertia is the resistance of a rotating pump body to the change of its rotational speed. The moment of inertia is defined in $[\text{kg}\cdot\text{m}^2]$.

PUMP TORQUE (optional)

This data entry is used to specify the ID of the pump torque curve. The pump torque is the relation between the pump torque (Newton meter) versus pump flow (flow units).

MOTOR TORQUE (optional)

This data entry is used to specify the ID of the motor torque curve. The motor torque is the relation between the motor torque (Newton meter) versus pump rotational speed (rotation per minute).

PUMP START-UP TIME (mandatory)

This data entry is used to define the time level at which the pump will start-up.

PUMP TRIP-OFF TIME (mandatory)

This data entry is used to define the time level at which the pump will trip-off. The pump is operated based on its operational schedule till that time.

Q-H CURVE FOR RPM = 0 (optional)

This data entry is used to define the Q-H curve for the rotational speed $N=0$ i.e. the relation between the flow and pump head when the pump is stopped. This data entry can help to define what happens for the positive and negative flow through the pump.

Pump Variable Speed Settings (optional)

MIKE URBAN is capable of modelling the variable pump speed during the extended period analysis. Within the Pump Variable Speed Editor, the user can define the required pump parameters as shown in Figure 1.12.

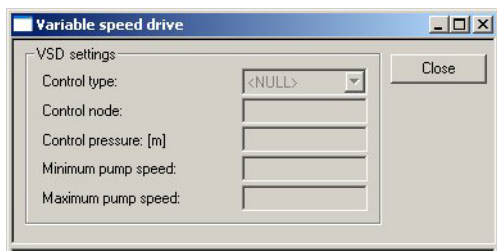


Figure 1.12 A variable pump speed settings

CONTROL TYPE (mandatory)

This list box data entry defines the variable pump speed control mode. There are two options available.

A) The pump is controlled by a pressure at the pump downstream node.

B) The pump is controlled by a pressure at any remote node (junction node or a tank with a variable water level).

Please note that it is possible to use this control also for the downstream node by specifying the downstream node ID as the "Control Node".

The difference between "downstream" and "remote" node control is in the numerical solution of the variable pump speed. In case of "downstream" node control, the control pressure settings are built into the main matrix solver while in the case of "remote" node control the program is using an iterative approach to find the pump speed.

CONTROL NODE (mandatory)

This data entry is used to define the control node ID.

CONTROL PRESSURE (mandatory)

This data entry is used to define the control node pressure.

MINIMUM PUMP SPEED (mandatory)

This data entry is used to define the minimum relative pump speed. Please note that this data entry is not used by in case of "downstream node control".

MAXIMUM PUMP SPEED (mandatory)

This data entry is used to define the maximum relative pump speed. Please note that this data entry is not used by in case of "downstream node control".

Variable Pumps

Many times, when performing an extended period simulation, it is desirable to model a variable pump. A variable pump can vary its speed setting and/or change its status to open or closed during a simulation. A variable pump is modelled by defining the pump's initial settings using the Pump Status previously described and then modifying the pump's operation during the



extended period simulation using the Control Editor. The Control Editor is available from the Extended Menu. See the section titled “Simple Control Editor” on page 74 for additional information.

A special case if a variable pump is VSD (variable speed drive) pump where the rotational speed of a pump is automatically adjusted by the program in order to maintain the control node pressure at the required level. MIKE URBAN WATER can model VSD pump using VSD settings in the Pump Editor or using the PID (Proportional Integral Differential Control) algorithm in the RTC (Real-Time Control) module.

Flow Direction

Note that the flow direction for pumps is assumed to be from the starting (upstream) node to the ending (downstream) node. The order in which these nodes are specified is critical for a proper network representation of the pump. To reverse the order of the nodes, choose «Swap Nodes».

Graphical Placement and Editing of Pumps

From the Map window, the ‘Add Pump’ tool can be used to place a pump. When you want to insert a pump onto an existing pipe, you must insert it as a new component.

To insert a pump onto the existing pipe:

1. Split the pipe by using Split tool from the Components toolbar.
2. Split the pipe once more to create a sections for 3 pipes.
3. Select the middle section by Select Feature tool and delete it by using the Delete Selected tool.
4. Select the Pump tool from the Components floating toolbar and click on the beginning pipe node in the Horizontal Plan window where you want to place the valve and double click on the ending node. This will create a new pump.

Another method of using the ‘Add Pump’ tool to add a pump is, from the Map window, select a *starting* node and then an *ending* node. Clicking on or near an existing node, MIKE URBAN will then snap to the selected node and a rubber-banding line will then be drawn from this node, representing the *starting* node, while the user selects the *ending* node. After selecting the *ending* node, MIKE URBAN will then place a pump between the two selected nodes.

In addition, if the user clicks with the ‘Add Pump’ tool in the Map window somewhere else other than a node or a pipe, MIKE URBAN will place a *starting* node at the clicked location. A rubber-banding line will then be drawn from this node, representing the pump, while the user selects the *ending* node. If the user clicks on or near an existing node, it will treat the selected node as the *ending* node. If the user clicks anywhere else, it will place an *ending* node



at the selected location. If the user selects an existing pipe, an error message will be displayed stating that an invalid *ending* node was selected.

To delete an existing pump graphically, select the pump using the Select tool and then select «Delete Selected». The selected pump will then be deleted. To edit a pump, click on the pump using the Open Editor tool. The Pump Editor will then be displayed allowing you to edit the attributes of the selected pump.

1.1.4 Valve Editor

Valves control the flow or pressure of water from one junction node to another. Valves are represented as short links of negligible length. Note that valve pressure settings are pressures (e.g., psi or m) and not total head (or hydraulic gradeline elevation). Also, valves cannot be directly connected to reservoir or storage tank nodes.

Valves are either defined interactively on the Map window using the Add Valve tool, or by manual data entry using the Valve Editor dialog box as shown in Figure 1.13. 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 EPANET|Valves.

Valve ID *	AssetID	Data source	Description	Diameter	Loss coeffi	Pressure z	Settini
wLink_1	<Null>	<Null>	<Null>	<Null>	<Null>	<Null>	<Null>

Figure 1.13 The Valve Editor allows you to define the valves used in the water distribution network

A list of the Valve Editor data entries for Figure 1.13 follows, with a short description given for each entry.



ASSET ID (optional)

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

VALVE ID (mandatory)

This data entry is used to specify an ID which uniquely identifies the valve link. The valve ID acts as a unique lookup key that identifies the link from all other links. A link can be a pipe, pump, or valve. Therefore, no two links may have the same link ID. However, a link and node (i.e., junction, reservoir, or tank) 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 URBAN whenever a new valve is placed into the list by pressing «Insert». When defining the valves graphically on the Map window using the Add Valve tool, the valve ID is automatically defined.

FROM NODE, TO NODE (mandatory)

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.

The Node Type pull-down selection list allows the user to specify what type of node is connected to the end of the pump. Then, choosing «...» will display the Select Node selection dialog box from which the user can select the appropriate node. Or, choosing «Pick» 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. To reverse the order of the nodes, choose «Swap Nodes».

DESCRIPTION (optional)

This data entry allows you to enter a description identifying the valve 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 valve location (such as database table or a database file name) in the asset management system.

STATUS (optional)

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, valve status is undefined.



PRESSURE ZONE ID (optional)

This data entry allows you to define the ID of the pressure zone that the valve lies within. The pressure zone ID can be used when distributing the zone demand as well as to graphically display different zones on the Map.

Selecting «...» allows you to display the Pressure Zone selection dialog box, where the appropriate pressure zone ID can be selected.

VALVE TYPE (mandatory)

This list box data entry defines the valve operating characteristics. There are six options available to define the valve operating characteristics. By default a pressure reducing valve (PRV) is selected.

A *Pressure Reducing Valve* (PRV) limits the pressure at the downstream node to not exceed a preset value when 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.

A *Pressure Sustaining Valve* (PSV) 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.

A *Pressure Breaker Valve* (PBV) forces a specified pressure loss to occur across the valve. Flow can be in either direction through the valve.

A *Flow Control Valve* (FCV) limits the flow through a valve to a specified amount. The program will produce a warning message if this flow cannot be maintained without having to add additional head at the valve.

A *Throttle Control Valve* (TCV) is used to simulate partially closed valves by adjusting the minor head loss coefficient of the valve. A relationship between the degree to which the valve is closed and the resulting head loss coefficient is typically available from the valve manufacturer.

A *General Purpose Valve* (GPV) 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 setting is the ID of a Headloss Curve.

SETTING (mandatory)

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 loss coefficients for TCVs.



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

VALVE CURVE (water hammer mandatory)

This data entry is used to specify an valve characteristics curve for the water hammer analysis. The valve characteristics curve is the relation between the valve flow coefficient (Cd or Kv) versus the valve opening (%). See the section titled “Specific Valve Data” on page 191 for additional information.

DIAMETER (mandatory)

This data entry is used to specify an valve diameter (inch or mm).

LOSS COEFFICIENT (optional)

This data entry specifies the minor loss coefficient for a fully opened valve. The default loss coefficient is 0. Choosing «...» will display Select Minor Loss Coefficient selection dialog box, allowing the user to select the appropriate minor loss coefficient to use.

VALVE STATE (optional)

This drop down list box data entry allows you to define whether the valve is *unmarked* (i.e. a valve already contained in the water distribution network), or is *marked* (i.e. one that is being considered for future expansion, etc.). By default, any valves added are unmarked.

CLOSED (optional)

This option check box data entry allows the user to toggle the OPEN and CLOSED status of the valve. Choosing CLOSED effectively removes the valve from the network system.

Valve Water Hammer Settings (optional)

MIKE URBAN is capable of modelling valves under the water hammer flow conditions. Within the Valve Water Hammer Editor, the user can define the required valve parameters as shown in Figure 1.11.

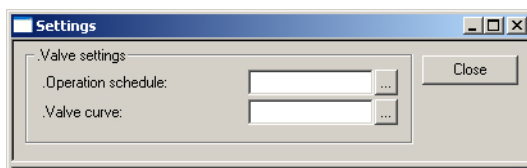


Figure 1.14 A valve water hammer settings

OPERATIONAL SCHEDULE (mandatory)

This data entry is used to select the valve operation schedule ID. The operational schedule is defined as valve opening (%) versus time. To define the valve schedule, use the Curves and Relations dialog box.



Variable Valves

Many times, when performing an extended period simulation, it is desirable to model a variable valve. A variable valve can vary its status to open or closed during a simulation. A variable valve is modelled by defining the valve's initial status using the Valve Editor and then modifying the valve's status during the extended period simulation using the Control Editor. The Control Editor is available from the Extended Menu.

See the section titled *Control Editor* or *Rule Based Control Editor* on -74 for additional information.

Check Valves

Note that check valves are defined within the Pipe Editor.

Flow Direction

Note that the flow direction for valves is assumed to be from the starting (upstream) junction node to the ending (downstream) junction node. The order in which junction nodes are specified for each valve is critical for a proper network representation of the valve. To reverse the order of the junction nodes, choose «Swap Nodes».

Graphical Placement and Editing of Valves

From the Map window, the Add Valve tool can be used to place a Valve. When you want to insert a Valve onto an existing pipe, you must insert it as a new component. To insert a Valve onto the existing pipe:

1. Split the pipe by using Split tool from the Components toolbar.
2. Split the pipe once more to create a sections for 3 pipes.
3. Select the middle section by Select Feature tool and delete it by using the Delete Selected tool.
4. Select the Valve tool from the Components floating toolbar and click on the beginning pipe node in the Horizontal Plan window where you want to place the valve and double click on the ending node. This will create a new Valve.

Another method of using the Add Valve tool to add a Valve is, from the Map window, select a *starting* node and then an *ending* node. Clicking on or near an existing node, MIKE URBAN will then snap to the selected node and a rubber-banding line will then be drawn from this node, representing the *starting* node, while the user selects the *ending* node. After selecting the *ending* node, MIKE URBAN will then place a Valve between the two selected nodes.

In addition, if the user clicks with the Add Valve tool in the Map window somewhere else other than a node or a pipe, MIKE URBAN will place a *starting*



node at the clicked location. A rubber-banding line will then be drawn from this node, representing the Valve, while the user selects the *ending* node. If the user clicks on or near an existing node, it will treat the selected node as the *ending* node. If the user clicks anywhere else, it will place an *ending* node at the selected location. If the user selects an existing pipe, an error message will be displayed stating that an invalid *ending* node was selected.

To delete an existing Valve graphically, select the Valve using the Select tool and then select «Delete Selected». The selected Valve will then be deleted. To edit a Valve, click on the Valve using the 'Open Editor' tool. The Valve Editor will then be displayed allowing you to edit the attributes of the selected Valve.

1.1.5 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 modelled 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 a infinite source of water (for the sake of the modelling 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, or by manual data entry using the Tank Editor dialog box as shown in Figure 1.15. The Tank Editor allows you to define the reservoir's ID, location, pressure zone, water surface elevation, and a description. The Tank Editor dialog box is reached by selecting **EPANET | Tanks**.

Figure 1.15 The Tank Editor allows you to define the storage tank nodes that supply water to the water distribution network

A list of the Tank Editor data entries for Figure 1.15 follows, with a short description given for each entry.

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).

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. 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 URBAN whenever a new tank node is placed into the list by pressing «Insert». When defining the tank nodes graphically on the Map window 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 URBAN 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.

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 junction node will be displayed in the Map window. When defining the junction nodes graphically on the Map window using the Add Tank tool, the X, Y location is automatically entered.

DESCRIPTION (optional)

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.

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.

PRESSURE ZONE ID (optional)

This data entry allows you to define the ID of the pressure zone that the tank node lies within. The pressure zone ID can be used when distributing the zone demand as well as to graphically display different zones on the Map.

Selecting «...» allows you to display the Pressure Zone selection dialog box, where the appropriate pressure zone ID can be selected.

TANK TYPE (mandatory)

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

TANK MIXING (optional)

MIKE URBAN 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 behaviour to assume, requires no extra parame-



ters 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 COEFF. (optional)

Locally defined reaction rates. Please refer to section on reaction rates for further.

BASE ELEVATION (mandatory)

This data entry defines the bottom elevation, in units of ft. or m, of the storage tank where the water level in the storage tank is zero, as shown in Figure 1.16. The default elevation is zero.

FIXED HGL (optional)

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 (optional)

The ID label of a time pattern used to model time variation in the tank's (reservoir's) total head. Leave blank if none applies. This property is useful if the



reservoir represents a tie-in to another system whose pressure varies with time.

TANK SIZE (mandatory)

This option button data entry selects the type of storage tank being defined. By default, a circular tank is defined. A the elevation-volume relationship for a tank of variable geometry can also 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.

DIAMETER or WIDTH and LENGTH (mandatory)

This data entry allows you to define the tank chamber size (ft or m).

VOLUME CURVE (mandatory)

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.

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 1.16.

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 1.16.

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 1.16.

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 1.16.

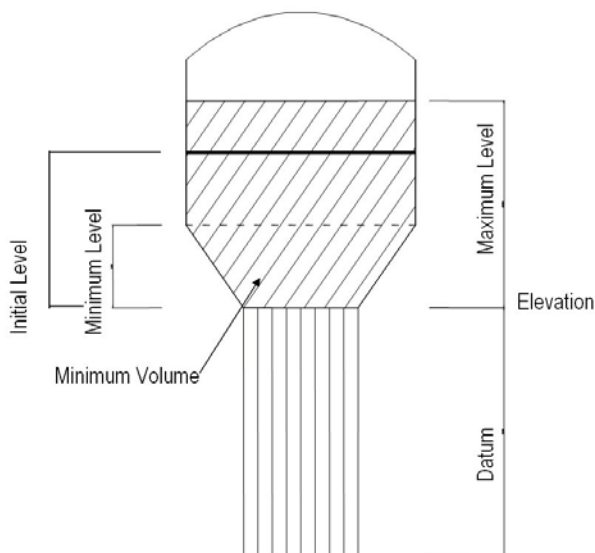


Figure 1.16 Definition of storage tank levels

Graphical Placement and Editing of Tanks

From the Map window, the Add Tank tool can be used in different ways to place a tank. If the user clicks on or near an existing junction node, MIKE URBAN will replace the selected node by a new tank.

If the user clicks with the Add Tank tool in the Map window somewhere else other than a node or a pipe, MIKE URBAN will place a *tank* node at the clicked location. If the user selects an existing pipe, a program message will be displayed allowing to split the pipe at the cursor location.

If desired, from the Map window, the user can move an existing tank using the Move Selected tool. By selecting the tank and holding down the left mouse button, the tank can be dragged to a new location. As the tank is dragged, its connecting links (i.e., pipes, pumps, and valves) rubber band along with it.

To delete an existing tank graphically, select the tank using the Select tool and then select «Delete Selected». The selected tank will then be deleted. To edit a tank, click on the tank using the Open Editor tool. The Tank Editor will then be displayed allowing you to edit the attributes of the selected tank.

1.1.6 Air-Chamber Editor

Air-Chamber nodes are also placed at points in the water distribution model where a water storage Air-Chamber is located. Storage Air-Chambers can be defined as Air-Chambers with the variable or fixed water level. The Air-Chamber with the variable water level are modelled as Air-Chambers where the



water surface level changes with time as water flows into and out of the Air-Chamber. The Air-Chambers with the fixed water level represent places (reservoir) within the water distribution model where a infinite source of water (for the sake of the modelling simulation) is available. Hence, the reservoir water level remains constant during the course of the simulation.

Air-Chamber nodes are either defined interactively on the graphical Map window using the 'Add Air-Chamber' tool, or by manual data entry using the Air-Chamber Editor dialog box as shown in Figure 1.15. The Air-Chamber Editor allows you to define the reservoir's ID, location, pressure zone, water surface elevation, and a description. The Air-Chamber Editor dialog box is reached by selecting **EPANET | Air-Chambers**.

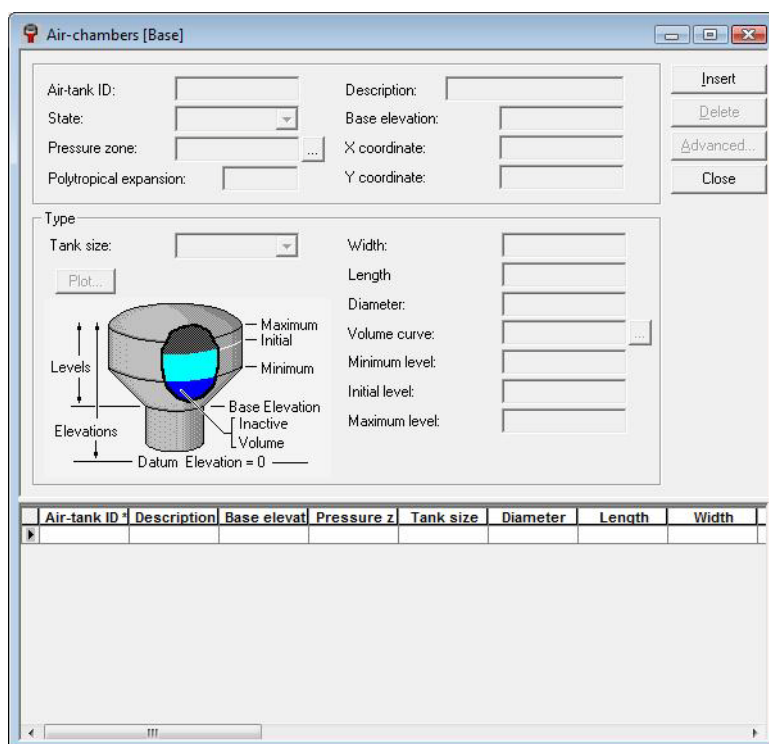


Figure 1.17 The Air-Chamber Editor allows you to define the storage tank nodes that supply water to the water distribution network

A list of the Air-Chamber Editor data entries for Figure 1.15 follows, with a short description given for each entry.

AIR-TANK ID (mandatory)

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



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

A new air-chamber ID is automatically suggested by MIKE URBAN whenever a new air-chamber node is placed into the list by pressing «Insert». When defining the tank nodes graphically on the Map window using the Add Air-Chamber tool, the air-chamber ID is automatically defined.

When importing (or merging) multiple water distribution network models into a single network model, MIKE URBAN 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.

X and Y COORDINATE (optional)

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

DESCRIPTION (optional)

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

STATE (optional)

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

PRESSURE ZONE ID (optional)

This data entry allows you to define the ID of the pressure zone that the air-chamber node lies within. The pressure zone ID can be used when distributing the zone demand as well as to graphically display different zones on the Map.

Selecting «...» allows you to display the Pressure Zone selection dialog box, where the appropriate pressure zone ID can be selected.

POLYTROPIC EXPANSION (mandatory)

This data entry allows you to define the polytropic exponent in the polytropic gas equation.

By default this data entry is $k=1.2$. See the section titled “Specific Valve Data” on page 191 for more information.



BASE ELEVATION (mandatory)

This data entry defines the bottom elevation, in units of ft. or m, of the storage tank where the water level in the storage tank is zero, as shown in Figure 1.16. The default elevation is zero.

TANK SIZE (mandatory)

This option button data entry selects the type of storage tank being defined. By default, a circular tank is defined. A the elevation-volume relationship for a tank of variable geometry can also 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.

DIAMETER or WIDTH and LENGTH (mandatory)

This data entry allows you to define the tank chamber size (ft or m).

VOLUME CURVE (mandatory)

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.

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 1.16.

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 1.16.

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 1.16.

Graphical Placement and Editing of Air-Chambers

From the Map window, the Add Tank tool can be used in different ways to place a air-chamber. If the user clicks on or near an existing junction node, MIKE URBAN will replace the selected node by a new air-chamber.

If the user clicks with the Add Air-Chamber tool in the Map window somewhere else other than a node or a pipe, MIKE URBAN will place a *air-chamber* node at the clicked location. If the user selects an existing pipe, a



program message will be displayed allowing to split the pipe at the cursor location.

If desired, from the Map window, the user can move an existing air-chamber using the Move Selected tool. By selecting the air-chamber and holding down the left mouse button, the air-chamber can be dragged to a new location. As the tank is dragged, its connecting links (i.e., pipes, pumps, and valves) rubber band along with it.

To delete an existing air-chamber graphically, select the air-chamber using the Select tool and then select «Delete Selected». The selected tank will then be deleted. To edit a air-chamber, click on the air-chamber using the Open Editor tool. The Tank Editor will then be displayed allowing you to edit the attributes of the selected air-chamber.

1.2 Other Editors

To help develop and setup a model, MIKE URBAN provides the user with several other editors including Project Options Editor, Report Editor, Water Source Editor, Energy Editor, Pressure Zone Editor, and others. This section discusses how to use these editors.

1.2.1 Project Options Editor

The project options dialog allows you to specify project configuration information for the pipe network system to be modelled. Project configuration data includes analysis type, friction loss formulation, simulation options, analysis options, and flowrate units. Alternatively, to display the Project Options dialog box, select **EPANET | Project Options**.

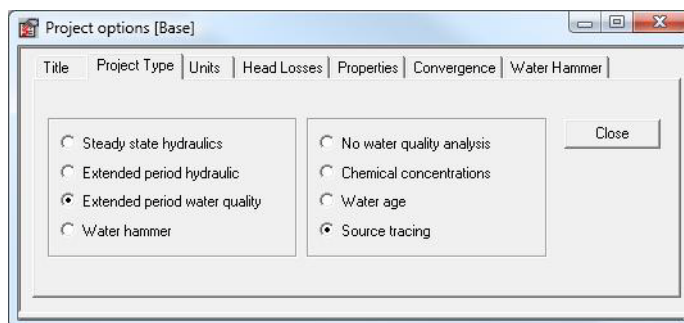


Figure 1.18 Project Options dialog box

The Project Options Editor dialog is used to define the project parameters.



TITLE (Optional)

This data entry allows you to enter a describe the current project. This description can be output in reports generated by the Report Generator.

PROJECT TYPE (Mandatory)

The project type is used to define the simulation type of the project.

- Steady State Hydraulics.
EPANET based steady state analysis.
- Extended Period Hydraulics.
EPANET based extended period simulation.
- Extended Period Water Quality.
- EPANET based extended period simulation with the water quality option such as *No Water Quality Analysis* (no quality analysis), *Chemical concentrations* (compute chemical concentration), *Water Age* (compute water age), *Source Tracing* (trace flow from a specific node).
- Water hammer
UNITS (Mandatory)

This selection data entry allows you to select one of the predefined project units environments such as CFS (cubic feet per second) or LPS (liter per second), for example.

HEAD LOSSES (Mandatory)

This selection data entry allows you to specify the formula used to compute headloss as a function of flow rate in a pipe. Choices are *Hazen-Williams*, *Darcy-Weisbach*, and *Chezy-Manning*.

PROPERTIES (Optional)

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 URBAN 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 (Optional)

These data entries allow you to determine 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.
- **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.
- **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 another 10 trials, with no link status changes allowed, in an attempt to achieve convergence.

- 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.

WATER HAMMER (Optional)

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

- Theta

Numeric parameter used to centre the high order finite difference scheme in time.

- Alpha

- Gravity

Gravity acceleration.

- Atmospheric pressure

The pressure of the atmosphere outside the pressurized pipeline system.

- Vapour cavity pressure
The pressure under which the gas particles starts to develop inside the water pipeline.
- Temperature
Temperature of water.

1.2.2 Pressure Zone Editor

Pressure zones are service areas defined by the hydraulic gradeline 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 Pressure Zone Editor dialog box is reached by selecting **EPANET | Pressure Zones**.

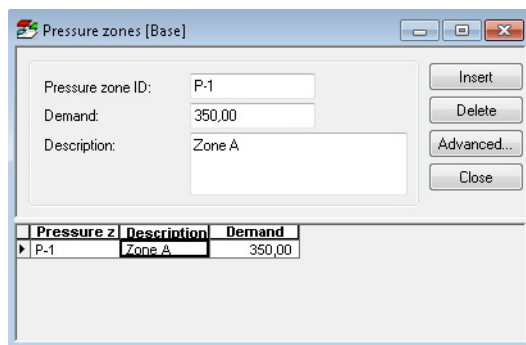


Figure 1.19 Pressure Zone Editor dialog box

The Pressure Zone Editor dialog is used to define the pressure zones for the pipe network system.

PRESSURE ZONE ID (mandatory)

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.

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 Generator.

DEMAND (optional)

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 Distribution dialog.



See “Distributed Demands” on page 59 for more details.

1.2.3 Energy Editor

Computation of pumping energy and cost can be performed in MIKE URBAN. Users can either choose pump-specific efficiency curves and electricity rate schedules or can select a set of default values for these computations. The Energy Editor dialog box is reached by selecting **EPANET | Energy**.

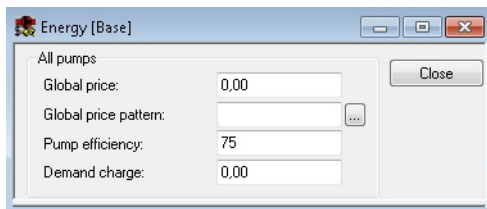


Figure 1.20 Energy Editor dialog box

The user can compute pumping energy and cost using Edit Energy Editor. The following parameters can be defined in Energy Editor:

GLOBAL PRICE (optional)

Average cost per kW/hour.

GLOBAL PRICE PATTERN (optional)

ID label of time pattern describing how energy price varies with time.

PUMP EFFICIENCY (optional)

Either a single percent efficiency for global setting or the ID label of an efficiency curve for a specific pump.

DEMAND CHARGE (optional)

Added cost per maximum kW usage during the simulation period.

1.2.4 Report Editor

Detailed report can be generated by EPANET engine when performing steady-state, extended period, and water quality analysis. The report parameters and settings can be specified in the Report Editor. Users can either choose reporting simulation status including error and warning messages or include the full simulation results of hydraulic and water quality parameters for each node, link, and time level. The Report Editor dialog box is reached by selecting **EPANET | Reports**.

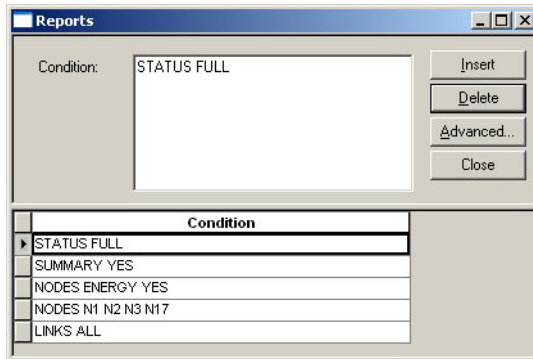


Figure 1.21 Reports Editor dialog box

The following parameters can be defined in Reports Editor:

CONDITION (optional)

The condition data entry is used to describe the contents of the output report produced from a simulation. The following keywords can be used:

STATUS (YES/NO/FULL)

STATUS determines whether hydraulic status messages are written to the Report file. If YES is selected the messages will identify those network components that change status during each time step of the simulation. If FULL is selected, then convergence information will also be included from each trial of each hydraulic analysis. This level of detail is only useful for de-bugging networks that become hydraulically unstable.

SUMMARY (YES/NO)

SUMMARY determines whether a summary table of number of network components and key analysis options is generated. The default is YES.

MESSAGES (YES/NO)

MESSAGES determines whether error and warning messages generated during a hydraulic/water quality analysis are written to the Report file. The default is YES.

ENERGY (YES/NO)

ENERGY determines if a table reporting average energy usage and cost for each pump is provided. The default is NO.

NODES (NONE/ALL/node1 node2...)

NODES identifies which nodes will be reported on. You can either list individual node ID labels or use the keywords NONE or ALL. Additional NODES lines can be used to continue the list. The default is NONE.



LINKS (NONE/ALL/link1 link2...)

LINKS identifies which links will be reported on. You can either list individual link ID labels or use the keywords NONE or ALL. Additional LINKS lines can be used to continue the list. The default is NONE.

Variable (YES/NO) or Variable (BELOW/ABOVE/PRECISION, value)

This reporting option is used to identify which variables are reported on, how many decimal places are displayed, and what kind of filtering should be used to limit output reporting. Node variables that can be reported on include:

- Elevation
- Demand
- Head
- Pressure
- Quality

Link variables include:

- Length
- Diameter
- Flow
- Velocity
- Headloss
- LinkQuality
- LinkStatus
- Setting (Roughness for pipes, speed for pumps, pressure/flow setting for valves)
- Reaction (reaction rate)
- F-Factor (friction factor)

The default reporting variables are Demand, Head, Pressure, and Quality for nodes and Flow, Velocity, and Headloss for links. The default precision is two decimal places.

Example:

The following example reports on nodes N1, N2, N3, and N17 and all links with velocity above 3.0. The standard node variables (Demand, Head, Pressure, and Quality) are reported on while only Flow, Velocity, and F-Factor (friction factor) are displayed for links.

```
NODES N1 N2 N3 N17
```

```
LINKS ALL
```



FLOW YES

VELOCITY PRECISION 4

F-FACTOR PRECISION 4

VELOCITY ABOVE 3.0

1.2.5 Curves and Relations Editor

The user can define data curves and their X, Y coordinate points in The Curves and Relations Editor. The Curves and Relations Editor dialog box is reached by selecting **EPANET | Curves and Relations**.

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 Kd versus time (only for Water Hammer Analysis).
- Dual-acting Valve 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).

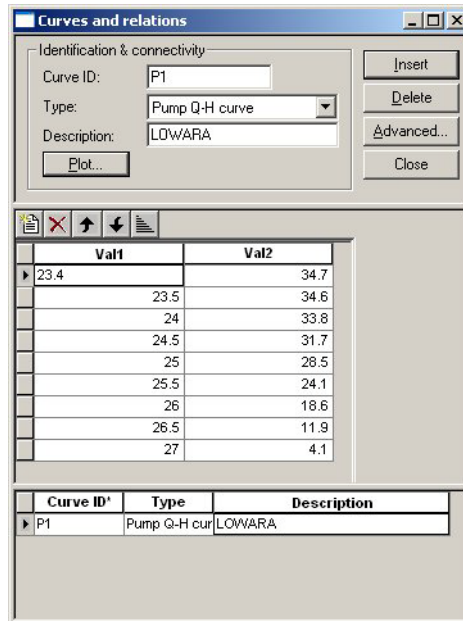


Figure 1.22 Curve Editor Window, Definition of data curves

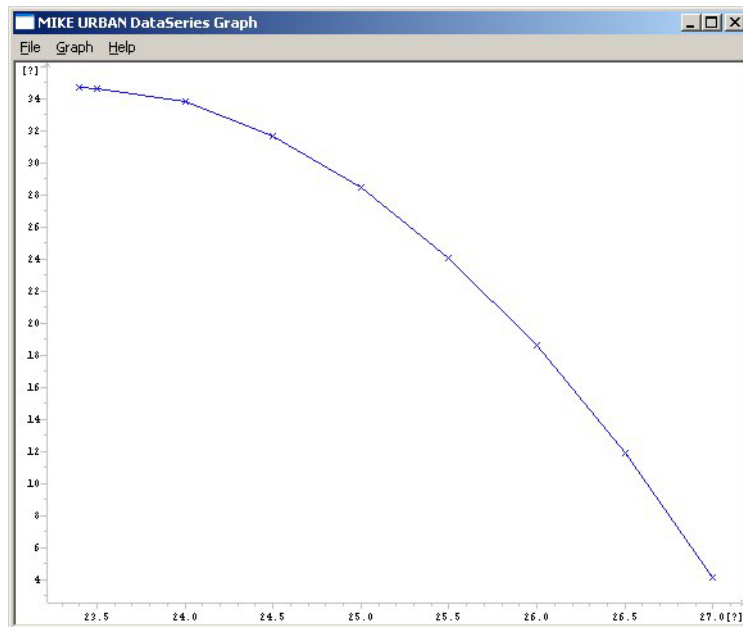


Figure 1.23 Preview of a defined curve in Curve Editor

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

1.2.6 Boundary Conditions Editor

Boundary conditions are used to define water level and flow changes during the water hammer analysis.

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 *Curves and Relations Editor* and stored in the database. These boundary conditions may be assigned to any node in the network. The boundary set point value for each time step is assigned from the corresponding time series curve. 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 (Water head) boundary.
- Q (Flow) boundary.
- Compatibility (default type).
- Hydraulic structure (storage tank, air-chamber).

The water hammer analysis ignores the diurnal patterns defined for the steady state and extended period simulation. Therefore, the boundary conditions using time series must be specified at nodes where the flow or water head changes need to be defined during the water hammer simulation. The Boundary Conditions Editor dialog box is reached by selecting **EPANET | Water Hammer | Boundary Conditions**.

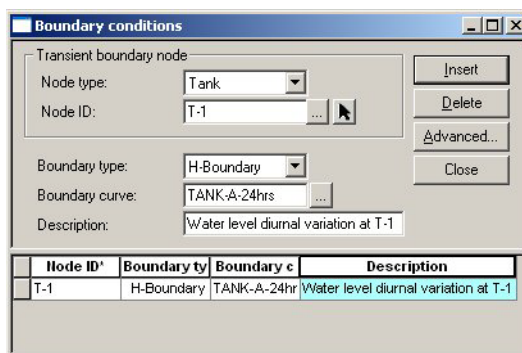


Figure 1.24 Boundary Conditions Editor dialog box

The following parameters can be defined in Boundary Conditions Editor:

NODE TYPE (optional)

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



NODE ID (mandatory)

This data entry is used to define the ID of the boundary node the boundary condition is being assigned to. Choosing «...» will display the Select Node selection dialog box from which the user can select the appropriate node type and ID. Or, choosing «Pick» allows the user to graphically select the node from the Map window.

BOUNDARY TYPE (mandatory)

This data entry is used to define the boundary type. Head versus time and flow versus time boundary types are available for selecting from. Note that constant head or flows (demands) do not need to be defined.

BOUNDARY CURVE (mandatory)

This data entry is used to define the boundary curve. Choosing «...» will display the Select Curve selection dialog box from which the user can select the appropriate boundary curve.

DESCRIPTION (optional)

This data entry allows you to enter a description identifying the boundary condition being defined. This description can be output in reports generated by the Report Generator.

1.3 Network Demand

Network demand for water is assigned at junction nodes, on a node by node basis. To help develop a model, MIKE URBAN WATER 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 URBAN 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 URBAN defines pressure zones.

1.3.1 Distributed Demands

Network demands are defined at junction nodes, on a node by node basis. For large network systems, assigning this demand data can be a very tedious job. Since many times the total demand is known for a particular network pressure zone or for the entire network system, MIKE URBAN provides the capability to distribute this total demand among the applicable junction nodes.

Pipe Demand Coefficients

MIKE URBAN computes the water demands for each node in the network system based upon the total network demand using two methods: 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 box, as shown in Figure 1.25, is used to automatically assign the demands at the appropriate junction nodes. The Distributed Demands dialog box is reached by selecting **EPANET | Distributed Demands**.

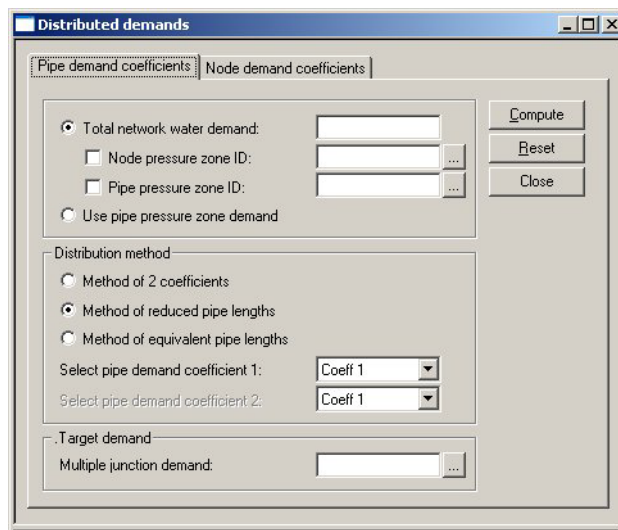


Figure 1.25 The Distributed Demands dialog box computes the demand at each junction node automatically based upon the total demand within a pressure zone or entire network system

A list of the Distributed Demands data entries for Figure 1.25 follows, with a short description given for each entry.

TOTAL NETWORK WATER DEMAND (mandatory)

This data entry is used to specify the total network demand for a particular network pressure zone or the entire network system. The flow units are user-specified.

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.

NODE PRESSURE ZONE ID, PIPE PRESSURE ZONE ID (optional)

This check box allows you to select whether the total network water demand corresponds to the entire network or a single pressure zone depending on



whether the pressure zone ID is assigned to nodes or pipes. Checking this box applies the specified water demand to a single specified pressure zone. Unchecking this box applies the specified water demand to the entire water distribution network.

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

METHOD OF TWO COEFFICIENTS or METHOD OF REDUCED PIPE LENGTHS or METHOD OF EQUIVALENT PIPE LENGTHS

MIKE URBAN 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*. This radio button group allows the user to select the method to be used.

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

$$q_{pi} = \frac{(Q)k_{1i}k_{2i}}{\Sigma(k_{1i}k_{2i})} \quad (1.4)$$

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

$$q_{pi} = \frac{(Q)l_i k_{1i}}{\Sigma(k_{1i}l_i)} \quad (1.5)$$

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

$$q_{pi} = \frac{(Q)l_i k_{Di}}{\Sigma(k_{Di}l_i)} \quad (1.6)$$

where:

q_{pi} = Total water demand applied to the pipe, split between the two end nodes.

Q = Total network water

l_i = Pipe length

k_{1i}, k_{2i} = pipe demand 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 6inch). 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, as described on page -18. The computed demands, which are assigned once «Compute» is selected, are stored at each individual node. These demands are stored in the Junction Editor, as described on page -12. Selecting «Reset» causes all of the nodal demand entries to be set to zero for the entire network, but it will leave the additional demand entries that are defined unaffected.

SELECT PIPE DEMAND COEFFICIENT 1,2 (mandatory)

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

TARGET DEMAND (mandatory)

This data entry allows you to specify the multiple demand category, 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 that such demand category already exists for each node used in the demand distribution.

Node Demand Coefficients

Node demand coefficient allows you 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.

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.

TOTAL NETWORK WATER DEMAND (mandatory)

This data entry is used to specify the total network demand for a particular network pressure zone or the entire network system. The flow units are user-specified.

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.

NODE PRESSURE ZONE ID (optional)

This check box allows you to select whether the total network water demand corresponds to the entire network or a single pressure zone. Checking this box applies the specified water demand to a single specified pressure zone.



Unchecking this box applies the specified water demand to the entire water distribution network.

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

EQUAL NODE DEMAND DISTRIBUTION (optional)

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

Selecting Equal Node Demand Distribution computes the junction demand:

$$q_{ni} = \frac{Q}{N} \quad (1.7)$$

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

TARGET DEMAND (mandatory)

This data entry allows you to specify the multiple demand category, 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 that such demand category already exists for each node used in the demand distribution.

Selecting the Method of Node Demand Coefficients, MIKE URBAN computes the total water demand assigned to each junction node as:

$$q_i = \frac{Q_t}{\sum_{i=0,n} C_i} \cdot C_i \quad (1.8)$$

where:

q_i = node demand

Q_t = total network demand

C_i = demand coefficient



1.3.2 Demand Allocation Editor

Junction node demands can be edited either within the junction editor for each particular junction node or in the Multiple Demand editor, which allows the user to display and edit all multiple demands. In addition to that, MIKE URBAN provides another way of developing junction node demands, which is based on geocoding the consumption data to the appropriate junctions or pipes and aggregating their set point demand values to the junction demands. This allows simplifies the demand development process and allows you, for example, to import consumption data from the consumption database systems and geocoded it based on its X, Y geographical coordinates.

The Demand Allocation Editor dialog box is reached by selecting **EPANET | Demand Allocation**.

MUID *	Reference ID	Junction ID	Pipe ID *	Demand	Minimum d	Average de	Maximum d	De

Figure 1.26 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 1.26 follows, with a short description given for each entry.

REFERENCE ID (optional)

This data entry is used to specify an ID which uniquely identifies the demand point. The reference ID acts as a unique lookup key that identifies the demand point from all other demand points. The reference 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.

PRESSURE ZONE ID (optional)

This data entry allows you to define the ID of the pressure zone that the demand point lies within. The pressure zone ID can be used when distributing the zone demand as well as to graphically display different zones on the Map. Selecting «...» allows you to display the Pressure Zone selection dialog box, where the appropriate pressure zone ID can be selected.

X, Y COORDINATE (mandatory)

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

JUNCTION ID or PIPE ID (mandatory)

This data entry is used to specify the node or pipe ID, to which the demand point is linked to. The user can specify this junction or pipe ID by editing or to use *Geocode* function, which will automatically geocode (link) each demand point to the appropriate junction node or a pipe (such as nearest node, for example).

DEMAND (mandatory)

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.

MINIMUM DEMAND (optional)

This data entry is used to specify the minimum demand value, which can be then assigned to *demand* for the process of the demand aggregation. The minimum demand value can be imported from the external database systems (such as CIS Customer Information System, for example) or can be developed within MU by applying a minimum hour factor to the average demand, for example.

AVERAGE DEMAND (optional)

This data entry is used to specify the average demand value, which can be then assigned to *demand* for the process of the demand aggregation. The average demand value can be imported from the external database systems (such as CIS Customer Information System, for example) and can be used within MU for development of actual demand, minimum demand, or maximum demand.

MAXIMUM DEMAND (optional)

This data entry is used to specify the maximum demand value, which can be then assigned to *demand* for the process of the demand aggregation. The maximum demand value can be imported from the external database systems (such as CIS Customer Information System, for example) or can be



developed within MU by applying a maximum hour factor to the average demand, for example.

DESCRIPTION (optional)

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

DEMAND CATEGORY (optional)

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.

CATEGORY TYPE (optional)

This data entry allows you to enter a category type identifying the consumption point being defined.

ADDRESS (optional)

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

OWNER (optional)

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 Generator.

EQUIVALENT PERSON(optional)

This data entry is used to specify the equivalent person value for each demand point. The EP value can be used when calculating the actual junction demand, which will be then used in the process of demand aggregation.

EQUIVALENT TENEMENT (optional)

This data entry is used to specify the equivalent tenement value for each demand point. The EP value can be used when calculating the actual junction demand, which will be then used in the process of demand aggregation.

Geocoding

Demand geocoding is an automatic process, where each demand point is linked to the appropriate junction node or a pipe (such as nearest node, for example). The user can also specify the linked junction or pipe ID manually by editing or importing from external database systems, such as GIS or CIS. There are several different geocoding methods available. The geocoding is done from a wizard. Press the 'Geocode' button on the Demand Allocation dialog to activate the wizard.

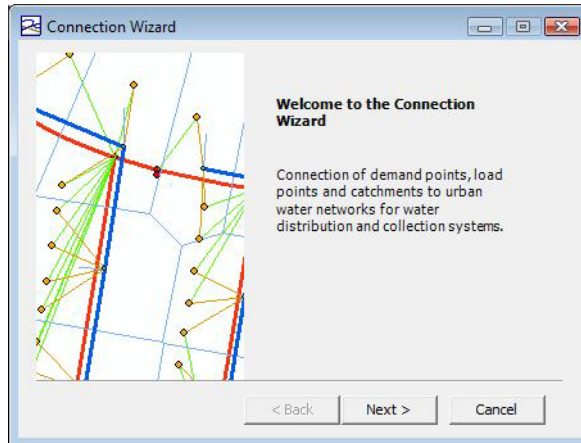


Figure 1.27 Demand geocoding wizard

Different options are available for the geocoding process.

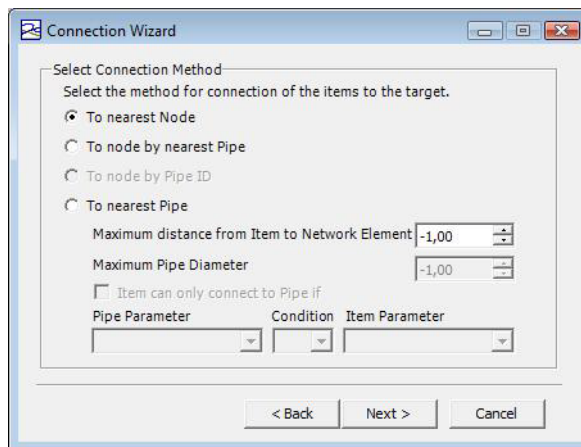


Figure 1.28 The different geocode options

- To nearest node
- To node by nearest pipe

The nearest pipe is located first and its pipe diameters can be optionally checked. The nearest of the pipe beginning or ending nodes is used as a demand node.
- To node by Pipe ID

The nearest pipe is already specified and the nearest of the pipe beginning or ending nodes is used as a demand node.
- To nearest pipe



MAXIMUM PIPE DIAMETER

It is possible to restrict the maximum diameter of pipes, used during the demand geocoding. This allows us to ignore long-distance pipes, for example.

Aggregation

Demand aggregation is used to develop node demands for junction nodes based on the geocoded node or link ID/

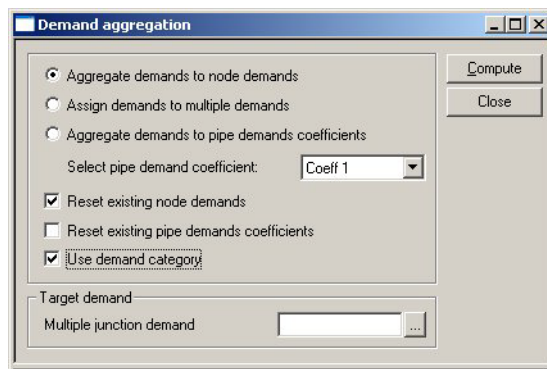


Figure 1.29 Demand Aggregation dialog box

AGGREGATE DEMANDS TO NODE DEMANDS or ASSIGN DEMANDS TO MULTIPLE DEMANDS or AGGREGATE DEMANDS TO PIPE DEMAND COEFFICIENTS (mandatory)

- Aggregate demands to node demands
Multiple node demands are developed by aggregating demand points to the appropriate junction nodes based on the matching junction ID. This option only works for demand allocations connected to nodes, not to pipes.
- Assign demands to multiple demands
Multiple node demands are created for demand points. For each demand point, the corresponding record is created in the multiple demands editor. Use this method if you want to populate multiple demands for demand points. This option only works for demand allocations connected to nodes, not to pipes.
- Aggregate demands to pipe demand coefficients
Pipe demand coefficient is calculated by aggregating demand points to the appropriate pipes based on the matching pipe ID. This option only



works for demand allocations connected to pipes. Note that this does not add to the Multiple Demands table (as is done for allocation assigned to nodes), but sets the demand coefficients of the pipe.

SELECT PIPE DEMAND COEFFICIENT (optional)

This data entry is used to specify the pipe demand coefficient, which will be used by *Aggregate demands to pipe demand coefficient* method.

RESET EXISTING NODE DEMAND (optional)

Existing junction node demands are deleted prior to demand aggregation.

RESET EXISTING PIPE DEMAND COEFFICIENTS (optional)

Existing pipe demand coefficient values are deleted prior to demand aggregation.

USE DEMAND CATEGORY (optional)

This option data entry is used if the user wants to aggregate demand points based on their category to the appropriate junction nodes.

TARGET (mandatory)

This data entry allows you to specify the multiple demand category, which will be used as a target demand for the aggregated 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 that such demand category already exists for each node used in the demand distribution.

1.3.3 Multiple Demand Editor

Multiple demands can be edited either within the junction editor for each particular junction 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 selecting **EPANET | Multiple Demands**.



Junction ID	Demand	Description	Demand cat	Cyclic profil	Demand co
10337	0,04	<Null>	<Null>	<Null>	<Null>
10480	0,05	<Null>	<Null>	<Null>	<Null>
10483	0,02	<Null>	<Null>	<Null>	<Null>
10484	0,03	<Null>	<Null>	<Null>	<Null>
10486	0,01	<Null>	<Null>	<Null>	<Null>
10487	0,00	<Null>	<Null>	<Null>	<Null>
10492	0,01	<Null>	<Null>	<Null>	<Null>
10496	0,01	<Null>	<Null>	<Null>	<Null>
10503	0,03	<Null>	<Null>	<Null>	<Null>
10577	0,08	<Null>	<Null>	<Null>	<Null>
11194	0,03	<Null>	<Null>	<Null>	<Null>
11196	0,01	<Null>	<Null>	<Null>	<Null>

Figure 1.30 Multiple Demand Editor dialog box

JUNCTION ID (mandatory)

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

DEMAND (mandatory)

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

DEMAND COEFFICIENT (optional)

This data entry allows you to define the demand coefficient, which can be used in the automatic demand distribution. See “Distributed Demands” on page 59 for more details.

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 Report Generator.

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.

Cyclic Profile ID (optional)

This field displays the cyclic profile ID associated with the Junction. The cyclic profiles are defined using the curve editor and can be assigned to selected junctions in the multiple demand editor by right clicking in the Cyclic ProfileID field.



1.3.4 Demand Statistics

MIKE URBAN 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. The Demand Statistics dialog box is reached by selecting **EPANET | Demand Statistics**.

RecTypeID	PZoneID	Category	MinDemand	MaxDemand	AvgDemand	SumDemand	NewAvgDe	NewSumDe	CatTypeID
Data 1			1.3	1.3	1.3	10.4	<Null>	<Null>	<Null>
Data 1		Leakage	6.56021	6.56021	6.56021	52.4817	<Null>	<Null>	<Null>
Data 1		Residential	50	50	50	400	<Null>	<Null>	<Null>
Zone 1			57.8602	57.8602	57.8602	462.882	<Null>	<Null>	<Null>
Data 2		<Null>	0.9	0.9	0.9	7.2	<Null>	<Null>	<Null>
Data 2		Leakage	3.43979	3.43979	3.43979	27.5183	<Null>	<Null>	<Null>
Data 2		Residential	30	30	30	240	<Null>	<Null>	<Null>
Zone 2			34.3398	34.3398	34.3398	274.718	<Null>	<Null>	<Null>
Network			92.2	92.2	92.2	737.6	<Null>	<Null>	<Null>

Figure 1.31 Demand statistics dialog box

A list of the Demand Statistics information for Figure 1.31 follows, with a short 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 the pressure zone), and *network* (demand statistics for the whole network).
- **Pressure Zone**
Pressure zone ID identifies the pressure 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 pressure 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 pressure zone. The maximum demand is calculated as maximum demand at junction nodes at specific time level



- **Average Demand**
This data entry represents the average demand per category per pressure zone. The average demand is calculated as average demand at junction nodes during the entire simulation period.
- **Sum Demand**
This data entry represents the total demand per category per pressure 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 average demand for selected category, pressure zone, or a network. All corresponding junction node demands will automatically be adjusted (scaled) in order to fit the new average demand value.
- **New Sum Demand**
This data entry allows the user to specify the new total demand for selected category, pressure 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 data entry allows you 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.



RecTypeID	PZoneID	Category	MinDemand	MaxDemand	AvgDemand	SumDeman	NewAvgDe	NewSumDe	CatTypeID
Data 1		<Null>	1.3	1.3	1.3	10.4	<Null>	<Null>	<Null>
Data 1		Leakage	6.56021	6.56021	6.56021	52.4817	<Null>	<Null>	<Null>
Data 1		Residential	50	50	50	400	<Null>	<Null>	<Null>
Zone 1			57.8602	57.8602	57.8602	462.882	<Null>	<Null>	<Null>
Data 2		<Null>	0.9	0.9	0.9	7.2	<Null>	<Null>	<Null>
Data 2		Leakage	3.43979	3.43979	3.43979	27.5183	<Null>	<Null>	<Null>
Data 2		Residential	30	30	30	240	<Null>	<Null>	<Null>
Zone 2			34.3398	34.3398	34.3398	274.718	<Null>	<Null>	<Null>
Network			92.2	92.2	92.2	737.6	<Null>	<Null>	<Null>

Figure 1.32 Demand statistics dialog box: Redistribute allows you to define the new average or total zone (or network) demand and redistribute it to junction nodes

1.4 Extended Period Simulations

Network models are analyzed either as steady state (static) simulations or extended period (continuous or dynamic) simulations. The data requirements for each type of model are basically the same; however, additional data is required for extended period simulations. Extended period data is defined as status, controls, patterns, and times.

Status represents the setting of various network components, which include the initial status of pipes, pumps, and valves. For example, a pipe’s status could be defined as being initially closed (as if it were defined as a valve).

Controls (sometimes called switches) allow the settings (or status) of various network components to change at a particular time, when a specific pressure is reached, or when a specific water tank level occurs. For example, a pump could be turned off once a particular water tank level has been reached.

Repetive profiles are used to describe demand fluctuations. Profile data (sometimes called hydrograph or curve data) is applied to nodal demands and include a multiplicative factor that are applied to be nodal demand base value.

Times are used define various time-step parameters, such as:

- The total duration of the modelling simulation. This is typically 24 hours, since the demand pattern is nearly always daily.
- The time step (sometimes called the time interval), which is used to model the simulation in steps. This is typically 1 hour.
- The starting time, which establishes the time at which the simulation begins.

The following sections describe available editors that allow you to define an extended period simulation.



1.4.1 Simple Control Editor

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

Using the Control Editor, as shown in Figure 1.33, the user can specify the operational controls for the pipe network system on a one-on-one component basis. The Control Editor can be used for both extended period simulations and steady state simulations.

The following situations exemplify the types of operational controls that can be specified:

- A pipe can be opened at a given time (based upon the beginning of the network simulation). This type of operational control has no effect in a steady state simulation.
- A pump can be turned on or off depending on the water level in a specified tank.
- A pump's baseline operating speed can be adjusted using a multiplier, thereby increasing or decreasing the pump's output.
- A valve can be opened or closed based upon the pressure in an adjacent node.

The initial status of the network's pipes, pumps, and valves default to an *open* status, and can be changed to a *closed* status using the Control Editor based upon some condition. Additionally, pumps have a default pump speed ratio of 1.0 (indicating that they are operating on their original characteristic operational curves), but which can be modified based upon a particular condition.

The Control Editor dialog box, as shown in Figure 1.33, is used to define the operational controls for running the pipe network system. The Control Editor dialog box is reached by selecting **EPANET | Extended Period|Simple Controls**

In order to export simple controls into ASCII file, open Simple Controls Editor, select Advanced and select "Export to ASCII file...". This will allow you to define the name of the ASCII file and export the content of the Simple Control Editor into the selected ASCII file. You can edit the controls within the ASCII file and import them back to MIKE URBAN by selecting Load from ASCII file. This is convenient in cases when you use Excel or other tools to create the list of controls for the model.

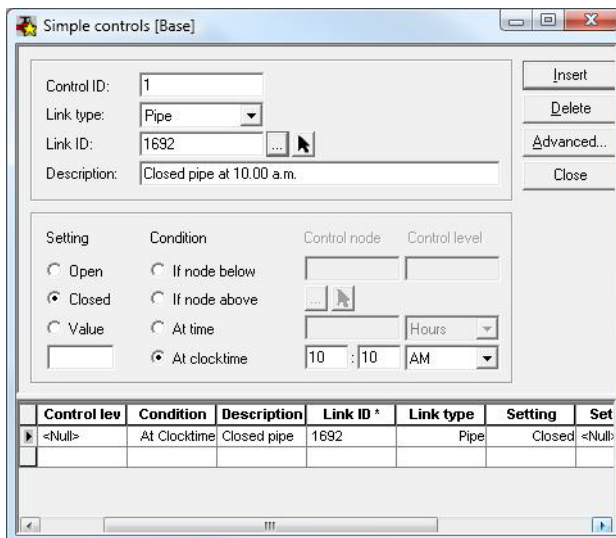


Figure 1.33 The Control Editor allows the user to define the network operational controls

A list of the Control Editor data entries for Figure 1.33 follows, with a short description given for each entry.

LINK TYPE (optional)

This pull-down selection list allows the user to select what type of link (i.e., pipe, pump, or valve) a control rule is being specified for.

LINK ID (mandatory)

This data entry is used to define the ID of the link to control. Choosing «...» will display the Select Link selection dialog box from which the user can select the appropriate link type and ID. Or, choosing «Pick» allows the user to graphically select the link from the Map window.

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 URBAN.

SETTING (mandatory)

This radio button selection entry is used to specify the OPEN or CLOSED status of the link being controlled, or VALUE which then allows the user to specify a multiplier to a pump's operating speed if a pump is being controlled. If VALUE is selected, then an additional data entry field will be presented to allow the user to specify the pump speed ratio.

CONDITION (mandatory)

This radio button selection entry is used to specify the control condition that then applies the operational rule onto the link 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 «Pick» 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. Note that this type of condition has no effect for steady state simulations, although can be defined should the user want to perform an extended period simulation at a later time.

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

CONTROL NODE ID (optional)

This data entry is used to define the ID of the control node. Choosing «...» will display the Select Node selection dialog box from which the user can select the appropriate node type and ID. Or, choosing «Pick» allows the user to graphically select the link from the Map window.

CONTROL LEVEL (optional)

This data entry is used to define control node level. The control node level represents pressure at junction nodes (psi or m) or water depth above a datum in storage tanks (feet or m).

ADVANCED - Import from Ascii File

This button allows the user to load simple controls that have been saved as a text file. This allows the user to use external editors to prepare and modify the control settings for the active project.

ADVANCED - Export to Ascii File

Allows the user to export the simple controls into an ASCII file. The user can define the file name and path that it is saved to.

Multiple Controls

During a simulation, a link (pipe, pump, or valve) can be operated by more than one operational control rule. For example, a link can be opened at a given time and then closed if a given nodal pressure is exceeded.

Not that it is necessary to re-apply the link setting (such as control valve setting or pump speed) after the link is re-open by the control rules.



Pipes with Check Valves

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

Available Operational Controls

The following operational controls can be defined by the user:

- A pipe, pump, or valve can be opened or closed.
- A pump's speed ratio can be changed.
- The pressure setting for PRV (pressure reducing), PSV (pressure sustaining), and PBV (pressure breaker) valves can be changed.
- The flow setting for FCV (flow control) valves can be changed.
- The minor loss coefficient for TCV (throttle control) valves can be changed.

Time Based Controls

The user can specify a specific time from the start of the simulation at which a change in a link's status is to occur. For example, pipe 15 can be opened 3 hours into the simulation and then closed at 4.2 hours into the simulation. This example is shown below:

```
LINK 15 OPEN AT TIME 3.0  
LINK 15 CLOSED AT TIME 4.2
```

Storage Tank Water Level Based Controls

The user can specify a storage tank level (*not* elevation) at which a control will occur. For example, pump 296 can be opened when the water level in storage tank 40 drops below 26 feet, and can be closed when the level rises above 40 feet. This example is shown below:

```
LINK 296 OPEN IF NODE 40 BELOW 26.0  
LINK 296 CLOSED IF NODE 40 ABOVE 40.0
```

Junction Node Pressure Based Controls

The user can specify a pressure level at which a control will occur if the specified pressure is above or below the pressure level. For example, valve 55 can be opened if the pressure at junction node 47 is below 50 psi, and pump 12 operating speed can be dropped to one-half of its normal speed when the pressure at junction node 12 goes above 75 psi. This example is shown below:



```
LINK 55 OPEN IF NODE 47 BELOW 50.0
LINK 12 0.5 IF NODE 12 ABOVE 75.0
```

1.4.2 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 **EPANET | Extended Period| Rule Based Controls**.

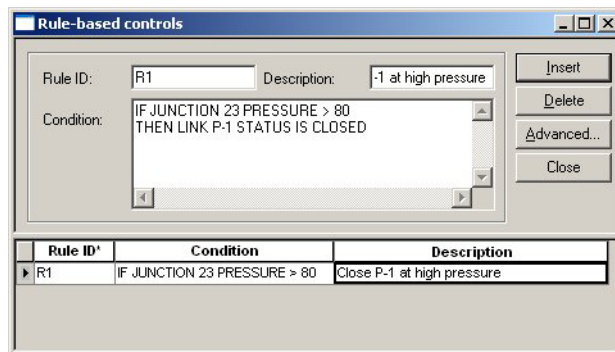


Figure 1.34 The Control Editor allows the user to define the network operational controls Rule Based Controls Editor Dialog Box

In order to rule based controls into ASCII file, open Rule Based Controls Editor, select Advanced and select "Export to ASCII file...". This will allow you to define the name of the ASCII file and export the content of the Rule Based Controls Editor into the selected ASCII file. You can edit the rules within the ASCII file and import them back to MIKE URBAN 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.

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

<=> =

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 it 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.



Multiple Pumps, Valves (WD-Tools)

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 it self. (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 ...



Control of valves

This set of rules opens and closes a Valve based on different water levels in tanks.

```
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
```

```
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
```

Control of pumps

This set of rules opens and closes a pump based on different water levels in tanks.

```
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
```

```
RULE AH_PS1_Stop ;ALEXHILL PS1 - AUTOMODE1
```

```
IF TANK 172 LEVEL < 6.55
```

```
OR TANK 170 LEVEL < 0.36
```

```
OR TANK 171 LEVEL < 0.36
```

```
THEN PUMP 14146 STATUS IS CLOSED
```



1.4.3 Repetitive Profiles (Pattern) Editor

MIKE URBAN assumes that water usage rates, external water supply rates, and constituent source concentrations (for water quality analysis) 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 1 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.

The definition of repetitive profiles consists of two main steps:

Step 1: Definition of Diurnal Profiles (24 hrs. duration)

Step 2: Definition of Cyclic Patterns, which are combining 1 or more diurnal profiles together.

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 tank water level). The duration of such diurnal profile curve is 1 day i.e. 24 hours. The Diurnal Profiles Editor dialog box is reached by selecting **EPANET | Extended Period|Repetitive Profiles | Diurnal Patterns**.



From	To	Multiplier
12:00:00 AM	1:00:00 AM	0.78
1:00:00 AM	2:00:00 AM	0.71
2:00:00 AM	3:00:00 AM	0.63
3:00:00 AM	4:00:00 AM	0.61
4:00:00 AM	5:00:00 AM	0.6
5:00:00 AM	6:00:00 AM	0.64
6:00:00 AM	7:00:00 AM	0.79
7:00:00 AM	8:00:00 AM	1.08
8:00:00 AM	9:00:00 AM	1.25
9:00:00 AM	10:00:00 AM	1.19
10:00:00 AM	11:00:00 AM	1.13
11:00:00 AM	12:00:00 PM	1.11
12:00:00 PM	1:00:00 PM	1.02

Pattern ID	Category	Description
P1	Residential	Zone A

Figure 1.35 Diurnal Profiles Editor, the Multipliers grid allows the user to define demand and/or constituent concentration multipliers for the current pattern ID

PATTERN ID (mandatory)

This data entry is used to specify the ID of the 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 (mandatory)

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 Time Settings Editor. The default value is 1 hour.

CATEGORY (optional)

This data entry allows you to enter a description identifying the demand pattern being defined. This description can be optionally included in reports.

DESCRIPTION (optional)

This data entry allows the user to enter a category that further define the demand pattern. For example, a demand might have the description of 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 reports generated by the Report Generator.



MULTIPLIERS (mandatory)

Selecting «Define» will display the Multipliers dialog box, as displayed in Figure 1.35. This dialog box allows you to define the baseline demand factors and constituent concentration levels (multipliers) for the current pattern ID. The complete pattern is then applied to the baseline demand (or source concentration) specified at each node (that corresponds with this pattern ID) over the extended period simulation.

Within each time period, the demand (or source concentration) remains constant—at a level equal to the multiplier times the baseline value.

There is no limit on the number of multipliers that can be defined for a pattern. However, since the diurnal profile duration is exactly 24 hours, the number of multipliers is determined by the pattern time step. The typical time period length is 1 hour, 24 multipliers are required to completely define the pattern for the entire simulation. For more information on defining the time step interval and analysis period, see the section titled *Time Setting Editor* which follows this section.

DISTRIBUTE

Select Distribute to let the program calculate the number of multipliers to fill entirely the diurnal pattern duration of 24 hours with the current pattern time step.

GRAPH

It is possible to create a GRAPH for pattern multipliers directly in the PATTERN EDITOR window, not only in the MULTIPLIERS window. This function allows the user to display the pattern graph quickly.

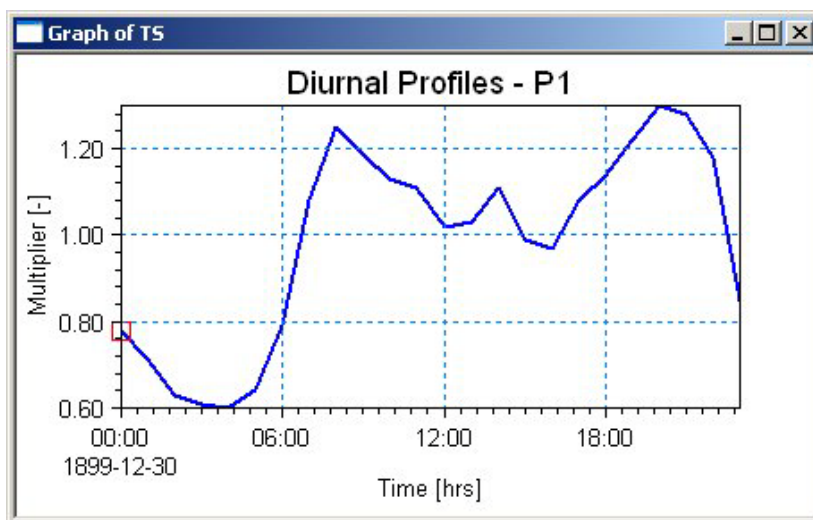


Figure 1.36 Graphical representation of the diurnal pattern multipliers

It is possible to import and export the diurnal pattern multipliers from an ASCII file. Select Advanced - Import or Advanced - Export to import or export the diurnal pattern multipliers from an ASCII text file.

Cyclic Profiles

The cyclic pattern editor is used to combine 1 or more diurnal profiles into one cyclic pattern. This allows the user to compose the cyclic patterns of two diurnal profiles one for e.g. working days and one for the weekend. If the total duration of the defined cyclic pattern is less than the total duration of the simulation, then the defined pattern will be repeated. For example, a 5 day simulation whose hourly demands repeat themselves on a daily cycle, only 24 multipliers would need to be defined for the cyclic pattern diurnal profile.

It is possible to display the graph for the cyclic profile, consisting of several diurnal patterns. In order to display the graph, open Cyclic Profile Editor, select the profile and click "Graph".

The Diurnal Profiles Editor dialog box is reached by selecting **EPANET | Extended Period| Repetitive Profiles | Cyclic Profiles**.

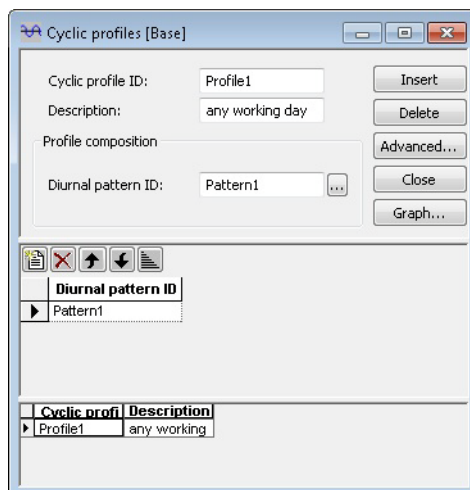


Figure 1.37 Cyclic Patterns Editor

CYCLIC PROFILE ID (mandatory)

This data entry is used to specify the ID of the cyclic pattern. The cyclic profile ID value can be any string value (up to 40 characters).

DIURNAL PATTERN ID (mandatory)

This data entry is used to specify the ID of the diurnal pattern. Choosing «...» will display the Select Diurnal Profile selection dialog box from which the user can select the appropriate diurnal pattern ID.



1.4.4 Time Settings Editor

For an extended period simulation, a simulation duration and time step must be specified. Using the Time Editor, as shown in Figure 1.38, the user can specify the extended period simulation time parameters. Only those time parameters that differ from the default values must be specified. The default time units are hours.

The Time Editor dialog box is reached by selecting **EPANET | Extended Period| Time Settings**.

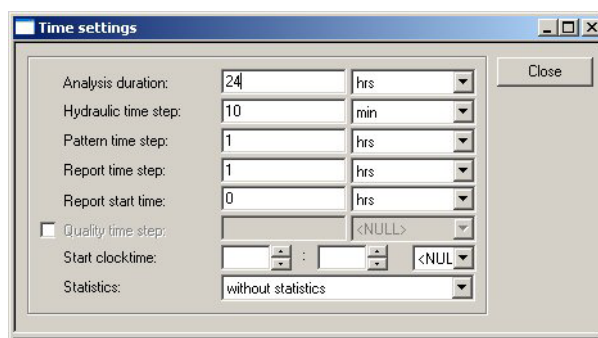


Figure 1.38 The Time Settings Editor allows the user to define the extended period simulation time parameters

A list of the Time Editor data entries for Figure 1.38 follows, with a short description given for each entry.

ANALYSIS DURATION (optional)

This data entry must be specified in order to perform an extended period simulation, and specifies the length of the entire simulation for both hydraulic extended period simulations and water quality simulations. The default value for this data entry is 0 hours, which implies that a steady state simulation will be performed. Note that a water quality analysis cannot be performed from a steady state simulation.

HYDRAULIC TIME STEP (optional)

This data entry must be specified in order to perform an extended period simulation, and specifies how often a new hydraulic computation of the pipe network system is to be computed. The default value is 1 hour.

PATTERN TIME STEP (optional)

This data entry is optional, and 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). If necessary, the software will adjust the specified Hydraulic Time Step so that it is not greater than the specified Pattern Time Step. The default value is 1 hour.



REPORT TIME STEP (optional)

This data entry is optional, and specifies the interval of time between which network conditions are reported. If necessary, the software will automatically reduce the specified value for the Hydraulic Time Step so that it is no greater than the Report Time Step. The default value is 1 hour.

REPORT START TIME (optional)

This data entry is optional, and specifies at what time into the simulation the analysis results should start to be reported. The default value is 0 hour.

QUALITY TIME STEP (optional)

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.

START CLOCK TIME (optional)

Start clock time is the time of day (e.g., 3:00 PM) at which the simulation begins. The default is 12:00 AM midnight.

STATISTICS (optional)

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

NONE (results reported at each reporting time step)

AVERAGE (time-averaged results reported)

MINIMUM (minimum value results reported)

MAXIMUM (maximum value results reported)

RANGE (difference between maximum and minimum results reported)

Statistical processing is applied to all node and link results obtained between the Report Start Time and the Total Duration

Time Units

Note that seconds, minutes, hours, and days can be used as time units, but that hours are generally used. It is recommended that the same time units be used for all entries within the Time Settings Editor—that you do not mix time units (i.e., hours and seconds).

Diurnal Curves and Patterns

Note that changing the pattern time step changes the number pattern multipliers, which define the 24 hrs. duration diurnal curve. MIKE URBAN will display a query dialog, as shown in Figure 1.39, asking the user whether to redistrib-



ute the pattern multipliers. If the user selects yes, the pattern number of pattern multipliers will be adjusted to fit the diurnal curve duration of 24 hrs.

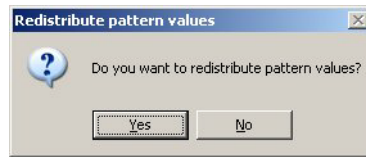


Figure 1.39 When changing the pattern time step, MIKE URBAN will ask whether to redistribute the pattern multipliers

1.5 Water Quality Simulations

MIKE URBAN 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.

1.5.1 Water Quality Analysis Selection

The first step in defining a water quality simulation is to select the type of water quality analysis to be performed. This is accomplished using the Project Options dialog box Project Type tab, as shown in Figure 1.40. Note that MIKE URBAN can only perform one type of water quality analysis during a simulation.

The Project Options dialog box is reached by selecting **EPANET | Project Options**

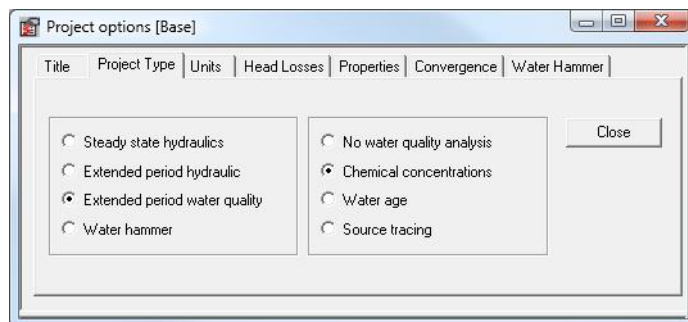


Figure 1.40 The Project Options dialog box allows the user to define the type of water quality analysis to be performed



A list of water quality options are available from the Project Options dialog box is shown in Figure 1.40, along with a short description given for each option.

NO WATER QUALITY ANALYSIS (default)

By selecting this option button entry, no water quality analysis will be performed. This is the default selection.

CHEMICAL CONCENTRATIONS

This option button entry is used to select that a chemical concentration analysis is to be performed. MIKE URBAN will report the concentration of the specified chemical at each time step for each network node. This type of analysis is typically performed to determine the amount of chlorine in the network to maintain safe drinking water standards.

WATER AGE

This option button entry is used to select that a water age analysis is to be performed. MIKE URBAN will report the water age at each time step for any network node, assuming an initial age of zero at the start of the simulation. This type of analysis is typically performed to determine dead ends (locations of stagnant water) within a pipe network design.

SOURCE TRACING

This option button entry is used to select that a source tracing analysis is to be performed. MIKE URBAN will report, for each time step, the percentage of water reaching each node from a selected source (origin) node. This type of analysis is typically performed for constituent tracking.

1.5.2 Water Quality Analysis Parameters

The Project Options dialog box Properties tab, as shown in Figure 1.41, allows you to specify the analysis parameters used to perform a water quality simulation.

The Project Options dialog box is reached by selecting **EPANET | Project Options**.

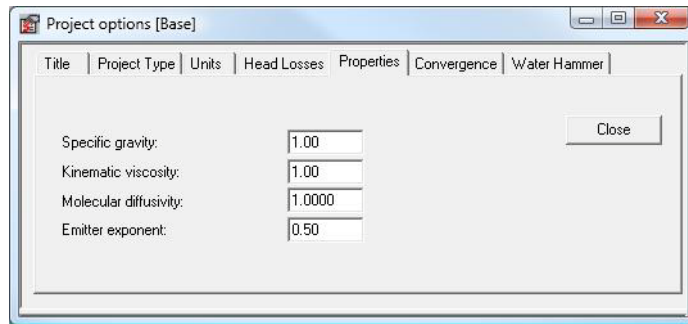


Figure 1.41 The Project Options dialog box allows the user to define the analysis parameters used in a water quality simulation

A list of the water quality analysis parameters from the Project Options dialog box is shown in Figure 1.41, along with a short description given for each parameter. These water quality modelling parameters are used when modelling the pipe-wall reaction mechanism.

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).

KINEMATIC 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 analyzed 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 URBAN to ignore mass transfer limitations.

1.5.3 Initial Water Quality Editor

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:

- Initial concentration for chemical constituents in a chemical propagation analysis.

- Initial hour for water age determination.
- Initial percentage of water originating at a specified source node for source tracing.

By default, all nodes are assigned with an initial water quality of zero. The Initial Water Quality Editor, as shown in Figure 1.42, is used to assign the initial water quality levels for the pipe network system. The Initial Water Quality Editor dialog box is reached by selecting **EPANET | Water Quality | Initial Water Quality**.

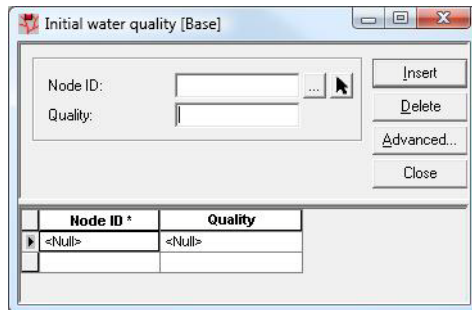


Figure 1.42 The Initial Water Quality Editor is used to define the initial water quality conditions of the pipe network system

A list of the Initial Water Quality Editor data entries for Figure 1.42 follows, with a short description given for each entry.

NODE ID

This data entry is used to specify an ID which uniquely identifies a node in which the initial water quality is being specified for.

A new entry is automatically inserted into the list by pressing «Insert». Choosing «...» will display the Select Node selection dialog box from which the user can select the appropriate node Or, choosing «Pick» allows the user to graphically select the node from the Map window.

QUALITY

This data entry is used to specify the initial water quality at the node (or group of nodes) being defined.

If a *chemical concentration* water quality simulation is being performed, then this entry denotes the water quality in mg/litres.

If a *water age* water quality simulation is being performed, then this entry denotes the initial age of the water in hours.

If a *source tracing* water quality simulation is being performed, then this entry denotes the initial percentage of water from the source (origin) node in percent.



1.5.4 Point Constituent Source Editor

The Point Constituent Source Editor, as shown in Figure 1.43, 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.

The Point Constituent Source Editor dialog box is reached by selecting **EPANET | Water Quality | Quality | Point Constituent Sources**. Note that a *Chemical Concentrations* water quality simulation must be specified in the Projects Options dialog box (see Figure 1.40) in order access the Point Constituent Source Editor dialog box.

Node ID	Concentration	Cyclic profile	Source type	Node type
T-100	0.100	C-Constant	Concentration	Tank

Figure 1.43 The Point Constituent Source Editor is used to specify at which nodes an external chemical constituent enters the pipe network system

A list of the Point Constituent Source Editor data entries for Figure 1.43 follows, with a short description given for each entry.

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.

NODE ID

This data entry is used to define the ID of the node the point constituent is being assigned to. Choosing «...» will display the Select Node selection dialog box from which the user can select the appropriate node type and ID. Or, choosing «Pick» allows the user to graphically select the node from the Map window.

CONCENTRATION

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



For a junction node, if there is no external inflow assigned to the node (such as a well—denoted by a negative demand), then the resulting water quality at the node always equals the specified concentration. This allows the user to simulate chlorine booster stations at a node, such as is used in satellite treatment in a network.

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 URBAN can model the following types of sources:

- 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.

CYCLIC PROFILE 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.

Selecting «Button» allows you to display the Select Pattern selection dialog box, where the appropriate pattern ID can be selected.

Point Constituent Source Time Patterns

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.

1.5.5 Reaction Rate Editor

The Reaction Rate Editor, as shown in Figure 1.44, allows you to specify the rate at which a constituent decays (or grows) by reaction as the constituent travels through the pipe network. Reaction rates can be defined at a global level where the same reaction rate applies to the entire network, and at a local level allowing the user to define reaction rates based upon a particular pipe type, etc. Note that locally defined reaction rates override globally defined reaction rates.

The Reaction Rate Editor dialog box is reached by selecting **EPANET | Water Quality | Reaction Rates**. Note that a *Chemical Concentrations* water quality simulation must be specified in the Projects Options dialog box (see Figure 1.41) in order access the Reaction Rate Editor dialog box.

Global Settings

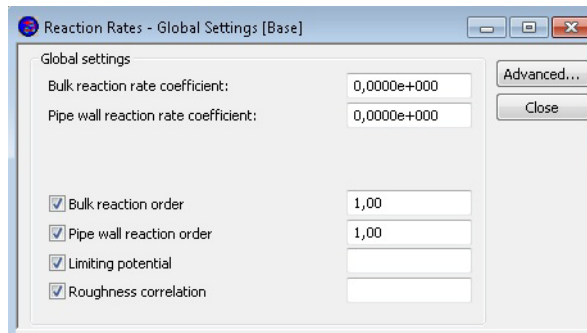


Figure 1.44 The Reaction Rate Editor is used to specify constituent reaction rates at both a global and local level

A list of the Reaction Rate Editor data entries for Figure 1.44 follows, with a short description given for each entry.

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

LIMITING POTENTIAL specifies that reaction rates are proportional to the difference between the current concentration and some limiting potential value.

ROUGHNESS CORRELATION

ROUGHNESS CORRELATION will make all default pipe wall reaction coefficients be related to pipe roughness in the following manner

1.5.6 Local Settings

Tank Reaction Rate Assignments

The locally defined tank reaction rate is specified in the Tank Editor.

Pipe Reaction Rate Assignments

The locally defined pipe wall and pipe bulk reaction rates are specified in the Pipe Editor. This allows the use of the field calculator to specify the reaction rate based on the pipe material, pipe diameter, and pipe construction year, for example.

Table 1.1 Roughness for Head Loss Equations

Head loss Equation	Wall reaction Formula
Hazen-Williams	F/C



Table 1.1 Roughness for Head Loss Equations

Head loss Equation	Wall reaction Formula
Darcy-Weisbach	$-F/\log(e/D)$
Chezy-Manning	$F*n$

where F = roughness correlation, C = Hazen-Williams C-factor, e = Darcy-Weisbach roughness, D = pipe diameter, and n = Chezy-Manning roughness coefficient. The default value computed this way can be overridden for any pipe by using the wall reaction rate format to supply a specific value for the pipe.

Note

Remember to use negative signs on all reaction coefficients that are to represent constituent decay (e.g., chlorine decay). Otherwise, a positive value denotes constituent growth.

1.5.7 Source Tracing

The Trace Node dialog box, as shown in Figure 1.45, allows you to track over time what percent of water reaching any node in the network had its origin from a specified node (i.e., junction, tank, or reservoir). Source tracing is a useful tool for analyzing a network distribution system that draws water from two or more different raw water supplies. It can be used to show to what degree water from a given source blends with that from other sources, and how the spatial pattern of this blending changes over time.

The Trace Node dialog box is reached by selecting **EPANET|Water Quality | Trace Node**. Note that a *Source Tracing* water quality simulation must be specified in the Projects Options dialog box (see Figure 1.41) in order access the Trace Node dialog box.



Figure 1.45 The Trace Node dialog box is used to specify a single node that acts like a tracer in determining what percent of its water reaches any other node in the network

A list of the Trace Node dialog box data entries for Figure 1.45 follows, with a short description given for each entry.



NODE TYPE

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

NODE ID

This data entry is used to define the ID of the node the trace node is being assigned to. Choosing «...» will display the Select Node selection dialog box from which the user can select the appropriate node type and ID. Or, choosing «Pick» allows the user to graphically select the node from the Map window.

1.6 Tools

The following section describes additional capabilities in MIKE URBAN that allow the user to work effectively with the program.

1.6.1 Send Data

You can transfer data between the asset and the model database of MIKE URBAN. See the section titled *Transferring data between 'Asset' and 'Model'* in the *MIKE URBAN Model Manager book*, chapter *Modelling Water Distribution networks* for more details on Send Data.

1.6.2 Engineering Tables

Included with MIKE URBAN are predefined engineering tables that include specified pipe roughness coefficients and minor loss coefficients, as shown in the Engineering Tables dialog box as shown in Figure 1.47. These values are used as lookup coefficient values when defining the pipe and valves that make up the network system. The Engineering Tables dialog box is reached by selecting **Tools | Engineering Tables**. Note that additional coefficients can be added to the provided engineering table values using this dialog box.



Minor loss coefficient	Component
5	Angle valve, fully open
0.05	Contraction - Conical D2/D1 = 0.80
0.18	Contraction - Sudden D2/D1 = 0.80
0.37	Contraction - Sudden D2/D1 = 0.50
0.16	Expansion - Sudden D2/D1 = 0.80
0.49	Contraction - Sudden D2/D1 = 0.20
0.92	Expansion - Sudden D2/D1 = 0.20
0.03	Expansion - Conical D2/D1 = 0.80
0.5	Cross - Line Flow
0.08	Expansion - Conical D2/D1 = 0.50
0.57	Expansion - Sudden D2/D1 = 0.50
0.13	Expansion - Conical D2/D1 = 0.20

Figure 1.46 The Engineering Tables define lookup table values for minor loss coefficients

Pipe roughness coefficients are specified as Hazen-Williams C , Darcy-Weisbach ε , and Manning's n values for various pipe material. Minor loss coefficients are used in pipe and valve definitions that contain additional headlosses due to pipe fittings and/or flow obstructions. When defining minor losses for a network component and more than one minor loss applies, then the sum of all the minor loss coefficients should be used to account for the total additional headloss.

Material	Darcy-Weisbach	Hazen-Williams C	Manning's n
Concrete	0.013	110	0.12
Copper	0.011	135	0.015
Cast iron	0.012	130	0.26
Riveted steel (new, rough)	0.021	80	9.1
Concrete (centrif. spun)	0.013	135	0.37
Riveted steel (new, smooth)	0.019	110	0.91
Galvanized iron	0.016	120	1.5
Best concrete	0.01	150	0.37
Aluminium	0.024	100	0.3
Brick in mortar	0.015	100	0.61
Brick sewer	0.015	100	0.61

Figure 1.47 The Engineering Tables define lookup table values for pipe roughness coefficients

Roughness coefficient and minor loss coefficient lookup table dialog boxes are displayed from either the Pipe Editor and Valve Editor by clicking on the «...» button, adjacent to the Roughness and Minor Loss data entries.



1.6.3 Convert Asset Valves/Pumps to Model Valves/Pumps

The Convert Asset Valves (or Pumps) to Model Valves (or Pumps) dialog box is tailored to convert asset valves/pumps (point feature class mwa_valve/mwa_pump) to the model valves/pumps (polyline feature class mw_valve/mw_pump). This tool is available when running MIKE URBAN in the WD Water Distribution mode.

The Convert Asset Valves (or Pumps) to Model Valves (or Pumps) dialog box is reached by selecting **Tools | Toolbox | Convert Asset Valves (Pumps) to Model Valves (Pumps)**. This tool will start converting asset valves to model valves by finding the nearest model pipe for each asset valve (or selected asset valve), which is located within a snap tolerance radius to the nearest model pipe. In order to import GIS valves to the asset valves (mwa_valve feature class) use File Import and Export dialog box; it is also possible to create asset valves by graphical editing.

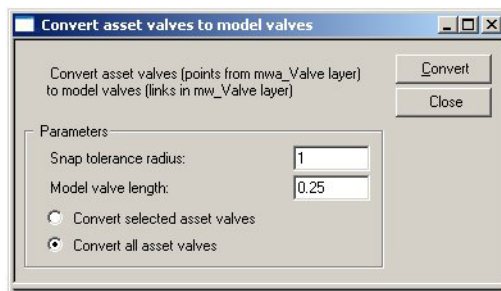


Figure 1.48 Convert asset valves to model valves dialog box

A list of the Convert Asset Valves to Model Valves dialog box data fields for Figure 34.57 follows, with a short description given for each entry.

SNAP TOLERANCE RADIUS

This data entry is used to specify the snap tolerance radius, which is used to convert asset valves close to the model pipes.

MODEL VALVE LENGTH/MODEL PUMP LENGTH

This data entry is used to specify the length of the model valve to be created.

CONVERT SELECTED ASSET VALVES/PUMPS or CONVERT ALL ASSET VALVES/PUMPS

This data entry choice is used to specify whether all asset valves will be converted to the model valves or whether only selected asset valves will be converted. Note that the snap tolerance radius will be applied in addition to this primary source data filter.

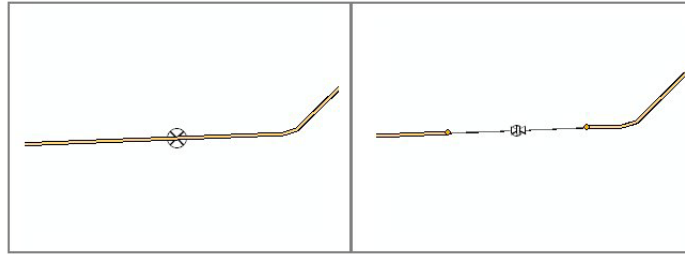


Figure 1.49 Conversion of the asset valve to the model valve; note that the pipe is split and two junction nodes are created nearby the asset valve location.

1.6.4 Duplicate Pipe Attributes

It is very common that you import the model based on the CAD files, for example, and that the pipe attributes (imported as CAD labels) were not available for every pipe. Similar situation will also happen when you start editing pipes manually and you do not really want to assign the pipe attributes one by one.

MIKE URBAN provides a tool, which allows you to select pipes from the Map and duplicate their attributes. Pipe material, as illustrated in the example below, is automatically duplicated to all pipes, which are adjacent to the selected pipe(s) until "T" or other complex junction exists.

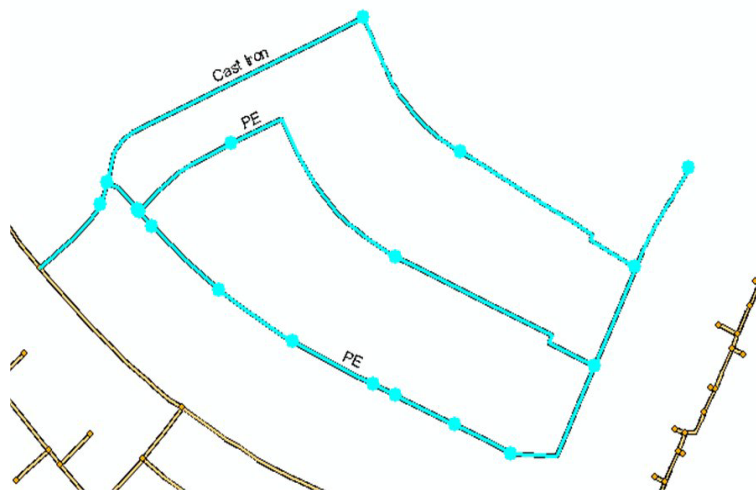


Figure 1.50 Step1: Select pipes which attribute(s) you want to duplicate.

Now, select attributes you would like to duplicate, such as pipe material, diameter, and others. Click "OK" where ready.

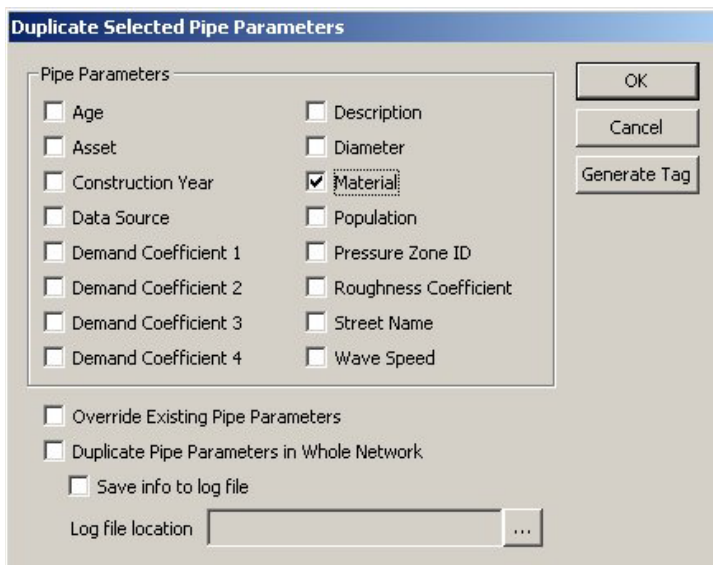


Figure 1.51 Step 2: Select what attributes you want to duplicate

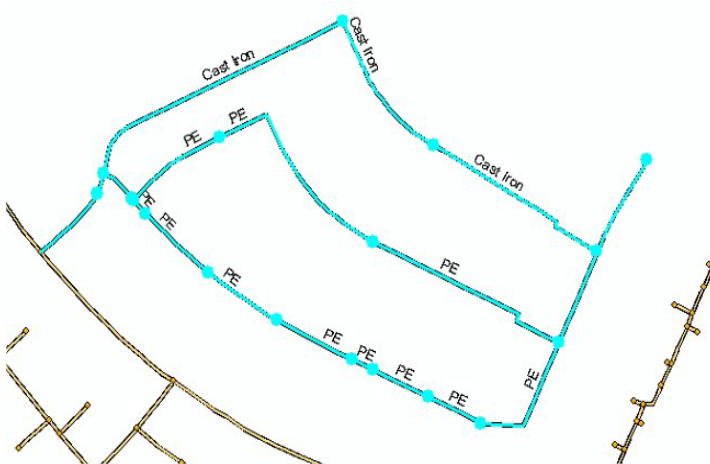


Figure 1.52 Step 3: Pipe attributes are duplicated

"Generate tag" tool allows you to generate unique identifier "Tag" for every stretch of pipes within the model. This is very useful for generating unique "GIS" tag for such "chain of pipes", which are linked together by just 1 single node.

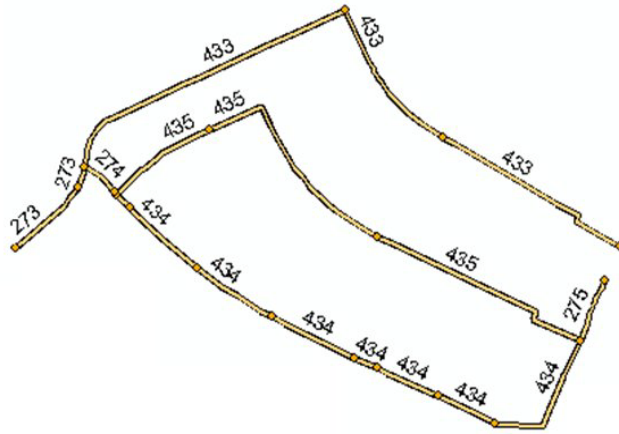


Figure 1.53 Illustration: GIS tag is generated for each stretch of pipes





MIKE URBAN WD

User Guide





2 Using the Model

MIKE URBAN™ is the most advanced, powerful, and comprehensive water distribution modelling package available. MIKE URBAN WD (Water Distribution) can analyse an entire water distribution system, or selected portions, under steady state, extended period and water hammer flow conditions, with water quality analysis if needed. You can enter model data suited for your choice of numerical engine EPANET (public domain EPANET engine developed by EPA, US), Water hammer engine (developed by DHI and a part of the WD-Tools) and/or WD-Tools (DHI suit of tools such as fire flow analysis, pipe roughness calibration, variable speed drive pump, extended controls based on the modified EPANET engine). You can also enter 'Asset WD' data suited for a water supply and water distribution system. The public domain EPANET engine can be run from the Model Manager, while running WD-Tools requires you to have the license for the MIKE URBAN Water Distribution system with WD-Tools.

MIKE URBAN allows extreme flexibility when developing a water distribution model. The user can develop a model from scratch using a variety of input methods including importation of data files from a GIS database or pre-existing water distribution model, schematically drawing the pipe network, or by direct data entry using the program editors.

If a map of the water distribution system is available, MIKE URBAN can import this map and display it as a background image—allowing the user to interactively construct and layout the pipe network system. Network components can be selected from a component toolbar, and then graphically placed on the screen at the precise location of each component.

Many times, existing water distribution systems do not have a detailed map that can be used to graphically construct a network model. For these situations, MIKE URBAN allows the user to develop a model by simply defining water distribution components (i.e., pipes, junction nodes, pumps, valves, tanks, and reservoirs) in interactive, easy-to-use dialog boxes. This allows the user to define a model when an accurate map is not available for the pipe network model.

After the pipe network has been defined, a hydraulic analysis of the network can be performed. Detailed reports can be generated from the analysis results and printed out. In addition, if a graphical representation of the pipe network has been created, the computed pipe flowrates and junction node pressures can be displayed and printed out.

This User Guide consists of the following chapters:

1. Using the Model: provides the fundamental understanding of the basic operation of MIKE URBAN Water Distribution modeling.
2. EPANET Input Descriptions: provides detailed descriptions of the data input dialog boxes used by this application.



3. Water Distribution Tools: provides detailed description of the additional modeling tools including fire flow analysis, pipe roughness calibration, and others.
4. Water Hammer: provides detailed descriptions of the water hammer simulations.
5. Program Methodology: presents the theory and methodology that was used in developing this program.

2.1 Overview

MIKE URBAN Water Distribution (WD) consists of the following 'modules'.

2.1.1 EPANET

Pipe Flow Module

MIKE URBAN applies the widely used and universally accepted "numerical engine" developed by US EPA - EPANET 2.0 - for the computation of steady state and quasi steady state flow, pressure conditions and water quality in pipe networks. The method used by EPANET to solve the system of node and link equations is known as the "gradient algorithm" and has several attractive features. First, the system of linear equations to be solved at each iteration of the algorithm is sparse, symmetric, and positive-definite. This allows highly efficient sparse matrix techniques to be used for their solution. Second, the method maintains flow continuity at all nodes after its first iteration. And third, it can readily handle pumps and valves without having to change the structure of the equation matrix when the status of these components changes.

The most important MIKE URBAN Pipe Flow features are:

- Steady state analysis.
- Extended period analysis.
- Rule based control.

Water Quality Module

Water quality analysis is used to compute the age of water, perform source tracking, calculate the fate of a dissolved substance, or determine the growth or decay of a substance. The water quality solver is based on the efficient Lagrangian Time Driven method. Water quality sources are defined in terms of mass inflow rate in addition to concentration level. Bulk flow reactions can be modeled with n-th order kinetics, pipe wall reactions can be modeled with either zero- or first-order kinetics, a water quality constituent's growth or decay can be modeled up to a limiting potential, and pipe wall reaction coefficients can be correlated to pipe roughness coefficients.



The most important MIKE URBAN Water Quality features are:

- Water age analysis.
- Chlorine concentrations analysis.
- Path and concentrations of pollutants analysis.

2.1.2 WD-Tools

Fire Flow Analysis

Calculating water availability for fire protection requirements is one of the most frequent modeling tasks of water utility. 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.

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.

Pipe Roughness Calibration

The Pipe Roughness Calibration module automatically adjusts pipe roughness coefficients to best match field pressure observations and best reflect what is actually occurring in your system. Both English and Metric units are fully supported in the Pipe Roughness Calibration. Pipe roughness coefficients can be calculated for the Hazen-Williams, Darcy-Weisbach (Colebrook-White), or Manning friction loss equations.

PID Control

The PID (Proportional-Integral-Differential) Control module provides advanced Real-Time Control simulation capabilities for water supply and water distribution systems. It permits description of various controllable devices and makes the definition of complex operational logic for interdependent regulators fully transparent and time efficient.

The following controllable devices can be specified:

- Control valve (PRV-Pressure Reducing Valve, PSV-Pressure Sustaining Valve, PBV-Pressure Breaker Valve, TCV-Throttle Control Valve, FCV-Flow Control Valve)
- Pump



Variable Speed Drive

A variable pump can vary its speed setting and/or change its status to open or closed during a simulation. VSD pump extension allows modeling of a pump where the rotational speed of a pump is automatically adjusted by the program in order to maintain the control node pressure at the required level. VSD pump using can also be modeled using the PID control module.

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. In order to allow for modeling of chains of pumps or valves chains in efficient way, the EPANET Rule-based control syntax was extended by adding LIKE keyword, for example.

Demand Adjusted Analysis

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

Pressure Zone Mapping

This tool helps to visualise how different network parts are hydraulically interconnected and where the HGL line breaks.

Water Hammer

Water hammer simulates transient (unsteady) flow in any fully pressurized system carrying liquids. MIKE URBAN 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 URBAN EPANET based steady state flow module.

Water Hammer allows you to model:

- Pump start-up and pump trip-off.
- Water hammer protection of pumping stations.
- Power failure events.



2.2 Application Basics

The following subsections discuss the application basics of MIKE URBAN Water Distribution. These subsections describe the steps involved in defining a water distribution model and what data is required for the model definition.

2.2.1 Defining the Model

It is generally more efficient to gather and organize the data required to define your pipe network model before you begin input of data into MIKE URBAN Water Distribution.

MIKE URBAN Water Distribution views the water distribution network as a collection of links connected together at their end-points by nodes. Links and nodes are identified with ID numbers and can be arranged in any fashion. Figure 2.1 shows an example water distribution network and its related components.

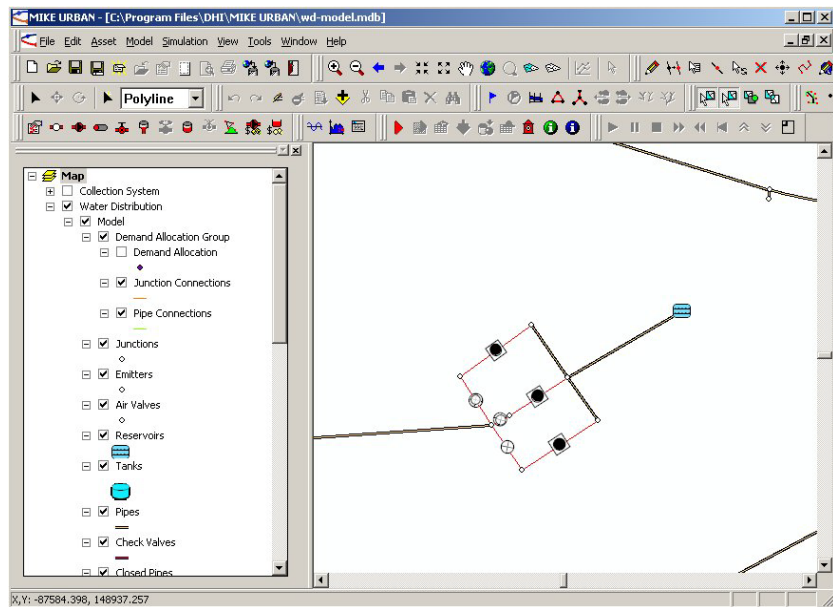


Figure 2.1 The network components that make up a typical water distribution model

For large networks, collecting the input data to define a pipe network model can involve considerable effort. Also, entering this input data into the computer can, at times, become tedious. This process can be prolonged by errors introduced while entering the data. To reduce the possibility of error, the following procedure is recommended for preparing and entering the input data and performing the network analysis.



1. Obtain a map, or create a schematic diagram of the pipe network to be modelled. Number all pipes, junction nodes, and control components in the network. These numbers will then be used as the numerical or string IDs by MIKE URBAN Water Distribution in its analysis. These IDs can be used in checking your work. In addition, pipes, nodes, pumps, valves, storage tanks, and all other network components can also be given text descriptions for reference purposes.
2. Prepare the input data for each pipe and junction node. The data for these components should be complete. For example, pipe data includes the pipe length, diameter, roughness coefficient, and minor loss coefficient for each pipe in the network. If the roughness coefficient is not available, the pipe material and age can be used to interpolate a roughness coefficient value using the roughness coefficient tables provided in MIKE URBAN Water Distribution. Junction node data includes the water demand and node elevation for each node in the network.
3. If there are control components in the network, prepare the input data for each component. Control components include pumps, check valves, regulating valves, sustaining valves, flow control valves, and storage tanks.
4. Enter the input data which defines the network model using the MIKE URBAN Water Distribution data input dialog boxes. The program will report an error if invalid data is entered. This immediate checking of input data helps prevent errors from being introduced into the pipe network model.
5. After the input data has been entered, MIKE URBAN Water Distribution can perform a complete check of the input data and a geometric verification of the network connectivity. In addition, it can print a summary of the input data to allow the user to verify the entered data. Many of the input data dialog boxes, such as the and Pipe Editor dialog boxes, can be placed side by side to allow the user to more easily verify model consistency.
6. After the input data has been verified, the pipe network can then be analysed. Analysis results should be carefully examined to make certain that the input data accurately defines the model and that the analysis results appear reasonable. Of particular importance are the junction node pressures. These values should lie within a reasonable range.
7. Calibrate the network model to make certain that the analysis provides accurate modelling results. For more information on how to calibrate your network model, see the section titled “Model Calibration” on page 243.



Constructing Complex Networks

When constructing a complex pipe network model, begin by first constructing a simplified version of the network. For example, leave out the network control components. Then, analyse the network and verify that the modelling results appear reasonable. If they are, then add the control components to the network.

Roughness Coefficient Data

Model calibration should be performed to properly adjust the pipe roughness and minor loss coefficients so that the defined computer model pipe network accurately models the actual physical pipe network.

However, as a pipe network ages, pipe roughness change due to corrosion and deposition. During a pipe network's design stage, it is important to know how age can affect the pipe roughness coefficients. Roughness coefficient data from textbooks and engineering handbooks may not accurately reflect the aged roughness coefficients. Generally, accurate aged roughness coefficients can be obtained from the pipe manufacturer. Otherwise, laboratory experiments or field measurements may need to be performed to estimate these values.

2.2.2 Input Data Requirements

Defining a pipe network model requires a variety of complex and interrelated input data. The input data required to define a pipe network model is categorized and described in this section.

Project Data

General data describing the pipe network model, such as project and output specification settings, is categorized as project data.

Project Specification Data

Included in the project specification data are general settings, such as friction loss formulation, units used in the analysis, and the type of analysis to be conducted (i.e., steady state, extended period, water quality, etc.). If the analysis is an extended period simulation, this data also defines the total simulation time and time step size.

Network Data

Network data consists of all components that make up the network model, such as pipes and junction nodes. This subsection lists all components that can be used in a steady state analysis.



Pipe Data

Pipe data includes the physical characteristics of each pipe, such as pipe length, diameter, roughness coefficient, and minor loss due to fittings along the pipe. In addition, check valves can be defined along a pipe to prevent flow reversal. The pipe data provides the basic information for the hydraulic analysis, and should be prepared carefully.

A text description can be defined for each pipe. Because pipes are associated with the roads and streets that they lie under, a pipe is generally labeled by its associated street name.

Junction Data

Junction data describes the physical characteristics of each junction node, such as external water demand and node elevation.

A text description can be defined for each junction node. Junction nodes are often labelled with names corresponding to the buildings they are located near.

Reservoir Data

Reservoir data specifies the reservoir's water surface elevation. In addition, a text description can be defined for each reservoir.

Storage Tank Data

Storage tank data includes the minimum, maximum, and initial water surface elevations for each storage tank. MIKE URBAN Water Distribution distinguishes between cylindrical, rectangular, and any shape storage tanks. The geometry of a cylindrical tank can be fully described by its diameter, while rectangular tanks are defined by a width and length. A text description can be defined for each storage tank.

Emitters Data

Emitters are devices associated with junctions that model the flow through a nozzle or orifice. In these situations the demand (i.e. the flow rate through the emitter) varies in proportion to the pressure at the junction raised to some power. The constant of proportionality is termed the "discharge coefficient". For nozzles and sprinkler heads the exponent on pressure is 0.5 and the manufacturer usually states the value of the discharge coefficient as the flow rate in gpm through the device at a 1 psi pressure drop. Emitters are used 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 and pressure exponent for the leaking crack or joint can be estimated) and compute a fire flow at the junction (the flow available at some minimum residual pressure).

Pump Data

Pump data is defined by selecting one of four pump types: constant power, standard pump curve, custom pump curve with no extended flow range, or custom pump curve with extended flow range. Each of these pump types



require specific data. If a pump is a constant power pump or a constant head pump, the pump is described by the power or head setting of the pump. If a pump has a pump curve associated with it, the pump is described by a pump characteristic curve consisting of pump head and discharge values. A text description can be defined for each pump.

Valve Data

Valve data includes the valve type, the HGL setting of the valve, and the reference junction node. In MIKE URBAN Water Distribution, there are six valve types: pressure reducing valves (PRV), pressure sustaining valves (PSV), pressure breaker valves (PBV), flow control valves (FCV), throttle control valves (TCV) and general purpose valves (GPV). The GPV allows the user to define the flow-headloss relationship through the valve. Most pipe networks use valves to regulate flowrates and pressures. Valves greatly affect the analysis results. Therefore, valves should be defined carefully. A text description can be defined for each valve. A check valve is not defined with the valve data, but is defined with the pipe data.

Consumption Data

Consumption data includes junction node demands and time patterns to additional categories of water users at a junction as well as demand allocation data, and pipe demand coefficients. For more information on how to develop node demands, see the section titled “Distributed Demands” on page 119.

Extended Period Data

If an extended period analysis is to be performed, extended period (dynamic) components such as pressure switches can be incorporated into the pipe network.

Control Data

Control data allows pipes, pumps, and valve settings to change at specific times or when specific pressures or tank levels are reached in the network. A text description can be defined for each control.

Water Quality Data

If a water quality analysis is to be performed, then specialized data is required.

Point Source Data

Point constituent source data includes the reservoir, storage tank, or junction node where the constituent (e.g., chlorine) is introduced into the pipe network. In addition, the substance's decay rate, current age, and the amount introduced is specified. A text description can be defined for each point source.



The decay/growth rate is defined using water quality reaction equations. MIKE URBAN Water Distribution can model either bulk flow reactions or reactions with the pipe wall. Bulk flow reactions are reactions that occur in the main flow stream of a pipe or in a storage tank, unaffected by any processes that might involve the pipe wall. Pipe wall reactions are reactions that may involve the pipe material or occur close to the pipe wall out of the main flow stream.

Fire Flow Data

MIKE URBAN Water Distribution can be used to perform a fire flow analysis for any junction node within the network system. Please refer to the *Tutorials Guide* for a complete discussion on how to perform a fire flow analysis with MIKE URBAN Water Distribution.

Water Hammer Data

MIKE URBAN Water Distribution can be used to perform a water hammer analysis for the network system. See the *Tutorials Guide* for a tutorial on how to run a Water Hammer analysis as well as *Water Hammer Analysis Chapter* for a complete discussion on how to perform a water hammer analysis with MIKE URBAN Water Distribution.

2.3 Entering Data

MIKE URBAN is extremely flexible in how a water distribution model can be developed. The user can develop a model from scratch using a variety of input methods, including importation of data files from a GIS database or pre-existing water distribution models, schematically drawing the pipe network, or by simple data entry.

2.3.1 Interactive Data Entry

MIKE URBAN has been designed to make developing a water distribution model easy and flexible by providing a variety of interactive input methods. For example, the user can graphically trace out an existing water distribution pipe network on top of a scanned-in aerial map. Or, the user can manually specify the pipe network system through the program's interactive network component editors. And any of these methods can be used simultaneously with any other, allowing complete flexibility while defining the network system. This translates into greater productivity for the user.

Most data used to describe a water distribution model is defined using interactive dialog boxes, an example of which is shown in Figure 2.2. These dialog boxes allow the user to quickly comprehend what data input requirements are needed to define the model.

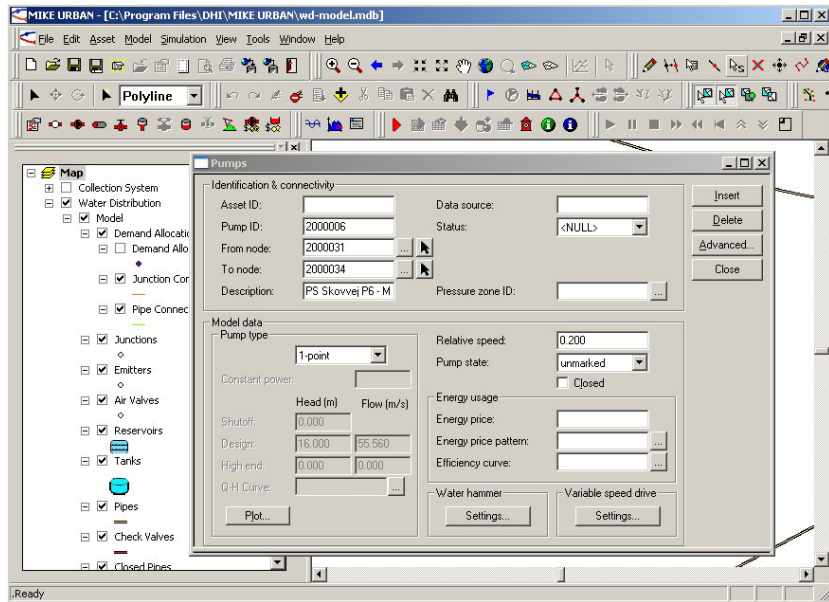


Figure 2.2 This interactive dialog box illustrates how data is entered to define a water distribution network model

The MIKE URBAN Water Distribution data input requirements, the methods available to describe this data, and the dialog boxes used to enter this data are discussed in detail in Chapter “EPANET Input Description” on page 11.

2.3.2 Graphical Input

As was discussed in the previous section, MIKE URBAN allows you to enter data interactively into the pipe network using interactive dialog boxes. This allows you to quickly define a model if a schematic layout of the pipe network is unavailable, or if developing a schematic layout is too expensive or time-consuming to construct. However, a powerful capability of MIKE URBAN is to allow the user to graphically construct the pipe network model by simply pointing and clicking with a mouse. This graphical construction of a pipe network system is shown in Figure 2.3.

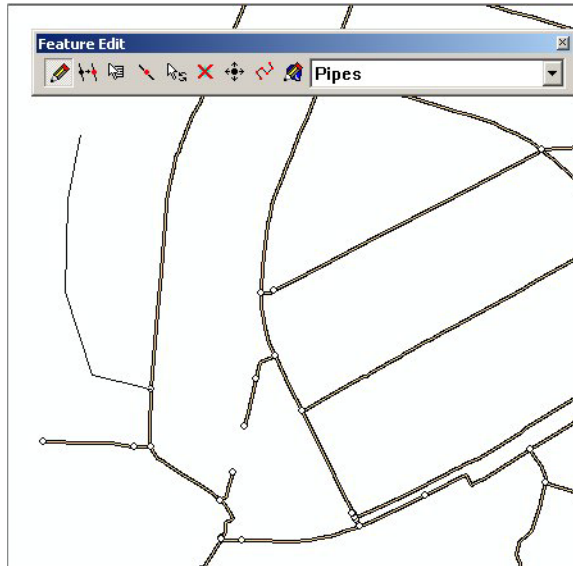


Figure 2.3 MIKE URBAN allows the user to interactively develop a pipe network model by simply pointing and clicking to graphically add the components to construct the water distribution network

Using the tools in the Component toolbar, for example, the user can select the junction node component tool (Add Junction icon) and then interactively locate and place junction nodes on the Map window. A background aerial image can be displayed in the Map window to aid in the placement of network components. While the user is moving the cursor in the Map window, a precise X,Y location is displayed on the status line at the bottom of the program.

To delete a network component, the *select* component tool can be used interactively to select network elements. Then, selecting «Delete Selected» will delete the selected elements. Note that you can undelete the deleted elements.

2.3.3 Importing Graphical Data

MIKE URBAN allows you to import network data, such as ESRI Shapefiles, DWG, DGN, DXF files and raster images (such as scanned-in quad maps or aerial photographs), and display them as a background layer in the Map window. This enables you to import a graphical background layer to facilitate laying out the pipe network. You can also import a DXF file containing lines and polylines representing the pipe network and MIKE URBAN will convert the lines and polylines to equivalent pipes.

To import a graphical data file, select *Add Data* tool. MIKE URBAN will then display the Add Data dialog box, as shown in Figure 2.4. Select the file format to import and then choose «Add».

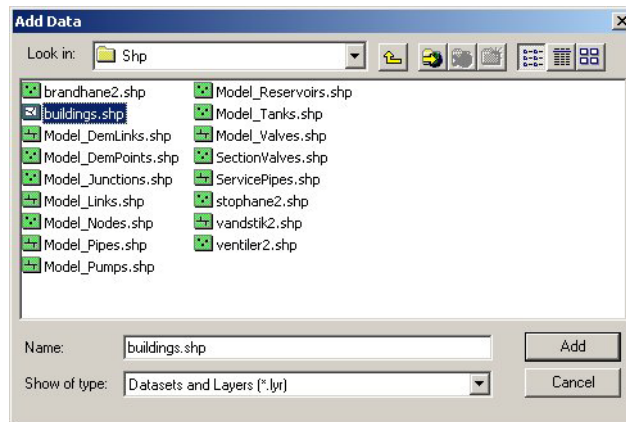


Figure 2.4 The Import dialog box provides support for importing vector files and raster image files

Hiding and Showing Graphical Data

Once graphical data has been imported and displayed as a background layer, it may be necessary to hide (or show) this layer or other layers. To do so, select **View | Table of Contents**. This will display the Table of Contents dialog box.. This dialog box allows you to control what layers are to be displayed in the Horizontal View window. Select the layer to hide (or show) and then choose «Close». MIKE URBAN will the update then display.

2.4 Demand Processing

MIKE URBAN Water Distribution provides the user with a wide range of tools suitable for network demand processing including individual node demand definition, multiple demands, diurnal demand curves, automatic demand distribution based on the node and pipe demand coefficients, and reusing data from the customer information system.

2.4.1 Distributed Demands

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

The Distributed Demands dialog box, as shown in Figure 2.5, is used to automatically assign the demands at the appropriate junction nodes. The Distributed Demands dialog box is reached by selecting **EPANET | Distributed Demands**.



Pipe Demand Coefficients

MIKE URBAN computes the water demands for each node in the network system based upon the total network demand using two methods: 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.

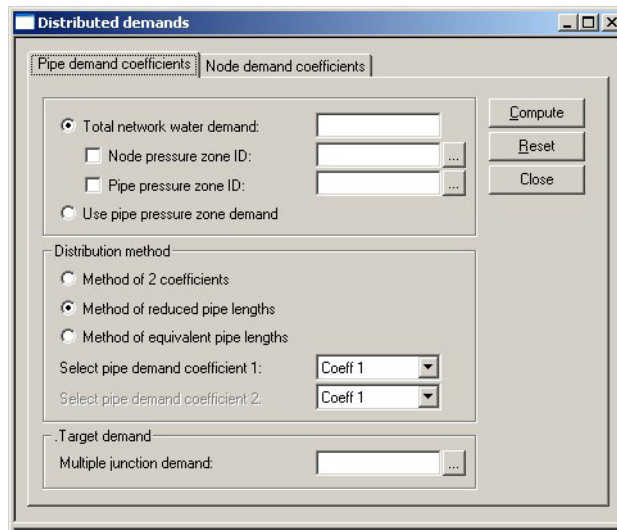


Figure 2.5 The Distributed Demands dialog box computes the demand at each junction node automatically based upon the total demand within a pressure zone or entire network system

See the lesson in Tutorials Guide on Distributed Demands for a complete description on how to perform a demand distribution with MIKE URBAN Water Distribution.

METHOD OF TWO COEFFICIENTS or METHOD OF REDUCED PIPE LENGTHS or METHOD OF EQUIVALENT PIPE LENGTHS

MIKE URBAN 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*. This radio button group allows the user to select the method to be used.



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

$$q_{pi} = \frac{(Q)k_{1i}k_{2i}}{\Sigma(k_{1i}k_{2i})} \quad (2.1)$$

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

$$q_{pi} = \frac{(Q)l_i k_{1i}}{\Sigma(k_{1i}l_i)} \quad (2.2)$$

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

$$q_{pi} = \frac{(Q)l_i k_{Di}}{\Sigma(k_{Di}l_i)} \quad (2.3)$$

where:

q_{pi} = Total water demand applied to the pipe, split between the two end nodes.

Q = Total network water

l_i = Pipe length

k_{1i} , k_{2i} = pipe demand 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 6inch). 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.

Node Demand Coefficients

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

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.



EQUAL NODE DEMAND DISTRIBUTION

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

Selecting Equal Node Demand Distribution computes the junction demand:

$$q_{ni} = \frac{Q}{N} \quad (2.4)$$

where:

Q = Total network water demand (or zone demand)

q_{ni} = calculated demand at each junction node

N = junction nodes count with the selected zone or a total network

NODE DEMAND DISTRIBUTION

Selecting the Method of Node Demand Coefficients, MIKE URBAN computes the total water demand assigned to each junction node as:

$$q_i = \frac{Q_t}{\sum_{i=0,n} C_i} \cdot C_i \quad (2.5)$$

where:

q_i = node demand

Q_t = total network demand

C_i = demand coefficient

Developing Pipe Demand Coefficients

There are many ways of developing pipe demand coefficients. Typically, pipe demand coefficients correspond to the amount of invoiced water along the specified pipe, which is the most accurate data source for the Two Coefficient Method. Alternatively, the number of inhabitants can be supplied. If such data is unavailable, it is also possible to classify pipes by the residential type (family houses, commercial, city centre) and to use such classification in the Pipe Length Method.

In cases, when node demands are retrieved from the Customer Information Systems, it is possible to calculate pipe demand coefficients in the form of aggregated demands for streets or the counted lots



Counted Lots

The pipe demand coefficient is derived from the X,Y position of the counted lots. In order to use X,Y position of a point-attributes for the geocoding process, it is necessary to create a database file, such as Dbase.DBF, Microsoft Access.MDB where X,Y and other attributes are stored, or create the X,Y position for each point in ArcGIS, for example. Use the import features of MIKE URBAN to import such data to the Demand Allocation editor.

Example: the pipe demand coefficient COEFF1 was updated (aggregated) from the X, Y points (demand lots), which are also displayed in the horizontal plan window for each corresponding demand allocation point.

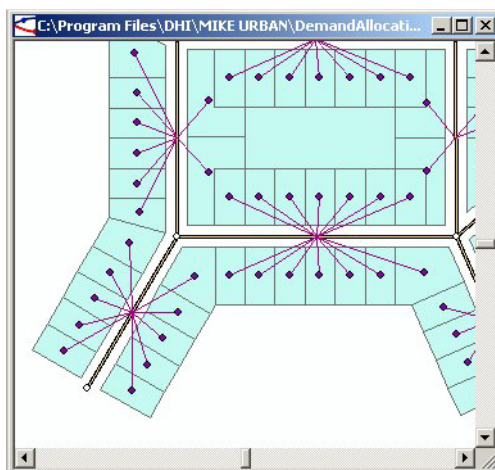


Figure 2.6 The demand coefficients (represented by circles) are assigned to the nearest pipe within the snapping radius defined in the demand allocation dialog box

Accumulated Street Demands-Polygons

In some cases, it is possible to create polygons, covering streets and to assign the aggregated demand to them. This solution assumes, that each street included in the model has its corresponding covering polygon in ArcView, and the name of a street is included in the model database together with its sum demand value from the billing database.

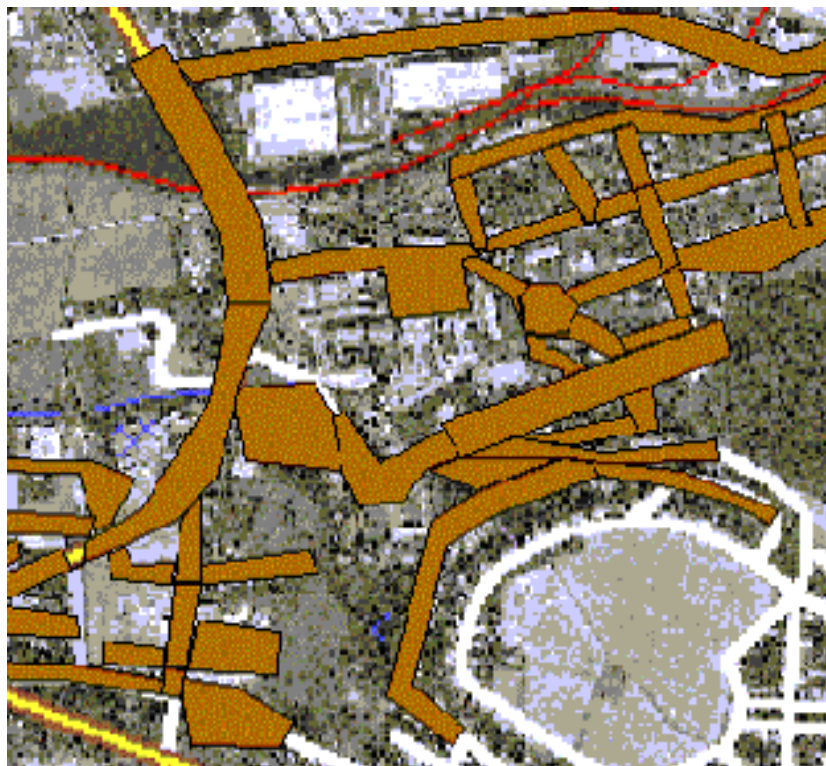


Figure 2.7 Polygons created to cover streets

Demand Allocation

Junction node demands can be edited either within the junction editor for each particular junction node or in the Multiple Demand editor, which allows the user to display and edit all multiple demands. In addition to that, MIKE URBAN provides another way of developing junction node demands, which is based on geocoding the consumption data to the appropriate junctions or pipes and aggregating their set point demand values to the junction demands. This allows simplify the demand development process and allows you, for example, to import consumption data from the consumption database systems and geocoded it based on its X, Y geographical coordinates.

See the Tutorials Guide for the tutorial on demand allocation for a complete description on how to perform a demand allocation with MIKE URBAN Water Distribution.

Demand Editing and Demand Scenarios

In addition to the automated demand processing; node demands can be edited individually within the Junction Editor and/or the Multiple Demand Editor. Each node can have unlimited number of demands and each demand



can be linked to its diurnal curve. For more details, see Junction Editor and Multiple Demand Editor.

Global Editing

Using the Field Calculator tool can change node demands. This allows us to select the part of the network and to increase the node demand by 20%, for example.

To do this, open the Multiple Demand Editor, select the column *Demand* and select the *Field Calculator* from the right-mouse click pop-up menu. Define the following statement:

```
DEMAND=DEMAND * 1.2
```

To increase node demands associated with the specific diurnal curve (Pattern), use the following statement:

```
DEMAND=DEMAND * 1.2 where PATTERN =P1
```

Convenient way of handling different scenarios is to change the Total Network Demand within the Distributed Demand dialog and to increase or decrease the node demands by redistributing another total consumption.

2.5 Performing an Analysis

Once you have defined a water distribution input model, you are ready to perform an analysis. The following tasks are required in performing an analysis.

First, you need to check the model for errors. MIKE URBAN contains a project check tool that can verify the data used to define an EPANET model (see more in the Model Manager User Guide).

Second, after the program has checked the model for errors, you can then execute the EPANET network hydraulic analysis. The EPANET network analysis program will read the input data and will then compute the flow rates, pressures, and water quality for the defined pipe network.

The following subsections discuss these tasks in detail.

2.5.1 Executing the Analysis

Once the project check tool has successfully checked the model for potential problems, you are ready to perform an analysis of the water distribution network model. To execute the analysis, select **Simulation | Run Simulation**, as shown in Figure 2.8.

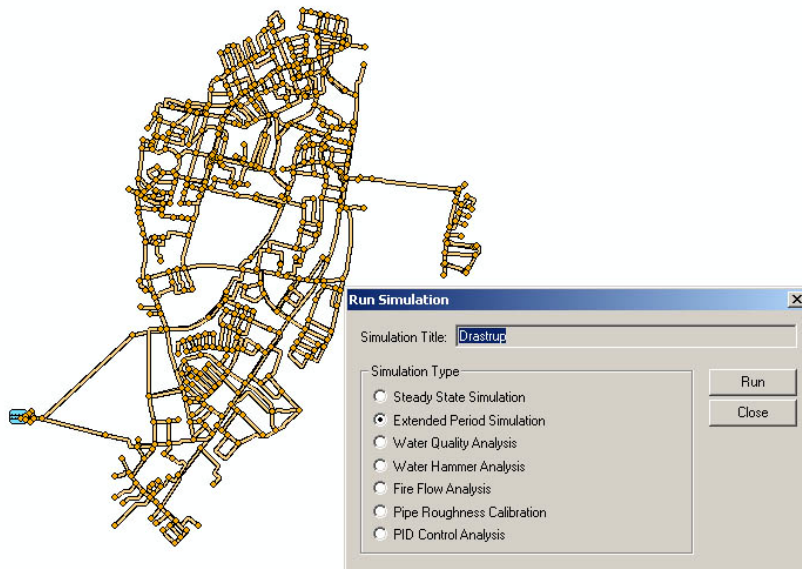


Figure 2.8 The File Menu is used to select PERFORM ANALYSIS to analyze the water distribution network

Please note that EPANET engine does not handle special characters such as < > ? " : | \ / * . Make sure that neither the database nor the scenario ID contain such characters.

After completing the analysis, the program will report the total number of warnings generated during the analysis and whether the analysis ended with an error. You can then view the analysis results. If the analysis ended with an error or if the warnings are serious, you can then make the appropriate corrections to the input data.

2.5.2 Batch simulations

The 'Simulation|Run Batch Simulation' allows you to select any number of available scenarios and use them for running the simulation. You can use ">", ">>", "<", and "<<" commands to select or deselect available scenarios. Once done, select "OK" to start the batch analysis.

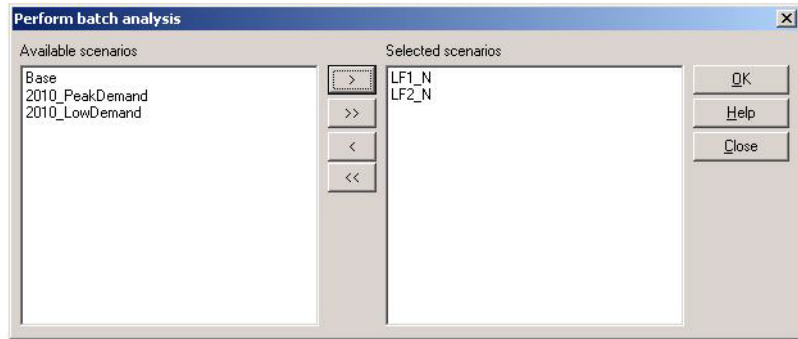


Figure 2.9 The LF1_N and LF2_N scenarios are selected for the batch analysis

2.6 Viewing Analysis Results

Once the analysis has been run successfully, the analysis results can be displayed. To load the computed analysis results, select Model Results| Load Simulation Results. MIKE URBAN will then load the computed analysis results.

The water distribution analysis results are written to an output file with the same filename (but with a RES file extension) as the input data file.

To load EPANET results, go to **EPANET | Load Results | EPANET Results**. A dialog with choices of which data to load from the result then appears as shown in

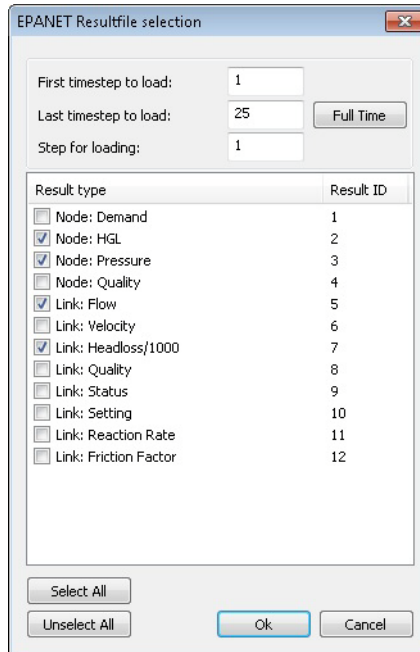


Figure 2.10 EPANET result file selection dialog

In addition to the standard EPANET analysis results, MIKE URBAN can display the output results in a summary table, generate custom output reports, and display many different graphical representations of the analysis results. The following sections describe these capabilities.

2.6.1 Comparing Alternative Solutions

MIKE URBAN can display multiple analysis result files for a network. In order to compare two analysis result files for a network (for example, to compare design alternatives of the same network), MIKE URBAN can subtract the two analysis result files from each other. MIKE URBAN will then display the difference between these two analysis results for both node and pipe values (such as node pressure, hydraulic gradeline, water quality, demand, flow, velocity, and hydraulic headloss). The result difference can be displayed in the same format as the standard analysis results.

To subtract two analysis result files, select Model Results| Results Comparison. Note that it is only possible to subtract two analysis result files if the number of nodes, pipes, and time-steps (if performing an extended period simulation) are the same. The following comparisons can be performed.



Different Demands

Network demands can be different (such as comparing a work and weekend day), or local nodal demands can increase (such as simulating fire flows or higher demands from what was predicted from the original master plan).

Different Pipe Roughness

Model calibration requires many changes in pipe roughness coefficients. Pipe aging can also be modeled (where the pipe roughness increases due to aging).

Different Pumping Scenarios

Different pump schedule scenarios can be compared. Network optimization may be based on reducing pumping to save money used in powering pump stations. By displaying the original and improved network results, it is easy to show that the new network pumping schedule meets the same criterion (flow and pressure specifications) with less pumping and/or pumping at more off-peak hours.

Effect of Pressure Reducing Valves

Decreasing the maximum pressure within the network has the positive effect of decreasing water leakage at pipe, pump, and valve joints. Optimizing pressure reducing valves (PRVs) settings can sometimes be a primary task of network optimization. Displaying the pressure differences between the original and optimized network provides a very good overview of potential water losses.

Water Quality Parameters

Modeling different water quality scenarios (such as different chlorine supply patterns from water treatment booster stations) allow the user to quickly examine water quality for the network.

When subtracting the two analysis result files, a positive resulting value means that the first analysis result file has a larger value than the second analysis result file.

2.6.2 Viewing the EPANET Analysis Results

When EPANET performs its analysis of the water distribution network, it generates two ASCII output files containing the analysis results—a summary output file and a complete output file. The summary output file contains a short description of the analysis results. The complete output file contains the contents of the summary output file in addition to output results for every component within the water distribution network.



To view the EPANET summary analysis results, open the .SUM file using Notepad, for example or select Summary from within Run Simulation dialog.

2.6.3 Result Browser Window

The Result Browser window allows you to graphically select any network component in the Map window, by simply clicking on it with the mouse, and the program will then display that component's input attributes and analysis results. This allows you to quickly examine the pipe network system at the component level (i.e., pipe, junction node, valve, pump, tank, and reservoir), check what is defined for the model, and determine the computed analysis results. For example, selecting a pipe from the Map window will display in the Browser window the pipe's ID, diameter, length, roughness coefficient, minor loss coefficient, reaction rate coefficient, open/close status, and flowrate.

Please also refer to the Model Manager User Guide for further on presenting simulation results.

2.6.4 Results Table

The Results Table allows you to display the tabular list of analysis results for junctions, tanks, pipes, pumps, and valves. It is possible to display the results for all elements or for the selection set. The selection set can be defined based on the rules (constraints) such as "pressure below 15", "velocity above 1.5", and similar. It is also possible to display the tabular results for nodes and links, which are graphically selected from the Map.

2.6.5 Thematic Mapping

The Map window allows you to graphically plot the analysis results directly onto the pipe network schematic. To add a layer to the Map window, select Model Results | Add Results Layer and pick up the selected result item, such as Link:Flow, from the list.

The user can change most of the display options. In the Map window, complete thematic mapping of the analysis results is available, including node elevation, HGL, pressure, demand, and any water quality constituent. This allows you to quickly interpret the modeling results and identify any trouble areas. And, directional flow arrows can be plotted on top of the pipes to show the flow direction for any time-step. In addition, MIKE URBAN provides automatic color-coding of pipes and nodes based upon any input or output property, allowing the network to be color-coded based upon pipe sizes, flowrates, velocities, headlosses, nodal pressures, nodal demands, hydraulic grades, elevations, water age, percent source contributions, water quality concentrations, and any other attribute. Numerical ranges for colors can be specified. Furthermore, pipes can be plotted with variable width and nodes with variable



radius, allowing you to quickly identify those areas of the network experiencing the most flow, headloss, water quality constituent concentration, etc.

The link arrows corresponding to the link orientation, pipe slope and/or to the flow direction can be drawn by the selected size. The node and link labels can be displayed for all elements or only for selected elements. It is possible in this way to display the labels just for filtered nodes and pipes, for nodes and pipes along a specific profile or a flow path, for the individually selected nodes etc. This is useful especially when we want to display very quickly results for several elements on the plan. The values of pipe diameter and pipe length can be displayed together with the prefix.

Please also refer to the Model Manager User Guide for further on presenting simulation results.

2.6.6 Profile Plots

Profile plots allow you to graphically plot the analysis results along any pipe-line path. To display a profile plot, a profile path must first be defined from the pipe network horizontal plan. Once the profile path has been defined and the profile plot displayed, the path can be saved for later re-use.

Profile plots allow you to graphically plot the analysis results along any pipe-line path. Profile plots can have two separate vertical axes to allow plotting of variables from two separate unit families, such as flow and pressure. Profile plots can be plotted along any user-specified route. Profile plots can be generated as line graphs, bar graphs, or mixed—along with complete graph customization. For example, profile plots can be plotted with an envelope to show the minimum and maximum values reached during an extended period simulation.

Please also refer to the Model Manager User Guide for further on presenting simulation results.

2.6.7 Time Series Plots

Time series plots allow you to graphically display the analysis results for any network element for an extended period simulation. Multiple time series plots can be generated for the various network elements, such as pipe flow, velocity, headloss, nodal demand, pressure, hydraulic grade, water age, water quality constituent concentration, pump characteristic operating curve, tank water level, total and net system demand, etc.

Please also refer to the Model Manager User Guide for further on presenting simulation results.



2.6.8 Link Q-H Plots

Time series scatter graphs can be used to display the Q-H relation for any selected link for an extended period simulation.

2.6.9 Fire Hydrant Q-H Plots

The Q-H relation plots allow you to graphically display the fire flow analysis results (calculate hydrant curve) for any fire hydrant node.

2.6.10 Current Operating Pump Q-H Point

The position of the current operating points can be displayed for any pump for an extended period simulation. In order to display this graph, load the simulation results, open the Pump Editor and display the graph.

2.6.11 Check for Pipes and Nodes

It is possible to detect pipes where the flow direction has changed during the hydraulic simulation. The same tool can also be used for detecting links and nodes, where the results (pressures, for example) changed more than a specified limit.

2.6.12 Animation

MIKE URBAN will automatically generate animations of extended period simulations for both horizontal plan plots and profile plot plots. Animations show values that change with respect to time for extended time period simulations. To display the Animate dialog box, select Animate from the Model Results menu. Note that in order to succeed, either map window or other plot must be displayed. Also we recommend setting the options of the animated plot on color or grade coding to get better results.

See the Model Manager book for more information on viewing simulation results.

2.6.13 EPANET Error and Warning Messages

In order to display EPANET error and warning messages, generated during the steady state or extended period simulation, select Model Results | EPANET Errors and Warnings.

The EPANET Errors and Warnings dialog box allows you to read the error and warning messages from the .SUM file, generated by EPANET and store this information in the database manner. All messages are classified based on their type, which makes it possible to filter out errors or warning related to



pumps or to filter out errors and warning for a specific time level or an model element, for example.

Message	Message ty	Link/Node I	Link/Node t	Simulation t	Simulation t
FCV 2000019	FCV	2000019	Valve	0:00:00	0.000
FCV 2000092	FCV	2000092	Valve	0:00:00	0.000
FCV 2000121	FCV	2000121	Valve	0:00:00	0.000
FCV 2000129	FCV	2000129	Valve	0:00:00	0.000
FCV 2000141	FCV	2000141	Valve	0:00:00	0.000
FCV 2000143	FCV	2000143	Valve	0:00:00	0.000
FCV 2000144	FCV	2000144	Valve	0:00:00	0.000
Pump 2000000	Pump	2000006	Pump	0:00:00	0.000
FCV 2000019	FCV	2000019	Valve	0:15:00	900.000
FCV 2000092	FCV	2000092	Valve	0:15:00	900.000

Figure 2.11 The EPANET Errors and Warnings dialog box is used to process simulation messages

A list of the EPANET Errors and Warnings dialog box data fields for Figure 33.29 follows, with a short description given for each entry.

MESSAGE TYPE

This list box classifies the error and warning message, such as FCV, Pump, Pipe, Negative Pressures, System Unbalanced and similar.

LINK/NODE ID

This data entry contains the model element ID, where applicable.

LINK/NODE TYPE

This data entry contains the model element type, where applicable, such as Junction, Tank, Pipe, Pump, or Valve.

MESSAGE

This is the error or warning message generated by EPANET engine.

SIMULATION TIME

Simulation time of the error or warning message is available in [days:hrs:min:sec] format as well as the number of elapsed seconds from the simulation start.

ADVANCED...

It is possible to Select or Select All messages and map highlight the corresponding network elements on the Map; it is also possible to open Select by Attribute dialog box and filter out messages based on their type, simulation time, model element identification or a type. HTM based report can also be generated.



2.6.14 Store Results to Database

It is possible to store the complete model results (for all time steps) and for the selected part of the network into the database. This is useful when you want to use Microsoft Access or Excel tools for analysing the model results and for using user-defined tools for the additional results processing. In order to store the model results to the database, load the simulation results into MIKE URBAN, and select nodes and links you want to use for storing the simulation results to the database. You can also store all nodes and links in case you do not use the selection set.



3 Water Distribution Tools (WD-Tools)

Water Distribution Tools provides the user with modules and extensions allowing for automatic fire flow analysis, pipe roughness calibration, PID control and demand adjusted analysis. Apart from that, several extensions to the standard EPANET such as VSD (variable speed drive pump) or extended syntax for Rule-Based Controls are available from within standard program dialogs. For information on Water Hammer, please refer to the separate chapter.

3.1 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:

1. Specify a design fire flow rate and compute the available fire flow pressure.
2. Specify a design fire flow pressure and compute the available fire flow rate.

A unique feature of MIKE URBAN is its capability of modelling a fire hydrant connecting pipe with the user defined resistance and other properties. This pipe connects the fire flow node with a fictitious reservoir whose water level equals the required residual pressure. This concept helps to calculate more accurate or realistic results.

The Fire Flow Analysis dialog box is reached by selecting EPANET| WD Tools| Fire Flow Analysis. Note that to run the fire flow analysis, you need to select "Fire Flow Analysis" from within the Run Simulation dialog box.

Figure 3.1 The Fire Flow dialog box is used to specify fire flow analysis parameters

A list of the Fire Flow Analysis dialog box data entries for Figure 3.1 follows, with a short description given for each entry.

Calculate available flow for design pressure, Calculate available pressure for design flow, 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 pressure

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



Maintain minimum residual pressure

This data entry allows you to define the minimum residual pressure for the fire flow analysis. The fire flow analysis is then iterated as long as the minimum residual pressure is maintained in the appropriate nodes (as specified in the 'report critical nodes and pipes section').

Maintain maximum velocity

This data entry allows you to define the maximum acceptable velocity in pipes during the fire flow analysis. The fire flow analysis is then iterated as long as the maximum velocity is maintained in the appropriate pipes (as specified in the 'report critical nodes and pipes section').

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 demand.

Simulate fire hydrant

This data entry allows you to model a fire hydrant 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 connecting pipe. A typical value is 6-10.

Connecting pipe diameter

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

Connecting pipe length

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

Connecting pipe roughness

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

Report critical nodes and pipes

This data entry allows you to report the critical junction nodes, where the minimum residual pressure is less than the critical pressure.

Critical node pressure

This data entry allows you to define the critical residual pressure.



Maximum velocity

This data entry allows you to define the maximum velocity in 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 minimum residual pressure is less than a critical pressure or pipe where the maximum velocity is higher than the maximum velocity limit.

Search within the same zone

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

Use only selected junction nodes, Use all junction nodes

This radio button selection allows you to define whether the fire flow analysis is performed for the selected junction nodes or for all junction nodes.

Select time level

This data entry allows you to define a time level at which the fire flow analysis is performed. Note that the default value is 0 i.e. the first time level corresponding with the steady state analysis.

3.1.1 Browsing Results

Select Model Results | Results Browser to display the Results Browser window and click the junction node to view the simulated fire flow results.

3.1.2 Tabular Results

The simulated fire flow results for “Calculate available flow for design pressure” and “Calculate available pressure for design flow” are reported to the *.FFA text file. Open this text file using any file processing program such as Windows Notepad to see the detailed list of the fire flow results for each simulated junction node. The FFA text file format is suitable for importing into Microsoft Excel, for example. The simulated fire flow results for “Calculate fire hydrant Q-H curve” are reported to the *.FQH text file.

3.1.3 Thematic Maps

The simulated fire flow results can be displayed using the colour coded maps. Select Add Results 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 flow: 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
- Critical events: the number n means that the pressure at the node was n times below the critical pressure during the fire flow simulation (for the surrounding nodes).
- Status: error code (“OK”, “Cannot find minimum pressure limit“, “Maximum pressure is lower then residual pressure“, “Cannot iterate flow for the residual pressure“).

3.2 Pipe Roughness Calibration

The Pipe Roughness Calibration module automatically adjusts pipe roughness coefficients to best match field pressure observations and best reflect what is actually occurring in your system.

One of the main calibration parameters in the pipe network hydrodynamic model is the roughness coefficients. Pipe roughness values may be estimated in two ways: using values from literature or directly from field measurements. To obtain initial estimates of pipe roughness through field testing, it is a good practice to divide the water distribution system into composite zones that contain pipes of like material and age. Additionally, several pipes of different diameters should be tested in each zone to obtain individual pipe roughness estimates. The process of calibration ideally requires simulation over an extended period of time, such as a time range for the maximum day - not just the maximum and minimum hour for the maximum day.

Both English and Metric units are fully supported in the Pipe Roughness Calibration. Pipe roughness coefficients can be calculated for the Hazen-Williams, Darcy-Weisbach (Colebrook-White), or Manning friction loss equations.

3.2.1 Evolutionary Algorithms - Theoretical Background

Evolutionary algorithms (EAs) are engines simulating grossly simplified processes occurring in nature and implemented in artificial media -- such as a computer. The fundamental idea is that of emulating the Darwinian theory of evolution. According to Darwin, evolution is best depicted as the process of the adaptation of species to their environment as one of "natural selection". Perceived in this way, all species inhabiting our planet are actually results of this process of adaptation.

Evolutionary algorithms effectively provide an alternative approach to problem solving - one in which solutions to the problem are evolved rather than the problems being solved directly. The family of evolutionary algorithms today is divided into four main streams: Evolution Strategies (Schwefel,

1981), Evolutionary Programming (Fogel, 1966), Genetic Algorithms (Holland, 1975) and Genetic Programming (Koza, 1992).

Although different and intended for different purposes, all EAs share a common conceptual base (schematized in Figure 3.2). In principle, an initial population of individuals is created in a computer and allowed to evolve using the principles of inheritance (so that offspring resemble parents), variability (the process of offspring creation is not perfect -- some mutations occur) and selection (more fit individuals are allowed to reproduce more often and less fit less often so that their "genealogical" trees disappear in time). One of the main advantages of EAs is their domain independence. EAs can evolve almost anything, given an appropriate representation of evolving structures. Similarly to processes observed in nature, one should distinguish between an evolving entity's genotype and its phenotype. The genotype is basically a code to be executed (such as a code in a DNA strand), whereas the phenotype represents a result of the execution of this code (such as any living being).

Although the information exchange between evolving entities (parents) occurs at the level of genotypes, it is the phenotypes in which one is really interested.

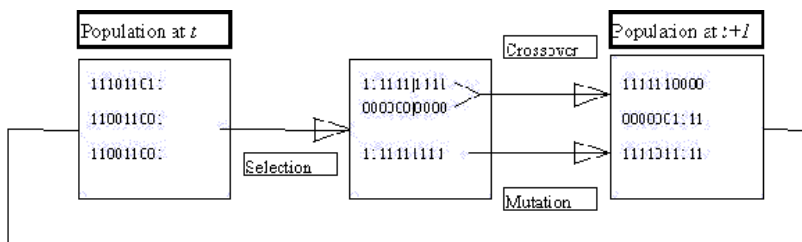


Figure 3.2 Schematic illustration of an evolutionary algorithm. The population is initialized (usually randomly). From this population, the most fit entities are selected to be altered by genetic operators exemplified by crossover (corresponding to sexual reproduction) and mutation. Selection is performed based on certain fitness criteria in which the more 'fit' are selected more often. Crossover simply combines two genotypes by exchanging sub-strings around randomly selected points. In the illustration above, parental genotypes are indicated as either all 1s or all 0s, for the sake of clarity. Mutation simply flips the randomly selected bit

The phenotype is actually an interpretation of a genotype in a problem domain. This interpretation can take the form of any feasible mapping. For example, for optimization and constraint satisfaction purposes, genotypes are typically interpreted as independent variables of a function to be optimized. Along these lines, one can employ mapping in which genotypes are interpreted as roughness coefficients in a free surface pipe flow model with the genetic algorithms (GAs) directed towards the minimization of the discrepancies between model output and measured water level and discharge values.



Resulting GA represents an automatic calibration model of hydrodynamic systems.

3.2.2 Introduction

The pipe roughness calibration will be demonstrated on the example of a gravity network consisting of approximately 600 nodes and 650 pipes. The water distribution network is supplied by water from the upstream reservoir with a fixed HGL = 257 m. The total network demand is approximately 140 l/s, additional 327 l/s supplies adjacent pressure zones from the main pipeline. Flow and pressure measurements were conducted on the network. The pipe age is ranging between 20-80 years and the main pipe material is cast-iron.

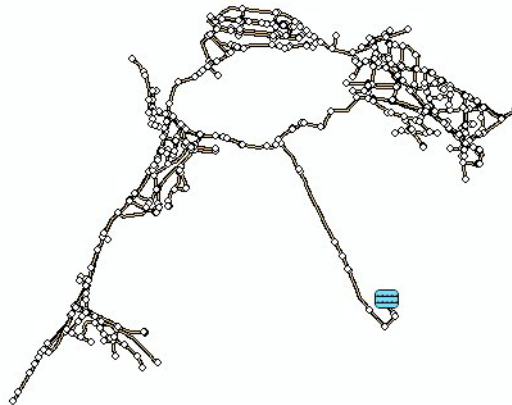


Figure 3.3 Map with the network layout

3.2.3 Pipe Roughness Calibration

The calibration of pipe roughness coefficients consists of several steps:

1. Definition of pipe roughness groups and pipe group assignment.
2. Definition of targeted pressure values.
3. Definition of targeted flow values.
4. Detection of closed pipes (optional).
5. Automated calibration of pipe roughness coefficients by Genetic Algorithms.
6. Assignment of calibrated pipe roughness values to the pipes.

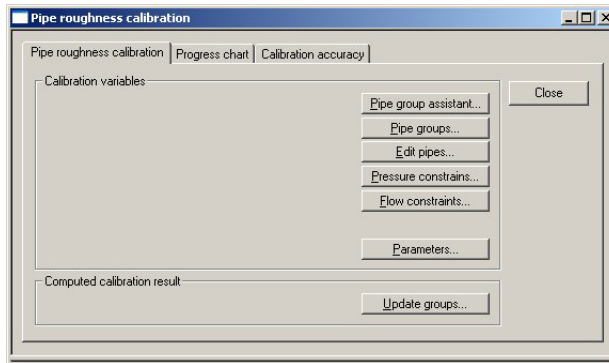


Figure 3.4 Network calibration and optimization dialog box

Pipe Groupings

Pipes can be lumped together in separate logical groups based on their known physical characteristics such as pipe material, age, and diameter. It is assumed that all pipes within a calibration group (where a single pipe may constitute a group) will possess an identical roughness coefficient. Any combination of pipe calibration groups can be specified and their fitness evaluated to match field observations. Selected pipes can be excluded from these groups and their roughness coefficients will remain unchanged during the calibration process. The Group ID can be assigned for each pipe the Pipe Editor. Pipes with undefined Group ID will be excluded from the automatic pipe roughness calibration.

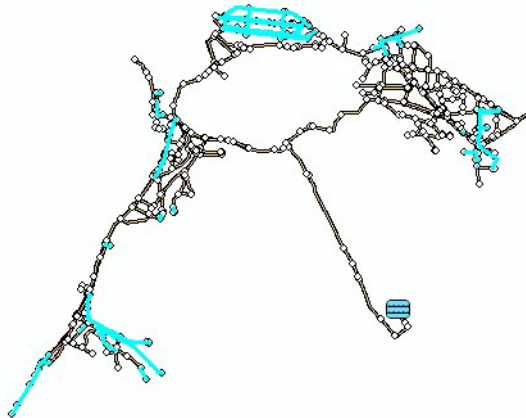


Figure 3.5 Pipes with fixed value of roughness coefficient ($k=3\text{mm}$)

Group Definition

We can create as many pipe groups as necessary and define their minimum and maximum roughness coefficients. Minimum and maximum pipe roughness coefficient is defined for each pipe. Any group can be also displayed on the Map.

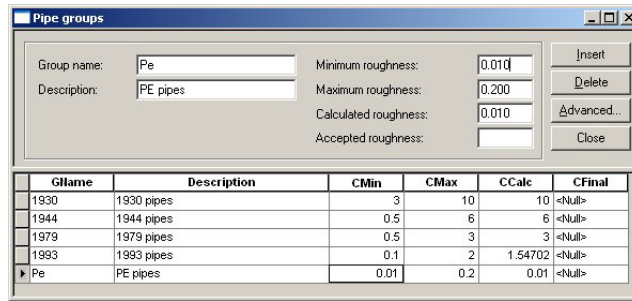


Figure 3.6 Pipe roughness groups define the roughness coefficient limits

Pressure and Flow Measurements

Pressure and flow measurements (fire flow tests, SCADA) can be graphically designated for any set of junction nodes, pipes in the system. Pipe roughness coefficients are automatically adjusted so that the model pressure/flow predictions correlate well with the targeted junction nodes/pipes.

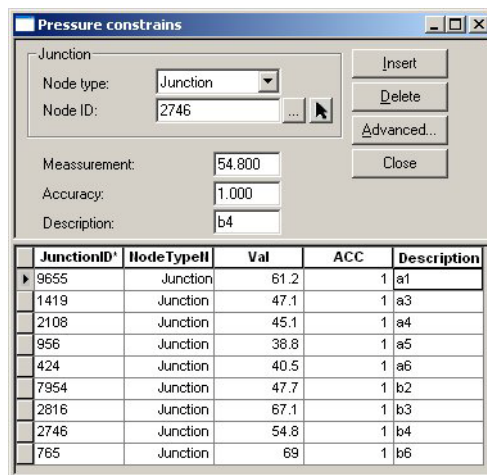


Figure 3.7 Measured pressure in five selected points

The measured flow and pressure values can be entered from within the corresponding editors.

Genetic Algorithms Calibration

Genetic Algorithms pipe roughness calibration can be run when the pipes are assigned to the roughness groups and when one of the flow and pressure targeted values is defined. Several parameters can be defined for the pipe calibration:

- Initial Population Size: 25 (the default).
- Additional Population Growth: 125 (the default).
- Maximum Generation Count: 20 (the default).



The initial population size is the number of initial generation members (the number of networks), which is used by the automated calibration.

The additional population growth is the number of generation members (the number of networks), which are used during the automated population selection and calibration.

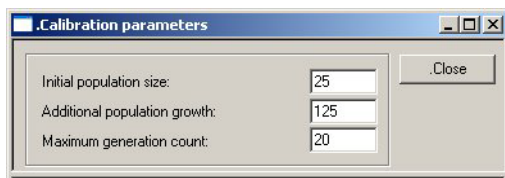


Figure 3.8 Calibration parameters

The maximum generation count is the maximum length of the generation cycle.

Each generation cycle consists of:

1. Initial population estimate.
2. Additional population is created from the initial population size.
3. The hydraulic analysis is performed for each population member (the network).
4. Initial population for the next generation is created from the best population members.

After step 4 has been completed, the cycle is repeated. Three thousand hydraulic analyses will be executed in this example.

[GROUPS]		Min	Max	Best
GroupID				
5	1	3.000	10.000	8.84942
	3	0.500	3.000	0.50000
	2	0.500	6.000	5.51223
	4	0.100	2.000	0.53395
	5	0.010	0.200	0.01000

Figure 3.9 Summary file with then calibrated pipe roughness coefficients

The calibration summary is available by selecting Summary from Simulation dialog, Figure 3.9. If the results are satisfactory, the calibrated pipe roughness coefficients can be loaded and assigned to pipes by selecting Update Pipes from Pipe Roughness Calibration window. To assign the calibrated pipe roughness coefficients to pipes, select Advanced-Assign Pipe Roughness



Coefficients to Pipes; note that this will assign the accepted pipe roughness to each pipe based on the calibration pipe GroupID.

3.3 PID Control

The MIKE URBAN WD-Tools PID Control module features advanced Real-Time Control (RTC) simulation capabilities for water supply and water distribution systems. It permits description of various controllable devices and makes the definition of complex operational logic for interdependent regulators fully transparent and time efficient. The following controllable devices can be specified:

- Control Valve (PRV-Pressure Reducing Valve, PSV-Pressure Sustaining Valve, PBV-Pressure Breaker Valve, TCV-Throttle Control Valve, FCV-Flow Control Valve).
- Pump.

The devices may be specified as settings or PID-controlled, with control function selection based on a global system analysis. Each regulator or pump operates under the control logic encapsulated into a set of simple logical rules and control functions. The system allows a schematized definition of any form and size of decision tree, featuring logical operators AND, OR, NOT and NOR, in any associate combination.

The process of selecting an appropriate control function under the current operational situations relies on the evaluation of system state conditions including measurable and derived hydraulic and WQ variables (e.g. water level, flow, pollutant concentration, level difference), devices statuses (e.g. valve position, pump settings) and the current control function.

The control functions range from the simplest constants for the operational variables (e.g. constant water level) to dynamic controlled variables, set in a continuous functional relation with any of the measurable variables in the system (e.g. pump START/STOP levels as functions of water level at strategic location in the system).

3.3.1 Architecture of RTC Systems

An RTC system includes monitors/sensors, which generate measurement values characterising states of the system. To be useful for RTC, the measurements must be available with the relatively insignificant time leg (delay). The sensors must be accurate and reliable.

The active control is performed by regulators - controllable movable devices (valves) and pumps. Regulators may take various forms and sizes, and the regulation may be continuous within the functional range, step-wise, or discontinuous (e.g. ON/OFF, OPEN/CLOSED). The regulators may be powered mechanically, hydraulically or pneumatically.



Controllers on the basis of a pre-programmed operational strategy determine the regulator movements (the control actions). The operational strategy may consist of two parts: the control function(s) and, if more control functions are specified, the control logic (rules), responsible for the selection of an appropriate control function. A control function establishes a relation between a control variable and a controlled variable. A controlled variable can be a regulator setting (e.g. valve opening, pump START or STOP level) or some of the flow variables (e.g. water level, flow).

In the latter case, the control decisions are derived by evaluating (comparing) the current value of the controlled flow variable and the pre-defined set-point value. The control algorithm is based on the numerical solution of the "continuous control problem" equation and is usually termed as PID (Proportional-Integral-Differential) control. The actuation signal for the regulator is generated by a PID controller, which usually appears as part of the operational strategy programmed in a Programmable Logical Controller (PLC).

Selection of a controlled variable is, however, subject to limits set by the variable's "controllability". Therefore, a controlled variable is usually selected among the flow variables (flow, water level), preferably in the vicinity of the regulator. As a controlled variable becomes more distant from the regulator, it becomes more difficult to control, due to time lags, diffusion and uncontrollable interference. Control of relatively distant controlled variables is difficult and often cannot give satisfactory results.

When a regulator setting is used as a controlled variable, the control algorithm is reduced to an explicit functional relation between the control variable and the regulator setting, which controls the system response indirectly. This is much simpler than PID control, but in turn, the control results are in many cases inherently inexact and only a rough flow control can be achieved. This type of control is most suitable for regulators of the ON/OFF (or OPEN/CLOSE) type, while the application to continuously controllable regulators should be carefully considered.

If the operational strategy is based on conditions local to the regulated device (for example the ON/OFF-control of a pump based on the water level in a groundwater well) it is called local control. A PLC receives signals (measurements) from local sensors and sends the control decisions (actuation signals) to the regulators.

If the operational logic is based on global conditions, it is then called global control. In such a situation, a global controller is required. A global controller is a computer program that makes the overall system state analysis in real time and provides additional input to the local controllers, which overrides or supplements the local logic with e.g. actuator signals, or by modified set-point values.

Additional component needed is then a data transmission system (UHF radio, leased or dialled telephone lines, GSM, etc.) to transfer data between sensors, controllers and the global controller. In connection with the global con-



troller function, an RTC system is usually equipped with the data management and storage facilities (databases) and the user interface. This is usually termed as SCADA (Supervisory Control And Data Acquisition) system.

Both local and global RTC based on sensor measurements is termed as reactive RTC.

The global control can be extended also to include forecast data in addition to real-time data, which is then called predictive RTC. The most comprehensive way to obtain forecast data is to include a model in the control system. Predictive control brings additional benefits in relatively inert systems, i.e. where the response time of an operational variable is long compared to the change of relevant disturbance (external input or control action).

3.3.2 MIKE URBAN vs. Real Life

MIKE URBAN RTC simulates reactive local and global RTC systems in water supply and water distribution networks. The software implementation is inherently a conceptualisation of real life, of which the user must be fully aware. Some conceptualisations applied in MIKE URBAN RTC are listed below.

- The program does not distinguish explicitly between local and global RTC. Per default, all elements of a modelled RTC system are assumed available for global control.
- Sensors are specified as operational devices with definition of sensor type and position in the MIKE URBAN network. Sensors with multiple functionality must be specified individually.
- When devices (valves and pumps) are specified as controllable in the MIKE URBAN interface, a number of additional physical parameters about the behaviour of the structure is required to describe e.g. the allowed change rates for the state of the structure.
- The actual controllers are not specified explicitly as physical devices, but their function (i.e. operational logic as a combination of operational conditions and control functions) is associated with the respective devices.
- MIKE URBAN RTC uses sampling and actuation (control loop) frequency identical to the simulation time step.
- Sensor readings are simulated as perfectly accurate and with 100% availability.
- Low-level logic of the pump START/STOP operation is built into the program and is controlled by the START and STOP levels.

The PID control algorithm is built into the program and is controlled by the PID constants and by factors for weighting the terms of the numerical solution of the control equation.



Figure 3.10 through Figure 3.12 show examples of how the actual variable (flow or water level) can fluctuate around the set-point as consequence of various choices of the PID constants. Each figure has three different graphs depending on whether the constant is too high, too low or adequate.

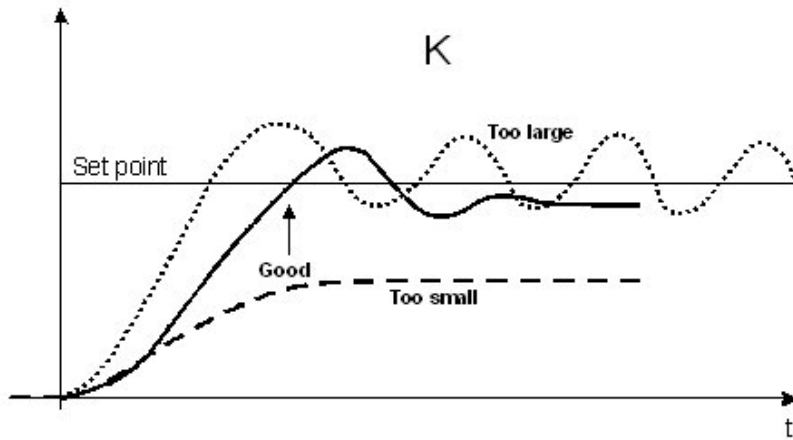


Figure 3.10 Fluctuations around the set point depending on the size of the proportionality factor, K

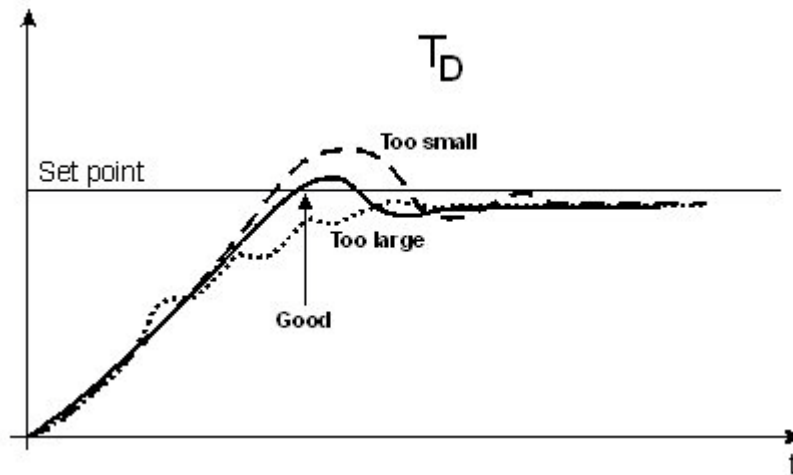


Figure 3.11 Fluctuations around the set point depending on the size of the derivation time, TD

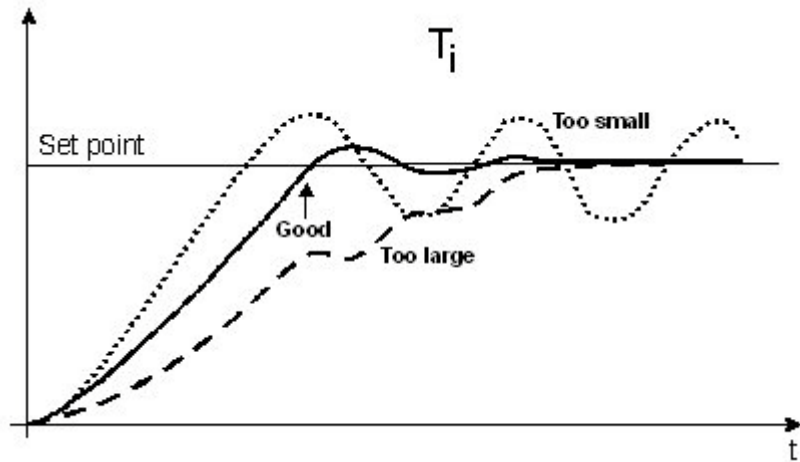


Figure 3.12 Fluctuations around the set point depending on the size of the integration time, T_i

3.3.3 The PID Algorithm

At each simulation time step, the set-point is evaluated against the actual value of the control variable (flow or water level, depending on the set-point type).

The actual value of the control variable is determined from the following equation:

$$u = K \left(e + \frac{1}{T_i} \int_0^{T_i} e \, dt + T_d \frac{de}{dt} \right) \tag{3.1}$$

where:

- u =the output signal i.e. value of the control variable,
- e =the error to be minimised,
- K =the proportionality factor,
- T_d =the derivation time,
- T_i =the integration time.



For numerical solution, a discrete form of this equation is required. By use of the Laplace transformation one obtains:

$$U(s) = K \left(1 + \frac{I}{T_i s} + T_d s \right) E(s) \quad (3.2)$$

where:

- $U(s)$ =the Laplace transform of u ,
- $E(s)$ =the Laplace transform of e .
- By use of the backward Euler transformation:

$$s = \frac{z - 1}{T_s z} \quad (3.3)$$

where T_s is the sampling period, one obtains the following discrete representation of the PID equation:

$$\begin{aligned} u(n) = & K \left(1 + \frac{T_s}{T_i} + \frac{T_d}{T_s} \right) (y_{ref}(n) - y(n)) - \\ & - K \left(1 + \frac{2T_d}{T_s} \right) (y_{ref}(n-1) - y(n-1)) + \\ & + K (y_{ref}(n-2) - y(n-2)) \frac{T_d}{T_s} + \\ & u(n-1) \end{aligned} \quad (3.4)$$

where:

- u =value of the control variable
- K =the factor of proportionality
- T_i =the integration time
- T_d =the derivation time
- T_s =sampling period, i.e. simulation time step
- y_{ref} =the set point for the regulated variable



- $a1..a3$ =factors for weighing each time step (index n , $n-1$ and $n-2$)
- y =actual value of the regulated variable

Indexes n , $n-1$ and $n-2$ denote the current, the previous and the second previous time step, respectively.

The weight factors $a1... a3$ have been added to the PID algorithm in order to give the user more ways of stabilising the algorithm in case of instability problems.

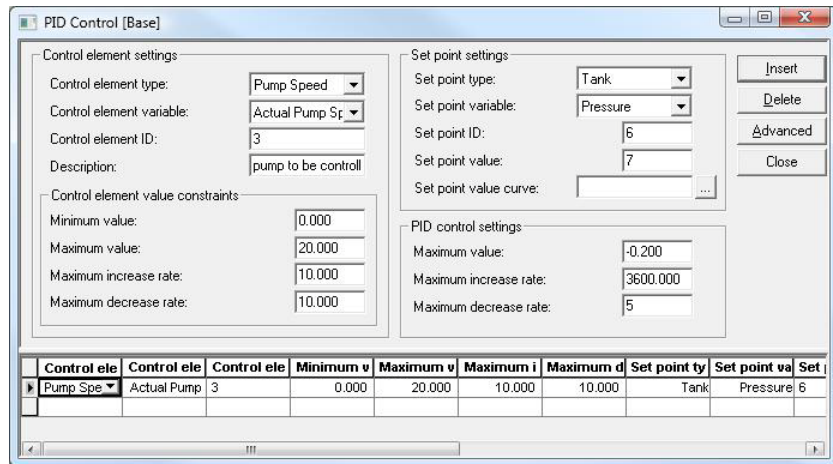


Figure 3.13 The PID Control Editor

The following is the list of parameters that are used to specify the PID based control.

Control Settings

Controlled element settings define the element, such as a pump or a valve which is controlled by the PID algorithm.

- Controlled element type: type of the controlled element, such as pump speed and control valve settings.
- Controlled element variable
- Controlled element ID: ID of the controlled element.
- Description: description of the controlled element (Optional).
- Minimum value: the minimum value constraint.
- Maximum value: the maximum value constraint.
- Maximum increase rate: the maximum rate at which the variable can increase. In units of seconds⁻¹.



- Maximum decrease rate: the maximum rate at which the variable can decrease. In units of seconds⁻¹.

Set Point Settings

Set point settings define the PID algorithm constants.

- Set point type: type of the set point node; such as junction, tank.
- Set point variable
- Set point ID: ID of the set point node.
- Set point value: the constant value.
- Set point value curve: pattern ID defining how the set point value changes in time.

PID Settings

- Maximum value: the proportionality constant of the PID algorithm (KP proportionality constant)
- Max. increase rate: the integration time used in the PID algorithm (TI integration time)
- Max. decrease rate: the derivation time used in the PID algorithm (TD derivation time)

The speed of the valve or pump control depends on the PID control settings. The PID controls and the Control Element Settings must reflect the physical limitations or restrictions of the real-life controlled element and the PID algorithm.

3.4 Demand Adjusted Analysis

Demand Adjusted 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 shutdown and maintenance.

The Wagner equation is used to adjust the node demand based on the available pressure.



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

where:

- Q_{new} = adjusted node demand
- P_{actual} = actual pressure
- $P_{required}$ = required pressure (such as 35 m), node demand is equal to the original demand if the pressure is above the required pressure
- $P_{minimum}$ = minimum pressure (such as 5 m), node demand is 0 if the pressure drops below the minimum pressure
- n = coefficient with recommended values between 1.7 and 2.0

Note that nodes with negative demand i.e. inflow nodes are excluded from this demand adjustment. In order to use Demand Adjusted Analysis, select EPANET | WD-Tools | Demand Adjusted Analysis and define the simulation settings.

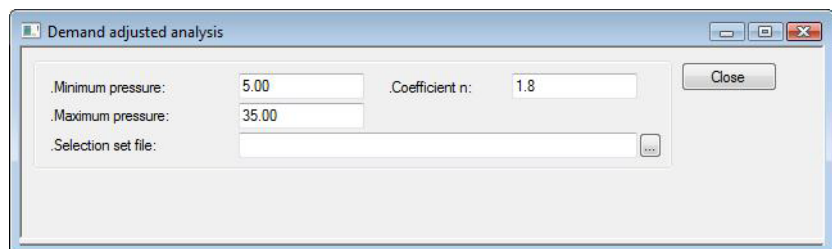


Figure 3.14 The simulation settings for the Demand Adjusted Analysis

A list of the Demand Adjusted Analysis entries with a short description given for each entry:

- Minimum pressure: minimum pressure (such as 5m), node demand is 0 if the pressure drops below the minimum pressure
- Maximum pressure: required pressure (such as 35m), node demand is equal to the original demand if the pressure is above the required pressure
- Selection set file: file with selected nodes (selection file from MIKE URBAN)
- Coefficient n: coefficient "n" for the exponent in Wagner equation ($Exp = 1/n$)



In order to run the Demand Adjusted Analysis, select "Demand Adjusted Analysis" simulation option from Run Simulation dialog.

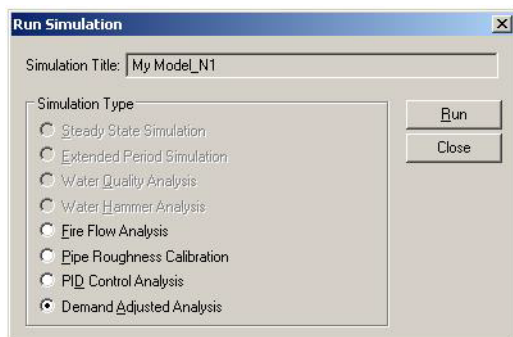


Figure 3.15 Run the Simulation with the Demand Adjusted Analysis choice

The simulation will run for any steady state or extended period model and the simulation results will be presented in the same way as standard results. In case that the node demands were adjusted, simulation report.INF file will provide more details. The INF-file is found in the project directory.

Time(hrs)	NoOfAllNodes	NoOfMin	NoOfMax	NoOfAdj	SumDemOrig	SumDemNew
0	2534	15	1218	853	165.96	130.53
1	2534	15	1227	844	137.84	105
2	2534	15	1231	840	131.61	99.33
3	2534	15	1279	792	133.3	101.8
4	2534	15	1273	798	169.62	135.67
5	2534	14	1240	832	283.47	239.83
6	2534	14	1173	900	472.13	410.94
7	2534	14	959	1114	511.62	423.75
8	2534	14	988	1085	443.29	368.1
9	2534	14	1064	1008	372.21	310.59
10	2534	14	1050	1022	366.78	304.9
11	2534	13	1049	1023	344.79	302.36
12	2534	15	1058	1013	366.12	294.98
13	2534	14	1057	1015	368.2	297.12
14	2534	13	1052	1020	343.68	302.07
15	2534	14	1046	1026	372.3	310.55
16	2534	14	1089	983	372.68	312.83
17	2534	14	1057	1015	416.68	351.97
18	2534	14	1005	1068	512.51	440.24
19	2534	14	918	1155	565.44	470.04
20	2534	15	936	1136	522.69	435.95
21	2534	15	990	1081	494.77	390.68
22	2534	15	1086	985	388.33	300.61
23	2534	15	1194	877	231.82	190.18
24	2534	15	1223	848	165.96	130.84

Figure 3.16 Content of the *.INF file.

Where:

- Time: time level (hrs)
- NoOfAllNodes: number of junction nodes
- NoOfMin: number of nodes where minimum pressure was exceeded



- NoOfMax: number of nodes where maximum pressure was exceeded
- NoOfAdj: number of node where the original node demand was adjusted
- SumDemOrig: original network demand (total for the whole network)
- SumDemNew: new network demand (total for the whole network)

Figure 3.17 illustrates how the original network demand (as defined in the Multiple Demand Editor) was adjusted based on the actual pressures; this graph was created based on the .INF file.

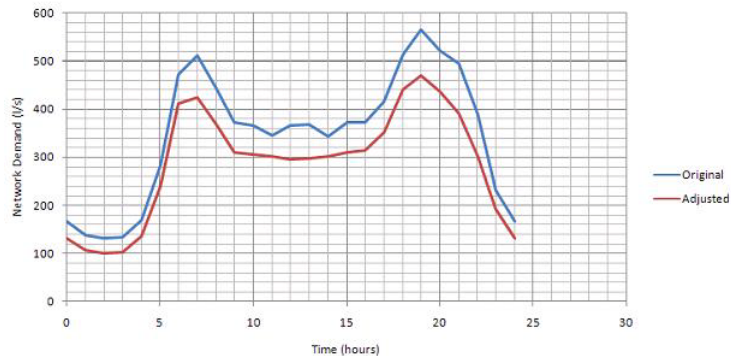


Figure 3.17 Comparison of the original network demand versus adjusted demands during 24 hours

The Demand Adjusted Analysis can be used to model systems with intermittent water supply or systems with low pressure conditions. We can also use this concept in modelling system breakdowns, such as closure of parts of the water distribution systems.

3.4.1 Low Pressure Systems

The example below illustrates a water distribution network with 3 water sources that cannot deliver the required water demands. As a result of that, low pressures (even "negative" in this case) are reported.

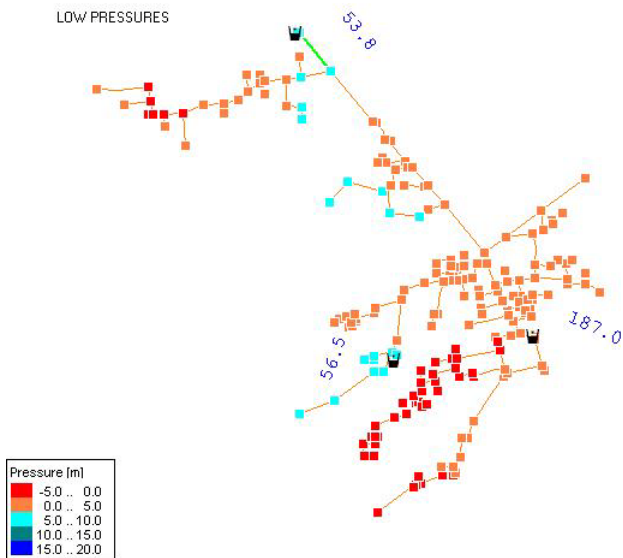


Figure 3.18 Low pressure scenario, total network demand ~ 300 l/s (given demand)

In order to see what is the network capacity, we can run the Demand Adjusted Analysis with the settings of minimum pressure = 5 m, for example. We can see that the network capacity is 83 l/s in order to provide minimum pressure of 5 m, see Figure 3.19.

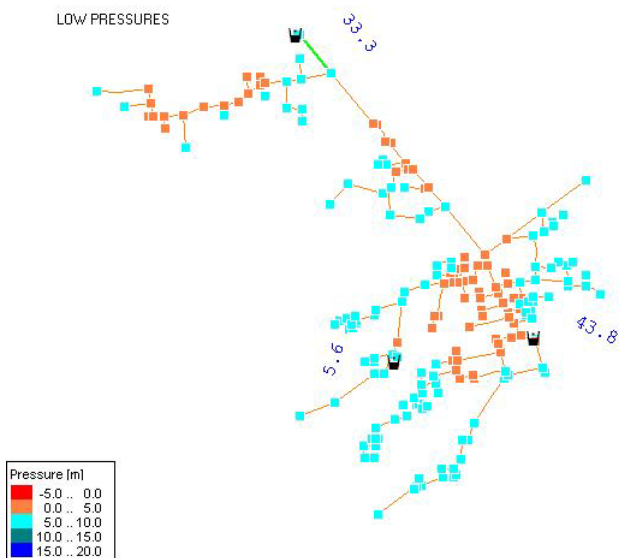


Figure 3.19 Pressure dependent demands, total network demand ~ 83 l/s (given demand)



3.4.2 Intermittent Water Supply Systems or Modelling System Shutdown and Maintenance

It is also possible to use the Demand Adjusted Analysis to analyze intermittent water supply systems or system shutdown and maintenance. Standard demand driven analysis will report negative pressures once a portion of the water distribution system has not water supply while the Demand Adjusted Analysis will automatically adjust the demands and turn them eventually off during non-supply periods.

3.5 Variable Speed Drive Pump

A variable pump can vary its speed setting and/or change its status to open or closed during a simulation. VSD pump extension allows modelling of a pump where the rotational speed of a pump is automatically adjusted by the program in order to maintain the control node pressure at the required level. VSD pump using can also be modelled using the PID control module.

See the section titled *Pump Editor* on -18 for additional information.

3.6 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. In order to allow for modelling of chains of pumps or valves chains in efficient way, the EPANET Rule-based control syntax was extended by adding LIKE keyword, for example.

See the section *Rule Based Control Editor*.

3.7 Pressure Zone Mapping

MIKE URBAN can graphically display different "pressure zones" based on the network topology and geometry, closed pipes, closed valves and pumps. Pressure zone connectivity breakdown rules can be defined. This tool helps to visualise how different network parts are hydraulically interconnected and where the HGL line breaks. In order to generate pressure zones, select EPANET | WD Tools | Pressure Zone Mapping... from the main program menu. The Pressure Zone Mapping wizard will guide you through the settings.

The Pressure Zone Mapping wizard allows you to define the separators, merge small groups of pipes into the same group, and it can automatically generate the colour legend to display the pressure zones.

- Separators: separators are links, which will be used to separate one zone from another.



- Merge zones smaller than: in case that there are many small zones (a typical example would be small pipes located in pumping stations and storage tanks), they will be all merged into the same zone for graphical display instead of creating a separate zone for each of them.

This tool help you to understand the hydraulic behaviour of the network prior to running the hydraulic simulation and it also helps you to detect possible errors in the network connectivity.

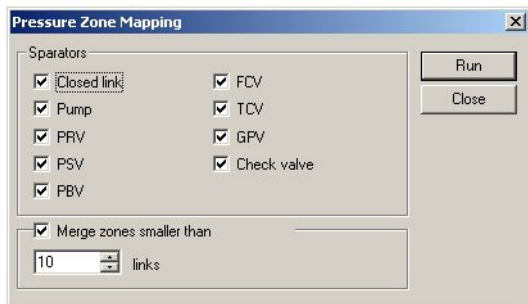


Figure 3.20 Pressure Zone mapping wizard

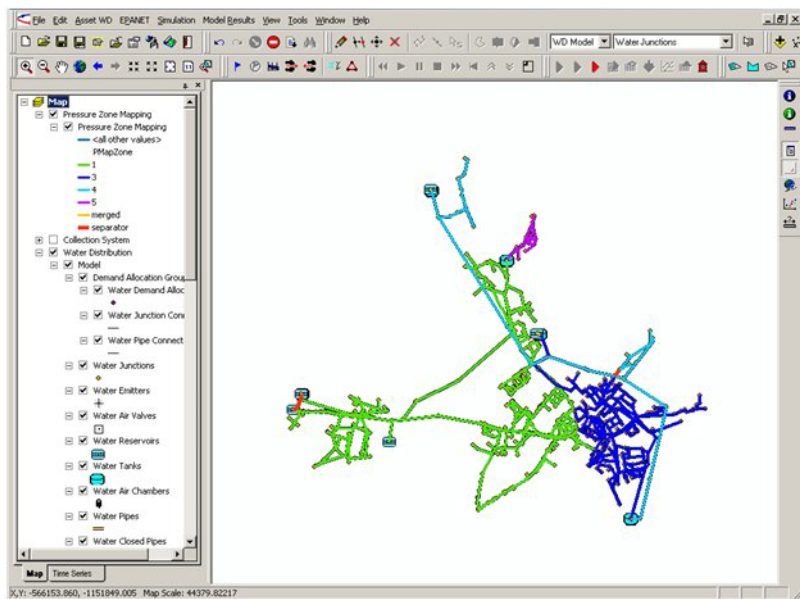


Figure 3.21 Pressure zones are automatically mapped based on the connectivity breakdown rules.



3.8 Advisor

Advisor is a set of tools designed to support the analysis of the water supply and water distribution systems. It helps the user to check the results of the hydraulic simulations, understand system deficiencies, improve design and operation. Advisor is available under EPANET|WD-Tools|Advisor and contains the following tools:

- Pipe criticality
- Sustainability analysis
- Cost analysis

3.8.1 Pipe Criticality

Pipe criticality modeling 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:

1. Water demand criteria (PI-1)
2. Service pressure criteria (PI-2)
3. Pipe flow criteria (PI-3)
4. Pipe length criteria (PI-4)

The combined pipe criticality is computed as an average of all above performance indicators, i.e. $C(\text{pipe } i) = \text{Average}(\text{PI-1} + \text{PI-2} + \text{PI-3} + \text{PI-4})$ (pipe i)

Water demand criteria (PI-1) 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 (PI-2) 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.

Pipe flow criteria (PI-3) is computed as the value of average hour demand $Q(\text{hour})$, the value of 1 corresponds to the total value of average hour demand.

Pipe length criteria (PI-4) 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.



It is recommended to use the maximum peak demand for the above simulations.

In order to use Pipe Criticality, select WD-Tools from the main program menu and then select Advisor | Pipe Criticality. The Pipe Criticality form where you define the simulation settings will then display.

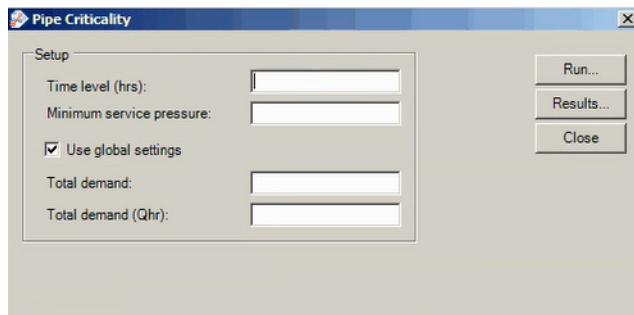


Figure 3.22 Pipe criticality setting (Global settings)

The interface allows you to define the following parameters:

- Time level: in case of Extended Period Simulation, this is the time level at which the pipe criticality analysis will be performed
- Minimum service pressure: minimum acceptable service pressure within the network

The pipe reliability settings can be defined on the level of the whole model area or per each specific zone. In case of Global settings:

- Total demand: network demand
- Total demand (Qhr): maximum hour network demand



Pipe Criticality

Setup

Time level (hrs):

Minimum service pressure:

Use global settings

Total demand:

Total demand (Qhr):

Run...
Results...
Close

Select all

Zone ID	Description	Q	Q(hr)	Selected
Airline	Airline zone	280	392	<input checked="" type="checkbox"/>
Portsmouth	Portsmouth zone	2000	2800	<input checked="" type="checkbox"/>
Northwest	Northwest zone	10700	14980	<input type="checkbox"/>
South_Norfolk	South Norfolk zone	2600	3640	<input type="checkbox"/>
IndianRiver	Indian River zone	870	1218	<input type="checkbox"/>
Cavalier	Cavalier zone	980	1372	<input type="checkbox"/>

Figure 3.23 Pipe criticality setting (Local settings)

In case of the local settings, the network demands are defined per each zone, please note that the zone demands are defined in the Pressure Zone Editor.

- Q: network demand
- Qhr: maximum hour network demand
- Selected: use this switch to active or deactivate zones; only selected zones will be used for the pipe criticality analysis.

In order to run the pipe criticality analysis or to load the results of the previous pipe criticality results, use the following controls

- Run: run pipe criticality analysis
- Results: load pipe criticality results and store them into the database



ID *	PipeID *	ZoneID *	Q	P1	SumNodes	P2	Qhr	P3	SumLength	P4	C
38527	447	Airline	2.94	0.01	2	0.02	2.94	0.01	8003.08	0.19	0.06
38610	467	Airline	27.83	0.1	0	0	27.83	0.07	7336.56	0.18	0.09
36687	1516	Airline	3.58	0.01	0	0	3.58	0.01	7336.56	0.18	0.05
38604	466	Airline	28.43	0.1	0	0	28.43	0.07	7336.56	0.18	0.09
36686	1515	Airline	5.09	0.02	0	0	5.09	0.01	7336.56	0.18	0.05
40722	8931	Airline	214.06	0.76	52	0.53	214.06	0.55	40561.33	0.98	0.71
38599	465	Airline	32.11	0.11	0	0	32.11	0.08	7336.56	0.18	0.09
38592	464	Airline	45.19	0.16	0	0	45.19	0.12	7336.56	0.18	0.11
38541	452	Airline	103.7	0.37	46	0.47	103.7	0.26	36989.48	0.9	0.5
41110	9583	Airline	0.02	0	0	0	0.02	0	7430.96	0.18	0.05
38537	451	Airline	54.11	0.19	3	0.03	54.11	0.14	9598.44	0.23	0.15
38534	450	Airline	8.58	0.03	1	0.01	8.58	0.02	8263.35	0.2	0.07
40762	8974	Airline	7.34	0.03	2	0.02	7.34	0.02	8817.04	0.21	0.07
36336	11393	Airline	0.45	0	1	0.01	0.45	0	7638.13	0.19	0.05
40763	8975	Airline	2.76	0.01	0	0	2.76	0.01	7660.46	0.19	0.05
38768	491	Airline	13.17	0.05	8	0.08	13.17	0.03	17122.23	0.42	0.14
38570	459	Airline	65.55	0.23	0	0	65.55	0.17	7336.56	0.18	0.14
38568	458	Airline	6.17	0.02	0	0	6.17	0.02	7336.56	0.18	0.05

Figure 3.24 Pipe criticality results, table view



Figure 3.25 Pipe criticality results, Map view (pipes are colour coded by the pipe criticality C)

3.8.2 Sustainability Analysis

Sustainability analysis helps you to understand the simulation results and analyze them for possible problems, anomalies, critical areas, and similar. Various predefined thematic maps are available including:

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

Detailed HTML reports can be generated to understand operation of pumps, storage tanks, and other facilities.

In order to use Sustainability Analysis, select WD-Tools from the main program menu and then select Advisor | Sustainability Analysis. Sustainability Analysis dialog where you define the simulation settings will display.



Please note, that the simulation results needs to be loaded in order to start the sustainability analysis.

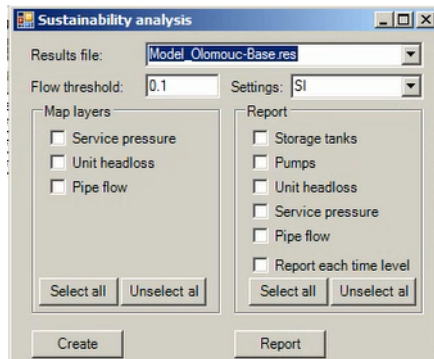


Figure 3.26 Sustainability Analysis dialog

You can define the following settings:

- Results file: name of the results file
- Flow threshold: minimum flow criteria used for reverse flow calculation
- Settings: unit selection
- Map layers: select what layers will be added to the Map
 - Service pressure
 - Unit headloss
 - Pipe flow
- Report: select what sections will be used for reporting
 - Storage tanks
 - Pumps
 - Unit headloss
 - Service pressure
 - Pipe flow
 - Report each time level: please note that selecting each time level can result in excessive processing time
- Create: perform the sustainability analysis
- Report: create a report

The program will provide a detailed analysis of the simulation results and it will create the following layers and reports:

- Service pressures
 - Pressure anomalies e.g. pressure too low or to high

- Pressure range: pressure distribution
- Pressure fluctuation: difference between the minimum and maximum pressure at every node during the simulation.
- Unit headloss: head loss per 1000
- Pipe flows
- Reverse flow: The layer will show how many times the flow direction has changed in every pipe. Note that a threshold value needs to be specified e.g. "0.1" meaning that if the absolute flow is smaller than that the pipe is not considered for reporting i.e. if the flow must be smaller than -0.1 and greater than 0.1 to be considered as a pipe with flow.
- Flow velocity: velocity distribution
- Flow velocity fluctuation

In addition to the Map layers, the HTML report creates a detailed report for the whole network as well as for the particular zone.

- Storage tanks: reports tanks that are either drained or overflows during the simulation, reports if the tanks are balanced within the simulation. Balanced tank is a tank where the water level at the beginning of the simulation is the same as at the end of the simulation.
- Pumps: reports pump that are operated near their maximum capacity
- Unit Headloss: reports pipes with too high unit head loss
- Service Pressures: reports excessive pressures
- Pipe flows: reports reverse flows
- Flow velocity: reports excessive flow velocity

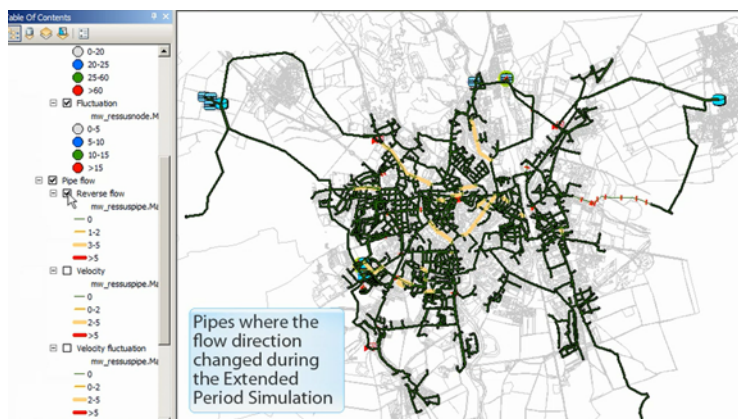


Figure 3.27 Sustainability analysis, Map view of reverse flows

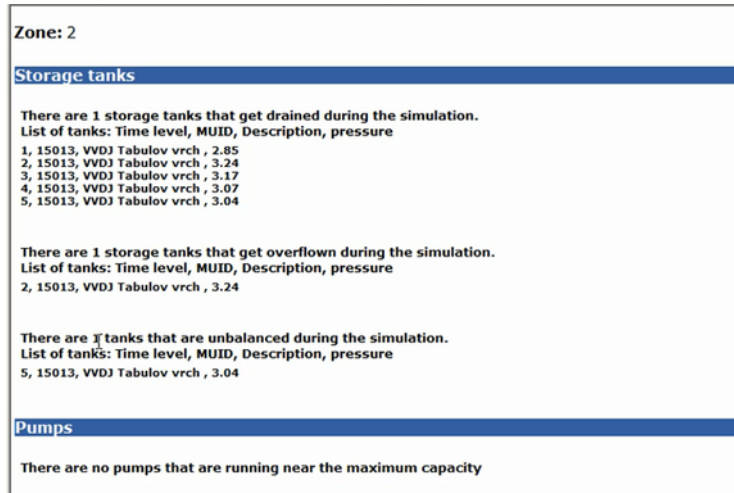


Figure 3.28 Sustainability analysis, report

3.8.3 Cost Analysis

Cost analysis allows you to review the energy consumption results on more details, create tabular outputs, and great graphs of pump utilization, average power consumption, and costs.

In order to use the Cost Analysis, load the simulation results and select EPANET | WD Tools | Advisor | Cost Analysis...

Pump Energy Table

The pump energy table is displayed on the first tab of Cost Analysis. It reports pump utilization, average efficiency, average power consumption, peak power consumption, and cost per day.

Pump ID	Utilization	Avg. Effic.	Avg. kW Per	Avg. kW	Peak kW	Cost/day
3842	16,497	75	0,137	95,229	95,648	50,616
3844	100	75	0,266	154,225	156,569	555,211
6981	24,546	75	0,277	149,45	149,559	132,064
Total Cost						737,8907318115...
Demand charge						211,3196105957...

Figure 3.29 Pump energy table



Pump Energy Graphs

Pump energy including pump utilization, power consumption, and costs can be displayed for every pump in the model. Such graphs help to analyze pump energy and evaluate or compare pumps.

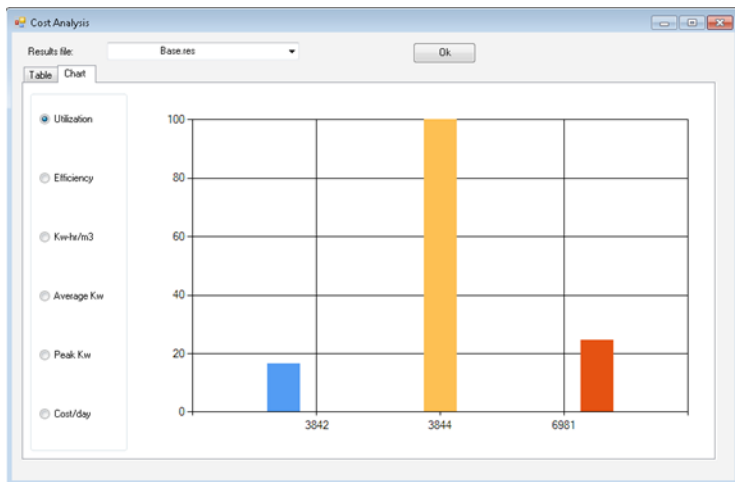


Figure 3.30 Pump energy graphs

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 of 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)

Note that the total cost and demand charge (if any) are listed at the bottom of the table.

To define energy price, price pattern, pump efficiency use EPANET | Energy Editor and EPANET | Pump Editor where you can define the following fields:

- PRICE = average cost per kW-hour,
- PATTERN = ID label of time pattern describing how energy price varies with time,
- EFFIC = either a single percent efficiency for global setting or the ID label of an efficiency curve for a specific pump,



- DEMAND CHARGE = added cost per maximum kW usage during the simulation period.

3.8.4 Alarms and Violations

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

In order to use the Alarms and Violations, select EPANET | WD Tools | Alarms and Violations ...

Alarms and Violations dialog

The Alarms and Violations dialog is used to define critical values and validate simulation results.

OBJECTID	Element Type	Element MUID	Description	Variable No	Criteria No	Alarm Value	Actual Value	Alarm At Time	Status No	Status Message	Comment
2	Tank	M-ZONE		Node: Pressu	<	2			OK	Validated at 2	Level too low
3	Tank	M-ZONE		Node: Pressu	>	5	5,4034	05:20:00	Failed	Validated at 2	Level too high
4	Junction	WEST-ZONE		Node: Pressu	<	30			OK	Validated at 2	Pressure too low
5	Junction	EAST-ZONE		Node: Pressu	<	30			OK	Validated at 2	Pressure too low
15	Pipe	73943		Link: Velocity	<	0,25	0,0016	00:30:00	Failed	Validated at 2	Velocity too low
19	Pump	2010_PUMP3		Link: Flow	<	30	0,0187	02:40:00	Failed	Validated at 2	Pump flow too low
20	Pump	2010_PUMP3		Link: Flow	>	60			OK	Validated at 2	Pump flir too high

Figure 3.31 Alarms and Violations table

The dialog table allows you to define or display the following parameters:

- Element Type: select from the following types:
 - Junction
 - Tank
 - Pipe
 - Pump
 - Valve
- Element MUID: enter the MUID of the junction node, pipe, pump, valve, or a tank
- Description: enter user defined description (optional)
- Variable: select the result item from the following types:
 - Node: Demand
 - Node: Head
 - Node: Pressure



- Link: Flow
 - Link: Absolute Flow
 - Node: Water Quality
 - Link: Velocity
 - Link: Unit Headloss
 - Link: Water Quality
 - Link: Status Code
 - Link: Setting
 - Link: Reaction Rate
 - Link: Friction Factor
 - Link: Start End Head
- Criteria: select the value criteria from the following types:
 - <
 - <=
 - =
 - >
 - >=
 - <>
 - Alarm Value: enter the critical (alarm) value
 - Actual Value: this is the highest or lowest of the actual values resulting in the alarm
 - Alarm at Time: this is the time of the simulation corresponding to the “Actual value”
 - Status: this is the status of the results validation, “OK” or “Failed”
 - Status Message: this is the message from the results validation
 - Comment: user defined comment

Note, that the alarm will be triggered if the criteria is fulfilled i.e. if the criteria is defined for a “tank level < 2” then the alarm will be reported once the computed level is below “2”.

Similarly, if you want to alarm for a pump operation during which the pump flow exceeds 300 l/s, enter the following criteria “pump flow > 300”.

The dialog interface allows you to use the following commands:

- Insert: add a new record to the table
- Delete: delete the highlighted record from the table
- Show: highlight nodes and links used in alarms in the Map
- Series: displays time series for the selected alarms (rows)
- Clipboard: copy the content of the table into a clipboard



- Validate: perform the results validation for the selected results file
- Clear: clear the results of the last validation
- On Top: forces the dialog to remain on top of other windows
- Close: close the dialog window





4 MIKE URBAN Water Hammer

Water Hammer (a part of the WD-Tools module) simulates transient (unsteady) flow in any fully pressurized system carrying liquids. MIKE URBAN 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 URBAN Steady State Pipe Flow Module.

Water Hammer allows you to model:

- Pump start-up and pump trip-off.
- Water hammer protection of pumping stations.
- Power failure events.

4.1 Water Hammer Calculation

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

$$\frac{\partial Q}{\partial x} + \frac{gA}{a^2} \frac{\partial H}{\partial t} = 0 \quad (4.1)$$

and the equation of motion:

$$\frac{\partial Q}{\partial t} + gA \frac{\partial H}{\partial x} + \frac{f}{2DA} Q|Q| = 0 \quad (4.2)$$

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

$$a = \sqrt{\frac{K}{\rho [1 + (K/E) \psi]}} \quad (4.3)$$



in which E is the Young's modulus of elasticity of the conduit walls, K - the bulk modulus of the fluid, ρ - 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.

4.2 Theoretical Background

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

4.2.1 Description of Water Hammer Model

The water hammer computation is based on the Continuity equation

$$\frac{\partial Q}{\partial x} + \frac{g A}{a^2} \frac{\partial H}{\partial t} = 0 \quad (4.4)$$

and the Momentum equation

$$\frac{\partial Q}{\partial t} + g A \frac{\partial H}{\partial x} + \frac{f}{2DA} Q |Q| = 0 \quad (4.5)$$

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

$$a = \sqrt{\frac{K}{\rho [1 + (K/E) \psi]}} \quad (4.6)$$



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

$$\psi = 0 \quad (4.7)$$

Thick-Walled Elastic Conduit ($D/e \leq 10$)

- anchoring at both ends = full restraint

$$\psi = 2 (1 + \nu) \left(\frac{R_o^2 + R_i^2}{R_o^2 - R_i^2} - \frac{2\nu R_i^2}{R_o^2 - R_i^2} \right) \quad (4.8)$$

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

- upstream anchoring = upper restraint

$$\psi = 2 \left(\frac{R_o^2 + 1.5R_i^2}{R_o^2 - R_i^2} + \frac{\nu(R_o^2 - 3R_i^2)}{R_o^2 - R_i^2} \right) \quad (4.9)$$

- frequent expansion joints = expansion joints

$$\psi = 2 \left(\frac{R_o^2 + R_i^2}{R_o^2 - R_i^2} + \nu \right) \quad (4.10)$$

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

- anchoring at both ends = full restraint

$$\psi = \frac{D}{e} (1 - \nu^2) \quad (4.11)$$

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



- upstream anchoring = upper restraint

$$\psi = \frac{D}{e}(1 - 0.5 \nu) \quad (4.12)$$

- frequent expansion joints = expansion joints

$$\psi = \frac{D}{e} \quad (4.13)$$

Tunnels Through Solid Rock, Parmakian 1963

- Unlined tunnel

$$\psi = 1 \quad E = G \quad (4.14)$$

where G is the modulus of rigidity of the rock.

- Steel - lined tunnel

$$\psi = \frac{DE}{GD + Ee} \quad (4.15)$$

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.

$$e_e = E_r e_c + \frac{A_s}{L_s} \quad (4.16)$$

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.

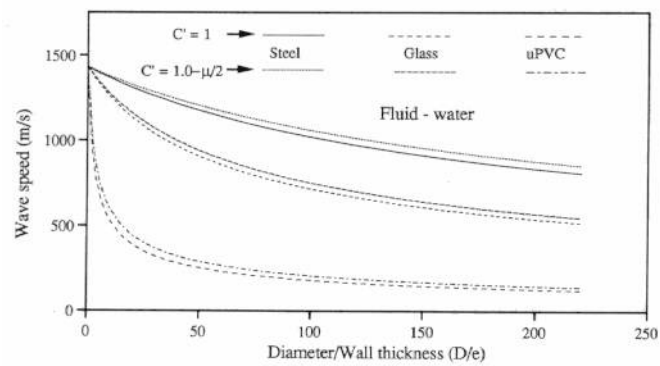


Figure 4.1 Fluid water

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

Table 4.1 Values of Young's Modulus of Elasticity and Poisson's Ratio for a range of common materials

Material	Young's Modulus (10E9 N/m ²)	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
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:

- $K = 2.05 \times 10^9 \text{ N/m}^2$ for water
- $K = 1.62 \times 10^9 \text{ N/m}^2$ for oil.

4.3 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

$$\frac{f}{2DA} |Q| Q \approx \frac{1}{2} \frac{f}{2DA} \left(|Q_{j-1}^n| Q_{j-1}^{n+1} + |Q_j^n| Q_j^{n+1} \right) \quad (4.17)$$

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



4.3.1 Coefficients for the Numerical Scheme

$$\alpha = \frac{gA}{a^2} \quad (4.18)$$

$$fric = \frac{\lambda}{2AD} \quad (4.19)$$

$$Cr = \frac{a \Delta t}{\Delta x} \quad (4.20)$$

$$\alpha_c = \left(6\psi^2 - 6\psi + 1\right) + Cr^2 \left(6\theta - 6\theta^2 - 1\right) \frac{\Delta x^2}{3a^2 \Delta t^2} \quad (4.21)$$

$$\alpha_m = \left(6\psi^2 - 6\psi + 1\right) + Cr^2 \left(6\theta - 6\theta^2 - 1\right) \frac{gA \Delta x^2}{3a^2 \Delta t^2} \quad (4.22)$$

$$A1 = \alpha (1 - \psi) \Delta x \quad (4.23)$$

$$B1 = -\theta \Delta t + \alpha_c \quad (4.24)$$

$$C1 = \alpha \psi \Delta x \quad (4.25)$$

$$D1 = \theta \Delta t - \alpha_c \quad (4.26)$$

$$E1 = -(1 - \theta) \Delta t (Q_{j-1}^n + Q_{j-1}^{n-1} - Q_j^n - Q_j^{n-1}) + \theta \Delta t (Q_{j-1}^n - Q_j^n) + \alpha_c (Q_j^{n-1} - 2Q_j^n - Q_{j-1}^{n-1} + 2Q_{j-1}^n) + \alpha_c (1 - \psi) \Delta x H_{j-1}^{n-1} + \alpha_c \psi \Delta x H_j^{n-1} \quad (4.27)$$

$$A2 = -g a \theta \Delta t + \alpha_m \quad (4.28)$$



$$B2 = (1 - \psi) \Delta x + \text{fric} \Delta t \Delta x |Q_{j-1}^n| \quad (4.29)$$

$$C2 = g a \theta \Delta t - \alpha_m \quad (4.30)$$

$$D2 = \psi \Delta x + \text{fric} \Delta t \Delta x |Q_j^n| \quad (4.31)$$

$$E2 = (1 - \psi) \Delta x Q_{j-1}^{n-1} + \psi \Delta x Q_j^{n-1} + gA(1 - \theta) \Delta t (H_{j-1}^n - H_j^n + H_{j-1}^{n-1} - H_j^{n-1}) + gA\theta \Delta t (H_{j-1}^n - H_j^n) + \alpha_m (H_j^{n-1} - 2H_j^n - H_{j-1}^{n-1} + 2H_{j-1}^n)$$

4.3.2 Looped Network Solution Algorithm

The main algorithm generates a set of grid points using a finite difference scheme, see Cunge, Holly, Verwey (1980). The grid is introduced in time and space, where at every point the values of H and Q are defined as the unknown variables. Between the two successive grid points in time and space both the continuity and the momentum equation are applied. Together with the necessary boundary data, a sufficient number of equations are obtained to solve H and Q at every grid point.

The general form of the governing equations is

$$A1_j H_{j-1}^{n+1} + B1_j Q_{j-1}^{n+1} + C1_j H_j^{n+1} + D1_j Q_j^{n+1} = E1_j \quad (4.33)$$

$$A2_j H_{j-1}^{n+1} + B2_j Q_{j-1}^{n+1} + C2_j H_j^{n+1} + D2_j Q_j^{n+1} = E2_j \quad (4.34)$$

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 4.2. For any computational grid point, equations (4.35), (4.36) may be written as

$$H_i = L1_i H_l + M1_i H_{jj} + N1_i \tag{4.35}$$

$$Q_i = L2_i H_l + M2_i H_{jj} + N2_i \tag{4.36}$$

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

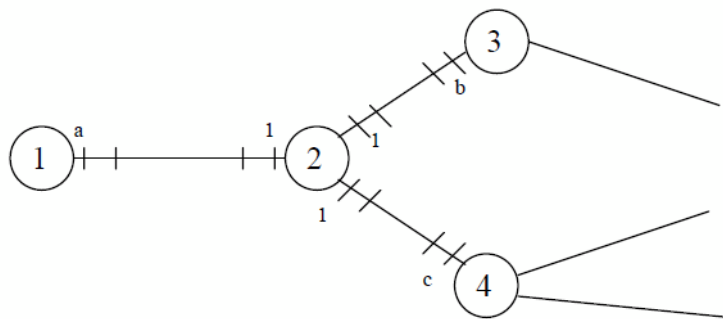


Figure 4.2 Part of a looped pipe network

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.

$$\sum_{k=1}^m Q_{lk}^{n+1} = 0 \tag{4.37}$$

$$h_{l1}^{n+1} = h_{l2}^{n+1} = K = h_{lk}^{n+1} = K = h_{lm}^{n+1} \tag{4.38}$$

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.

$$[S] \{h\} = \{T_L\} \quad (4.39)$$

where $[S]$ is a coefficient's matrix, $M \times M$ 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 (4.37) and (4.38).

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.

4.3.3 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.

4.4 Water Hammer Calculations

Water hammer simulates transient (unsteady) flow in any fully pressurized system carrying liquids. MIKE URBAN 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 URBAN Water Distribution Steady State Pipe Flow Module.

Water Hammer allows you to model:

- Pump start-up and pump trip-off.
- Water hammer protection of pumping stations.
- Power failure events.

Water Hammer Data Preparation

allows you to create all the input files interactively and save them for computation. 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, graphs 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. The presentation includes the ability to save screen displays into Windows AVI animation files or send screen displays to printers. It is also possible to generate hydraulic grade lines and pressures.



4.4.1 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 Project Options dialog, define transient boundary conditions and computational parameters. In addition to the specific pipe data, you need to know the wave speed in each pipe.

Initial Conditions

Initial conditions are computed with the use of the Steady State Model. 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.

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 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

$$Cr = \frac{a\Delta t}{\Delta x} \tag{4.40}$$



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 URBAN 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 URBAN 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).

4.4.2 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.

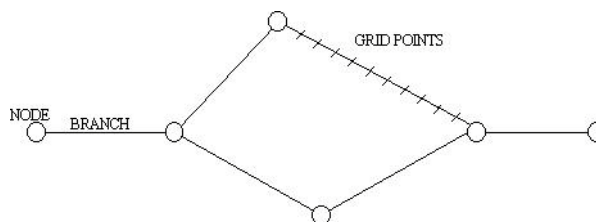


Figure 4.3 Definition Network Layout



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 4.2 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 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 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.

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 used 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.

Control Rules

Simple Control Rules and Rule Based Controls are not allowed during the water hammer analysis. Valve opening and pump scheduling is handled directly by the specific valve and pump data.

Specific Pump Data

Input of pipes is the same as in the case of steady state analysis. Then you have to specify rated rotational pump speed and its schedule - time series of the rotational pump speed versus time.

Pumps may be located inside pipeline systems (booster pumps) or they may be connected to a suction well. Pumps are frequently used for various pipeline systems, and may operate during hydraulic transients with constant pump speed. Alternatively, the pump speed can decrease and/or increase depending on pump shut-down and/or pump start-up. The greatest difficulties



come from hydraulic transient flows caused by turbopumps, since they may work in four quadrants. Four quadrants pumps are currently not supported.

There are in principle four dependent variables describing any state of a pump, namely:

- discharge - Q (m³/s)
- total dynamic head (tdh) - H (m)
- rotational speed - N (rpm)
- shaft torque - T (N.m)

The total dynamic head is defined as follows:

$$tdH = H = \left(\frac{V_d^2}{2g} + \frac{p_d}{\rho g} + z_d \right) - \left(\frac{V_s^2}{2g} + \frac{p_s}{\rho g} + z_s \right) \quad (4.41)$$

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

$$P = \frac{\rho g Q H}{\eta} = T \omega = \frac{T 2 \pi N}{60} \quad (4.42)$$

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 Fig. 3.2. 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:

$$H = b - aQ^2$$

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 GGM method and in all computation modules if the pump discharge is out of the $Q-H$ curve

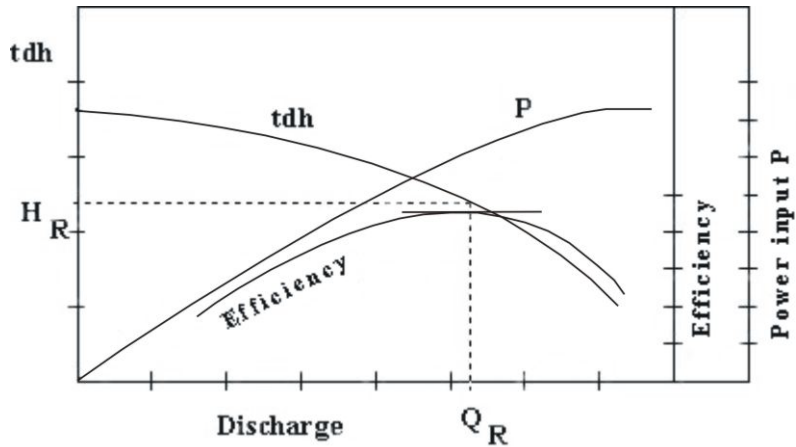


Figure 4.4 Q - H Curve

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 N_1, N_2

$$\frac{Q_1}{Q_2} = \frac{N_1}{N_2} \frac{H_1}{H_2} = \left(\frac{N_1}{N_2} \right)^2 \frac{P_1}{P_2} = \left(\frac{N_1}{N_2} \right)^3 \quad (4.43)$$

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

$$P = \frac{\rho g Q H}{\eta} = T \omega = \frac{T 2 \pi N}{60} \quad (4.44)$$

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

$$\Delta N = \frac{(T_m - T) \Delta t 30}{I \omega \pi} \quad (4.45)$$

where, T_m (N.m) is the motor torque.

The relation between the pump speed and the total pump dynamic head is described by the following equation:

$$tdH_t = \frac{tdH_{100\%}}{N_{100\%}^2} N_t^2 \quad (4.46)$$

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, r_i , from the axis of rotation, the moment of inertia of the body is given by:

$$I \omega \sum_{i=1}^n m r_j \quad (4.47)$$



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.

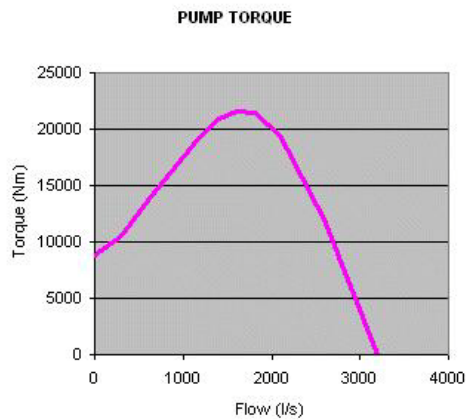


Figure 4.5 Pump torque curve

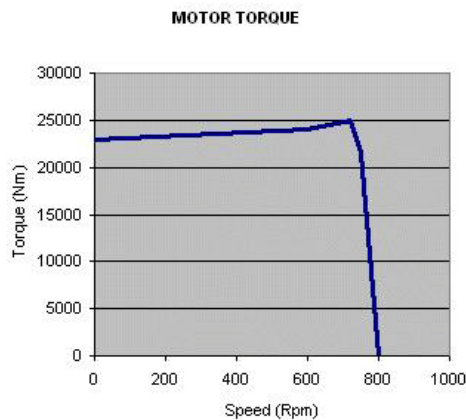


Figure 4.6 Motor torque curve



Pump

A Pump can operate in 3 quadrant with constant and/or variable rotational speed. Parameters which describe a Normal Pump are as follows:

[ID]

Pump ID

[N1]

Starting node ID

[N2]

Ending node ID

[rpm], [N]

Rated speed in r.p.m

[flow direction]

Direction of pumping, from beginning node to ending node, from ending node to beginning node

[operation type]

Define pump operation. Each pump is operated by the predefined operation schedule (always required) and pump start-up or trip-off can be modelled by specifying the time level in seconds.

[operational schedule]

N-time curve

[pump characteristics for 100% pump speed]

Q-H curve in 3 quadrants.

[pump characteristics for zero pump speed]

Q-H curve in 3 quadrants.

[Iw]

combined moment of inertia

[pump torque]

T(pump)-Q curve



[motor torque]

T(motor)-N curve

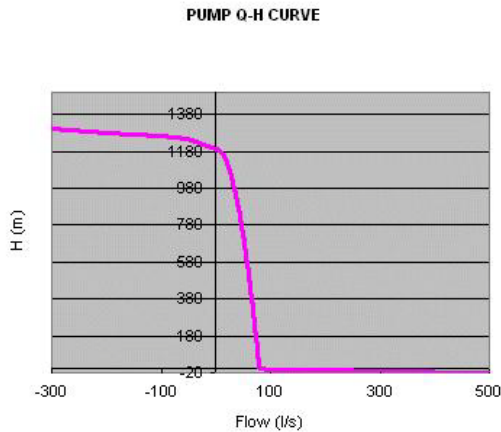


Figure 4.7 Pump Q-H curve defined in 3 quadrants for 100% rated speed

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

$$Q = Cd Av \sqrt{2g\Delta H} \quad Cd = \frac{1}{\sqrt{\xi}} \tag{4.48}$$

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

Free-discharge valve

$$Q = Cd Av \sqrt{2g\Delta H} \quad Cd = \frac{1}{\sqrt{\xi + 1}} \tag{4.49}$$

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

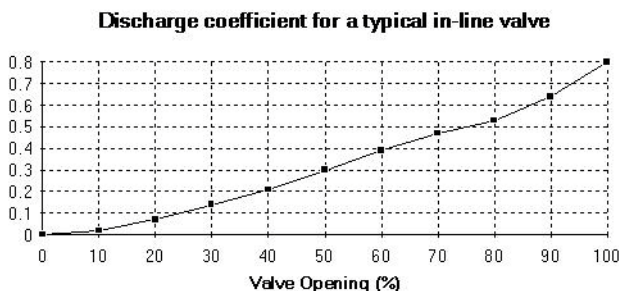


Figure 4.8 Discharge coefficient for a typical in-line valve

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.

Parameters describing Control Valves are as follows:

[ID]

Valve ID

[N1]

Starting node ID

[N2]

Ending node ID

[Dd]

Valve diameter

[Valve characteristic]

discharge coefficient C_d as a function of the relative valve opening A [%] is represented by table values [A- C_d table]. C_d units (-)

or,

discharge coefficient K_v as a function of the relative valve opening A [%] is represented by table values [A- K_v table]. K_v coefficient (m^3/hod) represents discharge (m^3) though the valve corresponding to the specific valve opening (%) under the pressure of 1 bar.



[Valve schedule]

tabulated values of the relative valve opening *A* during the opening and/or closing procedure

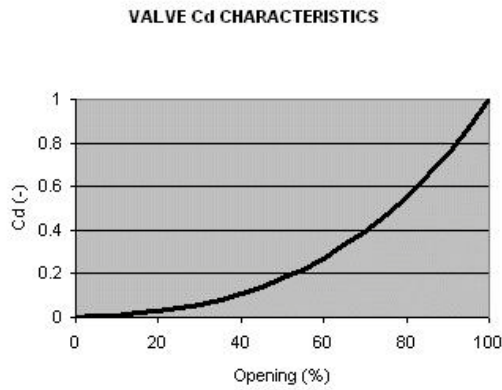


Figure 4.9 Valve Cd Characteristics

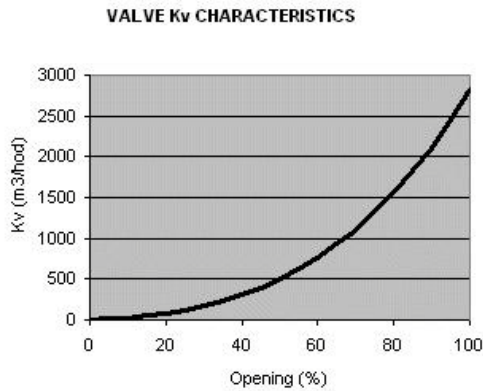


Figure 4.10 Valve Kv Characteristics (Example)

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

$$Cd = \frac{Kv}{3600A\sqrt{2g}} \tag{4.50}$$



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

$$Cd = \frac{1}{\sqrt{\xi - 1}} \quad (4.51)$$

or, for an in-line valve:

$$Cd = \frac{1}{\sqrt{\xi}} \quad (4.52)$$

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.

$$Cr = 1 = a \frac{\Delta t}{\Delta tx} \quad (4.53)$$

in which a - wave speed, dt - time step, dx - grid step, Cr - Courant number, a non dimensional parameter.



Specific Curves Data

Curve editor is extended by several new curve types, which are used in the fast transient analysis. The following new curve types are available in the Water Hammer simulations:

Table 4.3 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
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



List of Supported Components

Table 4.4 List of supported components

Component	Remark
Tank	Available
Pump	Available
Pressure reducing valve PRV	Not available
Pressure sustaining valve PSV	Not available
Pressure breaker valve PBV	Not available
Flow control valve FCV	Not available
Throttle control valve TCV	Available
Closed pipes	Available
Pipes with check valves CV	Available
Node demands	Multiple junction demands including their patterns are kept constant during water hammer analysis.

List of Unsupported Component

The following components are not supported by the Water Hammer simulations.

Table 4.5 List of unsupported components

Component	Remark
General purpose valve GPV	Not available
Emitter	Not available
Simple control rules	Not available
Rule base controls	Not available
Patterns	Demand and Reservoir patterns need to be entered as Transient Boundary Conditions

4.4.3 List of Additional Components

Several additional network components are used in Water Hammer simulations comparing to EPANET based simulation. These components (struc-



tures) are classified according to their location either in nodes or on branches.

Table 4.6 List of new components

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

4.4.4 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 4.11. illustrates an Air Chamber with a vertical cylindrical tank.

The hydraulic behavior of an Air Chamber is described by the relation between air pressure, its volume and continuity equation. It is assumed that the enclosed air follows the polytropic relation for a perfect gas

$$C = H_{air} * \forall_{air}^k \quad (4.54)$$

in which H_{air} and \forall_{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). The orifice losses are different for the inflow and outflow from the chamber.

Parameters:

- Node ID. Air chamber can be located only in the "dead-end" node, i.e. node with only one adjacent pipe.
- 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.
- Kappa: $k =$ the exponent in the polytropic gas equation (default value $k=1.2$).

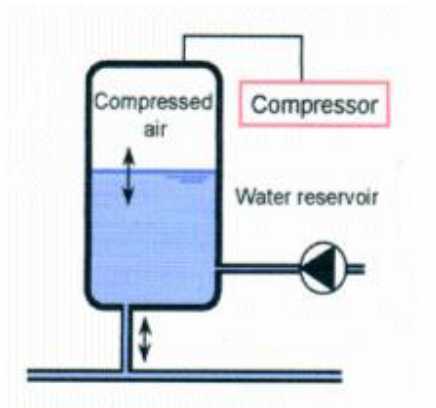


Figure 4.11 Air chamber

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

$$C = H_{air} * \forall_{air}^k \tag{4.55}$$

in which H_{air} and \forall_{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, p_{air} is the air pressure inside a chamber for which the Q_{air} (air discharge) is taken from the dual-acting valve characteristics. Next chart shows characteristics of Pont&Mousson, Ventex dual-acting valve, diameter of 150mm.

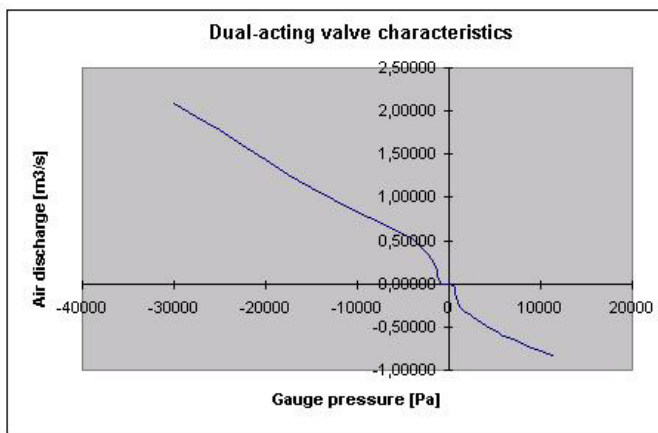


Figure 4.12 Dual-acting Valve Characteristics

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.

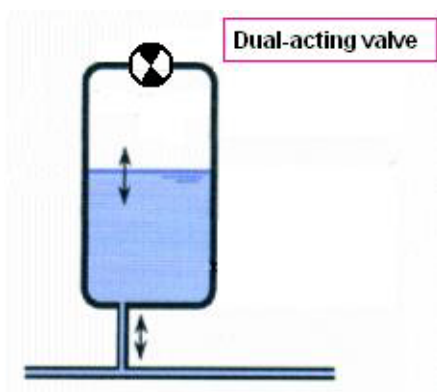


Figure 4.13 Vented Air-Chamber



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.

Parameters:

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

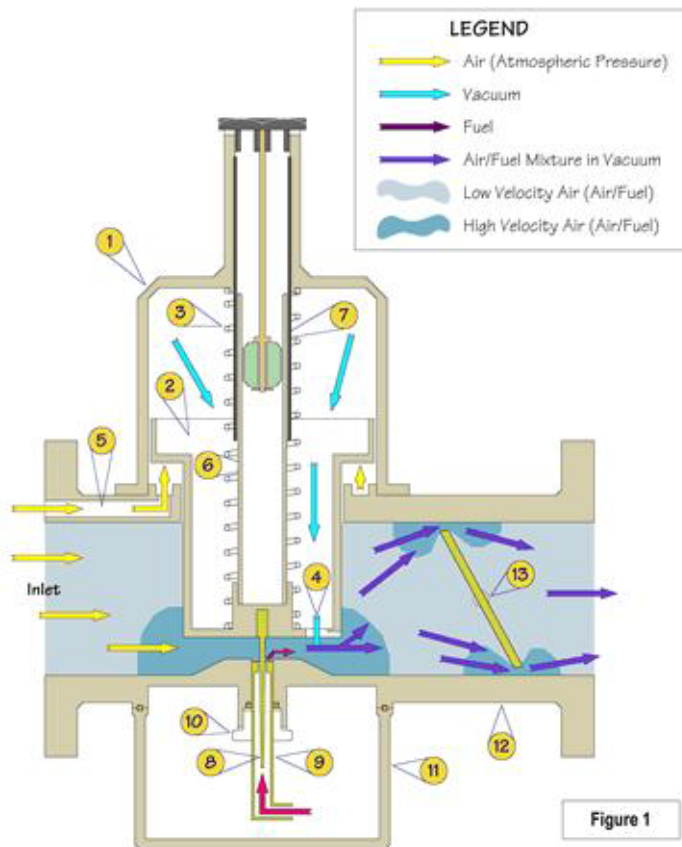


Figure 4.14 Air Valve

4.4.5 Tutorial

This section contains brief summary describing how to use Water Hammer when creating a new project.

1. Create new project as it is described for steady state analysis.
2. Change the project type to Transient Flow Analysis in the Project Options dialog. Use Edit | Project Options to open Project Options dialog.
3. Modify time settings by selecting Extended | Time Editor. Use seconds to define time settings. Typical time step for running the water hammer analysis would be 0.1 second. Short section pipes are automatically treated in a special way in order to avoid numerical problems associated with high values of Courant numbers Cr .
4. Define $Q = f(t)$ and $H = f(t)$ boundary conditions for the transient flow analysis. Use Edit | Water hammer | Boundary Conditions to assign the appropriate boundary conditions to nodes.



5. Add wave speed for each pipe in the model. Use the Field Calculator from within the Pipe Editor to assign the wave speed globally. Example of the command assigning the wave speed of 1200m/s to each pipe: `wavespeed=1200`.
6. Edit specific data to pumps, valves, air-chambers, vented air-chambers, or air valves where appropriate.
7. Run the water hammer analysis and load the analysis results.
8. Use standard way of reviewing the analysis results such as time series, animated plans, or animated profile plots.





5 EPANET

MIKE URBAN consists of EPANET numerical engines (public domain EPANET engine developed by EPA, US and EPANET engine modified by DHI Water&Environment by adding variable speed drive pump, extended controls, automated fire flow analysis, PID real-time control and other advanced modeling features) used for computing steady state, extended period, and water quality analysis and of WATER HAMMER numerical engine used for computing fast transient analysis.

The EPANET computer model used for water distribution network analysis is composed of two parts: (1) the input data file and (2) the EPANET computer program. The data file defines the characteristics of the pipes, the nodes (ends of the pipe), and the control components (such as pumps and valves) in the pipe network. The computer program solves the nonlinear energy equations and linear mass equations for pressures at nodes and flowrates in pipes.

EPANET Input Data File

The EPANET input data file, created automatically by MIKE URBAN WATER, includes descriptions of the physical characteristics of pipes and nodes, and the connectivity of the pipes in a pipe network system. The user can graphically layout the water distribution network, if desired. Values for the pipe network parameters are entered through easy-to-use dialog boxes. MIKE URBAN WATER then creates the EPANET input data file in the format required to run the analysis. The pipe parameters include the length, inside diameter, minor loss coefficient, and roughness coefficient of the pipe. Each pipe has a defined positive flow direction and two nodes. The parameters of nodes consist of the water demand or supply, elevation, and pressure or hydraulic grade line. The hydraulic grade line (HGL) is the summation of node elevation and pressure head at the node. The control components, which usually are installed on pipes, include control valves and booster pumps. They are also part of the input data file.

EPANET Computer Program

The EPANET computer program was developed by the U.S. EPA (Environmental Protection Agency). The program computes the flowrates in the pipes and then HGL at the nodes. The calculation of flowrates involves several iterations because the mass and energy equations are nonlinear. The number of iterations depends on the system of network equations and the user-specified accuracy. A satisfactory solution of the flowrates must meet the specified accuracy, the law of conservation of mass and energy in the water distribution system, and any other requirements imposed by the user. The calculation of HGL requires no iteration because the network equations are linear. Once the flowrate analysis is complete, the water quality computations are then performed.



5.1 History

Pipe network analysis of water distribution systems has evolved from a time-consuming process done infrequently to a quick and easy process done regularly on systems of all sizes.

Pipe network analysis initially started early in 1940. Years later, two network analysis programs were introduced by Shamir and Howard (1968) and Epp and Fowler (1970). Both programs used the Newton-Raphson method to linearize the nonlinear mass and energy equations. The major differences between these two programs are:

1. The Shamir-Howard program is based on node-oriented equations, while the Epp-Fowler program is based on loop-oriented equations.
2. The Shamir-Howard program solves for pressure, demand, and the parameters of pipes and nodes, while the Epp-Fowler program solves only for pressures and flowrates.

Since then, several programs have been developed, based on improved computing techniques as well as advances in computer hardware. Recently, several computer programs running on personal computers, such as EPANET, UNWB-LOOP, WADISO, U of K KYPIPE, and WATER have been created and made available. Of the four programs, only EPANET and U of K KYPIPE can perform dynamic simulation over an extended period of time and only EPANET can perform water quality analysis. In addition, WADISO can perform optimization analysis.

5.1.1 Analysis Methods

Three types of analysis may be conducted using MIKE URBAN WATER: steady state (static) analysis, extended period (dynamic) analysis, and water quality analysis. Steady state analysis is used to compute the pipe flowrates and the node HGL in a steady state pipe network system. Extended period analysis simulates the continuous flowrate and pressure changes over a period of time. Water quality analysis is used to compute the age of water, perform source tracking, calculate the fate of a dissolved substance, or determine the growth or decay of a substance. Each of these analysis types are discussed in detail in the following sections.

Steady State Hydraulics

The calculation of flowrates and pressures for a steady state pipe network system is called a steady state analysis. This analysis computes the pipe flowrates and the node hydraulic grade line elevations (HGL) so that the conservation of energy and mass are satisfied. The pipe network system can include pumps, check valves, and various types of control valves.



Extended Period Hydraulics

An extended period (dynamic) analysis is used to analyze a pipe network for an extended period of time. The total simulation time is usually divided into several time steps. At each time step an analysis is conducted for the pipe network based on the current network parameters and the pipe flowrates calculated from the previous time step.

In an extended period simulation, storage tanks and hydraulic switches are often present as part of the water distribution system. The system operating parameters at each time step depend on external conditions and the pipe flowrates from the previous time step. External conditions are operating parameters controlled by factors outside the system, such as external demand or pump power. The previous time step flowrates are also used to predict the storage tank HGL for the current time step.

Water Quality Analysis

A water quality analysis can be used by operations, planning, and engineering departments to study the flow and distribution of water. Source tracking, travel time determination, water age, and concentration levels of chemical constituents and contaminants are the primary concerns addressed by water quality models. In addition, tracking paths of flow and distribution provide the engineer insight to the origin and amount of water supplied to a particular location, as well as concentration levels of chemical constituents and contaminants.

Water quality models can also be used to study water retention time for reservoir operations, pipeline travel times, and the percentage of water supplied to a location from multiple sources (i.e., treatment plants, wells, and reservoirs). Additionally, water quality models can be used to develop a hydrant flushing program to reduce water stagnation at dead ends within the pipe network. And, site sampling locations, future rechlorination facility locations, cross-connection locations, and reservoir operating strategies can be designed and analyzed.

5.1.2 Applications of MIKE URBAN WATER

Water distribution analysis software, such as MIKE URBAN WATER, is typically used for three broad areas of analysis. These areas of analysis are generally referred to as planning, design, and operation applications (AWWA Manual M32, 1989).

Some examples of these applications include (1) analysis and design of booster pumps and storage tanks for municipal or rural water distribution systems, and (2) analysis and design of chlorination satellite stations.



Planning

MIKE URBAN WATER can be used in the planning of pipe network systems to meet forecasted demands of the next 10 years or 20 years. For example, the program can be used to develop long term capital-improvement plans for the existing pipe network system. These plans can include staging, sizing, and locating future pipe network and water chlorination facilities. The software can also be used in the development of a main rehabilitation plan or a system-improvement plan. And, a network analysis can provide suggestions and recommendations to prepare for the occurrence of any unusual events.

Design

MIKE URBAN WATER can be used to design a new pipe network system or improve on the existing pipe network system. For example, the analysis conducted using MIKE URBAN WATER could help users in selecting and sizing pipe network components, such as pipes, booster pumps, and pressure regulating valves. As a part of the analysis, the performance of the pipe network system can be analyzed to verify that the system satisfies fire-flow demand requirements.

Operation

The operating status of a water distribution system (e.g., the pipe flowrates and junction node pressures) can be determined by MIKE URBAN WATER. The analysis can then be used to develop operational strategies based on the guidelines for maximum use of available water and efficient management of electrical energy. MIKE URBAN WATER can also be used for system troubleshooting, such as finding the location of a pipe break.

5.1.3 Skeletonization

In the past, water distribution models have not included all of the pipes contained in the network system due to the fact that the numerical modeling schemes used and the memory requirements required could handle only a limited number of pipes and nodes. These limitations required skeletonization of the pipe network system, where only a subset of all the pipes contained within the network system was defined. However, these limitations have eroded over the past years as enhanced programming methods and increased computer hardware capabilities have come into being.

Skeleton network models typically include only those pipes that are considered significant to the flow and distribution of water. For example, a skeleton model might only consider 12 inch and larger diameter mains. Smaller diameter mains might also be included if they supply water to a significant area or complete a loop in the network. Examples of a complete network model and an equivalent skeleton network model are shown in Figure 5.1 and Figure 5.2.

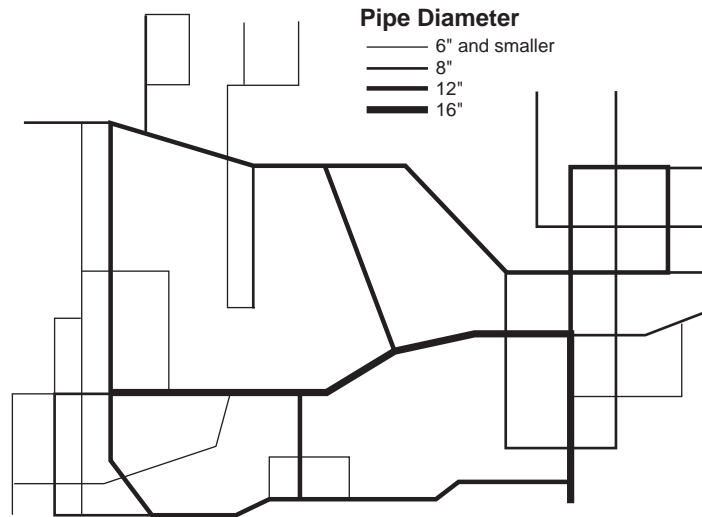


Figure 5.1 A complete water distribution network, showing all pipes contained within the network

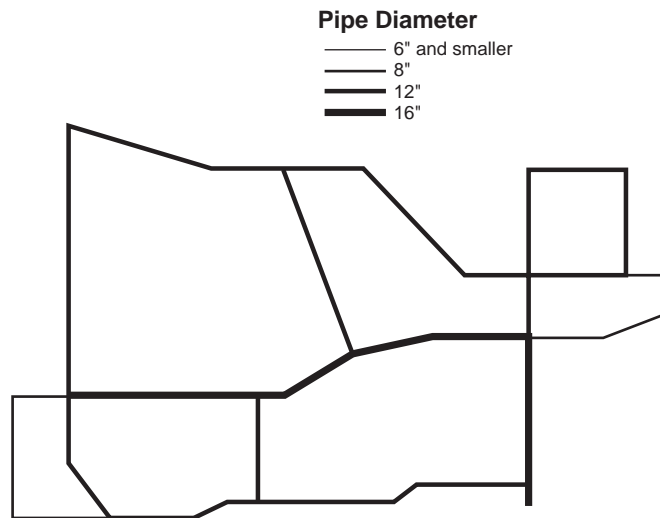


Figure 5.2 An equivalent skeleton water distribution network, showing those pipes that are considered significant to the flow and distribution of water for a steady state simulation

A major advantage with working with a skeleton model is that they are much easier to define since there is less data involved, and the simulation time is shorter since not as many pipes are involved in computing a solution. Also,



the displaying of results is quicker and more readable since less pipes are involved. In addition, the number of iterations required to converge to a solution is typically less as well.

Skeletonization, however, can adversely affect model accuracy. For example, skeletonization should not be used when performing water quality modeling since the flow rates, paths, and velocities for all pipes are critical components to a water quality simulation. For steady state water distribution simulations, though, skeletonization can produce results that have sufficient accuracy if the water consumption has been properly assigned to the defined nodes. For example, a skeleton model node might not only represent the water demand within the immediate area, but also demand for a smaller pipe that services an area much farther away. Also, if an existing calibrated model has been converted into a skeleton model, it may be necessary to recalibrate the new skeleton model since the nodal demands would be represented differently.

As was stated before, a skeleton model should not be used when performing water quality modeling. However, in other situations, the modeler must decide when it is appropriate to skeletonize a network model to produce results with sufficient accuracy to meet the modeling requirements. Therefore, when and where to skeletonize a network must be decided upon a case by case basis.

5.2 The Water Distribution Network

A water distribution system is a pipe network which delivers water from single or multiple supply sources to consumers. Typical water supply sources include reservoirs, storage tanks, and external water supply at junction nodes such as groundwater wells. Consumers include both municipal and industrial users. The pipe network consists of pipes, nodes, pumps, control valves, storage tanks, and reservoirs.

EPANET views the water distribution system as a network containing nodes and links, where the nodes are connected by links. Figure 5.3 illustrates a node-link representation of a simple water distribution network.

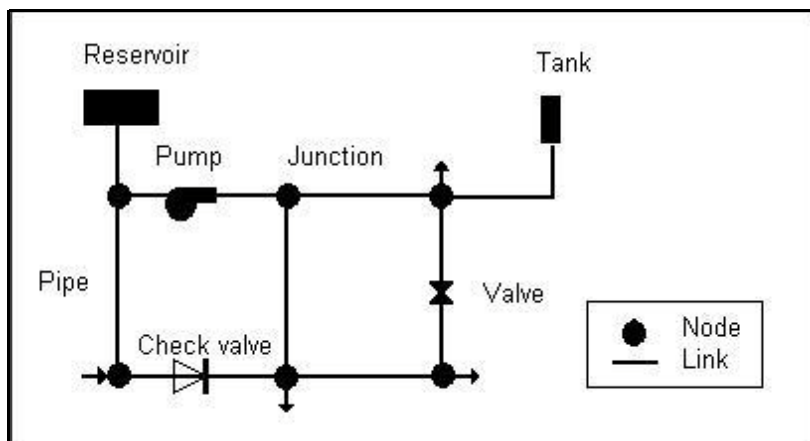


Figure 5.3 Node-link representation of a water distribution network

As shown in Figure 5.3, links can represent the following components in a network:

- Pipes
- Pumps
- Valves

Nodes, besides representing the connection point between pipes, can represent the following components in a network:

- Points of water consumption (demand nodes).
- Points of water input (source nodes).
- Locations of tanks or reservoirs (storage nodes).

How the EPANET program models the hydraulic behavior of each of these components is described in the following sections. All flow rates in this discussion will be assumed as cubic feet per second (cfs), although the program can also accept flow rates in gallons per minute (gpm), million gallons per day (mgd), and litres per second (L/s).

5.2.1 Pipes

Every pipe is connected to two nodes at its ends. In a pipe network system, pipes are the channels used to convey water from one location to another. The physical characteristics of a pipe include the length, inside diameter, roughness coefficient, and minor loss coefficient. The pipe roughness coefficient is associated with the pipe material and age. The minor loss coefficient is due to the fittings along the pipe.

When water is conveyed through the pipe, hydraulic energy is lost due to the friction between the moving water and the stationary pipe surface. This fric-



tion loss is a major energy loss in pipe flow and is a function of flowrate, pipe length, diameter, and roughness coefficient.

The head lost to friction associated with flow through a pipe can be expressed in a general fashion as:

$$h_L = aq^b \quad (5.1)$$

where

- hL = head loss, ft
- q = flow, cfs
- a = a resistance coefficient
- b = a flow exponent

EPANET can use any one of three popular forms of the headloss formula shown in Equation 5.1: the Hazen-Williams formula, the Darcy-Weisbach formula, or the Chezy-Manning formula. MIKE URBAN WATER allows the user to choose the formulation to use.

The Hazen-Williams formula is probably the most popular head loss equation for water distribution systems, the Darcy-Weisbach formula is more applicable to laminar flow and to fluids other than water, and the Chezy-Manning formula is more commonly used for open channel flow. Table 5.1 lists resistance coefficients and flow exponents for each formula. Note that each formula uses a different pipe roughness coefficient, which must be determined empirically. Table 5.2 lists general ranges of these coefficients for different types of new pipe materials. Be aware that a pipe's roughness coefficient can change considerably with age.

While the Darcy-Weisbach relationship for closed-conduit flows is generally recognized as a more accurate mathematical formulation over a wider range of flow than the Hazen-Williams formulation, the field data on e values (required for the Darcy-Weisbach formulation) are not as readily available as are the C values for the pipe wall roughness coefficient (used in the Hazen-Williams formulation).

Table 5.1 Pipe head loss formulas

Formula	Resistance Coefficient (a)	Flow Exponent (b)
Hazen-Williams	$4.72 C^{-1.85} d^{-4.87} L$	1.85
Darcy-Weisbach	$0.0252 f(\epsilon, d, q) d^{-5} L$	2
Chezy-Manning (full pipe flow)	$4.66 n^2 d^{-5.33} L$	2



Notes:

- C = Hazen-Williams roughness coefficient
- e = Darcy-Weisbach roughness coefficient, ft
- f = friction factor (dependent on e, d, and q)
- d = pipe diameter, ft
- L = pipe length, ft

Table 5.2 Roughness coefficients for new pipe

Material	Hazen-Williams C	Darcy-Weisbach ϵ , millifeet	Manning's n
Cast Iron	130 - 140	0.85	0.012 - 0.015
Concrete or Concrete Lined	120 - 140	1.0 - 10	0.012 - 0.017
Galvanized Iron	120	0.5	0.015 - 0.017
Plastic	140 - 150	0.005	0.011 - 0.015
Steel	140 - 150	0.15	0.015 - 0.017
Vitrified Clay	110	----	0.013 - 0.015

Pipes can contain check valves in them that restrict flow to a specific direction. They can also be made to open or close at pre-set times, when tank levels fall below or above certain set-points, or when nodal pressures fall below or above certain set-points. The normal initial condition for a pipe containing a check valve or a pump is to be in open mode. The pipe will then switch to closed mode only when flow is reversed.

In addition to the energy loss caused by friction between the fluid and the pipe wall, energy losses also are caused by obstructions in the pipeline, changes in flow direction, and changes in flow area. These losses are called minor losses because their contribution to the reduction in energy is usually much smaller than frictional losses. Head loss, which is the sum of friction loss and minor losses, reduces the flowrate through the pipe.

Modeling Parallel Pipes

MIKE URBAN WATER can model parallel pipes. To define a set of parallel pipes, simply draw in pipes with the same starting and ending nodes. However, to prevent the pipes from being drawn exactly on top of one another, it is suggested that one of the pipes have at least one vertex (or bend point) inserted in it so that the pipes are displayed slightly separated (see Figure 5.3). For information on defining curved pipes, see the section titled Pipe Editor on -18.

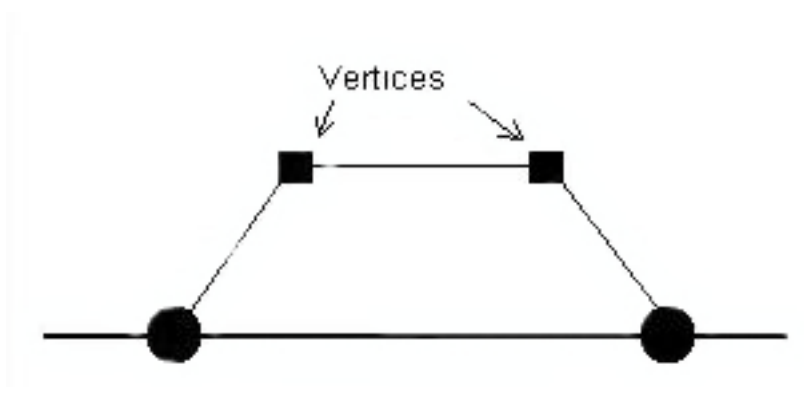


Figure 5.4 Vertices

Bend points are inserted to separate two parallel pipes to allow them to be selectable.

5.2.2 Pumps

A pump is a device that raises the hydraulic head of water. EPANET represents pumps as links of negligible length with specified upstream and downstream junction nodes.

Pumps are described with a pump characteristic curve. The pump curve describes the additional head imparted to a fluid as a function of its flow rate through the pump. EPANET is capable of modeling several types of user defined pumps, constant energy pumps, single-point pump curves, three-point pump curves, multiple point pump curves, and variable speed pump.

Constant energy pumps operate at the same horsepower or kilowatt rating over all combinations of head and flow. In this case the equation of the pump curve would be:

$$h_G = \frac{8,81 H_p}{q} \quad (5.2)$$

where

- hG = head gain, ft
- Hp = pump horsepower
- q = flow, cfs

A single point pump curve is defined by a single head-flow combination that represents a pump's desired operating point. EPANET fills in the rest of the curve by assuming:

1. A shutoff head at zero flow equal to 133% of the design head.
2. A maximum flow at zero head equal to twice the design flow.

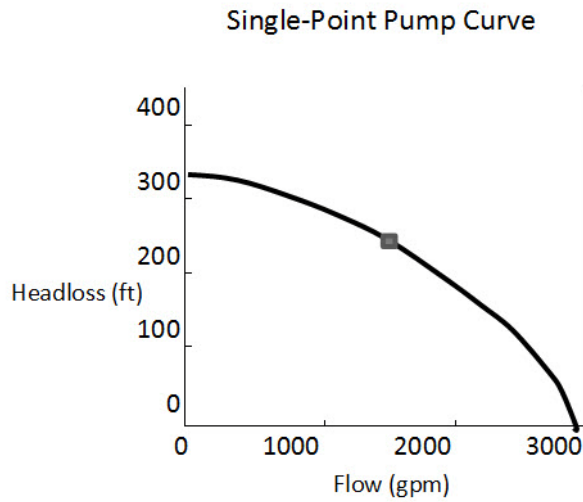


Figure 5.5 Single Point Pump Curve

A three-point pump curve is defined by three operating points:

1. Low Flow (flow and head at low or zero flow conditions).
2. Design Flow (flow and head at desired operating point).
3. Maximum Flow (flow and head at maximum flow).

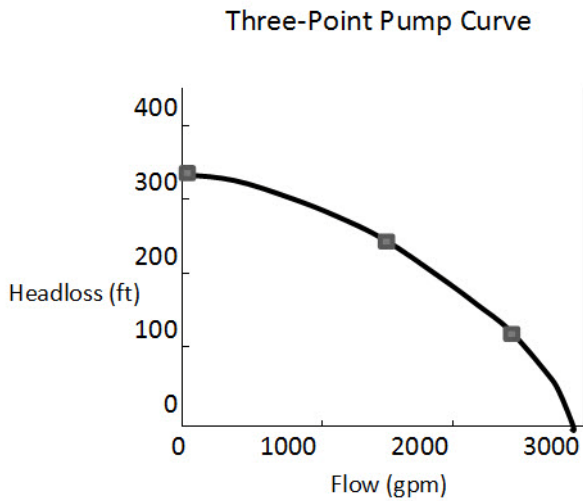


Figure 5.6 Three Point Pump Curve



A multi-point pump curve is defined by providing either a pair of head-flow points or four or more such points. EPANET creates the complete curve by connecting the points with straight line segments.

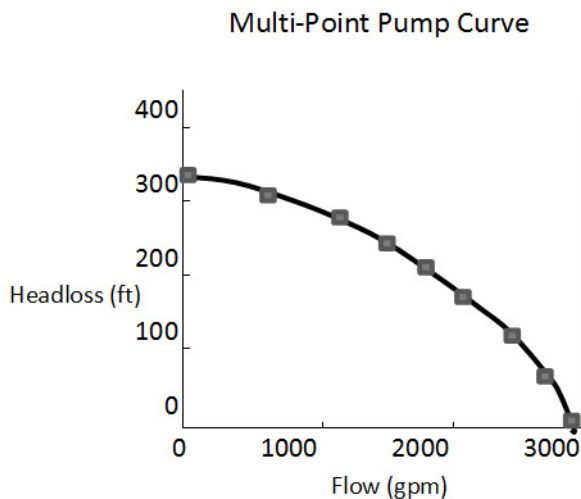


Figure 5.7 Multi Point Pump Curve

For variable speed pumps, the pump curve shifts as the speed of the pump changes. The relationships between flow (Q) and head (H) at speed N_1 and N_2 are:

$$\frac{Q_1}{Q_2} = \frac{N_1}{N_2} \quad (5.3)$$

$$\frac{H_1}{H_2} = \left[\frac{N_1}{N_2} \right]^2 \quad (5.4)$$

EPANET will shut a pump down if the system demands a head higher than the first point on a curve (i.e., the shutoff head.) A pump curve must be supplied for each pump the system unless the pump is a constant energy pump.

Pumping Rate Limits

Flow through a pump is unidirectional (i.e., one flow direction only) and pumps must operate within the head and flow limits imposed by their characteristic curves. If the system conditions require that the pump produce more than its shutoff head, EPANET will attempt to close off the pump and will issue a warning message.



Controlling Pumps

EPANET allows you to turn pumps on or off at pre-set times, when tank levels fall below or rise above certain set-points, or when nodal pressures fall below or rise above certain set-points. Variable speed pumps can also be considered by specifying using the Control Editor (see the section titled Control Editor on -74 for further information) that their speed setting be changed under these same types of conditions. By definition, the original pump curve supplied to the program has a relative speed setting of 1. If the pump speed doubles, then the relative setting would be 2; if run at half speed, the relative setting is 0.5 and so on. Figure 5.5 illustrates how changing a pump's speed setting affects its characteristic curve.

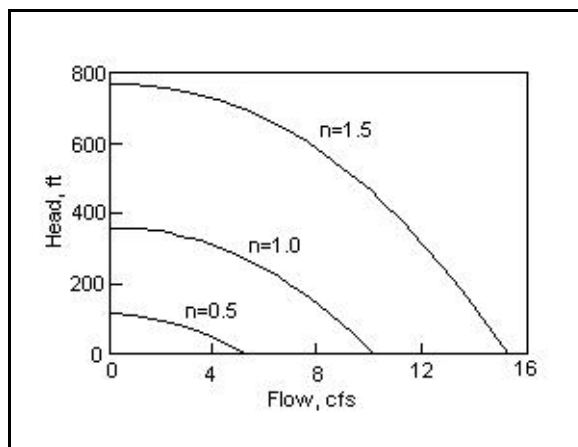


Figure 5.8 Effect of relative speed (n) on a pump curve

Modeling Pumps in Parallel and Series

As shown in Figure Figure 5.9, MIKE URBAN WATER can model parallel pumps and pumps in series. To model parallel pumps, it is necessary to insert the pumps on the same from and to nodes. To model pumps in series (where the outlet of the first pump directly discharges into the inlet of the second pump), it is necessary to insert the pumps one after the other on the same pipe.

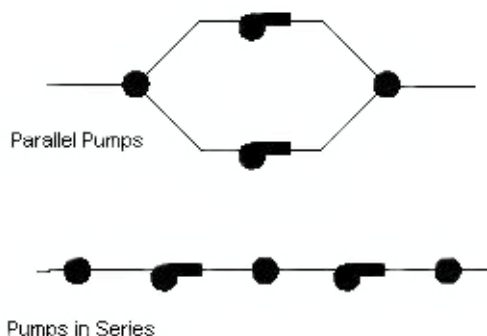


Figure 5.9 MIKE URBAN WATER can model both parallel pumps and pumps in series

If desired, the two or more pumps can be modeled as an equivalent composite single pump that has a characteristic curve equal to the sum of the individual pump curves. For pumps that are in parallel, the discharge values for the individual pump curves are added together to end up with the equivalent single pump curve. If the pumps are connected together in series, then the head values for the individual pump curves are added together to end up with the equivalent single pump curve.

For example, as shown in Figure 5.10, if two pumps are connected in parallel and one pump operates at a flow rate of 50 gpm with a head of 75 ft and the other pump operates at a 60 gpm with a head of 75 ft, then the resultant single equivalent pump curve flow rate of 110 gpm would be available with a head of 75 ft.

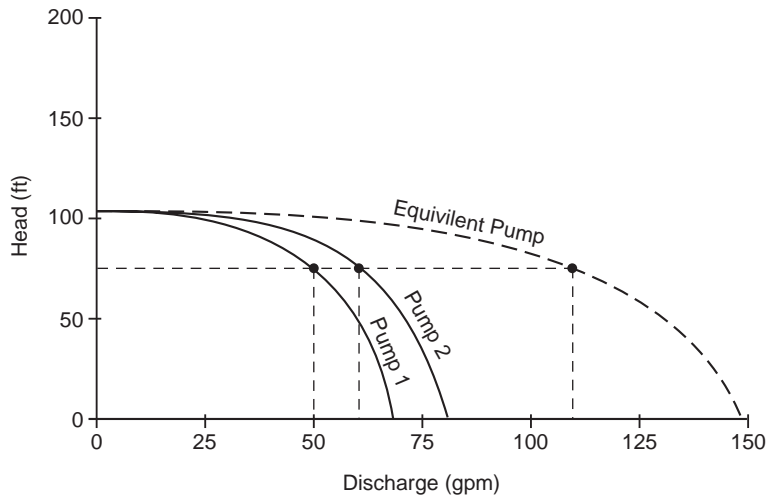


Figure 5.10 When modeling pumps in parallel, an equivalent single pump curve can be determined by adding together the characteristic pump curve discharge values from the other individual pump curves

Similarly, as shown in Figure 5.11 if two pumps are connected in series and one pump provides a head of 75 ft at a flow rate of 50 gpm and the other pump provides a head of 65 feet at a flow rate of 50 gpm, then the resultant single equivalent pump curve head of 140 ft would be available at a flow rate of 50 gpm.

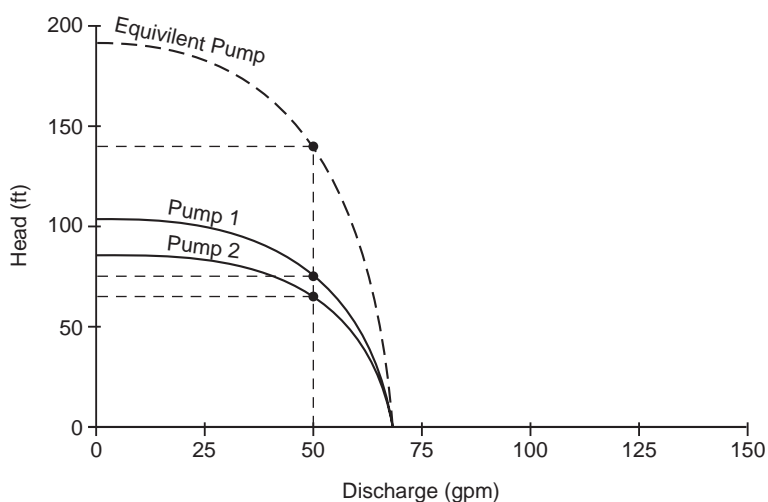


Figure 5.11 When modeling pumps in series, an equivalent single pump curve can be determined by adding together the characteristic pump curve head values from the other individual pump curves

When modeling pumps in series, it is preferable to use an equivalent composite pump to represent the multiple pumps in series. This is because it is much easier to control a single pump than multiple pumps simultaneously when the pumps turn on and off. In addition, multiple pumps in series can cause numerical disconnections in the pipe network when EPANET checks to see if the upstream grade is greater than the downstream grade plus the available pump head.

5.2.3 Valves

Aside from the valves in pipes that are either fully opened or closed (such as check valves), EPANET can also represent valves that control either the pressure or flow at specific points in a network. Such valves are considered as links of negligible length with specified upstream and downstream junction nodes. The types of valves that can be modeled are described below.

Pressure Reducing Valves (PRV)

Pressure reducing valves (PRV) limit the pressure on their downstream end to not exceed a pre-set value when the upstream pressure is above the setting. If the upstream pressure is below the setting, then flow through the valve is unrestricted. Should the pressure on the downstream end exceed that on the upstream end, the valve closes to prevent reversal of flow.



Pressure Sustaining Valves (PSV)

Pressure sustaining valves (PSV) try to maintain a minimum pressure on their upstream end when the downstream pressure is below that value. If the downstream pressure is above the setting, then flow through the valve is unrestricted. Should the downstream pressure exceed the upstream pressure then the valve closes to prevent reverse flow.

Pressure Breaker Valves (PBV)

Pressure breaker valves (PBV) force a specified pressure loss to occur across the valve. Flow can be in either direction through the valve.

Flow Control Valves (FCV)

Flow control valves (FCV) limit the flow through a valve to a specified amount. The program produces a warning message if this flow cannot be maintained without having to add additional head at the valve.

Throttle Control Valves (TCV)

Throttle control valves (TCV) simulate a partially closed valve by adjusting the minor head loss coefficient of the valve. A relationship between the degree to which the valve is closed and the resulting head loss coefficient is usually available from the valve manufacturer.

General Purpose Valves (GPV)

A General Purpose Valve (GPV) provides the capability to model devices and situations with unique headloss - flow relationships, such as reduced pressure backflow prevention valves, turbines, and well drawdown behavior. The valve setting is the ID of a Headloss Curve.

A Headloss Curve is used to describe the headloss (Y in feet or meters) through a General Purpose Valve (GPV) as a function of flow rate (X in flow units). It provides the capability to model devices and situations with unique headloss - flow relationships, such as reduced flow backflow prevention valves, turbines, and well drawdown behavior.

5.2.4 Minor Losses

Minor head losses (also called local losses) can be associated with the added turbulence that occurs at bends, junctions, meters, and valves. The importance of such losses will depend on the layout of the pipe network and the degree of accuracy required. EPANET allows each pipe and valve to have a



minor loss coefficient associated with it. It computes the resulting head loss from the following formula:

$$h_L = \frac{0,0252Kq^2}{d^4} \quad (5.5)$$

where

- hL = head loss, ft
- d = diameter in, ft
- K = loss coefficients
- q = flow rate, cfs

Table 5.3 gives values of K for several different kinds of pipe network components.

Table 5.3 Loss coefficients for common pipe network components

Component	Loss Coefficient
Globe valve, fully open	10.0
Angle valve, fully open	5.0
Swing check valve, fully open	2.5
Gate valve, fully open	0.2
Short-radius elbow	0.9
Medium-radius elbow	0.8
Long-radius elbow	0.6
45° elbow	0.4
Closed return bend	2.2
Standard tee - flow through run	0.6
Standard tee - flow through branch	1.8
Square entrance	0.5
Exit	1.0

5.2.5 Nodes

Nodes are the locations where pipes connect. Two types of nodes exist in a pipe network system, (1) fixed nodes and (2) junction nodes. Fixed nodes are nodes whose HGL are defined. For example, reservoirs and storage tanks are considered fixed nodes, because their HGL are initially defined. Junction



nodes are nodes whose HGL are not yet determined and must be computed in the pipe network analysis.

Degree of freedom, elevation, and water demand are the three important input parameters for a node (see Figure 5.3). A node's degree of freedom is the number of pipes that connect to that node. In EPANET, a junction node may be connected to more than one pipe, but a fixed node (i.e., storage tank or reservoir) must be connected to exactly one pipe. Therefore, a fixed node's degree of freedom is always one, and a junction node's degree of freedom may be greater than one. The elevation of a node can sometimes be obtained from system maps or drawings. More often, it is approximated using topographic maps. Water demand at a junction node is the summation of all water drawn from or added to the system at that node.

All nodes should have their elevation specified above sea level (i.e., greater than zero) so that the contribution to hydraulic head due to elevation can be computed. Any water consumption or supply rates at nodes that are not storage nodes must be known for the duration of time the network is being analyzed. Storage nodes (i.e., tanks and reservoirs) are special types of nodes where a free water surface exists and the hydraulic head is simply the elevation of water above sea level. Tanks are distinguished from reservoirs by having their water surface level change as water flows into or out of them—reservoirs remain at a constant water level no matter what the flow is.

EPANET models the change in water level of a storage tank with the following equation:

$$\Delta y = \frac{q}{A} \Delta t \quad (5.6)$$

where

- Dy = change in water level, ft
- q = flow rate into (+) or out of (-) tank, cfs
- A = cross-sectional area of the tank, ft²
- Dt = time interval, sec

For each storage tank, the program must know the cross-sectional area as well as the minimum and maximum permissible water level. Reservoir-type storage nodes are usually used to represent external water sources, such as lakes, rivers, or groundwater wells. Storage nodes should not have an external water consumption or supply rate associated with them.

Modeling Pumped Groundwater Wells

The water supply to the water distribution network is furnished by reservoirs and storage tanks. Reservoirs are treated as inexhaustible sources of water, and as such their water level never varies. However, as a storage tank empties, its water level lowers and it has to be refilled by pumping from either a

reservoir or a groundwater well. In EPANET, a pumped groundwater well is modeled the same as a pumped reservoir, as shown in Figure 5.12.

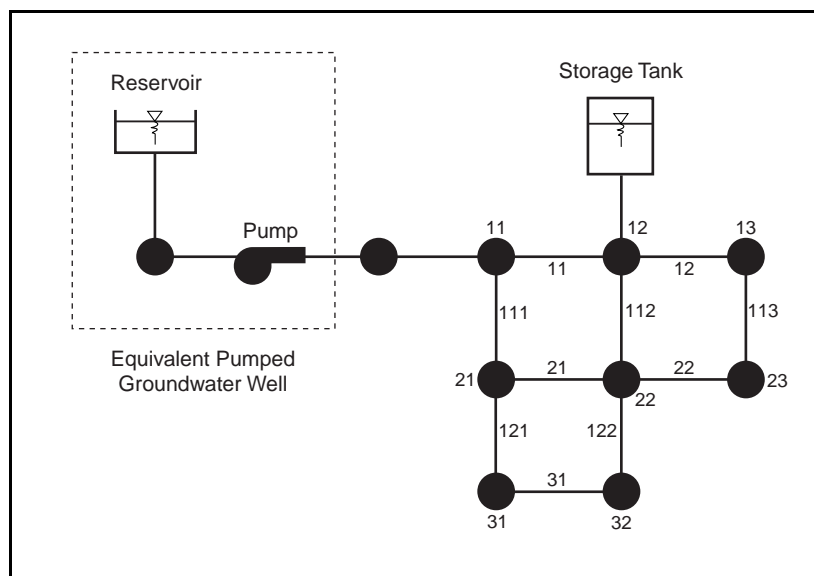


Figure 5.12 A pumped groundwater well is modeled as a reservoir with an attached pump in EPANET

In order to model a pumped groundwater well, an equivalent pumped reservoir must be defined (as shown in Figure 5.12) since a pumped groundwater well cannot be explicitly defined with EPANET. Note that a pump cannot be directly attached to reservoir or storage tank, therefore junction nodes are automatically inserted by MIKE URBAN WATER when assigning a pump to the link attached to a reservoir or storage tank.

As the storage tank empties and its water level falls to a certain amount, control rules turn on the groundwater (reservoir) pump to start refilling of the storage tank. The control rules used to regulate the starting and stopping of the groundwater pump are defined within the Control Editor (see the section titled Control Editor on - 74 for further information).

As pumping of the groundwater occurs, drawdown of the water table elevation at the groundwater well can occur, as shown in Figure 5.13. At higher pumping rates, the groundwater well may not be able to recharge fast enough to maintain the pumping rate specified by the defined groundwater well pump curve. To account for this effect, the groundwater well pump curve may need to be adjusted downward, as shown in Figure Figure 5.13, if this effect is a significant factor in modeling the refilling of the storage tank.

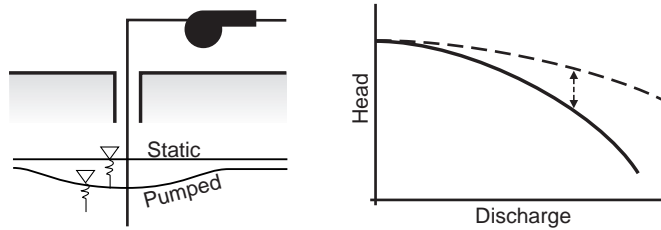


Figure 5.13 The groundwater well pump curve may need to be adjusted downward to account for groundwater well drawdown and recharge effects

Modeling Hydraulically Adjacent Storage Tanks

When performing an extended period simulation, if two or more storage tanks are hydraulically adjacent to each other it is possible that oscillations can occur between the tanks as the water bounces back and forth between them. This fluctuation is caused by small differences in flow rates as the tanks refill individually, causing the water level in the tanks to differ over time thereby causing the oscillation between the tanks. To prevent this effect from occurring, it is recommended that hydraulically adjacent tanks be modeled as a single composite tank with an equivalent total surface area and storage volume equal to the sum of the individual tanks.

5.3 Time Patterns

EPANET 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 desired value using the Time Editor (see the section titled Time Editor on - 87 for further information). The value of any of these quantities in a time period equals a baseline value multiplied by a time pattern factor for that period. Figure 5.14.1 illustrates a pattern of factors that might apply to daily water demands, where each demand period is of two hours duration. Different patterns can be assigned to individual nodes or groups of nodes.

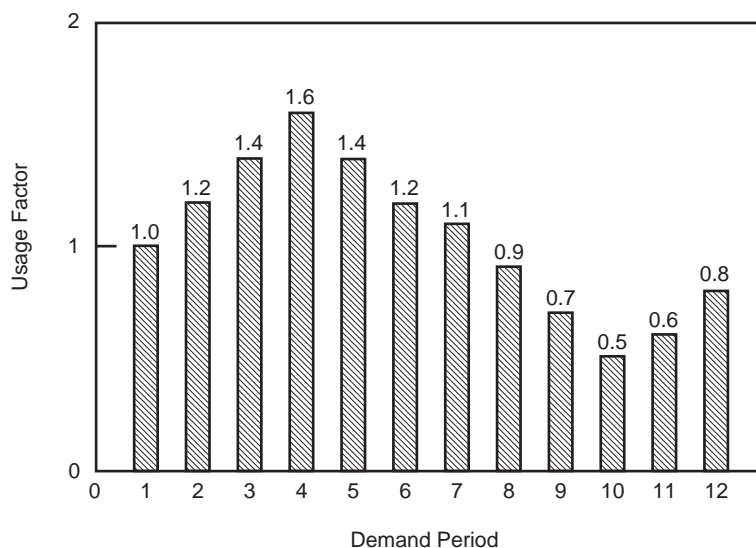


Figure 5.14 Time pattern for water usage

The Pattern Editor is used to define the time patterns to be used in an extended period simulation. See the section titled Pattern Editor on -83 for information on how to define time patterns.

5.4 Hydraulic Simulation Model

The method used in EPANET to solve the flow continuity and headloss equations that characterize the hydraulic state of the pipe network at a given point in time can be termed a hybrid node-loop approach. Todini and Pilati (1987) and later Salgado et al. (1988) chose to call it the "Gradient Method". Similar approaches have been described by Hamam and Brameller (1971) (the "Hybrid Method) and by Osiadacz (1987) (the "Newton Loop-Node Method"). The only difference between these methods is the way in which link flows are updated after a new trial solution for nodal heads has been found. Because Todini's approach is simpler, it was chosen for use in EPANET.

Assume we have a pipe network with N junction nodes and NF fixed grade nodes (tanks and reservoirs). Let the flow-headloss relation in a pipe between nodes i and j be given as:

$$H_i - H_j = h_{ij} = rQ^n_{ij} + mQ^2_{ij} \quad (5.7)$$

where H = nodal head, h = headloss, r = resistance coefficient, Q = flow rate, n = flow exponent, and m = minor loss coefficient. The value of the resistance coefficient will depend on which friction headloss formula is being used (see



below). For pumps, the headloss (negative of the head gain) can be represented by a power law of the form

$$h_{ij} = -\omega^2 (h_0 - r (Q_{ij} / \omega)^n) \quad (5.8)$$

where h_0 is the shutoff head for the pump, ω is a relative speed setting, and r and n are the pump curve coefficients. The second set of equations that must be satisfied is flow continuity around all nodes:

$$\sum_j Q_{ij} - D_i = 0 \quad (5.9)$$

for $i = 1, \dots, N$.

where D_i is the flow demand at node i and by convention, flow into a node is positive. For a set of known heads at the fixed grade nodes, we seek a solution for all heads H_i and flows Q_{ij} that satisfy Eqs. (5.8) and (5.9).

The Gradient solution method begins with an initial estimate of flows in each pipe that may not necessarily satisfy flow continuity. At each iteration of the method, new nodal heads are found by solving the matrix equation.

$$AH = F \quad (5.10)$$

where A = an $(N \times N)$ Jacobian matrix, H = an $(N \times 1)$ vector of unknown nodal heads, and F = an $(N \times 1)$ vector of right hand side terms.

The diagonal elements of the Jacobian matrix are:

$$A_{ii} = \sum_j p_{ij} \quad (5.11)$$

while the non-zero, off-diagonal terms are:

$$A_{ij} = -p_{ij} \quad (5.12)$$

where p_{ij} is the inverse derivative of the headloss in the link between nodes i and j with respect to flow. For pipes,

$$p_{ij} = \frac{1}{nr|Q_{ij}|^{n-1} + 2m|Q_{ij}|} \quad (5.13)$$



while for pumps

$$P_{ij} = \frac{1}{n\omega^2 r \left(\frac{Q_{ij}}{\omega}\right)^{n-1}} \quad (5.14)$$

Each right hand side term consists of the net flow imbalance at a node plus a flow correction factor:

$$P_{ij} = \frac{1}{nr|Q_{ij}|^{n-1} + 2m|Q_{ij}|} \quad (5.15)$$

where the last term applies to any links connecting node i to a fixed grade node f and the flow correction factor y_{ij} is: P

$$Y_{ij} = p(r|Q_{ij}|^n + m|Q_{ij}|^2) \text{sgn}(Q_{ij}) \quad (5.16)$$

for pipes and

$$y_{ij} = -p_{ij}\omega^2 \left(h_0 - r(Q_{ij}/\omega)^n \right) \quad (5.17)$$

for pumps, where sgn(x) is 1 if x > 0 and -1 otherwise. (Q_{ij} is always positive for pumps.)

After new heads are computed by solving Eq. (5.10), new flows are found from:

$$Q_{ij} = Q_{ij} - (y_{ij} - p_{ij}(H_i - H_j)) \quad (5.18)$$

If the sum of absolute flow changes relative to the total flow in all links is larger than some tolerance (e.g., 0.001), then Eqs. (5.9) and (5.10) are solved once again. The flow update formula (5.11) always results in flow continuity around each node after the first iteration.

EPANET implements this method using the following steps:

1. The linear system of equations (5.10) is solved using a sparse matrix method based on node re-ordering (George and Liu, 1981). After re-ordering the nodes to minimize the amount of fill-in for matrix A, a symbolic factorization is carried out so that only the non-zero elements of A need be stored and operated on in memory. For extended period simulation this re-ordering and factorization is only carried out once at the start



of the analysis.

2. For the very first iteration at time 0, the flow in a pipe is chosen equal to the flow corresponding to a velocity of 1 ft/sec. The flow through a conventional pump equals the design flow specified for the pump curve. An initial flow of 1 cfs is assumed for other types of pumps. (All computations are made with head in feet and flow in cfs).
3. The resistance coefficient for a pipe (r) is computed as described in Table 5.2. For the Darcy-Weisbach headloss equation, the friction factor f is computed by different equations depending on the flow's Reynolds Number (Re):

Hagen - Poiseuille formula for $Re < 2,000$ (Bhave, 1991):

$$f = \frac{64}{Re} \quad (5.19)$$

Swamee and Jain approximation to the Colebrook - White equation for $Re > 4,000$ (Bhave, 1991):

$$f = \frac{0.25}{\left[\ln \left(\frac{\varepsilon}{3.7d} + \frac{5.74}{Re^{0.9}} \right) \right]^2} \quad (5.20)$$

Cubic Interpolation From Moody Diagram for $2,000 < Re < 4,000$ (Dunlop, 1991):

$$f = (X1 + R(X2 + R(X3 + X4))) \quad (5.21)$$

$$R = \frac{Re}{64} \quad (5.22)$$

$$X1 = 7FA - FB \quad (5.23)$$

$$X3 = -0,128 + 13FA - 2FB \quad (5.24)$$

$$X4 = R(0,032 - 3FA + 0,5FB) \quad (5.25)$$

$$X3 = -0,128 + 13FA - 2FB \quad (5.26)$$



$$X4 = R(0,032 - 3FA + 0,5FB) \quad (5.27)$$

$$FA = (Y3)^{-2} \quad (5.28)$$

$$FB = FA \left(2 - \frac{0,00514215}{(Y2)(Y3)} \right) \quad (5.29)$$

$$Y2 = \frac{\varepsilon}{3,7d} + \frac{5,74}{Re^{0,9}} \quad (5.30)$$

$$Y3 = -0,86859Ln \left(\frac{\varepsilon}{3,7d} + \frac{5,74}{4000^{0,9}} \right) \quad (5.31)$$

where e = pipe roughness and d = pipe diameter.

4. The minor loss coefficient based on velocity head (K) is converted to one based on flow (m) with the following relation:

$$m = \frac{0,02517K}{d^4} \quad (5.32)$$

5. Emitters at junctions are modeled as a fictitious pipe between the junction and a fictitious reservoir. The pipe's headloss parameters are $n = (1/g)$, $r = (1/C)n$, and $m = 0$ where C is the emitter's discharge coefficient and g is its pressure exponent. The head at the fictitious reservoir is the elevation of the junction. The computed flow through the fictitious pipe becomes the flow associated with the emitter.
6. Open valves are assigned an r -value by assuming the open valve acts as a smooth pipe ($f = 0.02$) whose length is twice the valve diameter. Closed links are assumed to obey a linear headloss relation with a large resistance factor, i.e., $h = 108Q$, so that $p = 10^{-8}$ and $y = Q$. For links where $(r+m)Q < 10^{-7}$, $p = 10^{-7}$ and $y = Q/n$.
7. Status checks on pumps, check valves (CVs), flow control valves, and pipes connected to full/empty tanks are made after every other iteration, up until the 10th iteration. After this, status checks are made only after convergence is achieved. Status checks on pressure control valves (PRVs and PSVs) are made after each iteration.
8. During status checks, pumps are closed if the head gain is greater than the shutoff head (to prevent reverse flow). Similarly, check valves are



closed if the headloss through them is negative (see below). When these conditions are not present, the link is re-opened. A similar status check is made for links connected to empty/full tanks. Such links are closed if the difference in head across the link would cause an empty tank to drain or a full tank to fill. They are re-opened at the next status check if such conditions no longer hold.

9. Simply checking if $h < 0$ to determine if a check valve should be closed or open was found to cause cycling between these two states in some networks due to limits on numerical precision. The following procedure was devised to provide a more robust test of the status of a check valve (CV):

- if $|h| > H_{tol}$ then
- if $h > -H_{tol}$ then status = CLOSED
- if $Q < -Q_{tol}$ then status = CLOSED
- else status = OPEN
- else
- if $Q < -Q_{tol}$ then status = CLOSED
- else status = unchanged

where $H_{tol} = 0.0005$ ft and $Q_{tol} = 0.0001$ cfs.

10. If the status check closes an open pump, pipe, or CV, its flow is set to 10^{-6} cfs. If a pump is re-opened, its flow is computed by applying the current head gain to its characteristic curve. If a pipe or CV is re-opened, its flow is determined by solving Eq. (5.9) for Q under the current headloss h , ignoring any minor losses.
11. Matrix coefficients for pressure breaker valves (PBVs) are set to the following: $p = 108$ and $y = 108H_{set}$, where H_{set} is the pressure drop setting for the valve (in feet). Throttle control valves (TCVs) are treated as pipes with r as described in item 6 above and m taken as the converted value of the valve setting (see item 4 above).
12. Matrix coefficients for pressure reducing, pressure sustaining, and flow control valves (PRVs, PSVs, and FCVs) are computed after all other links have been analyzed. Status checks on PRVs and PSVs are made as described in item 7 above. These valves can either be completely open, completely closed, or active at their pressure or flow setting.
13. The logic used to test the status of a PRV is as follows:
- If current status = ACTIVE then
- if $Q < -Q_{tol}$ then new status = CLOSED
 - if $H_i < H_{set} + H_{ml} - H_{tol}$ then new status = OPEN
 - else new status = ACTIVE
- If current status = OPEN then



- if $Q < -Q_{tol}$ then new status = CLOSED
- if $H_i > H_{set} + H_{ml} + H_{tol}$ then new status = ACTIVE
- else new status = OPEN

If current status = CLOSED then

- if $H_i > H_j + H_{tol}$
- and $H_i < H_{set} - H_{tol}$ then new status = OPEN
- if $H_i > H_j + H_{tol}$
- and $H_j < H_{set} - H_{tol}$ then new status = ACTIVE
- else new status = CLOSED

where Q is the current flow through the valve, H_i is its upstream head, H_j is its downstream head, H_{set} is its pressure setting converted to head, H_{ml} is the minor loss when the valve is open ($= mQ^2$), and H_{tol} and Q_{tol} are the same values used for check valves in item 9 above. A similar set of tests is used for PSVs, except that when testing against H_{set} , the i and j subscripts are switched as are the $>$ and $<$ operators.

14. Flow through an active PRV is maintained to force continuity at its downstream node while flow through a PSV does the same at its upstream node. For an active PRV from node i to j :
 - $P_{ij} = 0$
 - $F_j = F_j + 108H_{set}$
 - $A_{jj} = A_{jj} + 108$

This forces the head at the downstream node to be at the valve setting H_{set} . An equivalent assignment of coefficients is made for an active PSV except the subscript for F and A is the upstream node i . Coefficients for open/closed PRVs and PSVs are handled in the same way as for pipes.

15. For an active FCV from node i to j with flow setting Q_{set} , Q_{set} is added to the flow leaving node i and entering node j , and is subtracted from F_i and added to F_j . If the head at node i is less than that at node j , then the valve cannot deliver the flow and it is treated as an open pipe.
16. After initial convergence is achieved (flow convergence plus no change in status for PRVs and PSVs), another status check on pumps, CVs, FCVs, and links to tanks is made. Also, the status of links controlled by pressure switches (e.g., a pump controlled by the pressure at a junction node) is checked. If any status change occurs, the iterations must continue for at least two more iterations (i.e., a convergence check is skipped on the very next iteration). Otherwise, a final solution has been obtained.
17. For extended period simulation (EPS), the following procedure is implemented:



After a solution is found for the current time period, the time step for the next solution is the minimum of:

- The time until a new demand period begins.
- The shortest time for a tank to fill or drain.
- The shortest time until a tank level reaches a point that triggers a change in status for some link (e.g., opens or closes a pump) as stipulated in a simple control.
- The next time until a simple timer control on a link kicks in.
- The next time at which a rule-based control causes a status change somewhere in the network.

In computing the times based on tank levels, the latter are assumed to change in a linear fashion based on the current flow solution. The activation time of rule-based controls is computed as follows:

- Starting at the current time, rules are evaluated at a rule time step. Its default value is 1/10 of the normal hydraulic time step (e.g., if hydraulics are updated every hour, then rules are evaluated every 6 minutes).
- Over this rule time step, clock time is updated, as are the water levels in storage tanks (based on the last set of pipe flows computed).
- If a rule's conditions are satisfied, then its actions are added to a list. If an action conflicts with one for the same link already on the list then the action from the rule with the higher priority stays on the list and the other is removed. If the priorities are the same then the original action stays on the list.
- After all rules are evaluated, if the list is not empty then the new actions are taken. If this causes the status of one or more links to change then a new hydraulic solution is computed and the process begins anew.
- If no status changes were called for, the action list is cleared and the next rule time step is taken unless the normal hydraulic time step has elapsed.

Time is advanced by the computed time step, new demands are found, tank levels are adjusted based on the current flow solution, and link control rules are checked to determine which links change status.

A new set of iterations with equations are begun at the current set of flows.

5.4.1 Water Age and Source Tracing

Water age is the time spent by a parcel of water in the network. It provides a simple, non-specific measure of the overall quality of delivered drinking water. New water entering the network from reservoirs or source nodes enters with age of zero. As this water moves through the pipe network it splits apart and blends together with parcels of varying age at pipe junctions and storage facilities. EPANET provides automatic modeling of water age. Internally, it



treats age as a reactive constituent whose growth follows zero-order kinetics with a rate constant equal to 1 (i.e., each second the water becomes a second older).

Source tracing tracks over time what percent of water reaching any node in the network had its origin at a particular node. The source node can be any node in the network, including storage nodes. Source tracing is a useful tool for analyzing distribution systems drawing water from two or more different raw water supplies. It can show to what degree water from a given source blends with that from other sources, and how the spatial pattern of this blending changes over time. EPANET provides an automatic facility for performing source tracing. The user need only specify which node is the source node. Internally, EPANET treats this node as a constant source of a non-reacting constituent that enters the network with a concentration of 100.

5.5 Water Quality Simulation Model

The governing equations for EPANET's water quality module are based on the principles of conservation of mass coupled with reaction kinetics. The following phenomenon are represented (Rossman et al., 1993; Rossman and Boulos, 1996):

5.5.1 Advective Transport in Pipes

A dissolved substance will travel down the length of a pipe with the same average velocity as the carrier fluid while at the same time reacting (either growing or decaying) at some given rate. Longitudinal dispersion is usually not an important transport mechanism under most operating conditions. This means there is no intermixing of mass between adjacent parcels of water traveling down a pipe. Advective transport within a pipe is represented with the following equation:

$$\frac{\partial C_i}{\partial t} = -u_i \frac{\partial C_i}{\partial x} + r(C_i) \quad (5.33)$$

where C_i = concentration (mass/volume) in pipe i as a function of distance x and time t , u_i = flow velocity (length/time) in pipe i , and r = rate of reaction (mass/volume/time) as a function of concentration.

5.5.2 Mixing at Pipe Junctions

At junctions receiving inflow from two or more pipes, the mixing of fluid is taken to be complete and instantaneous. Thus the concentration of a sub-



stance in water leaving the junction is simply the flow-weighted sum of the concentrations from the inflow pipes. For a specific node k one can write:

$$C_{i|x=0} = \frac{\sum_{j \in I_k} Q_j C_{j|x=L_j} + Q_{k,ext} C_{k,ext}}{\sum_{j \in I_k} Q_j + Q_{k,ext}} \quad (5.34)$$

where i = link with flow leaving node k , I_k = set of links with flow into k , L_j = length of link j , Q_j = flow (volume/time) in link j , $Q_{k,ext}$ = external source flow entering the network at node k , and $C_{k,ext}$ = concentration of the external flow entering at node k . The notation $C_{i|x=0}$ represents the concentration at the start of link i , while $C_{i|x=L}$ is the concentration at the end of the link

5.5.3 Mixing in Storage Facilities

It is convenient to assume that the contents of storage facilities (tanks and reservoirs) are completely mixed. This is a reasonable assumption for many tanks operating under fill-and-draw conditions providing that sufficient momentum flux is imparted to the inflow (Rossman and Grayman, 1999). Under completely mixed conditions the concentration throughout the tank is a blend of the current contents and that of any entering water. At the same time, the internal concentration could be changing due to reactions. The following equation expresses these phenomena:

$$\frac{\partial}{\partial t}(V_s C_s) = \sum_{i \in I_s} Q_i C_{i|x=L_i} - \sum_{j \in O_s} Q_j C_s + r(C_s) \quad (5.35)$$

where V_s = volume in storage at time t , C_s = concentration within the storage facility, I_s = set of links providing flow into the facility, and O_s = set of links withdrawing flow from the facility.

5.5.4 Bulk Flow Reactions

While a substance moves down a pipe or resides in storage it can undergo reaction with constituents in the water column. The rate of reaction can generally be described as a power function of concentration:

$$r = kC^n \quad (5.36)$$

where k = a reaction constant and n = the reaction order. When a limiting concentration exists on the ultimate growth or loss of a substance then the rate expression becomes



for $n > 0$, $K_b > 0$

$$R = K_b(C_L - C)C^{(n-1)} \quad (5.37)$$

for $n > 0$, $K_b < 0$

$$R = K_b(C - C_L)C^{(n-1)} \quad (5.38)$$

where C_L = the limiting concentration.

Some examples of different reaction rate expressions are:

Simple First-Order Decay ($C_L = 0$, $K_b < 0$, $n = 1$)

$$R = K_b C \quad (5.39)$$

The decay of many substances, such as chlorine, can be modeled adequately as a simple first-order reaction.

First-Order Saturation Growth ($C_L > 0$, $K_b > 0$, $n = 1$):

$$R = K_b(C_L - C) \quad (5.40)$$

This model can be applied to the growth of disinfection by-products, such as trihalomethanes, where the ultimate formation of by-product (C_L) is limited by the amount of reactive precursor present.

Two-Component, Second Order Decay ($C_L \neq 0$, $K_b < 0$, $n = 2$):

$$R = K_b C(C - C_L) \quad (5.41)$$

This model assumes that substance A reacts with substance B in some unknown ratio to produce a product P. The rate of disappearance of A is proportional to the product of A and B remaining. C_L can be either positive or negative, depending on whether either component A or B is in excess, respectively. Clark (1998) has had success in applying this model to chlorine decay data that did not conform to the simple first-order model.

Michaelis-Menton Decay Kinetics ($C_L > 0$, $K_b < 0$, $n < 0$):

$$R = \frac{K_b C}{C_L - C} \quad (5.42)$$



As a special case, when a negative reaction order n is specified, EPANET will utilize the Michaelis-Menton rate equation, shown above for a decay reaction. (For growth reactions the denominator becomes $CL + C$.) This rate equation is often used to describe enzyme-catalyzed reactions and microbial growth. It produces first-order behavior at low concentrations and zero-order behavior at higher concentrations. Note that for decay reactions, CL must be set higher than the initial concentration present.

Koechling (1998) has applied Michaelis-Menton kinetics to model chlorine decay in a number of different waters and found that both K_b and CL could be related to the water's organic content and its ultraviolet absorbance as follows:

$$K_b = -0,32UVA^{1,365} - \frac{(100UVA)}{DOC} \quad (5.43)$$

$$C_L = 4.98UVA - 1.91DOC \quad (5.44)$$

where UVA = ultraviolet absorbance at 254 nm (1/cm) and DOC = dissolved organic carbon concentration (mg/L).

Note: These expressions apply only for values of K_b and CL used with Michaelis-Menton kinetics.

Zero-Order growth ($CL = 0$, $K_b = 1$, $n = 0$)

$R = 1.0$

This special case can be used to model water age, where with each unit of time the "concentration" (i.e., age) increases by one unit.

The relationship between the bulk rate constant seen at one temperature (T_1) to that at another temperature (T_2) is often expressed using a van't Hoff - Arrhenius equation of the form:

$$K_{b2} = K_{b1}\theta^{T_2-T_1} \quad (5.45)$$

where q is a constant. In one investigation for chlorine, q was estimated to be 1.1 when T_1 was 20 deg. C (Koechling, 1998).

5.5.5 Pipe Wall Reactions

While flowing through pipes, dissolved substances can be transported to the pipe wall and react with material such as corrosion products or biofilm that are on or close to the wall. The amount of wall area available for reaction and the rate of mass transfer between the bulk fluid and the wall will also influ-



ence the overall rate of this reaction. The surface area per unit volume, which for a pipe equals 2 divided by the radius, determines the former factor. The latter factor can be represented by a mass transfer coefficient whose value depends on the molecular diffusivity of the reactive species and on the Reynolds number of the flow (Rossman et. al, 1994). For first-order kinetics, the rate of a pipe wall reaction can be expressed as:

$$r = \frac{2k_w k_f C}{R(k_w + k_f)} \quad (5.46)$$

where k_w = wall reaction rate constant (length/time), k_f = mass transfer coefficient (length/time), and R = pipe radius. For zero-order kinetics the reaction rate cannot be any higher than the rate of mass transfer, so

$$r = \text{MIN}(k_w, k_f C)(2 / R) \quad (5.47)$$

where k_w now has units of mass/area/time.

Mass transfer coefficients are usually expressed in terms of a dimensionless Sherwood number (Sh):

$$k_f = Sh \frac{D}{d} \quad (5.48)$$

in which D = the molecular diffusivity of the species being transported (length²/time) and d = pipe diameter. In fully developed laminar flow, the average Sherwood number along the length of a pipe can be expressed as

$$Sh = 3.65 + \frac{0.0668(d / L) \text{Re} Sc}{1 + 0.04[(d / L) \text{Re} Sc]^{2/3}} \quad (5.49)$$

in which Re = Reynolds number and Sc = Schmidt number (kinematic viscosity of water divided by the diffusivity of the chemical) (Edwards et.al, 1976). For turbulent flow the empirical correlation of Notter and Sleicher (1971) can be used:

$$Sh = 0.0149 \text{Re}^{0.88} Sc^{1/3} \quad (5.50)$$

5.5.6 Lagrangian Transport Algorithm

EPANET's water quality simulator uses a Lagrangian time-based approach to track the fate of discrete parcels of water as they move along pipes and mix



together at junctions between fixed-length time steps (Liou and Kroon, 1987). These water quality time steps are typically much shorter than the hydraulic time step (e.g., minutes rather than hours) to accommodate the short times of travel that can occur within pipes. As time progresses, the size of the most upstream segment in a pipe increases as water enters the pipe while an equal loss in size of the most downstream segment occurs as water leaves the link. The size of the segments in between these remains unchanged.

The following steps occur at the end of each such time step:

1. The water quality in each segment is updated to reflect any reaction that may have occurred over the time step.
2. The water from the leading segments of pipes with flow into each junction is blended together to compute a new water quality value at the junction. The volume contributed from each segment equals the product of its pipe's flow rate and the time step. If this volume exceeds that of the segment then the segment is destroyed and the next one in line behind it begins to contribute its volume.
3. Contributions from outside sources are added to the quality values at the junctions. The quality in storage tanks is updated depending on the method used to model mixing in the tank (see below).
4. New segments are created in pipes with flow out of each junction, reservoir, and tank. The segment volume equals the product of the pipe flow and the time step. The segment's water quality equals the new quality value computed for the node.

To cut down on the number of segments, Step 4 is only carried out if the new node quality differs by a user-specified tolerance from that of the last segment in the outflow pipe. If the difference in quality is below the tolerance then the size of the current last segment in the outflow pipe is simply increased by the volume flowing into the pipe over the time step.

This process is then repeated for the next water-quality time step. At the start of the next hydraulic time step the order of segments in any links that experience a flow reversal is switched. Initially each pipe in the network consists of a single segment whose quality equals the initial quality assigned to the downstream node.

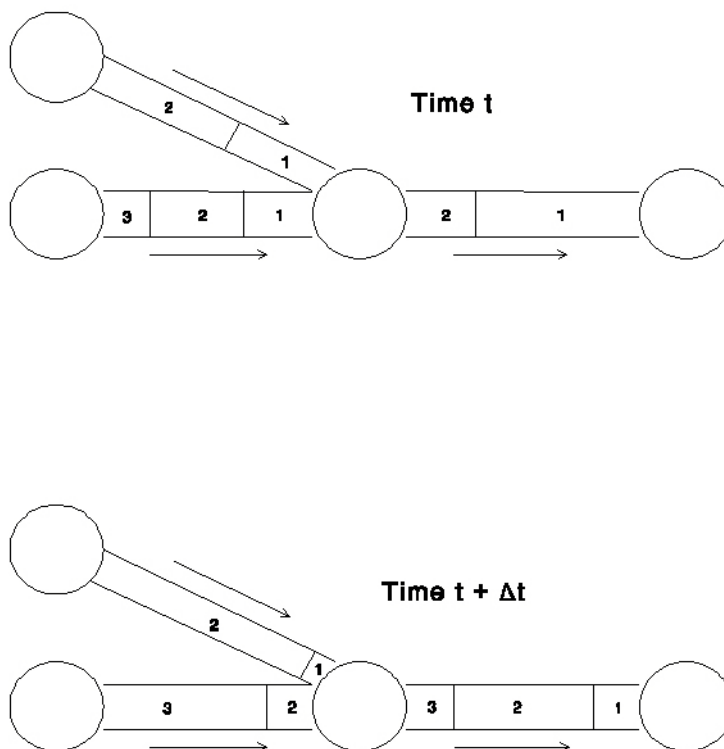


Figure 5.15 Behavior of Segments in the Lagrangian Solution Method

5.5.7 Tank Mixing Models

EPANET can use four different types of models to characterize mixing within storage tanks: Complete Mixing, Two-Compartment Mixing, First In First Out (FIFO) Plug Flow and Last In First Out (LIFO) Plug Flow. Different models can be used with different tanks within a network.

Complete Mixing assumes that all water that enters the tank is instantaneously and completely mixed with the water already in the tank.

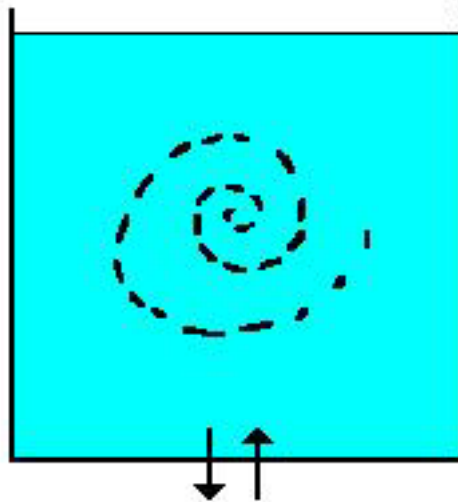


Figure 5.16 Complete mixed tank mixing model schematic

It is the simplest mixing behavior to assume, requires no extra parameters to describe it, and seems to apply well to a large number of facilities that operate in a fill and draw fashion.

Two-Compartment Mixing divides the available storage volume into two compartments.

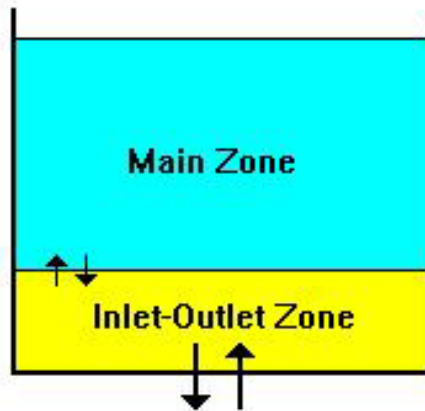


Figure 5.17 Two compartment tank mixing model schematic

The inlet/outlet pipes of the tank are assumed to be in the first compartment. New water that enters the tank is mixed with the first compartment. If this compartment is full, then it sends the overflow water to the second compartment where it completely mixes with the water stored there. When water leaves the tank, it exits from the first compartment, which if full, receives an



equal volume of water from the second compartment to make up the difference. The first compartment is capable of simulating short circulating between inflow and outflow, while the second compartment represents dead zones. The user must define a single parameter, which is the fraction of the volume of the tank to be dedicated to the first compartment.

FIFO Plug Flow assumes that there is no mixing during the residence time of the water in the tank.

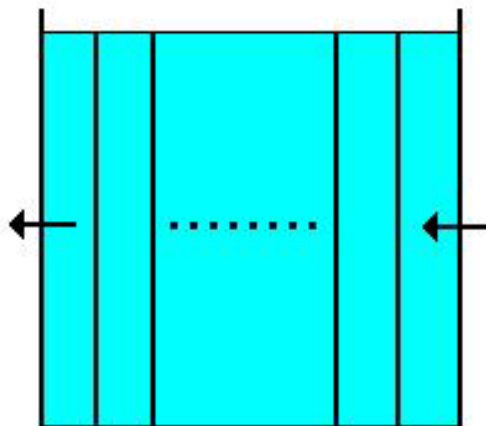


Figure 5.18 FIFO tank mixing model schematic

Water parcels move through the tank in a segregated fashion, where the first parcel to enter the tank is also the first to leave. Physically speaking, this model best represents baffled tanks with constant inflow and outflow. There are no additional parameters to describe in this model.

LIFO Plug Flow assumes that there is no mixing between parcels of water that enter the tank.

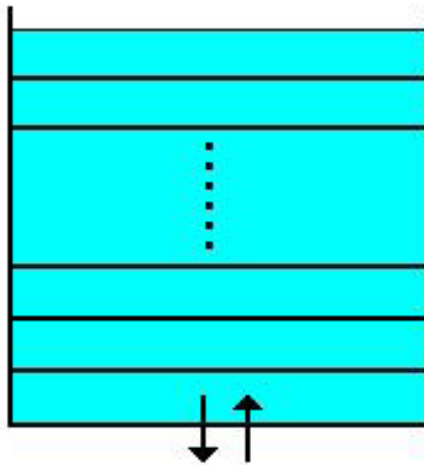


Figure 5.19 LIFO tank mixing model schematic

However, in contrast to the FIFO Plug Flow model, the parcels stack up one on top of another, where water enters and leaves from the bottom. In this fashion, the first parcel to enter the tank is also the last to leave. Physically speaking this type of model might apply to a tall narrow standpipe with an inflow/outflow pipe at the bottom and a low momentum inflow. It requires no additional parameters be provided

5.6 Model Calibration

Once a water distribution model has been developed, it must be calibrated so that it accurately represents the actual working real-life water distribution network under a variety of conditions. This involves making minor adjustments to the input data so that the model accurately simulates both the pressure and flow rates in the system. Note that both the pressure and flow rates need to be matched together, since pressure and flow rates are dependent upon each other. Therefore, matching only pressures or flow rates is not sufficient enough. Also, ideally the model should be calibrated over an extended period of time, such as a time range for the maximum day—not just the maximum and minimum hour for the maximum day.

This section discusses the concepts and steps involved in calibrating a water distribution model.

5.6.1 Accuracy Concerns

The computed pressure and measured field (actual) pressure will not exactly match for every node contained within the network system—there will be differences within the system. However, the maximum amount of these differences needs to be considered when performing a model calibration.



Generally there are three types of measurements used to determine the degree of accuracy reached when calibrating a model.

Pressure Differential Method

The simplest and easiest method in determining how accurate a simulated network model is to the actual network is to look at the maximum pressure differential between simulated and actual node pressures. For example, in a large system of several hundred or thousands of nodes, the model is generally considered calibrated if the pressure difference between the simulated and actual node pressures is within 5 to 10 psi of each other. For smaller network systems, with maybe a 100 or less nodes, this pressure differential would likely be considered too high.

Percent Pressure Differential Method

A more precise determination of accuracy is to look at the pressure differential as a percentage. For example, at a base pressure of 100 psi a 4 psi pressure differential between simulated and actual pressures represents a 4 percent differential, but at a base pressure of 40 psi this represents a 10 percent differential. Therefore, looking at the pressure differential as a percentage is a more precise measure of accuracy.

Head Loss Differential Method

The most precise determination of accuracy is to examine the difference in pipe head loss between simulated and actual. Head loss is more sensitive to calibration errors than pressure. As was discussed previously, it is better to look at the percent differential in head loss rather than the actual head loss values.

Although the head loss method is better than the other methods in determining accuracy, from a practical standpoint the pressure differential method and the percent pressure differential method are far easier to use and understand. Therefore, since the main point is to establish a common unit of measure to determine the degree of accuracy reached when calibrating a model, any of these methods can be used.

During the calibration process there should be a point where the modeler has decided that enough time has been spent on calibrating the model to the actual network and he should move on to analyzing the pipe network system.

5.6.2 Reasons for Calibrating a Model

Calibration is an important process that should be performed. A calibrated model establishes model credibility, sets a benchmark, can be used to predict potential problems, establish understanding of the system operation, and uncover errors with the system.



Model Credibility

Through the process of calibrating a model, credibility of the model is established. The data and modeling assumptions of an uncalibrated model are unlikely to match the actual system. On the other hand, a calibrated model is known to simulate a network system for a range of operating conditions. Its input data has been examined and adjusted to insure that the model can be used as an accurate, predictive tool. Hence, use of an uncalibrated model is not good engineering practice, since it will most likely lead to inaccurate model results and poor engineering decisions based upon these results.

Benchmark

Once a network model has been calibrated to a known range of operating conditions, it can be used as a benchmark. Pressure and flow rates computed by this model become the benchmark from which pressure and flow rates computed by subsequent, modified models can be compared. The differences between the two models can then be used to analyze the changes brought about in the modified system.

Predict Potential Problems

A calibrated model can be used to predict any potential problems due to changes in the system operation. A modified model, for example, might include additional valves or pumps to increase the number of pressure zones in a system. These changes can then be compared with the benchmark calibrated model to see what changes in pressure and flow rates have occurred.

Understand System Operations

In the process of calibrating a model, by collecting and analyzing data used to define the model and studying the existing network system, the engineer gains valuable insight and knowledge of the network system. The engineer needs to simulate many of the same system settings and operations that a network operator makes in order to calibrate the model to the actual operation of the network system. In addition, by analyzing the system operation, possible improvements to system operation may be determined.

Uncover Errors

Calibration requires collecting data about the system. Many times during the process of calibrating a model, questionable model results are investigated. Inconsistencies between the modeled results and the actual field conditions are examined, with additional field data being collected and analyzed. Incorrect pipe diameters may be determined, or even incorrectly closed valves discovered.



5.6.3 Calibration Model Data Requirements

The data requirements for performing a model calibration is made up with the fact that some data is fixed and unchanging (e.g., pipe diameter, length, etc.), some data is variable with time (e.g., demand patterns, pump rates, discharge pressures, reservoir elevations, etc.), and some data is assumed (e.g., pipe roughness values, consumption rates, etc.). Some data, such as consumption, is measured but sometimes also assumed.

Fixed data is easily obtained or measured. Variable data is more difficult to obtain and measure, but generally can be gathered from SCADA (Supervisory Control And Data Acquisition) system information or manual chart and graph systems. And, although it would be ideal to gather this information in order to perform an extended period simulation, it is extremely difficult to do for a large network system. This task is easily performed on a small system. But, for a large system, it may only be possible to gather this data for a single time period (i.e., maximum hour).

Simulation Type Considerations

Before collecting the data, it is important to decide whether a steady state or an extended period calibration simulation is to be performed. A steady state simulation is used to calibrate the maximum (peak) hour demand, whereas an extended period simulation is needed to calibrate a maximum day demand or when performing water quality modeling.

Complexity of the network system determines the degree of difficulty in calibrating the model. The more pipes, pumps, tanks, reservoirs, valves, pressure zones, and demand patterns there are, the more difficult the calibration process is and the less accurate the calibrated model will be.

The availability of data is also a limiting factor. For example, calibrating to the maximum hour requires data for only one time period. Calibration should also be performed for the minimum hour of the maximum day. Therefore, data would be required for two time periods. Extending the calibration process to an extended period simulation for the maximum day increases the data requirements to 24 time periods.

What can be done to simplify the calibration process is to approach the calibration process incrementally. To elaborate, first calibrate the model for the maximum hour. This will provide a good understanding of the network system, and can usually be accomplished with a reasonable degree of accuracy—even for large network systems. Then, the minimum hour can be calibrated. Knowledge learned in calibrating the maximum hour can be used to speed-up the calibration process for the minimum hour. Once both of these simulations have been calibrated, calibration of the maximum day extended period simulation can be performed if time permits and the data is readily available.



Data Acquisition

Calibration data should be compiled and organized so that it can be used in an efficient manner. In addition, a map of the network system is essential to properly calibrate the system. All pipes, pumps, valves, tanks, and reservoirs should be identified, including nodal elevations, pressure zone boundaries, and other important information.

SCADA data and graphs are useful since they contain data that will be referenced when developing the calibrated model. Even though the maximum hour is typically of primary interest, values for the entire day are also important since they provide an overall understanding of the operation of the network system. This information, for example, allows the user to compare the discharge pressure value at a particular location on the system during the day with the maximum hour value.

Operational Review

Operational rules must be determined for the network system for all major system components. Discussions with system operators should be performed to determine the operational rules and strategy behind how specific operations are performed. For example, under what conditions do system operators turn on a pump, close a control valve, or adjust a pressure-regulating valve? Is water pressure at particular key locations in the network act as a flag to turn on a particular pump? Are pumping schedules ever changed to try to minimize power costs? Are all facilities currently available for use, or are some facilities off-line for maintenance repair?

An operational review is essential in order to accurately calibrate the model and to make recommendations to improve system operation. In addition, by discussing and reviewing this information with system operators, a good working relationship is generally established between the modeler and the operators. And, the modeler and operators can teach each other about the system. The modeler can then use this information toward planning and improving the water distribution system so that the operators can then more easily manage the system.

Recalibration Frequency

Whenever changes occur in the water distribution network, such as operational changes, network configuration, or increases in water consumption, the degree of accuracy for the calibrated model is reduced. If these changes are severe enough, the model will need to be recalibrated. In practice this means that the model should be recalibrated whenever major new facilities are added to the network system, a new record for maximum hour is set, or operational procedures change significantly. Therefore, recalibration may be necessary every few years.



Many experts feel that calibration should be performed on an annual basis, regardless of what changes have occurred to the network system. Many times the primary reason to recalibrate the system is so that knowledge learned about the network can be used to improve the system operation. And, if the system changes are rather insignificant, then the recalibration process can be quickly and easily performed. By updating the model calibration annually, changes to the model are less extensive and are easier to implement in order to get the model back in tune with the actual network system.

5.6.4 Calibration Simulations

Once all of the necessary data has been gathered, the network model can be constructed. It is important that the network model accurately represents the physical layout of the system. Pipes and nodes must be accurately located in the model. Pipe roughness values should be estimated—based upon the age of the pipe.

Next, consumption values must be defined at the nodes. To save time initially, MIKE URBAN WATER provides a method of globally applying the total network demand to each of the individual network nodes using the Distributed Demand dialog box. See the section titled Distributed Demands on - 59 for more information on computing distributed demands. Also, operational data must be defined for all pumps, valves, storage tanks, and reservoirs. Finally, a network simulation can be performed with EPANET.

Initial Simulation

An initial simulation is performed to simply determine what the resulting pressures and flow rates are in the pipe network. This simulation may be simplified by using single operational values rather than complete operational data. At this stage of the network calibration, exact operational data is not required. For example, a reservoir elevation could be entered as rounded to the nearest whole number elevation, rather than to a tenth of a foot. Similarly, a single value pump definition could be used rather than a pump curve for each of the pumps defined in the network. As long as the simplified data is generally accurate, a balanced run can be produced so that the pipe network and operational input data can be checked.

Comparing Model Output

After the initial analysis output has been checked and verified, the modeled results can then be compared with actual measured field values. A table of actual flow rates and pressures should be prepared at key locations in the system so that comparisons with the computed analysis results can be quickly performed. Table 5.4 and Table 5.5 illustrate a comparison of flow rates and pressures between actual field measurements and computed values.



Table 5.4 Comparison of actual field flow rates and computed flow rates

Location	Actual Pipe ID	Actual Flow Rate (mgd)	Computed Pipe ID	Computed Flow Rate (mgd)	Actual Difference (mgd)	Percent Difference (%)
University Avenue @ Hilldale Shopping Center	346	32	597	34	+6	+2
East Johnson at State Capitol Building	1067	42	234	40	-2	-5
Middleton Heights Pump Station	2734	11	112	12	+1	+8



Table 5.5 Comparison of actual field pressures and computed pressures

Locations	Actual Node ID	Actual Pressure (psi)	Computed Node ID	Computed Pressure (psi)	Actual Difference (psi)	Percent Difference (%)
Pump station discharge pressure at 6300 University Avenue	2934	55	132	60	+5	+8
Pressure regulating valve downstream pressure at 6612 Mineral Point Road	1367	59	253	56	-3	-5
Control valve downstream pressure at State Capitol Building	4589	87	306	90	+3	+3

Note that the comparison process is best handled by comparing values on a broad scale, and then working downward to a more detailed examination of values on a more localized level. For example, examine the flow rates at the network supply points first. The computed and actual flow rates from these supply points should be compared. Next, flow rates and discharge pressures at pump stations should be examined. Continue this narrowing of scope as the calibration process progresses.

5.6.5 Model Adjustments

Once the differences between the computed and actual measured values can be determined, adjustments to the model data can be performed to make the



computed results match more closely to that of the actual data. When performing comparisons, differences in flow rates and pressures should be examined.

The computed flow rates and pressures may be lower or higher from the measured values. Table 5.6 allows you to quickly determine the cause for these differences—allowing you to adjust the model input to more closely match the measured values.

Table 5.6 Causes for differences between computed and measured flow rates and pressures

Network Parameter	Computed Flow Rate and Pressure Too Low	Computed Flow Rate and Pressure Too High
Total System Demand	High	Low
Pipe Roughness	Low	High
Pump Lifts	Low	High
Pressure Regulating Valve Settings	Low	High
Reservoir Elevation Settings	Low	High
Flow Control Valves	Not opened enough	Opened too much
Individual Nodal Demand	High	Low

In adjusting the model input data, there are four areas where adjustments can be considered. These input data adjustments are discussed in the following sections.

Operational Data Adjustments

Possible operational data adjustments to increase or decrease flow might include raising or lowering of reservoir elevations, or increasing or decreasing pump lifts. However, if the original operational data has been collected to a reasonable degree of accuracy, then major adjustments may not be valid. However, minor adjustments can be made to this data since there is always some degree of uncertainty.

Consumption Data Adjustments

Generally, a system-wide consumption value is known to reasonable accuracy due to pumping records. However, the distribution of this consumption may not be known to the same degree of accuracy. Therefore, a redistribution of nodal consumption may need to be performed.



Network Data Adjustments

There always exists the chance of input errors when defining a network model. For example, pipe diameters or lengths could be defined in error. Or, perhaps a pressure regulating valve was missed in the initial model definition. Therefore, it is important to check over the defined network input data to make certain errors were not made.

Pipe Roughness Adjustments

Pipe roughness values are typically estimated based upon material type and age. Sometimes pump tests are conducted to more accurately estimate pipe roughness, but this is typically for a single pipe. Therefore there exists a great deal of uncertainty in the assigned pipe roughness values. Therefore, pipe roughness values should be adjusted after the previously identified adjustments have been performed.

Number of Simulation Iterations

After performing these input data adjustments, a new computational analysis must be performed and the computed results compared again with the actual measured results. Numerous iterations may be necessary until the desired degree of accuracy is achieved.

For a small network system with fairly reliable input data, a calibrated model might be achieved within 10 or fewer iterations. For more complicated network systems, a calibrated model may require more than 100 iterations.



APPENDIX A

Literature

Abbott, M.B. and Basco, DE.R., "Computational Fluid Dynamics".



Abbott, M.B., "Hydroinformatics - Information Technology and Aquatic Environment".

Altman, T. and Boulos, P. F., Convergence of Newton Method in Nonlinear Network Analysis, *Journal Mathematical and Computer Modeling*, Vol. 21, No. 4, 1995, pp. 35-41.

Altman, T. and Boulos, P. F., On Solving Flow Constrained Networks: The Inverse Problem, *Journal Hydraulic Engineering ASCE*, Vol. 121, No. 5, 1995, pp. 427-431.

AWWA Manual M32, *Distribution Network Analysis for Water Utilities*, First Edition, American Water Works Association, Denver, CO, 1989.

Babbitt, H. E., *Plumbing*, Third Edition, McGraw-Hill, Inc., New York, NY, 1960.

Barlow, J. F., and Markland, E., Converting the Hazen-Williams Equation to the Colebrook Function, *Water Power & Dam Construction*, September, 1975, pp. 331-333.

Bhave, P. R., *Analysis of Flow in Water Distribution Networks*, Technomic Publishing Co., New York, NY, 1991.

Bhave, P. R., Calibrating Water Distribution Network Models, *Journal of Environmental Engineering*, Vol. 114, No. 1, February, 1988, pp. 120-136.

Bhave, P. R., Rules for Solvability of Pipe Networks, *Journal of Indian Water Works Association*, January-March, 1990, pp. 7-10.

Bhave, P. R., Unknown Pipe Characteristics in Hardy Cross Method of Network Analysis, *Journal of Indian Water Works Association*, April-June, 1990, pp. 133-135.

Bishop, M. M., Morgan, J. M., Cornwell, B., and Jamison, D. K., Improving the Disinfection Detention Time of a Water Plant Clearwell, *Journal of American Water Works Association*, Vol. 85, No. 3, 1993, pp. 68-75.

Boulos, P. F., Altman, T., and Liou, J. C. P., On the Solvability of Water Distribution Networks with Unknown Pipe Characteristics, *Journal Applied Mathematical Modelling*, Vol. 17, July, 1993, pp. 380-387.

Boulos, P. F., Altman, T., and Sadhal, K. S., Computer Modelling of Water Quality in Large Multiple Source Networks, *Journal Applied Mathematical Modelling*, Vol. 16, 1992, pp. 439-445.

Boulos, P. F., Altman, T., Bowcock, R. W., Dhingra, A. K., and Collevati, F., An Explicit Algorithm for Modeling Distribution System Water Quality with Applications, 2nd BHR International Conference on Water Pipeline Systems, D. S. Miller ed., Edinburgh, Scotland, 1994, pp. 405-423.



- Boulos, P. F., Altman, T., Jarrige, P. A., and Collevati, F., An Event-Driven Method for Modeling Contaminant Propagation in Water Networks, *Journal Applied Mathematical Modelling*, Vol. 18, 1994, pp. 84-92.
- Boulos, P. F., Altman, T., Jarrige, P. A., and Collevati, F., Discrete Simulation Approach for Network Water Quality Models, *Journal Water Resources Planning and Management ASCE*, Vol.121 No. 1, 1995, pp. 49-60.
- Boulos, P. F., and Altman, T, A Graph-Theoretic Approach to Explicit Nonlinear Pipe Network Optimization, *Journal Applied Mathematical Modelling*, Vol. 15, September, 1991, pp. 459-486.
- Boulos, P. F., and Altman, T., An Explicit Approach for Modelling Closed Pipes in Water Networks, *Journal Applied Mathematical Modelling*, Vol. 17, August, 1993, pp. 437-443.
- Boulos, P. F., and Altman, T., Explicit Calculation of Water Quality Parameters in Pipe Distribution Networks, *Journal Civil Engineering Systems*, Vol. 10, 1993, pp. 187-206.
- Boulos, P. F., and Ormsbee, L. E., Explicit Network Calibration for Multiple Loading Conditions, *Journal Civil Engineering Systems*., Vol. 8, 1991, pp. 153-160.
- Boulos, P. F., and Wood, D. J., An Explicit Algorithm for Calculating Operating Parameters for Water Networks, *Journal Civil Engineering System*, Vol. 8, 1991, pp. 115-122.
- Boulos, P. F., and Wood, D. J., Explicit Calculation of Pipe-Network Parameters, *Journal Hydraulic Engineering ASCE*, Vol. 116, No. 11, 1990, pp. 1329-1344.
- Boulos, P. F., Rossman, L. A., and Vasconcelos, J. J., A Comparison of Methods for Modeling Water Quality in Distribution Systems, *Proceedings of the AWWA 1994 Annual Conference and Exposition*, New York, NY, pp. 523-541.
- Cesario, A. L., *Modeling, Analysis, and Design of Water Distribution Systems*, American Water Works Association, Denver, Colorado, 1995.
- Clark, R. M. and Colye, J. A., Measuring and Modeling Variations in Distribution System Water Quality, *Journal of American Water Works Association*, Vol. 82, No. 8, 1990, pp. 46-53.
- Clark, R. M., Grayman, W. M., Goodrich, J. A., Deininger, R. A., and Hess, A. F., Field-Testing Distribution Water Quality Models, *Journal of American Water Works Association*, Vol. 83, No. 7, 1991, pp. 67-75.
- Clark, R. M., Smalley, G., Goodrich, J., Tull, R., Rossman, L. A., Vasconcelos, J. J., and Boulos, P. F., Managing Water Quality in Distribution Systems: Simulating TTHM and Chlorine Residual Propagation, *Journal Water SRT-AQUA*, Vol. 43, No. 4, 1994, pp. 182-191.



- Cole, G. B., Leak Detection -- Two Methods That Work, AWWA Seminar on Operational Techniques for Distribution Systems, Los Angeles, California, 1980.
- Cormen, T. H., Leiserson, C. E., and Rivest, R. L., Introduction to Algorithms, The MIT Press, Cambridge, MA, 1990.
- Cross, H., Analysis of Flow in Networks of Conduits or Conductors, Bulletin No. 286, University of Illinois Engineering Experimental Station, Urbana, Illinois, 1936.
- D. & L. George Ltd., Herts, England, 1991.
- Daugherty, R.L. and Franzini, J.B. and Finnemore, E.J., "Fluid Mechanics with Engineering Applications".
- Dennis, J. E. and Schnabel, R. B., Numerical Methods for Unconstrained Optimization and NonLinear Equations, Englewood Cliffs, New Jersey, Prentice-Hall, 1983.
- Duff, I. S., Erisman, A. M., and Reid, J. K., Direct Methods for Sparse Matrices, Oxford University Press, Inc., New York, NY, 1992.
- E. Cabrera and F. Martínez, "Water Supply Systems - State of the art and future trends".
- Edwards, D. K., Denny, V. E., and Mills, A. F., Transfer Processes, McGraw-Hill, New York, NY, 1976.
- Epp, R., and Fowler, A. G., Efficient Code for steady state Flows in Networks, Journal of the Hydraulics Division, Proceedings of the American Society of Civil Engineers, Vol. 96, No. HY1, January, 1970, pp. 43-56.
- Fawell, J. K., and Miller, D. G., UK Drinking Water: A European Comparison, Journal of the International Water and Environmental Management, Vol. 8, February, 1994, pp. 76-83.
- FEB Press, Ann Arbor, MI, U.S.A., 1967, 1983.
- Fletcher, R. and Reeves, C. M., Function Minimization by Conjugate Gradients, The Computer Journal, Vol. 7., July, 1964, pp. 149-154.
- Fletcher, R., Function Minimization without Evaluating Derivatives -A Review, The Computer Journal, Vol. 8., April, 1965, pp. 33-41.
- Fok, A.T.K, "Design Charts for Air Chamber on Pump Pipe Lines".
- George, A., and Liu, J., Computer Solution of Large Sparse Positive Definite Systems, Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1981.



Gofman, E. and Rodeh, M., Loop Equations with Unknown Pipe Characteristics, Journal of the Hydraulics Division, ASCE, Vol. 107, No. HY9, September, 1981, pp. 1047-1060.

Grayman, W. M., and Clark, R. M., Using Computer Models to Determine the Effect of Storage on Water Quality, Journal of American Water Works Association, Vol. 85, No. 7, 1993, pp. 67-77.

Grayman, W. M., Clark, R. M., and Males, R. M., Modeling Distribution-System water Quality Dynamic Approach, Journal of Water Resources Planning and Management, Vol. 114, No. 3, May, 1988, pp. 295-311.

Gupta, R. S., Hydrology and Hydraulic Systems, Prentice Hall, 1989.

Hamam, Y. M., and Brameller, A., Hybrid Method for the Solution of Piping Networks, Proceedings IEE, Vol. 118, No. 11, 1971, pp. 1607-1612.

Housing and Home Finance Agency, The Uniform Plumbing Code for Housing, Section 8.3.5, February, 1948.

Chaudhry, M.H., "Applied Hydraulics Transients".

Chow, V. T., Maidmant, D. R., and Mays, L. W., Applied Hydrology, McGraw-Hill, Inc., New York, NY, 1988.

International Conference on Computer Application for Water Supply and Distribution, Leicester Polytechnic, UK, September 8-10, 1987.

Jeppson, R. W., Analysis of Flow in Pipe Networks, 5th Printing, Ann Arbor Science, Ann Arbor, Michigan, 1981.

John Wiley and Sons, New York, 1989.

Journal of the Hydraulic Division, Vol. 104, No. HY9, September, 1978, pp. 1289-1303.

Journal of the Hydraulic Division, Vol. 107, No. HY10, October, 1981, pp. 1145-1161.

Lansey, K. E. and Mays, L. W., Optimization Model for Water Distribution System Design, Journal of the Hydraulic Engineering, Vol. 115, No. 10, October, 1989, pp. 1401-1418.

Lemieux, P. F., Efficient Algorithms for Distribution Networks, Journal of the Hydraulics Division, ASCE, Vol. 98, No. HY11, November, 1972, pp. 43-56.

Lindeburg, M. R., Civil Engineering Reference Manual, Sixth Edition, Professional Publications, Inc., Belmont, CA, 1992.

Longman, Essex, 1989.



Male, J. W., and Walski, T. M., *Water Distribution Systems: A Troubleshooting Manual*, Lewis Publishers, Inc., 1991.

Nayyar, M. L., *Piping Handbook*, Sixth Edition, McGraw-Hill, Inc., New York, NY, 1992.

Nielsen, H. B., *Methods for Analyzing Pipe Networks*, *Journal of the Hydraulic Engineering*, Vol. 115, February, 1989, pp. 139-157.

Nielsen, L. S., *Standard Plumbing Engineering Design*, McGraw-Hill, Inc., New York, NY, 1963.

Ohtmer, O., *NonLinear Flow Analysis in Pipe Networks*, *International Journal for Numerical Methods in Engineering*, Vol. 19, 1983, pp. 373-392.

Ormsbee, L. E. and Wood, D. J., *Hydraulic Design Algorithms for Pipe Networks*, *Journal of Hydraulic Engineering*, Vol. 112, No. 12, December, 1986, pp. 1195-1207.

Ortega, J. M. and Rheinbolt, W. C., *Iterative Solution of Non-Linear Equations in Several Variables*, Academic Press, New York, NY, 1970.

Osiadacz, A. J., *Simulation and Analysis of Gas Networks*, E & F Spon Ltd., London, UK, 1991.

Osiadacz, A. J., *Simulation and Analysis of Gas Networks*, Gulf Publishing Company, Book Division, Houston, TX, 1987.

Pierre, D. A., *Optimization Theory with Application*, Wiley, New York, NY, 1969.

Pitman, London, 1991.

Powell, M. J. D., *An Efficient Method for Finding the Minimum of a Function of Several Variables without Calculating Derivatives*, *The Computer Journal*, Vol. 7, July, 1964, pp. 155-162.

Press, W. H., Teukolsky, S. A., Vetterling, W. T., and Flannery, B. P., *Numerical Recipes in C*, Second Edition, Cambridge University, New York, NY, 1992.

Proceedings of the 1st International Conference on Hydroinformatics, Delft, The Netherlands, September 1994.

Proceedings of the Conference on Hydraulics in Civil Engineering, Brisbane, 1994.

Proceedings of the XXV th IAHR Conference, Tokyo, Japan, August 1993.

Richardson, A. K., *Estimating Water Storage Requirements*, Lectures presented at *Water Distribution System Analysis and Design Workshops* at the University of Wisconsin- Extension, University of Wisconsin-Madison CEE



325 class handout, summer semester, 1991, Alan K. Richardson is with Black & Veatch Consulting Engineers at Kansas City, Missouri.

Rossman, L. A., and Boulos, P. F., Numerical Methods for Modeling Water Quality in Distribution Systems: A Comparison, *Journal Water Resources Planning and Management ASCE*, Vol. 122, No. 2, 1996, pp 137-146.

Rossman, L. A., Boulos, P. F., and Altman, T., Discrete Volume-Element Method for Network Water-Quality Models, *Journal Water Resources Planning and Management ASCE*, Vol. 119, No. 5, September/October, 1993, pp. 505-517.

Rossman, L. A., Clark, R. M., and Grayman, W. M., Modeling Chlorine Residuals in Drinking-Water Distribution Systems, *Journal Environmental Engineering ASCE*, Vol. 120, No. 4, 1994, pp. 803-820.

Rossman, L. A., EPANET -An Advanced Water Quality Modeling Package for Distribution System, *Proceedings of American Water Works Association 1993 Annual Conference*.

Rossman, L. A., EPANET Users Manual, EPA-600/R-94/057, Environmental Protection Agency, Risk Reduction Engineering Laboratory, Cincinnati, Ohio, 1994.

Second ed., Van Nostrand Reinhold, New York, NY, 1987.

Shames, I. H., *Mechanics of Fluids*, Third Edition, McGraw-Hill, Inc., New York, NY, 1992.

Shamir, U., and Howard, D. D., Water Distribution Systems Analysis, *Journal of the Hydraulic Division, ASCE*, Vol. 94, No. HY1, January, 1968, pp. 219-234.

Schweitzer, P. A., *Handbook of Valves*, Industrial Press, Inc., New York, NY, 1972.

SI Metric Edition, Mc-Graw Hill Book Company, Singapore, 1989.

Streeter, V. L., and Wylie, E. B., *Fluid Mechanics*, McGraw-Hill, Inc., Seventh Edition, 1979.

Thorley, A.R.D., "Fluid Transients in Pipeline Systems".

Todini, E. and Pilati, S., "A Gradient Method for the Analysis of Pipe Networks".

Todini, E., and Pilati, S., A Gradient Method for the Analysis of Pipe Networks, *Proceeding of the International Conference on Computer Applications for Water Supply and Distribution*, Leicester Polytechnic, UK, September 8-10, 1987.



Travers, K., The Mesh Method in Gas Network Analysis, Gas Journal, November, 1967, pp. 167-177.

Tullis, J.P., "Hydraulics of Pipelines".

University of Kentucky, Lexington, KY, 1980.

Valencia, 19-23 October 1992, Computational Mechanics Publications, Southampton, ISBN 1-85312-253-X.

Vanicek, S. and Verwey, A. and Abbott, M., "HYPRESS: An Exercise in Object-Orientation for Water Hammer and Water Distribution Simulation in Pipe Networks",

Verwey, A. and Yu, J.H., "A Space-Compact High-Order Scheme for Water Hammer Simulations".

Verwey, A., "Linkage of Physical and Numerical Aspects of Models Applied in Environmental Studies",

Walski, T. M., Analysis of Water Distribution Systems, Van Nostrand Reinhold Company, Inc., New York, NY, 1984.

Walski, T. M., Gessler, J., and Sjostrom, J. W., Water Distribution Systems: Simulation and Sizing, Lewis Publishers, Inc., 1990.

Warga, J., Determination of steady state Flows and Currents in a Network, Proceedings of Instrument Society of America, Vol. 9., Pt. 5, September, 1954, Paper No. 54-43-4.

Warring, R. H., Handbook of Valves, Piping and Pipelines, First Edition, Trade & Technical Press Ltd., England, 1982.

Watters, G.Z., "Analysis and Control of Unsteady Flow in Pipe Networks Including Extended Period Simulations".

Webler, T., and Brown, H. S., Exposure to Tetrachloroethylene via Contaminated Drinking Water Pipes in Massachusetts: A Predictive Model, Vol. 48, No. 5, September/October, 1993, pp. 293-297.

Wood, D. J., 1991, Comprehensive Computer Modeling of Pipe Distribution Networks, KYPIPE Manual, Civil Engineering Software Center, College of Engineering, University of Kentucky, Lexington, KY, 1991.



INDEX

**Numerics**

1xQ-H Point Pump Curve	27
3xQ-H Point Pump Curve	27

A

Accumulated Street Demands-Polygons	123
Accuracy	50
Accuracy Concerns	243
Additional Components	196
Advective Transport in Pipes	234
Aggregate Demands	68
Aggregation	68
Air-Chamber	197
Air-Chamber Editor	44
Air-CHamber ID	45
Air-Valve	16, 201
Analysis	125
Analysis Duration	87
Analysis Methods	206
Animation	132
Architecture of RTC Systems	145
Asset ID	12, 19, 26, 35, 40
Atmospheric Pressure	51
Available Operational Controls	77

B

Base Elevation	42, 47
Benchmark	245
Boundary Conditions	58, 182
Boundary Curve	59
Boundary Type	59
Branches	184
Browser	130
Browsing Results	138
Bulk Flow Reactions	235
Bulk Reaction Order	96

C

Calculation	171
Calibration Model Data Requirements	246
Calibration Simulations	248
Category	71, 84
Category Type	72
CHeck Valve	22
Check Valves	38
CHemical Concentrations	90
Closed	23, 29, 37

Comparing Alternative Solutions	128
Completely Mixed	41
Computational Grid	185
Computational Parameters	182
Concentration	93
Concentration Source	94
Condition	54, 75
Constant Power	27
Construction Year	20
Consumption Data	115, 251
Control Editor	74
Control Level	76
Control Node	32
Control Pressure	32
Control Rules	185
Control Settings	151
Control Type	32
Control Valve	145
Controlling Pumps	217
Controls	73
Convergence	50
Counted Lots	123
Critical Node Pressure	137
Curve and Relations Editor	56
Cyclic Patterns	86

D

Darcy-Weisbach	139
Data Acquisition	247
Data Source	13, 19, 26, 35, 41
Defining Curved Pipes	24
Defining the Model	111
Definition of Network Layout	183
Demand	52, 65, 70, 71, 72, 129
Demand Allocation	64, 124
Demand Category	15, 66, 70
Demand Coefficient	14, 21, 70
Demand Description	15
Demand Editing	124
Demand Pattern ID	15, 70
Demand Processing	119
Demand Scenarios	124
Demand Statistics	71
Design	208
Design Fire Flow	137
Design Fire Pressure	136
Design Flow	27
Design Head	27
Developing Pipe Demand Coefficients	122



Diagrams 175
Diameter 20, 37, 43, 47
Distributed Demands 59, 119
Diurnal Curves and Patterns 88
Diurnal Pattern 86
Diurnal Profiles 83
Dual-Acting Valve Characteristics Curve 17
Dual-Acting Valve Characteristics 56

E

Effect of Pressure Reducing Valves . . . 129
Efficiency 53
Efficiency Curve 30
Elevation 14
Emitter 16, 114
Emitter Exponent 50
Energy 53, 54
Energy Editor 53
Energy Price 30
Energy Price Pattern 30
Engineering Tables 98
Entering Data 116
EPANET 108, 205
EPANET Analysis Results 129
EPANET Computer Program 205
EPANET Input Data File 205
Equivalent Person EP 66
Equivalent Tenement ET 66
Estate Height 14
Evolutionary Algorithms 139
Executing the Analysis 125
Extended Period Data 115
Extended Period Hydraulics 49, 207
Extended Period Simulations 73
Extended Period Water Quality 49
Extended Rule-Based Controls . . . 110, 157

F

FCV 221
FIFO 41, 42
Fire Flow Analysis 109, 135
Fire Flow Data 116
Fixed HGL 42
Flow Coefficient 16
Flow Control Valve 36, 115, 145, 221
Flow Direction 23, 33, 38
Flow Paced Booster Source 94
Forward Tracking 100

G

General Purpose Valve 36, 115, 221
Genetic Algorithms Calibration 143
Geocoding 66
Getting Started 111
Global Bulk Reaction Rate Coefficient . . 95
Global Editing 125
Global Pipe Wall Reaction Rate Coefficient 95
GPV 221
Graph 85
Graphical Input 117
Graphical Placement 17, 23, 33, 38, 44, 47
Gravity 51
Grid Points 185
Groundwater Wells 223
Group Definition 142

H

Hazen-Williams 139
Head Loss Differential Method 244
Headlosses 49
Hiding and Showing Graphical Data . . 119
High End Flow 28
High End Head 28
History 206
Hydraulic Simulation Model 226
Hydraulic Structures 180, 185
Hydraulic Time Step 87

I

Importing Graphical Data 118
Inactive Volume 43
Initial Conditions 182
Initial Level 43, 47
Initial Simulation 248
Initial Water Quality 91
Input Data Requirements 113
Interactive Data Entry 116
Introduction 141

J

Junction 12
Junction Node Data 114
Junction Node Demands 185
Junction Node Pressure Based Controls 77



L	
Lagrangian Transport Algorithm	238
Length	20, 43, 47
LIFO	41, 42
Link Type	75
Links	55
Looped Network Solution Algorithm	178
Loss Coefficient	21, 37
M	
Manning	139
Mass Booster Source	94
Material	20
Maximum Flow	28
Maximum Level	43, 47
Messages	54
Method of Pipe Lengths 21, 22, 59, 61, 120	
Method of Reduced Pipe Lengths .61, 120	
Method of Two Coefficients .21, 22, 60, 61, 120	
MIKE NET	107
Minimum Level	43, 47
Minor Losses	221
Mixing	234, 235
Molecular Diffusivity	50, 91
Moment of Inertia	31
Motor Torque	31, 56
Multiple Controls	76
Multiple Demand	15, 69
Multiple Demand Editor	48
Multipliers	85
N	
Network	210
Network Component Editor	11
Network Data	113, 252
Network Demand	59
Node Demand Coefficients	62
Node Demand Multiplier	137
Node Pressure Zone ID	60, 62
Node Type	58, 93, 98
Nodes	184, 222
Number of Simulation Iterations	252
Numerical Scheme and Algorithm	176
O	
Operation	208
Operational Data Adjustments	251
Operational Review	247
Operational Schedule	31, 37
Operational Schedule Type	30
Owner	66
P	
Parallel Pipes	213
Pattern	53
Pattern Editor	83
Pattern ID	84
Pattern Time Step	87
Patterns	73
PBV	221
PID	56, 147
PID Control	109, 145
Pipe Data	114
Pipe Demand Coefficients	59, 120
Pipe Diameter	68
Pipe Editor	18
Pipe Flow Module	108
Pipe Groupings	142
Pipe Reaction Rate Assignments	96
Pipe Roughness	129, 252
Pipe Roughness Calibration	109, 139
Pipe Wall Reactions	237
Pipes	211
Pipes with Check Valves	77
Planning	208
Point Source Data	115
Polytropic Expansion	46
Pressure and Flow Measurements	143
Pressure Breaker Valve 36, 115, 145, 221	
Pressure Differential Method	244
Pressure Normal	21
Pressure Reducing Valve 36, 115, 145, 220	
Pressure Sustaining Valve . .36, 115, 145, 221	
Pressure Zone	71
Pressure Zone Editor	52
Pressure Zone ID 13, 20, 27, 36, 41, 46, 52,	65
Price	53
Profile Plots	131
Project Data	113
Project Options	48
Project Specification Data	113
Project Type	49
Properties	49
PRV	220



PSV	221
Pump	114, 145, 190
Pump Editor	25
Pump Efficiency	56
Pump Energy Settings	29
Pump ID	26
Pump Operational Schedule	56
Pump Q-H Curve	56
Pump Speed	32
Pump Start-Up Time	31
Pump State	29
Pump Torque	31, 56
Pump Trip-Off Time	31
Pump Type	27
Pump Variable Speed	31
Pumping Rate Limits	216
Pumping Scenarios	129
Pumps	214
Q	
Q-H Pump Curve	28
Quality	92
Quality Menu	111
Quality Time Step	88
R	
Reaction Rate Editor	95
Recalibration Frequency	247
Redistribute	72, 85
Reference ID	64
Refresh	72
Reinforced Concrete Pipe	174
Relative Speed	29
Report Critical Node	137
Report Editor	53
Report Start Time	88
Report Time Step	88
Reservoir Data	114
Rigid Conduit	173
Rotational Pump Speed	31
Roughness	21
Roughness Coefficient	113
Roughness Correlation	96
RTC	145, 147
Rule Based Controls	78
Running Water Hammer Module	182
S	
Save	76
Scenario Manager	98
Select Time Level	138
Send Data	98
Set Point Settings	152
Setpoint Booster Source	94
Setting	36, 75
Shutoff Head	27
Simple Control	74
Simulate Fire Hydrant	137
Simulation Menu	111
Simulation Type Considerations	246
Skeletonization	208
Snap Tolerance	24
Source Tracing	90, 97, 233
Source Type	94
Specific Curves Data	195
Specific Gravity	49, 91
Specific Pipe Data	185
Specific Project Options Settings	194
Specific Pump Data	185
Specific Time Settings	194
Start Clock Time	88
State	14, 22
Statistics	88
Status	13, 20, 26, 35, 41, 46, 54, 73
Steady State Hydraulics	49, 206
Storage Tank	77, 114, 225
Street Name	22
Summary	54
Supported Components	196
Surface Elevation	14
T	
Tabular Results	138
Tank Depth-Volume Curve	56
Tank Editor	39
Tank Mixing	41
Tank Mixing Models	240
Tank Size	43, 47
Tank Type	41
Tanks	197
Target	69
Target Demand	62, 63
TCV	221
Temperature	52
Thematic Mapping	130
Thematic Maps	138
Theoretical Background	172
Theta	51



Thick-Walled Elastic Conduit	173
Thin-Walled Elastic Conduit	173
Throttle Control Valve	36, 115, 145, 221
Time	87
Time Based Controls	77
Time Editor	87
Time Patterns	94, 225
Time Series Plot	131
Time Step	84
Time Units	88
Times	73
Title	49
Tools	98
Tools Menu	111
Total Network Water Demand	60, 62
Tracking	100
Transient Flow Analysis	49
Transient H-Boundary	56
Transient Q-Boundary	56
Tunnels Through Solid Rock	174
Tutorial	202
Two-Compartment mixing	42
Type	14, 71
U	
Unbalanced System	50
Uncover Errors	245
Understand System Operations	245
Units	49
Unsupported Component	196
V	
Valve Characteristics Cd	56
Valve Characteristics Kv	56
Valve Curve	37
Valve Data	115, 191
Valve Diameter	17
Valve Editor	34
Valve Head Loss	56
Valve ID	35
Valve Operation Schedule	56
Valve State	37
Valve Type	36
Valve Water Hammer	37
Valve Water Hammer Settings	37
Valves	220
Vapour Cavity Pressure	52
Variable	55
Variable Pumps	32
Variable Speed Drive	110
Variable Speed Drive Pump	157
Variable Valves	38
Vented Air Chambers	199
View Menu	111
Viewing Analysis Results	127
Viscosity	49, 91
Volume Curve	43, 47
VSD pump	157
W	
Wall Reaction Order	96
Wall Speed	20
Wall Thickness	20
Warning	23
Water Age	90, 233
Water Hammer	51, 109, 171
Water Hammer Calculation	171, 181
Water Hammer Data	116
Water Hammer Data Preparation	181
Water Hammer Model	172
Water Hammer Result Presentation	181
Water Quality Analysis	89, 90, 207
Water Quality Data	115
Water Quality Module	108
Water Quality Parameters	129
Water Quality Simulation	89
Water Quality Simulation Model	234
Water Quality Tolerance	50
Water Source Price	56
Wave Speed	172
WD-Tools	1, 81, 109, 135
Width	43, 47
X	
X and Y Coordinate	13, 41, 46, 65