

MIKE URBAN Model Manager

User Guide





PLEASE NOTE

COPYRIGHT

This document refers to proprietary computer software which is protected by copyright. All rights are reserved. Copying or other reproduction of this manual or the related programs is prohibited without prior written consent of DHI. 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.





MIKE URBAN Model Manager	User Guide	15
1	Welcome to MIKE URBAN	17
1.1	Model Manager	18
1.1.1	EPANET	18
	Pipe Flow	18
	Water Quality	19
1.1.2	SWMM	19
1.2	Water Distribution	19
1.2.1	Water Distribution Tools	19
1.2.2	Water Hammer	21
1.3	Collection Systems	21
1.3.1	CS-Pipeflow	21
1.3.2	CS-Rainfall-Runoff	22
1.3.3	CS-Control	23
1.3.4	CS-Pollution Transport	23
1.3.5	CS-Biological Processes	25
2	Getting Started	27
2.1	How to start MIKE URBAN	27
	Desktop Workspace	27
	Map Windows	27
	Floating Toolbars	28
	Identify	28
2.2	Online Help	28
2.3	MIKE URBAN Examples	29
2.4	The Table Of Contents	30
	Working Modes	31
2.4.1	Boundary conditions displayed on the map	32
	MOUSE	32
	SWMM	33
	EPANET	34
2.4.2	Properties dialog	34
	The General tab	35
	The Selection tab	35
	The Labels tab	36
	The Symbology tab	37
2.5	The main menu	37
	File menu	38
	Edit menu	38
	Asset menu	38
	MOUSE menu (CS)	39
	SWMM 5 menu (CS)	39
	Model menu (WD)/EPANET menu	39
	Simulation menu	39
	Model Results menu	39
	View menu	39
	Tools menu	39



	Window menu39
	Help menu39
2.6	The toolbars	40
2.7	Common Editor Features	40
	Select, Select, All, Unselect All41
	Show All, Show Selected41
	Copy to Clipboard and Paste from Clipboard41
	Report to XML42
	Synchronize dialog/Default layout42
2.8	Languages	42
2.9	Selecting a Coordinate System	43
2.10	Starting a New Project	44
2.11	Tools in MIKE URBAN	47
	2.11.1 Selection tools	47
	Selection Methods47
	New Selection47
	Add to Selection47
	Remove from Current Selection47
	Select from Current Selection47
	Select Features47
	Select by Polygon47
	Select by Graphics47
	Select by Attributes48
	Select Selectable Layers48
	Clear Selected Features48
	Select Dead-End Nodes and Select Dead-End Pipes48
	Load Selection48
	Save Selection48
	Merge selected pipes - WD mode only48
	2.11.2 Tracing	49
	Find Circular Connection49
	Network Connectivity50
	Forward Flow Tracking50
	Backward Flow Tracking50
	2.11.3 General SQL command	51
	2.11.4 Rebuild network	53
2.12	Various WD Tools	54
	2.12.1 Duplicate selected pipe attributes	54
2.13	Saving Animations to AVI Files	56
2.14	Export of Maps	57
2.15	Images in MIKE URBAN	57
	2.15.1 World files	57
	2.15.2 World file naming conventions	58
	2.15.3 How georeferencing information is accessed	58
	2.15.4 World file contents	58
2.16	3D Viewer	59
2.17	Overview Mode	62



3	Customizing MIKE URBAN	65
3.1	Units, Default Values and Numeric Formats	65
3.1.1	Selecting an appropriate unit environment	65
3.1.2	Customizing unit environment	67
3.1.3	Saving and Retrieving User Defined Unit Environment	68
3.2	Languages	69
3.3	General Settings	69
3.4	Toolbars	70
3.5	User Defined Tools	72
3.6	Menus	73
3.7	Symbols	73
3.8	User Written Extensions	74
3.8.1	Developing extensions	75
3.8.2	Interfaces	75
	ICommand	75
	ITool	76
	ImuShellTool	78
3.8.3	Example	80
	Creating the COM library	80
	Implementing the ICommand & ITool	81
	Registering the myTool with MIKE URBAN	83
	Using the tool	84
4	Internet Links and updates	85
5	Linking to ArcMap	87
5.1	Typical ArcMap Tasks	88
5.1.1	Example	88
6	MIKE URBAN data model	93
6.1	MIKE URBAN Networks	93
6.2	Data Model Structure	95
6.2.1	Terminology	96
	Geodatabase	96
	The project (MUP) file	96
	Configuration and import bridges	96
	Time series and result data	96
6.2.2	Geodatabase basics	96
6.2.3	The Geodatabase and the Scenario Manager	97
6.2.4	The MIKE URBAN database contents	97
	Naming Convention	97
7	Import and Export	99
7.1	Introduction to MIKE URBAN Import/Export	99
7.2	Technical Background for Import and Export Functionality	99
7.2.1	General Description	100
7.2.2	MIKE URBAN Storages	101
7.2.3	MIKE URBAN Bridges	102



7.2.4	Assignments	103
	Simple Assignments	103
	Advanced Assignments	104
7.2.5	MIKE URBAN Transfer Modes	117
	'Replace existing data'	117
	'Only append to existing data'	117
	'Append and update existing data'	118
	'Only update existing data'	118
	Using transfer modes	118
	Examples of different transfer modes for jobs	119
	Examples of different transfer modes for table configurations	126
7.3	Import/Export - Wizard	133
7.3.1	Overall Description	133
7.3.2	"Use Wizard" option	135
7.3.3	Job loop (outer loop)	136
7.3.4	Table configuration loop (inner loop)	138
	Selection of source, target and transfer mode	139
	Assignment specification	140
7.3.5	Activation/Deactivation of jobs and table configurations	142
7.3.6	Final wizard page	143
7.4	Import/Export - Advanced dialog	143
7.4.1	Overall Description	143
7.4.2	The 'Action Button'	144
7.4.3	Job Local Menu	145
	Expand/Collapse	146
	Cache	146
	Create New Job	147
	Copy Job	147
	Create New Table Configuration	147
	Delete Job	147
7.4.4	Table Configuration Local Menu	147
	Table Configuration Tools	147
7.4.5	Transfer Modes	148
7.4.6	Pre-defined Import and Export Jobs, Backup/Restore Jobs and Connectors 149	
	Pre-defined Import and Export Jobs	149
	Pre-defined Backup and Restore Jobs	150
	Connectors	150
7.4.7	Detailed Description of Import/Export Job Properties	151
	Basic	152
	Source	152
	Target	155
	Bridge	156
	Topology	157
7.4.8	Detailed Description of Table Configurations Properties	159
	Basic	160
	Assignment	162
	Source	163



	Target	164
7.4.9	Defining Custom Import/Export Jobs	165
	Creating a New Job	165
	Creating New Table Configurations	165
8	Flagging	167
8.1	Introduction to the MIKE URBAN Flagging System	167
	8.1.1 What are flags?	167
	8.1.2 What can be flagged?	167
8.2	Defining Flag Values	167
8.3	Setting the Flags	168
8.4	Using the Flags	171
9	Editing Tools	173
9.1	Overview	173
9.2	Graphical Editing	173
	9.2.1 Geometric Network Selection	175
	9.2.2 Toolbars	176
	9.2.3 Keyboard Shortcuts	178
10	Modeling Collection Systems	181
10.1	Getting Started on a Collection System Project	181
	10.1.1 Entering data, Edit mode	182
	10.1.2 Activating menus to display	183
	10.1.3 Project information	183
10.2	Collection System (CS) Asset Network	184
	10.2.1 CS Asset Structures Editor	185
	10.2.2 Asset Pipes and Canals Editor	186
	10.2.3 CS Asset Weirs and Orifices Editor	187
	10.2.4 CS Asset Pumps Editor	188
	10.2.5 CS Asset Link types Editor	189
	10.2.6 CS Asset Node types Editor	190
	10.2.7 CS Asset CRS & Topography	191
	10.2.8 CS Asset Curves & Relations	192
	10.2.9 CS Asset Materials	193
10.3	Transferring data between 'CS Asset', 'MOUSE' and 'SWMM5'	194
10.4	SWMM 5	195
11	Modeling Water Distribution Systems	199
11.1	Getting Started on a Water Distribution System	199
	11.1.1 Entering data, Edit mode	200
	11.1.2 Activating menus to display	200
	11.1.3 Project information	201
11.2	Water Distribution (WD) Asset	201
	11.2.1 WD Asset Pipe Editor	202
	11.2.2 WD Asset Service Pipe Editor	208
	11.2.3 WD Asset Hydrants Editor	210
	11.2.4 WD Asset Valves Editor	212
	11.2.5 WD Asset Pumping Stations Editor	214



11.2.6	WD Asset Water Tanks Editor	215
11.3	Transferring data between 'Asset' and 'Model'	217
12	Common Editors for CS and WD Asset Data	219
12.1	Asset Structures and Buildings Editor	219
12.2	Asset Conduits Editor	221
12.3	Asset Owners and Operators Editors	223
12.4	Asset Systems, Sub-Systems and Streets/Sites	224
13	Catchments and Catchment Tools	229
13.1	MIKE URBAN Catchments	229
13.2	Management of MIKE URBAN Catchments	229
13.3	The Catchment Editor	229
13.3.1	The Editor Fields	230
13.3.2	Calculated vs. User Specified Values	232
13.3.3	Tools for Graphical Catchment Editing	232
13.3.4	Create Catchment Feature	232
13.3.5	Edit Catchment Feature	233
13.3.6	Move Catchment	234
13.3.7	Split Catchment	234
13.3.8	Append Catchment	235
13.3.9	Merge Catchments	236
13.4	Connecting catchments to the drainage/wastewater collection network	236
13.4.1	MOUSE/SWMM Catchment Connection Editor	237
13.5	Graphical Tools for Connecting Catchments to MOUSE and SWMM5 Networks	238
13.5.1	Catchment Dialog	238
13.5.2	Find Catchment Overlays/Gaps	238
13.5.3	Connect to Node	239
13.5.4	Show All Connected Catchments	239
13.5.5	Show All Disconnected Catchments	240
13.5.6	Show Nodes Connected To Catchments/Catchments Connected To Nodes	240
13.6	Automated Catchment Tools	241
13.6.1	The catchment delineation wizard	242
	Catchments generated as Thiessen polygons	245
	Catchments boundaries based on DEM method	245
13.6.2	The catchment connection wizard	246
13.6.3	The catchment parameter processing wizard	249
13.6.4	Catchment Slope and Length Tool	252
14	Load allocation through geo-coding	255
14.1	Management of Point Loads	255
14.2	The Load Allocation Editor	256
14.2.1	The Editor Fields	257
14.3	Importing Load Points	258
14.3.1	Importing Load Points From MIKE URBAN Water Distribution	258
14.3.2	Importing Load Points From External Sources	259



14.4	Graphical Editing of Load Points	259
14.4.1	Create a Load Point	259
14.4.2	Move Load Point	260
14.4.3	Delete Selected Load Point	260
14.5	Allocating the load points to the Model Network	260
14.5.1	Manual Load Point Allocation	260
14.5.2	Graphical Load Point Allocation	261
14.5.3	Automatic Load Points Allocations by GIS Geo-Coding	262
14.6	Transfer of Data to Model and Load Aggregation	264
15	Interpolation and Assignment Tool	267
15.1	Introduction	267
15.2	Select Target for the Assignment	268
15.3	Selection of Method and Data Source	269
15.4	Network Assignment Options	270
15.5	Network Interpolation Parameters	272
15.6	Overall Selection Options	272
15.7	Finishing the Wizard	273
16	Simplification Tool	275
16.1	Introduction	275
16.2	Launching the Tool	275
16.3	Simplification Procedure - Wizard	276
16.3.1	Selection of MIKE URBAN model network	276
16.3.2	Selection of simplification method	277
	Scrubbing	278
	Trimming	278
	Merging	278
16.3.3	Selection of model area of interest	278
16.3.4	Specification of criteria for elements to be selected	280
	Scrubbing	280
	Trimming	282
	Merging	282
16.3.5	Specification of merge methods - valid only for Merging	285
16.3.6	Reconnection of connected Catchments and Demands	287
16.3.7	Save Options	288
16.3.8	Manually editing of selection	289
16.3.9	Multiple Configurations	290
17	Scenario management	293
17.1	What is a Scenario Manager	293
17.2	Design of the MIKE URBAN scenario manager	294
17.2.1	Data Groups, Alternatives and Scenarios	294
17.2.2	Alternatives	295
	Collection System Alternatives	295
	Water Distribution Alternatives	296
17.2.3	Base data vs. child data	297
17.2.4	Inheritance principles	298



17.2.5	Data not specific to any alternative/scenario	298
17.3	Managing Scenarios and Alternatives	299
17.3.1	Scenarios	299
17.3.2	Alternatives	300
17.3.3	Example	302
17.3.4	Reporting of the changes	302
17.4	How to Start Working with Scenarios?	304
17.4.1	Creating alternatives and scenarios	304
18	Climate Change Scenarios	307
18.1	Introduction	307
18.2	Model Location	308
18.3	Choosing the Climate Model and CO2 Emission Scenario	308
18.4	Delta Change Values	310
18.5	Creating Climate Change Scenarios	311
19	Project Check Tool	313
19.1	Introduction	313
19.2	Launching the Tool	313
19.3	Running the Check	314
19.3.1	Computed values and topology	314
19.4	Using the Error Log	314
19.4.1	Additional functionality	316
19.5	Typical Use Cases	317
20	Submodel Manager	319
20.1	Introduction	319
20.2	Create Selection File	320
20.3	Split Submodels	321
20.4	Merge Submodels	323
20.5	Model Requirements	324
21	Model Configuration Tool	325
21.1	Introduction	325
21.2	Launching the Tool	325
21.3	New Model Configuration	326
21.4	Apply Model Configuration	328
21.5	Create Layer File	331
22	CS Toolbox	333
22.1	Introduction	333
22.2	Cross Section Generation Tool	334
22.3	Lateral Snapping Tool	336
22.4	Auto Connection Tool	338
22.5	Sequential Labeling Tool	340
23	Presenting results	343
23.1	Loading results	343



23.2	Displaying results on the horizontal plan	344
23.3	Results browser	345
23.4	Results manager	346
23.5	Derived results	347
	MOUSE results	347
	SWMM Results	348
	EPANET Results	349
23.6	Result statistics	349
23.7	Result conversion	350
23.8	Labelling and symbology	351
	General	352
	Label Appearance	352
	Graduated Color	352
	Flow Arrows	353
23.9	Thematic maps and statistics	354
23.10	Time series	359
23.11	Results table	359
23.12	Reports	360
	23.12.1 Setting up a report	360
	Join of tables and results	364
	Using filters	366
	Overall layout	367
	Saving the configuration file	370
23.13	Interpolate Raster	370
23.14	Profile plots	372
	23.14.1 The context menu options	373
	Properties	374
	Swap Profile	374
	Select	374
	LP Edit	374
	Refresh LP	375
	Zoom In, Zoom Out, Pan, Previous Zoom, Next Zoom, Zoom to Full Extent 375	
	Load Layout, Save Layout	375
	Set Current Layout as Default	375
	Reset Default Settings	375
	Export to DXF (CAD file format)	375
	Copy to clipboard	375
	23.14.2 Longitudinal profile properties	376
	Axis	377
	Symbols and Fonts	378
	Graphical Items	379
	Tabular Items	381
	Labels	383
	23.14.3 Selecting	388
	23.14.4 LP Edit	388
	23.14.5 Refresh LP	389



23.14.6	Export to CAD	389
23.15	Animations	389
23.16	Result comparison	391
23.17	Export of results to ESRI shapefiles	392
23.18	Layout and advanced GIS	392
23.18.1	Adding and animating results in ArcMap	393
23.19	Presentation of Water Distribution results	394
23.19.1	Select Nodes or Pipes where ...	394
23.19.2	Display link flow Q-H plots	396
23.19.3	Display fire hydrant Q-H plots	397
23.19.4	Generate HTML Report	397
23.19.5	Store results in the database	399
23.19.6	Display current operating pump points on the Q-H curve	400
23.19.7	Reliability Analysis	401
Calculate		403
Report		403
24	Calibration Plots	407
24.1	Measurement Stations	407
24.2	Stations and Station Connections Layer	409
24.3	Calibration Plots and Reports	409
Report Group		413
24.3.1	Calibration Plots	413
Plot Button		413
Report / Report All Button		414
WD Simulations		415
Index		417



MIKE URBAN MODEL MANAGER

User Guide





1 Welcome to MIKE URBAN

MIKE URBAN is a flexible system for modeling and design of water distribution networks and collection systems for waste water and storm water.

MIKE URBAN is based on a database for storing network as well as hydraulic modeling data. This database is based upon the ESRI GeoDatabase. The GeoDatabase is a structure on top of a normal relational database - meaning that the underlying technology can be based upon any of the major database vendors (Microsoft, Oracle, IBM etc.). The GeoDatabase is a native storage for GIS data and may also be operated upon directly by standard GIS applications.

With MIKE URBAN you have:

- GIS-based model building and management
- Powerful hydraulic simulation engines
- Integrated water quality, fire flow, real time control and water hammer simulation (water distribution)
- Integrated water quality, sediment transport, real-time control, dynamic pipe design and long-term statistics (collection systems)
- Scenario management
- Full undo and redo capability in all editors
- Thematic mapping and integrated dynamic result visualization
- Open data models - easy integration with other applications
- Worldwide support and training
- Integrates directly with online and real time control systems

The modular structure of MIKE URBAN is seen in Figure 1.1. The core module, MIKE URBAN, contains the possibility to model both Water Distribution and Collection Systems.

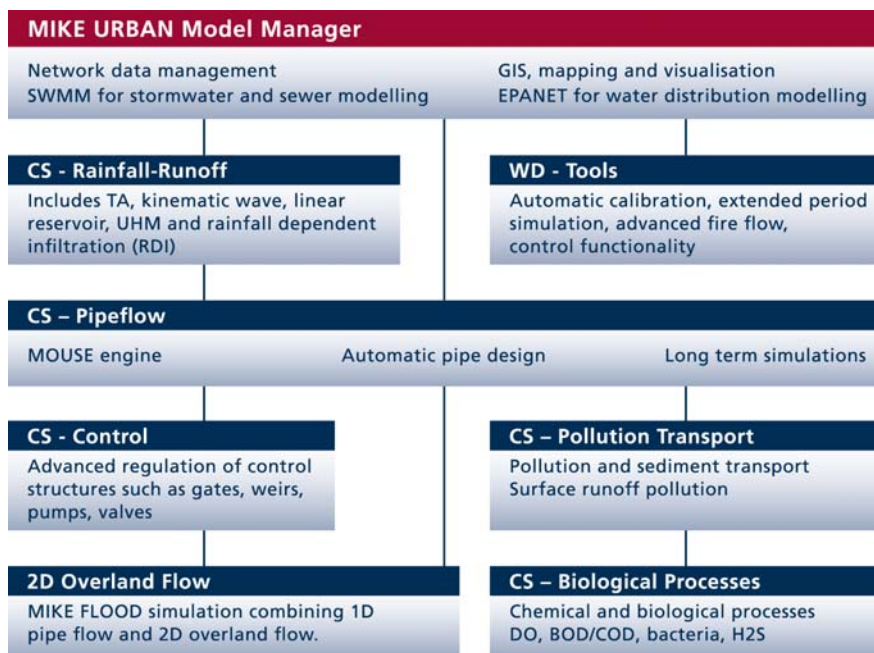


Figure 1.1 The modular structure of MIKE URBAN

1.1 Model Manager

The Model Manager is the main module of MIKE URBAN and includes a common data module for both Water Distribution and Collection Systems. Input editors for all simulation engines are included, irrespective of the installed simulation engines. It is possible to load SWMM 5 and EPANET files and result files of any size. The SWMM5 engine is included for the modeling of storm water and sewer modeling, while EPANET is included for the water distribution modeling. It is thus possible to make simulations with these numerical engines and view the results.

1.1.1 EPANET

Pipe Flow

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 itera-



tion. 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 Pipe Flow features are:

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

Water Quality

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 Water Quality features are:

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

1.1.2 SWMM

MIKE URBAN includes the SWMM (EPA's Storm Water Management Model) engine for storm water modelling. SWMM allows for the hydrodynamic simulation of flows and water levels in urban storm drainage and wastewater collection networks, thus providing an accurate information about the network functionality under a variety of boundary conditions. The hydrodynamic simulations can be extended with pollution and water-quality simulations. The model can also be enhanced by the variety of real-time control functions. The simulations can be carried out for single events.

1.2 Water Distribution

1.2.1 Water Distribution Tools

With MIKE URBAN Water Distribution Tools you get the following capabilities:



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.



1.2.2 Water Hammer

Water hammer module simulates transient (unsteady) flow in any fully pressurized system carrying liquids. MIKE URBAN Water Hammer module provides a cost effective tool for engineers seeking fast answers to questions about rapid operation of piping systems. Water hammer module 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 module allows you to model:

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

1.3 Collection Systems

MIKE URBAN for Collection System comes with the following modules:

- CS-Pipeflow
- CS-Rainfall-Runoff
- CS-Control
- CS-Pollution Transport
- CS-Biological Processes

1.3.1 CS-Pipeflow

MIKE URBAN CS-Pipeflow includes DHI's MOUSE engine. The public domain SWMM5 engine is already included with the Model Manager. With MIKE URBAN CS-Pipeflow you get access to:

- Hydrodynamic simulation of networks with MOUSE
- Pipe design (MOUSE engine)
- Long term statistics (MOUSE engine)

Hydrodynamic simulation

The MOUSE Hydrodynamic Pipe Flow Model solves the complete St. Venant (dynamic flow) equations throughout the drainage network (looped and dendritic), which allows for modeling of backwater effects, flow reversal, surcharging in manholes, free-surface and pressure flow, tidal outfalls and storage basins. The MOUSE hydrodynamic engine has been designed to handle any type of pipe network system with alternating free surface and pressurized flows as well as open channel network and pipes of any shape.



Virtually any construction can be described including pumps, weirs, orifices, inverted siphons etc.

The computational scheme uses an implicit, finite-difference numerical solution of the St. Venant flow equations. The numerical algorithm uses a self-adapting time-step, which provides efficient and accurate solutions in multiple connected branched and looped pipe networks. This computational scheme is applicable to unsteady flow conditions that occur in pipes ranging from small-profile collectors for detailed urban drainage, to low-lying, often pressurized, sewer mains affected by varying outlet water levels. Both sub-critical and supercritical flows are treated by means of the same computational scheme that adapts to the local flow conditions. In addition, flow phenomena, such as backwater effects and surcharges, are precisely simulated.

Pipe design

MOUSE Pipe Design Module is a tool, which makes it fast and easy to design pipe diameters for new systems as well as for pipes in existing systems. The basis for the design can be a rough input of a new system with geometric information about where the pipes should be and initial values for diameter and roughness (material) or it can be an existing MOUSE model for a complex system where some pipes should be re-designed.

In both cases the design module will be an excellent help to speed up the process by designing the exact pipe sizes which is needed to fulfil the design criteria and to finally verify that the design system is correctly designed.

Long term statistics

MOUSE Long-Term Statistics (LTS) allows that a MOUSE network with intermittent hydrological inputs can be setup for a long-term simulation, covering a continuous historical period, possibly over several years. The system automatically combines dynamic pipe flow simulations during wet weather and simple hydrological simulation during dry weather periods, which results in accurate computation of wastewater treatment plant loads, CSOs and other system outputs, while preserving rationality in use of computational resources. The results are presented both in the form of time series and a range of statistical parameters for selected variables. By running simulations with the current system configuration and the planned upgrade, impacts of the planned investments (e.g. new sewers, retention tanks, RTC schemes) on the system performance can be tested. This allows the user to develop the optimal rehabilitation / upgrade strategy, e.g. for achieving the consent with the environmental regulators' requirements.

1.3.2 CS-Rainfall-Runoff

MIKE URBAN CS-Rainfall-Runoff includes the MOUSE engine for rainfall-runoff. With MIKE URBAN CS-Rainfall-Runoff you get access to:

- Several types of surface runoff models (MOUSE)



- Rainfall dependent infiltration (MOUSE)

Surface Runoff

MOUSE Surface Runoff includes several types of surface runoff computation for the description of the urban catchment surfaces. This means that the surface runoff computations can be adjusted according to the amount of available information. The models run with well proven default hydrological parameters, which can be adjusted for better accuracy. The computed hydrographs are used as input to the MOUSE Pipe Flow model.

Rainfall dependent infiltration

MOUSE Rainfall Dependent Infiltration provides detailed, continuous modeling of the complete land phase of the hydrologic cycle, providing support for urban, rural and mixed catchments analyses. Precipitation is routed through four different types of storage: snow, surface, root zone and ground water, resulting in more accurate hydrographs. Instead of performing hydrological load analysis of the sewer system only for short periods of high intensity rainstorms, a continuous, long-term analysis can be used to look at periods of both wet and dry weather, as well as inflows and infiltration to the sewer network. This provides a more accurate picture of actual loads on treatment plants and combined sewer overflows. Further enhancements of groundwater-sewer interactions are possible by linking the MOUSE Pipe flow model with DHI's distributed groundwater model MIKE SHE.

1.3.3 CS-Control

MIKE URBAN CS-Control for MOUSE allows real-time control devices to be included in defining the urban drainage sewer network model. A selection of controllable devices is provided, along with a fully generic specification of control rules for any simple or complex global control scheme. The system allows the application of setting or set point (PID controller) based control functions, selected on the basis of logical evaluation of the actual system states (reactive control) or after the specified time series.

1.3.4 CS-Pollution Transport

Under the common names CS-Pollution Transport and CS-Biological Processes, the MOUSE engine provides several modules for the simulation of sediment transport and water quality for both urban catchments surfaces and sewer systems. Since pollutants are carried by sediment, sediment transport processes and water quality in sewer systems are closely interconnected. This is important for understanding phenomena like the first flush effect, which can only be simulated with a description of the temporal and spatial distribution of sediment deposits on the catchment surface and in the sewer system. MOUSE can model these complex mechanisms using its Surface Runoff Quality (SRQ), Pipe Sediment Transport (ST), Pipe Advection-Dispersion (AD), and Pipe Water Quality (WQ) Modules. Output from these modules, such as pollutant graphs from combined sewer overflows, can then be



applied directly to DHI's receiving waters models MIKE 11 and MIKE 21. Using MIKE 11 or MIKE 21 in conjunction with MOUSE allows assessment of water quality for the water bodies receiving these sewer overflows, such as rivers, streams, lakes and coastal waters.

In MIKE URBAN CS-Pollution transport the following can be modelled:

- Surface Runoff Quality (SRQ)
- Sediment Transport (ST)
- Advection-Dispersion (AD)

Surface Runoff Quality

The primary role of the Surface Runoff Quality (SRQ) module is to provide a physically-based description of the relevant processes associated with sediments and pollutants due to surface runoff, and then provide surface runoff sediment and pollutant data for the other pipe sewer network sediment transport and water quality modules. The following processes can be accounted for:

- Build-up and wash-off of sediment particles on the catchment.
- Surface transport of pollutants attached to the sediment particles.
- Build-up and washout of dissolved pollutants in potholes and stilling basins.

Sediment deposits can greatly reduce the hydraulic capacity of sewer pipes by restricting their flow area and increasing the bed friction resistance. The Pipe Sediment Transport (ST) module can account for these problems, by simulating pipe sewer network sediment transport-including deposition and erosion from non-uniform (graded) sediments. Contributions from rainstorm wash-off and dry-weather wastewater flow can be included. The ST module runs in conjunction with the dynamic flow routing, thereby simulating dynamic deposition of sediment and providing feedback due to the change in pipe area and resistance caused by sediment deposition. The following issues can be addressed:

- Prediction of sediment deposit locations and associated pollutants and metals in the sewer system.
- Prediction of reduction in hydraulic capacity due to observed and simulated sediment deposits.
- Analysis of the sewer system due to modified regulation strategies.

Pipe Advection-Dispersion (AD)

The Pipe Advection-Dispersion (AD) module simulates the transport of dissolved substances and suspended fine sediments in pipe flow. Conservative materials as well as those that are subject to a linear decay can be simulated. The computed pipe flow discharges, water levels, and cross-sectional flow areas are used in the AD Module computation. The solution of the advection-dispersion equation is obtained using an implicit, finite-difference scheme



which has negligible numerical dispersion. Concentration profiles with very steep fronts can be accurately modeled. The computed results can be displayed as longitudinal concentration profiles and pollutant graphs, which could be used at the inflow to a sewage treatment plant or an overflow structure.

The AD module can be linked to the Long Term Statistics module to provide long-term simulations of pollutant transport.

CS-Pollution Transport is only applicable with the MOUSE engine.

1.3.5 CS-Biological Processes

The MIKE URBAN CS-Biological Processes module works in conjunction with the Advection-Dispersion part of MIKE URBAN CS-Pollution Transport, thereby providing many options for describing the reaction processes of multi-compound systems, including degradation of organic matter, bacterial fate, exchange of oxygen with the atmosphere and oxygen demand from eroded sewer sediments. This allows realistic analysis of complex phenomena related to water quality in sewer systems. The module includes diurnal variation of foul flow discharges and user-specified concentrations of foul flow components. The sediment types included in the interaction with the CS-Biological Processes are foul flow organic sediments, and fine and coarse mineral in-pipe sediments originating from catchment runoff, potholes and stilling basins. The following can be account for with module:

- Decay of BOD/COD in bio-film and water phase
- Hydrolysis of suspended matter
- Growth of suspended biomass
- Oxygen consumption from decay of BOD/COD, bio-film and erosion of sediment
- Re-aeration
- Bacterial fate
- Interaction with sediments for nutrients and metals

The CS-Biological Processes module is only applicable with the MOUSE engine.



Welcome to MIKE URBAN



2 Getting Started

2.1 How to start MIKE URBAN

During the installation of MIKE URBAN, a shortcut is placed in the Programs' section of the Windows 'Start' menu (found under MIKE by DHI). You can also choose to create a MIKE URBAN icon on the Desktop and launch MIKE URBAN from this by creating a short-cut to MU.exe (found in the bin-directory of your MIKE URBAN installation).

Desktop Workspace

After MIKE URBAN is started, the MIKE URBAN desktop workspace will appear, as shown in Figure 2.1. This desktop workspace displays the MIKE URBAN menu, and displays any activated toolbars, the network browser window, and other enabled windows.

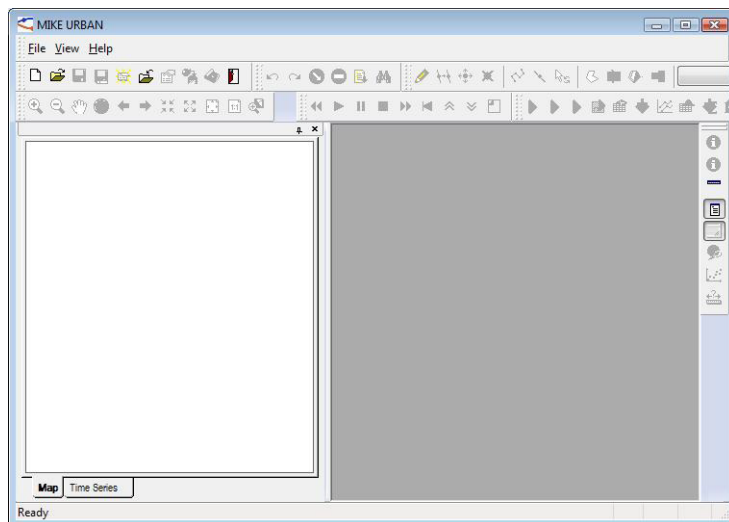


Figure 2.1 MIKE URBAN desktop

Map Windows

The Map window displays a layout plot of the pipe network system. The individual model elements (nodes, pipes, pumps etc.) are displayed. The Map window also allows the user to graphically layout the pipe network system.

Multiple instances of the MAP window can be displayed. This allows the user to display different zoom levels, display options and thematic mapping of the analysis results. Automatic graphical updating is performed when the user selects a different Map window as the active window.



Floating Toolbars

MIKE URBAN provides several task-specific toolbars to allow you to be more efficient. These toolbars can be docked underneath the menu bar, along any of the main window borders, or be allowed to float inside or outside of the main window. You can choose to create your own toolbars with your favourite tools (Tools|Customize|Customize Toolbars).

Identify



When you want information about a feature displayed in Map Window, you can use the Identify tool. The Identify tool allows you to see the attributes of your data. Clicking the Identify tool on a location inside a data frame will display the attributes of element at that location. The Identify tool is the easiest way to learn something about a location in a map. To display the Identify window, select either the tool from the toolbar or Tools|Identify. The Identify window will be displayed, as shown in

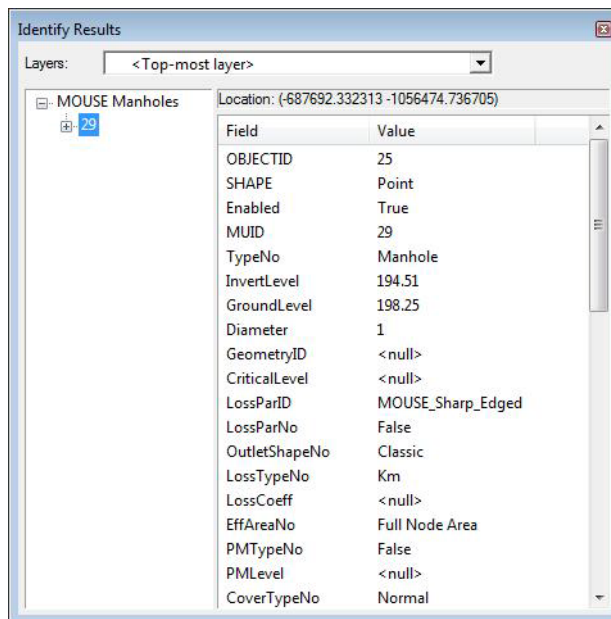


Figure 2.2 The identify tool displays information on the element chosen in the map

2.2 Online Help

On-line help is available for MIKE URBAN. The MIKE URBAN help system utilizes the Microsoft help technology known as HTML Help.



The Help system can be accessed in two ways:

- Choose 'Contents and Index' from the Help menu.
- By pressing F1 the relevant online help page will be displayed in the active window.

MIKE URBAN help is viewed in the HTML Help Viewer (see Figure 2.3) which consists of the following:

- Topic Pane - where the help topics are viewed
- Navigation Pane - where you can navigate in the help file. Index and search for instances of e.g. a word are possible.

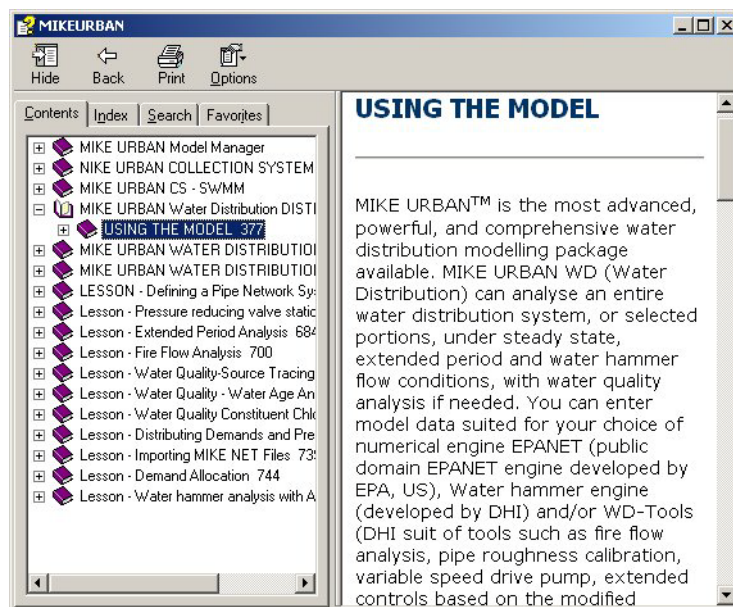


Figure 2.3 The online help

Status Line and tooltips

As you move your cursor in the map, the status line will display the coordinates and map scale. Pausing your cursor over an edit field will display a tooltip with the name of table and attribute name where data for that field is saved.

2.3 MIKE URBAN Examples

MIKE URBAN includes a number of examples demonstrating how to use the various modules. These are initially placed in the Program Files directory. The examples can be installed/copied from here to a user defined location by



going to Help|Install Examples, see Figure 2.4. It is possible to choose which examples to install/copy.

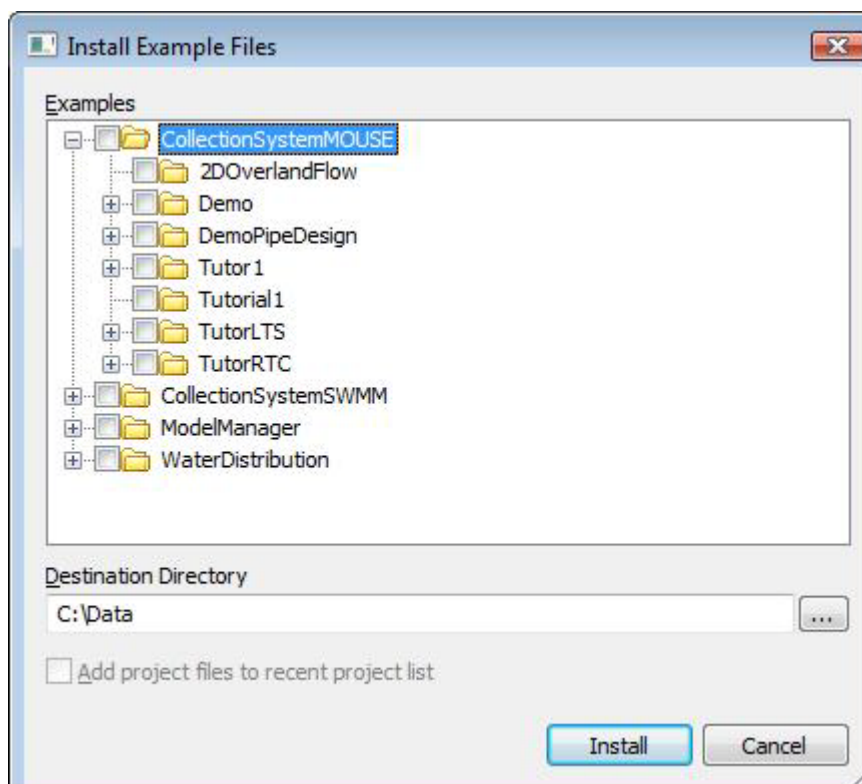


Figure 2.4 Installing/copying the examples to a different folder

2.4 The Table Of Contents

The table of contents window is both a navigation and a configuration tool that defines a view of the pipe network including the properties of each individual layer. The table of contents lists all the layers on the map and shows what the features in each layer represent. The check box next to each layer indicates whether or not the layer is currently drawn on the map. By default, the table of contents is located on the left side of the Map window. The table of contents allows you to show and/hide predefined MIKE URBAN layers such as Collection Systems and Water Distribution. The layer order, as it is in the Map tab, affects what is displayed in the horizontal view and can be easily rearranged by the user. Layers can in fact be moved up and down in the list and top layers are always drawn on top of the lower ones.

The TOC, composed of Map tab and Time Series tab, is mainly used for turning layers on and off and for accessing layer properties.

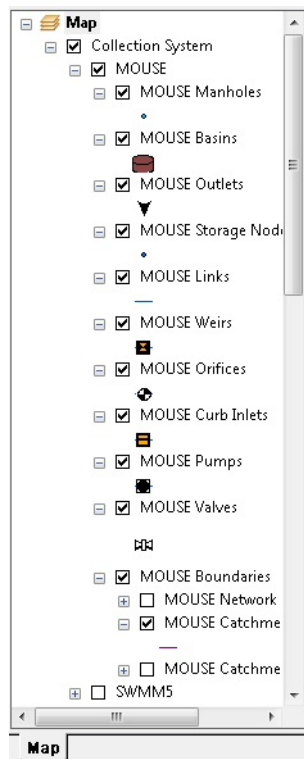


Figure 2.5 Table Of Content (TOC)

Working Modes

Choosing a specific working mode affects also the horizontal view: generally layers belonging to the redundant data models are, per default, hidden. However, regardless of the selected working mode, any layer contained in the geodatabase can be displayed by ticking both group and layer appropriate check boxes in the table of contents.

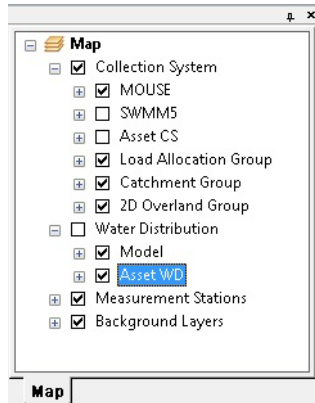


Figure 2.6 Working modes in the TOC

2.4.1 Boundary conditions displayed on the map

Boundary conditions are per default unchecked (one exception: catchment connections) and not displayed in the horizontal view but they can be activated at any time during the project.

To be displayed, boundary conditions must be applied and contain at least one 'Boundary Item'.

Changes in the 'Boundary Conditions' dialogs are not immediately displayed in the horizontal view. First make sure to save your edits and then layers must be regenerated. This is done by right-clicking on the layer that needs to be updated and then by choosing the 'Regenerate' option.

MOUSE

The different boundary conditions that can be visualized for MOUSE networks are seen in Figure 2.7.

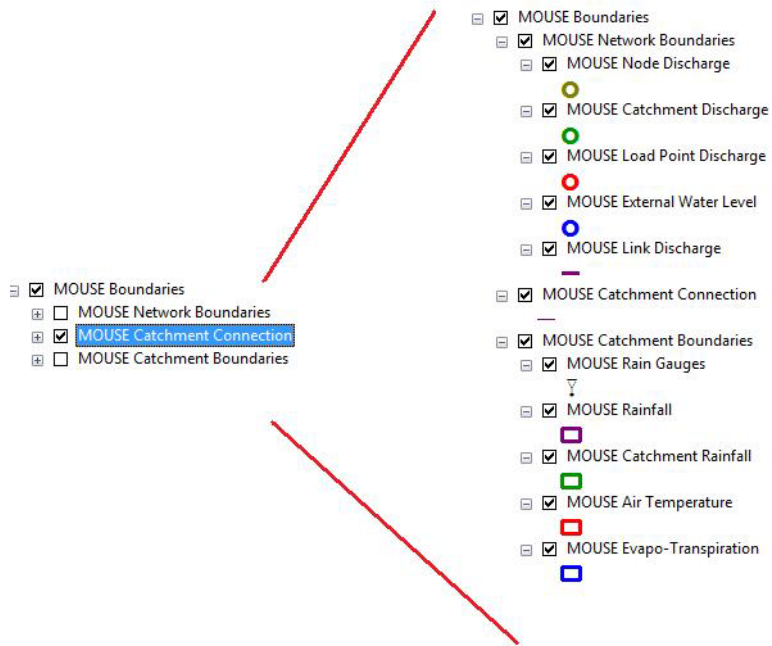


Figure 2.7 MOUSE boundaries

The 'MOUSE Boundaries' group includes three main sub-groups: Network boundaries (from the Network loads and the External water level), Catchment connections and Catchment boundaries (Catchment loads).

For further, please refer to the relevant chapter in the Collection System User Guide.

SWMM

The different boundary conditions that can be visualized for SWMM networks are seen in Figure 2.8.

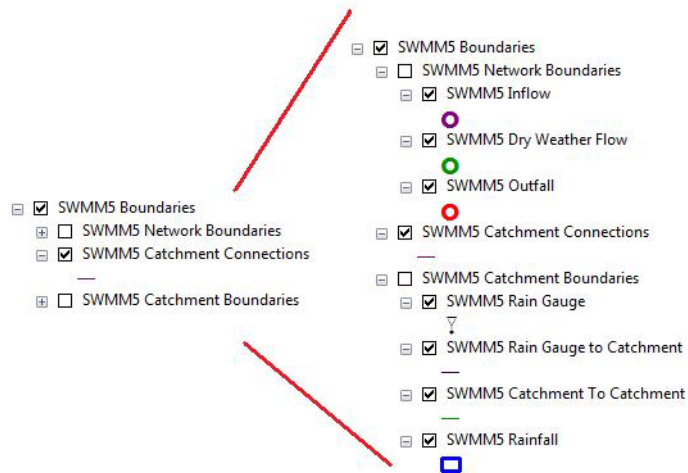


Figure 2.8 SWMM boundaries

The 'SWMM5 Boundaries' group includes three main sub-groups: Network boundaries (from the Dry Weather Flow, the Inflow and the Outfall properties), catchment connections and catchment boundaries (mostly from the Routing dialog).

For further, please refer to the relevant chapter in the Collection System User Guide.

EPANET

Node demands can be displayed by ticking the Water Node Demands layer. Per default, the different demand categories will be differentiated when displayed.

2.4.2 Properties dialog

The 'Properties' dialog can be reached by right clicking on one layer from the TOC. For the pre-existing items the dialog consists of four tabs: General, Selection, Labels and Symbology.

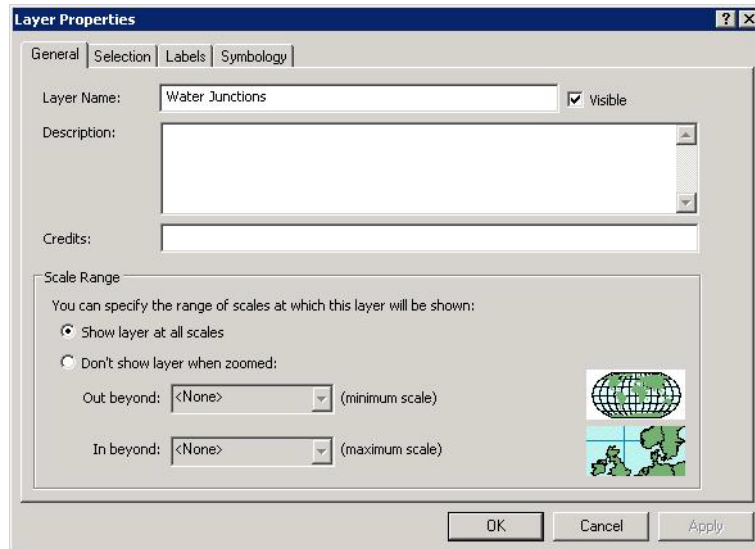


Figure 2.9 Properties dialog from the TOC

The General tab

Layer name and description can be modified in the General Tab (see Figure 2.9). The visualization of the layer in the horizontal view is controlled by 'Visible' check box: if ticked, the layer is displayed. Scale Range visualization options are also available within the General tab.

The Selection tab

In this tab, selection symbol and color can be modified.

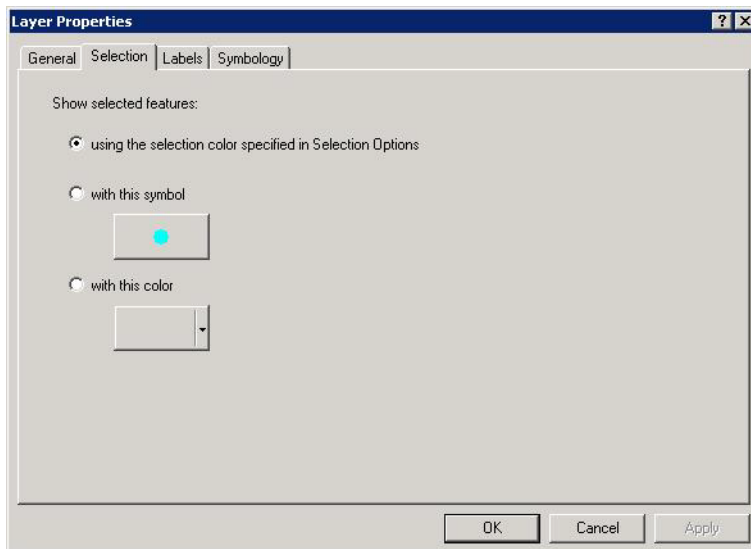


Figure 2.10 Properties dialog - Selection tab

The Labels tab

The label tab is used to define and display labels.

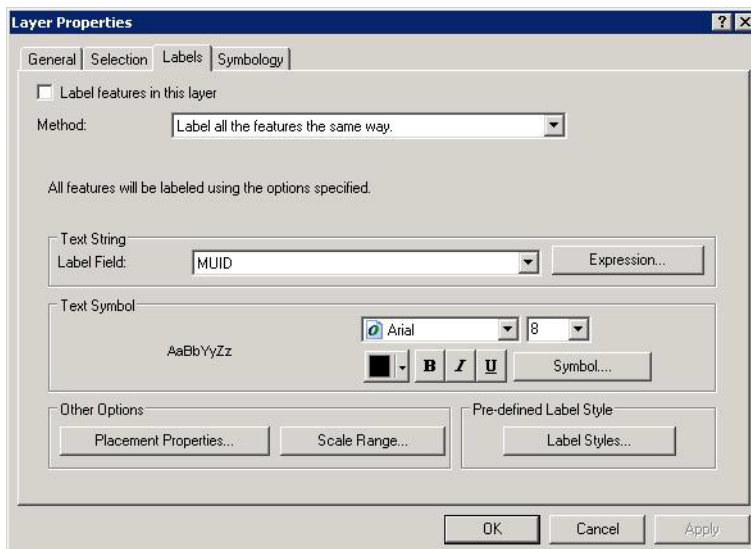


Figure 2.11 Properties dialog - Labels tab

- The 'Label features in this layer' allows the user to turn on and off the labels.



- The displayed labels can either be the same for all the elements, by using the method 'Label all the features the same way', or be different according to classes.
- The label content can be chosen from the drop-down list of the 'Label Field'
- Font, location and displaying criteria can be adjusted accordingly.

The Symbology tab

The symbology can be modified and it can vary depending on the value of a field or be constant.

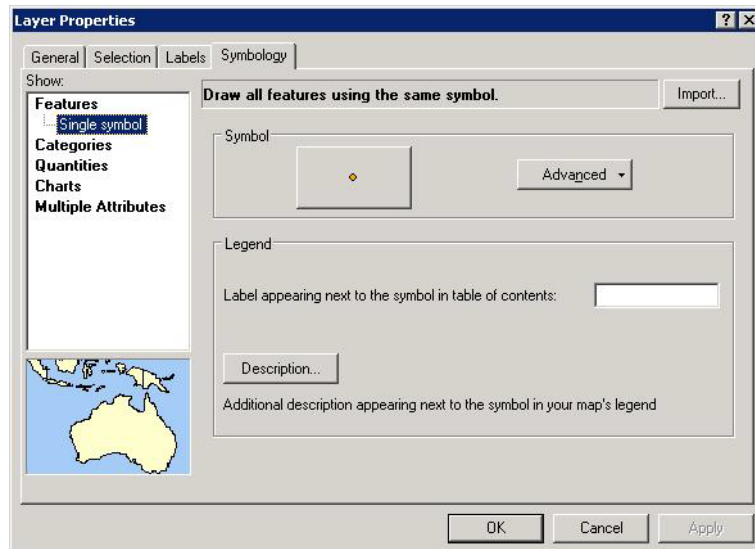


Figure 2.12 Properties dialog - Symbology tab

2.5 The main menu

Initially when launching MIKE URBAN three menus are seen. The 'File', 'View' and 'Help' menu. The File menu can then be used to create or open a project.

The main menu when running in Collection System mode:

- File
- Edit
- (Asset CS)
- (MOUSE)
- (SWMM5)



- Simulation
- Model Results
- View
- Tools
- Window
- Help

The main menu when running in Water Distribution mode:

- File
- Edit
- (Asset WD)
- (EPANET)
- Simulation
- Model Results
- View
- Tools
- Window
- Help

File menu

On the file menu it is possible to create, open, save and close MIKE URBAN projects. The menu also provides access to import and export functionality. This menu is also used to switch between Collection System and Water Distribution mode.

Edit menu

The edit menu provides a set of editing functions. Edit mode is entered from here and multiple undo/redo can be done from here. A 'Find' command is also found here.

Asset menu

The content of the Asset menu depends on the work mode of MIKE URBAN. When in Collection System mode the Asset dialogs relevant for a collection system are available, while in Water Distribution mode the Asset dialogs relevant for a water distribution system are available. These editors allow you define e.g. the asset pipes, pumping stations, storage tanks and other data that make up the asset part of the collection or water distribution system.



MOUSE menu (CS)

Provides access to all MOUSE model input dialogs. The MOUSE menu can only be made visible when in Collection System mode.

SWMM 5 menu (CS)

Provides access to all SWMM 5 model input dialogs. The SWMM 5 menu can only be made visible when in Collection System mode.

Model menu (WD)/EPANET menu

The EPANET (Model) Menu provides access to the network component editors. These editors allow you define the junction nodes, pipes, valves, pumps, reservoirs, tanks, and valves that make up the water distribution system. The menu can only be made visible when in Water Distribution mode.

Simulation menu

From the simulation menu you can, depending on the work mode, launch simulations for your Collection System or your Water Distribution system.

Model Results menu

Provides access to loading of results files, adding of statistical results layers and animation of result files.

View menu

Zoom features as well as toggling on and off of view related items (e.g. Table of Contents and Status Bar).

Tools menu

The Tools Menu provides access to many of the powerful tools provided with MIKE URBAN. E.g. Data transfer between various models, language and unit choices, scenario management, project checking, customizing of toolbars etc.

Window menu

The Window Menu provides methods for arranging the displayed windows.

Help menu

Offers access to the MIKE URBAN online help as well as e.g. providing access to updates on the web.

2.6 The toolbars

MIKE URBAN toolbars can be placed where you prefer. The toolbars provide shortcuts for program functions. For any tool on a toolbar, the same functionality can always be found in one of the main pull-down menus.

Individual toolbars can be switched on and off. MIKE URBAN saves the current toolbar combination for the next MIKE URBAN session. The displayed toolbars get automatically activated or de-activated (grayed-out) according to the presently active graphical window or dialog. For customizing, please see 3.4.

2.7 Common Editor Features

All of the network component editors provided within MIKE URBAN have similar features to help in defining a collection system or water distribution network. This section describes these common features. A typical network component editor (the pipe editor from the Water Distribution is used as an example) is shown below in Figure 2.13.

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

Figure 2.13 The network component editors all have similar features to aid the user in defining a network

Under the 'Advanced' button a number of features can be found, see Figure 2.14.

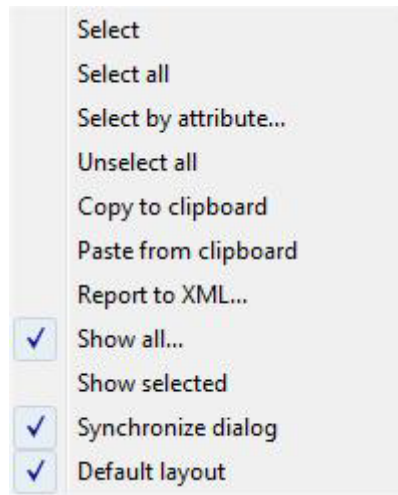


Figure 2.14 The features found under the Advanced button

Select, Select, All, Unselect All

Choosing 'Select' will select the element that is currently the 'active' record. Choosing 'Select All' will select all records in the editor. Selected records are default selected across MIKE URBAN, i.e. also on the Map window. 'Unselect All' will deselect all records in the editor and it will also deselect them from the Map window.

Show All, Show Selected

Choosing 'Show All' will display all records in the editor (selected and unselected).

Choosing 'Show Selected' will display only selected records in the editor.

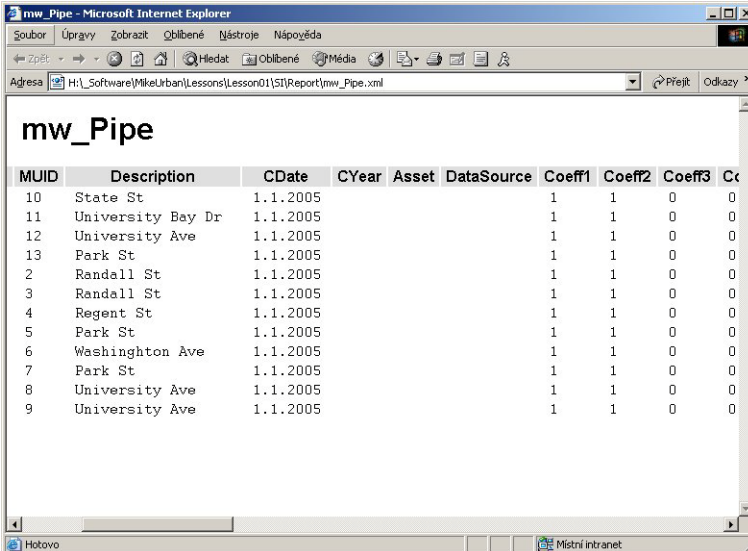
Copy to Clipboard and Paste from Clipboard

Choosing 'Copy to clipboard' will copy the editor data into the clipboard and the data will be available for pasting into another Windows applications. In case that the editor has two grids, a sub-menu with the grid selection will be displayed.

Choosing 'Paste from clipboard' will paste the clipboard data into the editor. In case that the editor has 2 grids, a sub-menu with the grid selection will be displayed. The clipboard data used for pasting must have the same format as the original clipboard data used for copying. In case that there are existing record with the same MUID, the clipboard data will update the existing data. In case that the clipboard contains records with new MUID, the data will be appended to the existing records in the editor. Note, that it is not possible to paste new records for data with geometrical representation such as pipes, nodes, tanks etc.

Report to XML

Choosing 'Report to XML' will generate an HTML report, as shown in Figure 2.15. Please note that a more advanced tool where you can set up reports from several tables and including simulation results can be found under Model Results|Reporting Tool.



MUID	Description	CDate	CYear	Asset	DataSource	Coeff1	Coeff2	Coeff3	Cc
10	State St	1.1.2005				1	1	0	0
11	University Bay Dr	1.1.2005				1	1	0	0
12	University Ave	1.1.2005				1	1	0	0
13	Park St	1.1.2005				1	1	0	0
2	Randall St	1.1.2005				1	1	0	0
3	Randall St	1.1.2005				1	1	0	0
4	Regent St	1.1.2005				1	1	0	0
5	Park St	1.1.2005				1	1	0	0
6	Washington Ave	1.1.2005				1	1	0	0
7	Park St	1.1.2005				1	1	0	0
8	University Ave	1.1.2005				1	1	0	0
9	University Ave	1.1.2005				1	1	0	0

Figure 2.15 Example of an HTML report generated by MIKE URBAN

Synchronize dialog/Default layout

Toggling the 'Synchronize dialog' off means that the default behaviour of MIKE URBAN (where a selection made in e.g. the map window will also select them in the relevant editor) is turned off. Closing the editor will restore the default behaviour.

Toggling the 'Default layout' off allows you to see all attributes for elements, including those that are normally hidden. If you have added attributes for elements that are not a part of the MIKE URBAN default attributes, you can view these in the grid when the default layout is toggled off. E.g. if you have added a column for nodes with an extra description this can be seen in the grid.

2.8 Languages

MIKE URBAN can be run in a number of languages. You can switch language on the fly in your MIKE URBAN application by choosing 'Tools|Language' and then toggling the language of your choice on. The languages that may be switched to are dependent of the operating system. E.g. only on a Japanese operating system may one switch to Japanese.



2.9 Selecting a Coordinate System

Choosing a coordinate system for your MIKE URBAN database may be an important part of setting up a MIKE URBAN database. The default coordinate system is 'Unknown' and allows you to operate most of MIKE URBANs features, it is basically treated as a rectangular system in unknown units.

However, several features of MIKE URBAN will only work when the coordinate system is properly defined. So if any of the below points apply to you, you should consider setting up the coordinate system correctly:

- You need to calculate pipe lengths and other geographic measures in a specific unit.
- You have a corporate GIS and will be using data from this in MIKE URBAN.
- You receive data in different coordinate systems and projections and need to overlay them correctly.
- You want to use MIKE URBAN features that requires the system to calculate the scale. (maps to scale, switching on and off features/labels depending on scale etc.)

These functions will only work properly if MIKE URBAN knows about the units and other definitions in your coordinate system. If you do not already know your coordinate system, your GIS department or GIS data vendor will normally be able to tell you.

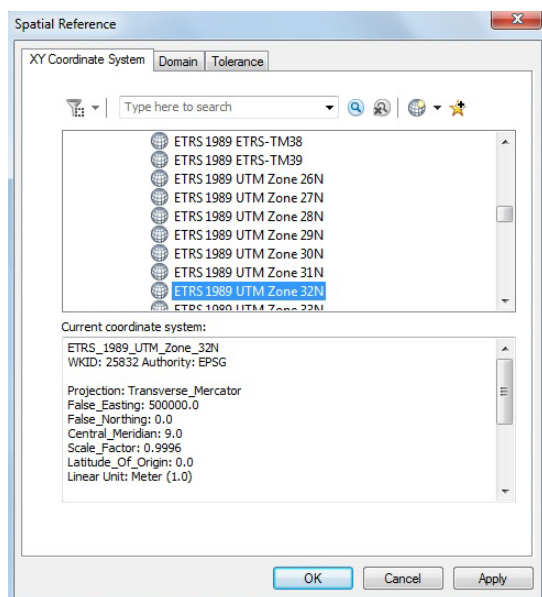


Figure 2.16 Properties of the spatial reference

The dialog defining the properties for spatial reference, see Figure 2.16, handles the setting of coordinate systems, there are three tabs:

XY Coordinate system

Here you select the coordinate system and define all parameters related to the coordinates.

Domain

The coordinate range or domain extent defines the minimum and maximum coordinate values which can be stored. On the domain page these can be seen. The Z and M domains are not used by MIKE URBAN.

Tolerance

The XY tolerance is the minimum distance between coordinate before they are considered equal.

In many cases you will simply select one of the predefined coordinate systems. Most often coordinate systems used in MIKE URBAN will be of the type 'Projected coordinate systems', and you simply browse through the available coordinate systems options and select the one matching your data. Click OK and the coordinate system is defined, after selecting the system you should double-check the values in the domain as the defaults are changed when you select the coordinate system.

You can, however, also create a new one or by importing a coordinate system from e.g. another geodatabase. See Figure 2.17 on where to access the tools for doing this. The concept of creating and editing coordinate system is described in ESRI documentation.

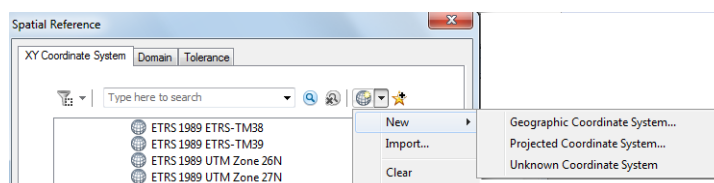


Figure 2.17 Defining a new or importing a coordinate system

Tip

If you do not know your coordinate but still want to utilize the GIS functionality for length and scale calculations then use a rectangular system (e.g. UTM) in the unit (typical meter or feet) that you are working in. This will provide scaling and calculations, but unexpected results may occur if you open the database together with other (correctly defined) data.

2.10 Starting a New Project

- Start MIKE URBAN



- Go 'File|New'
- On the dialog that comes up, see Figure 2.18 choose Working Mode to be either Collection System or Water Distribution
- Choose a coordinate system among the list of available coordinate systems, see section 2.9
- Select either .mdb or .gdb. MIKE URBAN supports 3 different database formats. The geodatabase is a "container" used to hold a collection of datasets. There are three types:
 - File Geodatabases - Stored as folders in a file system. Each dataset is held as a file that can scale up to 1 TB in size. This option is recommended over personal geodatabases.
 - Personal Geodatabases - All datasets are stored within a Microsoft Access data file, which is limited in size to 2 GB
 - ArcSDE Geodatabases - Stored in a relational database using Oracle, Microsoft Server, IBM DB2, or IBM Informix. These multi user geodatabases require the use of ArcSDE and can be unlimited in size and numbers of users.
- Choose a unit system among the list of available unit systems, see section 3.1
- Then press 'Save'

A new empty database is now created. The database can be populated with data either by importing data from various data sources or by entering data manually.

The URBAN Project file, *.mup file, contains the data as represented in the database along with references to background layers that may be loaded. Change of symbology will be saved into the *.mup file. It is thus possible to have several *.mup files referring to the same database.

In the title bar of the MIKE URBAN application you can see whether reference is made to either the database or to the project file (*.mup). Per default a project file with the same name as the database will be saved if no project file name is specified.

On the File menu the list of the last opened projects and/or databases is found. It is possible to create a backup of your MIKE URBAN database by using 'File| Backup Database'. It is necessary to leave the editing session before creating a backup of your database.

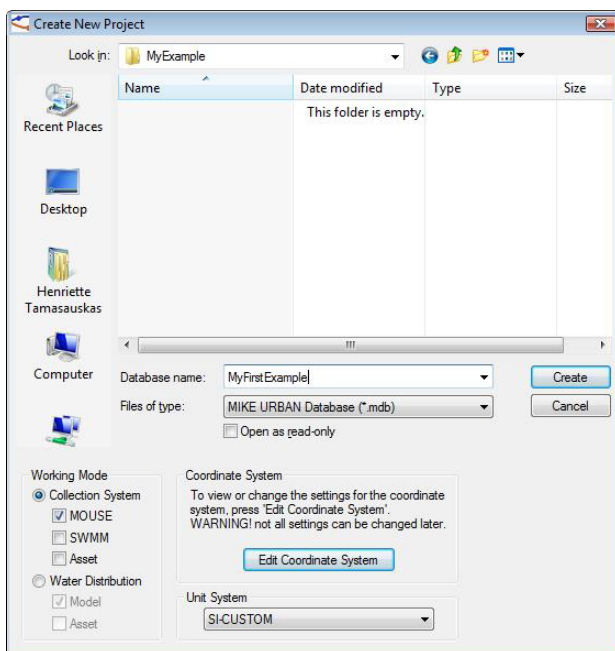


Figure 2.18 Creating a new project

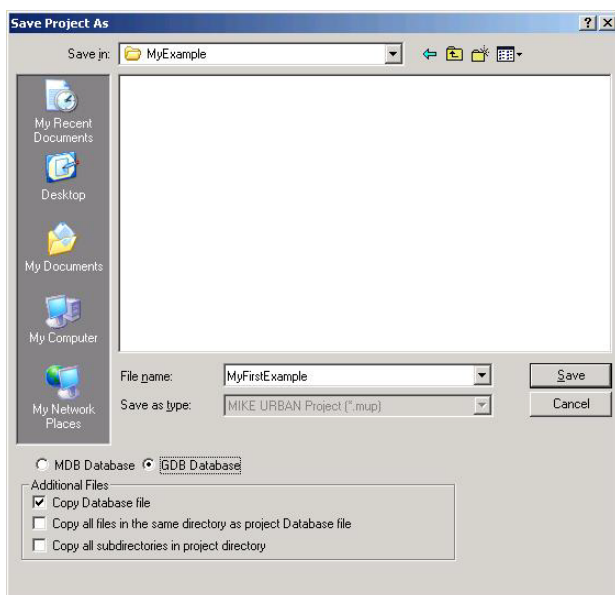


Figure 2.19 Save as dialog

It is possible to save a setup stored in a .mdb database to a .gdb database and the other way around. Effectively the 'save as' functionality offers conversion as well as a way to save a project under a different name.



2.11 Tools in MIKE URBAN

Under the 'Tools' menu a number of tools are found.

2.11.1 Selection tools

MIKE URBAN provides many different ways of selecting data. It is possible to use graphical selections, selections by attribute, and it is possible to choose from several pre defined selection sets. Under Tools|Selection a number of selection tools are found.

Selection Methods

Various selection methods exist: New Selection, Add to Selection, Remove from Current Selection and Select from Current Selection.

New Selection



To deselect all selected features at once and to start selecting new features, click New Selection.

Add to Selection



To add features into the current selection, click Add to Selection. Click a selected feature to add it to the current selection set.

Remove from Current Selection



To remove a feature from the selected set, click Remove From Current Selection. Click a selected feature and it deselects.

Select from Current Selection



Click a selected feature to select it from the current selection set.

Select Features



Click the feature you want to select or click and drag a box around the features you want to select.

Select by Polygon



Create a polygon to select the features you wish to select

Select by Graphics



Choose Edit|Graphics|Create Graphical Element and draw on the Map window. Then choose Edit|Graphics|Select Graphical Element and point to the



graphical element that you wish to perform the selection from. If you then choose the Select by Graphics Tool the elements from the selectable layers that are intersecting the graphical element will become selected.

Select by Attributes



The Select by Attributes dialog box allows you to select features on a given layer using an expression. The Select by Attributes uses the current feature class (such as pipe, node) from the current network as it is specified in the Feature Edit toolbar.

Select Selectable Layers



Check the boxes next to the layers from which you want to select, and uncheck the boxes you don't want to select.

Clear Selected Features



To deselect all selected features at once, click the map where there are no features, or click the Selection menu and click Clear Selected Features.

Select Dead-End Nodes and Select Dead-End Pipes

Allows you to select all nodes with only one connected pipe as well as pipes, which are adjacent to the dead-end nodes. This is helpful in checking the model geometry for errors.

Load Selection



To load the selected features from a list, click Load Selection and select the selection list file name.

Save Selection



To save the selected features to a list, click Save Selection and specify the selection list file name.

Merge selected pipes - WD mode only

In addition to the model simplification (Tools | Simplification), MIKE URBAN provides very fast and easy way of merging selected pipes - for WD mode only. In order to merge selected pipes, select such pipes from the Map window and select Tools | Selection | Merge selected pipes. Selected pipes, in case that there are no "T" or other complex junctions, will be merged together. Pipe length (geometrical and/or user defined) will be equal to the sum of merged pipe lengths while other pipe attributes (such as diameter, material, friction) will be equal to the first pipe used for merging (you should



therefore merge pipes with similar parameters or revise the merged pipe attributes for changes).

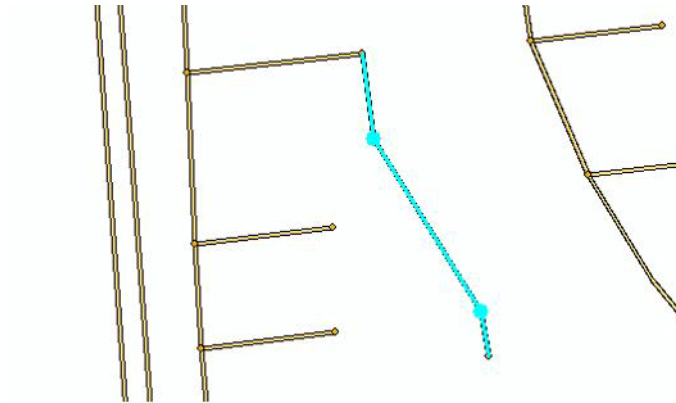


Figure 2.20 Select pipes you would like to merge together

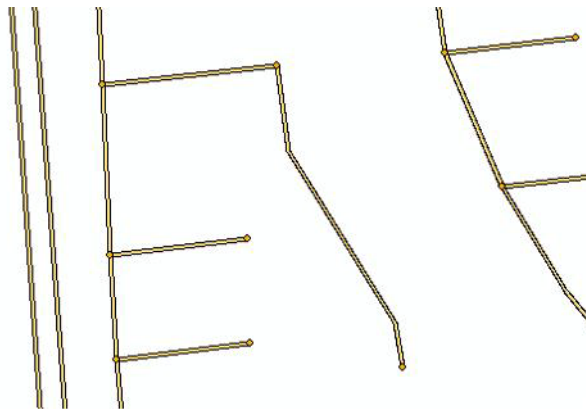


Figure 2.21 Selected pipes are merge together

2.11.2 Tracing

The following describes features in MIKE URBAN that allow the user to perform interactive forward and backward tracing of flow and to check the network connectivity.

Find Circular Connection



To select all loops in the pipe network, select **Tools | Tracing | Loops** and click any network node.



Network Connectivity



In defining a large network system, it is sometimes possible to define a disjointed network where a small part of the network is somehow not connected to the main network. To check the network connectivity, click **Tools | Tracing | Network**. Click any network node to select all pipes connected to the selected network node. If any nodes or links are not connected to the network, they will not be highlighted.

In the event that the network is quite large and the possibility exists that un-highlighted nodes or links might not be readily visible, then the Invert Selection icon on the toolbar palette can be clicked on. Then, all non-highlighted nodes and links will become highlighted, and the highlighted nodes and links will become un-highlighted. Any nodes or links that are not connected to the network will be highlighted.

After running a network simulation and loading the analysis results, the user can perform interactive forward and backward tracking of the flow to show where water goes and where water originates from for a particular node.

Forward Flow Tracking



To perform forward tracking of the flow, select **Tools | Tracing | Define Flags**. Then, from the Map window, click on the node you are interested to perform forward tracking of flow from. Select **Tools | Tracing | Forward Flow Tracing** and define the threshold value of a flow; pipes with a flow below the threshold value will be handled as closed. MIKE URBAN will display the path(s) that the flow takes from the selected node.

Backward Flow Tracking



To perform backward tracking of the flow, select **Tools | Tracing | Define Flags**. Then, from the Map window, click on the node you are interested to perform backward tracking of flow from. Select **Tools | Tracing | Backward Flow Tracing** and define the threshold value of a flow as shown in Figure 2.22; pipes with a flow below the threshold value will be handled as closed. MIKE URBAN will display the path(s) that the flow takes to get to the selected node from the source nodes (i.e. tanks and reservoirs) as shown in Figure 2.23.

Note that it is possible to define more than one flag to flag the node selected for the flow tracing.

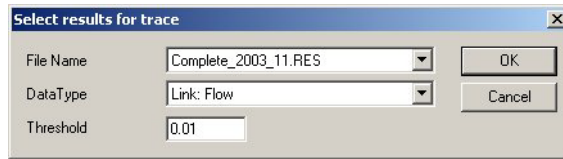


Figure 2.22 Defining the threshold value

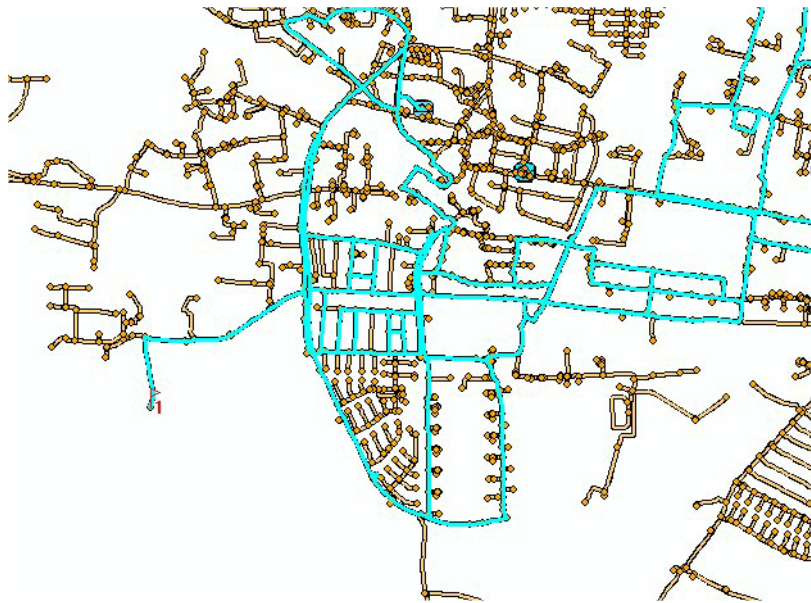


Figure 2.23 An example of forward flow tracing showing how flow is traced forward from the selected node

2.11.3 General SQL command

The 'General SQL Command' tool allows you to define, save, load, and execute an unlimited number of SQL commands.

The syntax you use to build SQL commands differs depending on the data source. This is because although SQL is a standard, not all database software implements the same dialect of SQL. To query file geodatabases (*.gdb) you use a dialect of SQL implemented within ArcGIS that supports a subset of the features and functions available in personal geodatabases. To query personal geodatabases (*.mdb), you use the same syntax used in Microsoft Access. INSERT and UPDATE in file geodatabases are e.g. limited to simple statements containing literal values only. Compound expressions such as SET DIAMETER = (DIAMETER - 0.50) as well as subqueries are not supported in INSERT and UPDATE statements. DELETE statements, on the other hand, are not limited; they support any WHERE clause capabilities supported by the file geodatabase. If your application contains subqueries, they



may not work with file geodatabases since support for them is limited. You can learn more on this on www.esri.com.

With the SQL commands you can for instance execute SQL UPDATE statements that will change the pipe diameter from the nominal (outside) diameter to the inside diameter or, you can define the pipe friction coefficient based on the pipe material and the pipe age.

General SQL commands can work on any MIKE URBAN geodatabase table such as "mw_*", "msm_*", "mss_*" and you can use standard SQL commands including UPDATE (to update table field values) ⁽¹⁾. Please note the previous sections on the differences between the commands that can be used on mdb and gbd files. Multiple SQL statements need to be separated by ";" (see Figure 2.24).

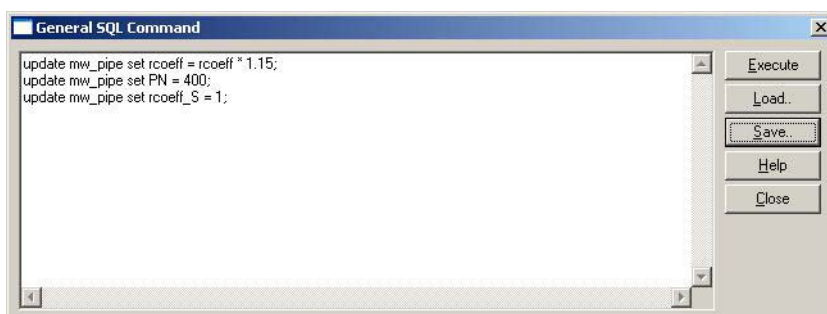


Figure 2.24 Example of SQL commands

The functionality of the buttons to the right are given below:

- Execute: executes the SQL statements
- Load: loads a text file with previously saved SQL statements
- Save: saves SQL statements to a text file for later re-use

¹ Although the use of ALTER (to add or remove table fields) is possible, DHI recommends to change table structure through geodatabase operations only (i.e. in ArcMap/ArcCatalog).



Some more examples are provided in below table. Please note that SQL commands are dependent on the type of database (e.g. mdb or gdb). Below examples can be used for setups saved in mdb.

Table 2.1 SQL command examples (mdb-files)

Command	Explanation
UPDATE mw_pipe SET rcoeff*1.15	General update of table: rcoeff value in mw_pipe table is multiplied by 1.15
UPDATE msm_node SET diameter = 1.0 WHERE diameter = 0.99	Specific update of table: diameter in msm_node table is set to 1.0 all the places where the diameter is currently 0.99
UPDATE msm_node SET AssetName = MUID WHERE MUID in (SELECT MUID FROM msm_node WHERE AssetName is null)	Updates table with use of a sub-selection: AssetName in msm_node table is updated with MUID in the cases where AssetName field is not filled (Null)
UPDATE msm_node, msm_CatchCon SET msm_node.Description = msm_CatchCon.CatchID WHERE msm_CatchCon.NodeID = msm_node.MUID	Updates table with information from other table: Description field in msm_node table is updated with ID of connected catchment (msm_CatchCon.CatchID)

UPDATE msm_node SET AssetName = MUID WHERE MUID in (SELECT MUID FROM msm_node WHERE AssetName is null)

2.11.4 Rebuild network

The rebuild network tool rebuilds the network connectivity where this might have been lost, i.e. if a pipe is missing nodes at either end the tool will link the pipe to nodes that are found within a specified snap tolerance at the end of the pipe. The tool will not add nodes but only link to existing nodes. If two or more nodes are found within the snap tolerance one of these is chosen. The rebuild network tool requires that MIKE URBAN is outside an edit session. When activating the tool a warning message will occur as seen in below figure.

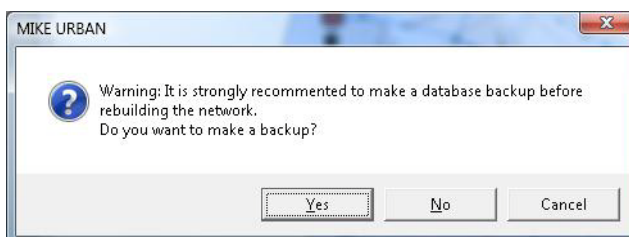


Figure 2.25 Warning message when activating the rebuild network tool.



After choosing to either create a backup or not, you will be prompted to specify the snap tolerance to use when rebuilding the network. The snap tolerance will be specified in map units.

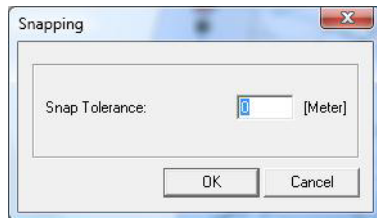


Figure 2.26 Specifying the snap tolerance

Please note that the rebuild of the network may take some time. A progress bar will be displayed during the process.

2.12 Various WD Tools

2.12.1 Duplicate selected pipe attributes

It is common that when importing a model based on e.g. CAD files, where the pipe attributes (imported as CAD labels) were not available for every pipe. A similar situation occurs when you start editing pipes manually and 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 2.27 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.

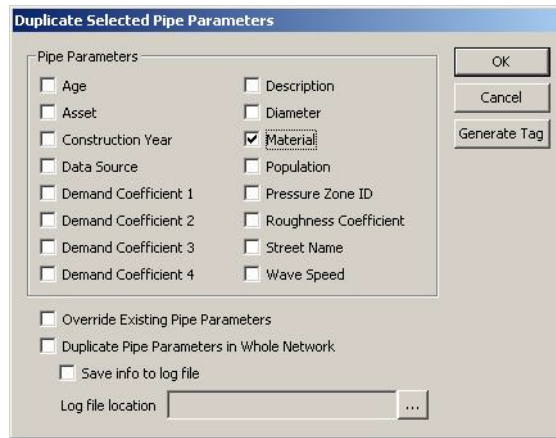


Figure 2.28 Select which attributes you want to duplicate

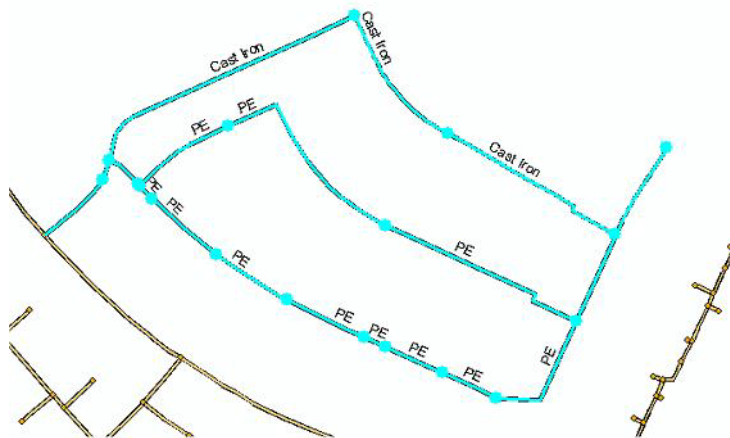


Figure 2.29 Selected attribute(s) were duplicated to selected pipes

"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.

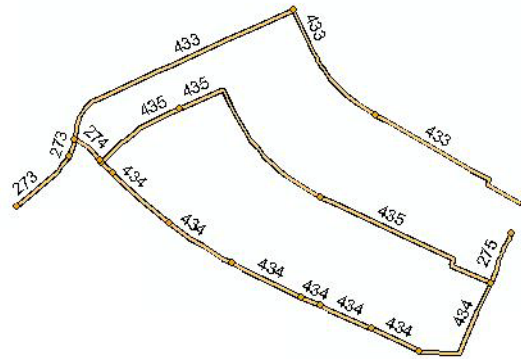


Figure 2.30 GIS tag is generated for each stretch of pipes

2.13 Saving Animations to AVI Files

It is possible to save the animation of the results to Windows animation AVI file. This option is available for saving both Map and Profile Plot results.

In order to save Map results to the animation file, load the simulation results to MIKE URBAN, display them on the Map and select File | Save AVI file... .Options dialog box will be displayed, from within you specify the interval between frames as a multiplication of 100 milliseconds.



Figure 2.31 Save animation to AVI file - Options setting

MIKE URBAN will display the standard "Save file" dialog box where you specify the location and name of the animation file and then you can define the video compression mode based on your Windows settings. Note that using Full Frames (Uncompressed) will create a video file, which you can play on any Windows regardless the local settings. If you wish to stop the animation recording, press the 'Esc' button.

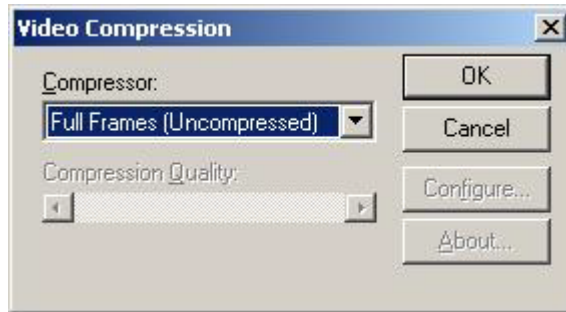


Figure 2.32 Save animation to AVI file - Video Compression mode

In order to save Profile Plot results to the animation file, load the simulation results to MIKE URBAN, define the Profile Plot path and display the Profile Plot. Select File | Save AVI file... from within the Profile Plot window and follow the same steps as in saving the Map results to the animation file.

2.14 Export of Maps

It is possible to export the Map window in two ways:

- Export Map (exports the map into an image file e.g. BMP)
- Export Map to DXF (exports all layers including result layers into a DXF)

For the latter export: MIKE URBAN will export will export the layers found in the TOC into separate DXF layers. In addition to this, it will also create separate layers for all annotations, flow arrows, and similar information. It will also parse pipe into separate layers based on the pipe diameter.

Such DXF file can be then easily imported into CAD programs and used for professional mapping.

2.15 Images in MIKE URBAN

A background image can e.g. be added in MIKE URBAN by choosing Edit|Insert Layer.

2.15.1 World files

Images are interpreted as raster data, where each cell in the image has a row and column number. In order to display images with GIS-data, it is necessary to establish an image-to-world transformation that converts the image coordinates to real-world coordinates.



This transformation information is stored with the image.

Some image formats, such as GeoTIFF, and ESRI grids, store the georeferencing information in the header of the image file. MIKE URBAN uses this information if it is present. However, other image formats store this information in a separate ASCII file. This file is generally referred to as the world file, since it contains the real-world transformation information used by the image. World files can be created with any editor.

2.15.2 World file naming conventions

It is easy to identify the world file which should accompany an image file: world files use the same name as the image, with a "w" appended. For example, the world file for the image file mytown.tif would be called mytown.tifw.

2.15.3 How georeferencing information is accessed

The image-to-world transformation is accessed each time an image is displayed (e.g., when you pan or zoom). The transformation is calculated from one of the following sources, listed in order of priority:

- the world file
- the header file (if the image type supports one)
- from the row/column information of the image (an identity transformation)

Because a world file has higher priority, you can override the header file transformation information by creating your own world file.

2.15.4 World file contents

The contents of the world file will look something like this:

```
20.17541308822119
0.000000000000000
0.000000000000000
20.17541308822119
424178.11472601280548
4313415.90726399607956
```

When this file is present, MIKE URBAN performs the image-to-world transformation. The image-to-world transformation is a six-parameter affine transformation in the form of:



$$x1 = Ax + By + C$$

$$y1 = Dx + Ey + F$$

where:

$x1$ = calculated x-coordinate of the pixel on the map

$y1$ = calculated y-coordinate of the pixel on the map

x = column number of a pixel in the image

y = row number of a pixel in the image

A = x-scale; dimension of a pixel in map units in x direction

B, D = rotation terms

C, F = translation terms; x, y map coordinates of the center of the upper-left pixel

E = negative of y-scale; dimension of a pixel in map units in y direction

NOTE: The y-scale (E) is negative because the origins of an image and a geographic coordinate system are different. The origin of an image is located in the upper-left corner, whereas the origin of the map coordinate system is located in the lower-left corner. Row values in the image increase from the origin downward, while y-coordinate values in the map increase from the origin upward.

The transformation parameters are stored in the world file in this order:

20.17541308822119 - A

0.0000000000000000 - D

0.0000000000000000 - B

-20.17541308822119 - E

424178.11472601280548 - C

4313415.90726399607956 - F

2.16 3D Viewer

This feature, which is invoked by choosing View|View 3D, provides a 3-dimensional view to the network whether it is a MOUSE, SWMM or EPANET



model. Using this feature requires the Visualizer add on module and the 3D Analyst Extension to the ArcMAP license.

If any network features are selected only those will be shown in the 3D Viewer, otherwise all network data. For large networks 3D viewing is computationally demanding and response in the view may be too slow. Selecting part of the network will speed up the response.

The 3D Viewer will visualize the following features:

MOUSE and SWMM

- Links
- Nodes
- Weirs
- Pumps

EPANET

- Pipes
- Junctions
- Pumps
- Valves
- Tanks

Additionally, the 3D View will also show raster layers shown in the map view. This is mostly useful if the raster data holds elevation data such as a DEM. Showing non-elevation raster data such as aerial photos will lead to a distorted vertical scaling.

Once View|3D view is selected a new window for 3D visualization will appear, see Figure 2.33.

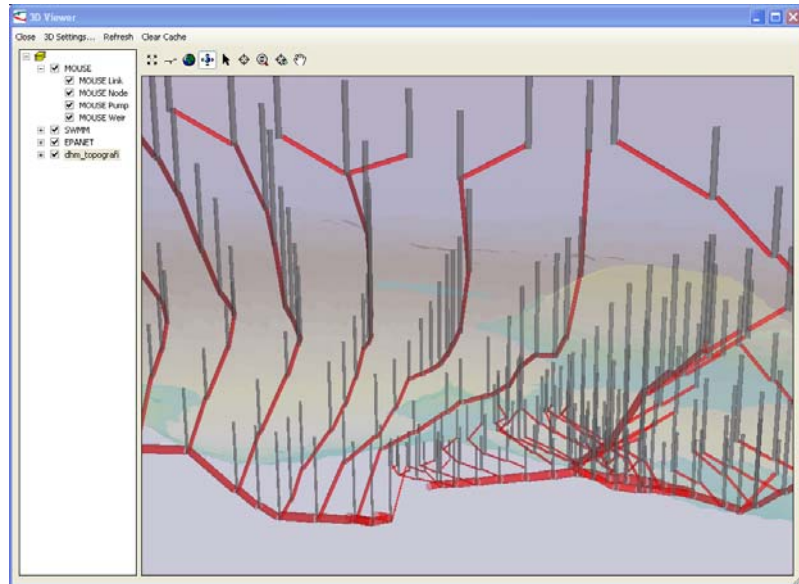


Figure 2.33 3D View of network

As for the map view, drawing of features can be switched on and off through the TOC.

The buttons in the toolbar above the 3D view can be used for panning, zooming and rotating the view. Pressing the 'sea gull' button on the toolbar will allow you to 'fly' through the system. Pressing the 'Esc' button on the keyboard will stop the flying.

The main menu bar has the following options:

- Close. This will close the window.
- 3D Settings. This will open the dialog shown in Figure 2 allowing for adjusting colors, transparency and scaling factors for diameters and elevations.
- Refresh. This will redraw the view using data stored in the cache.
- Clear cache. This will clear the cache holding information on data shown. Subsequent clicking on the refresh button will force data to be reloaded from the MIKE URBAN data tables.

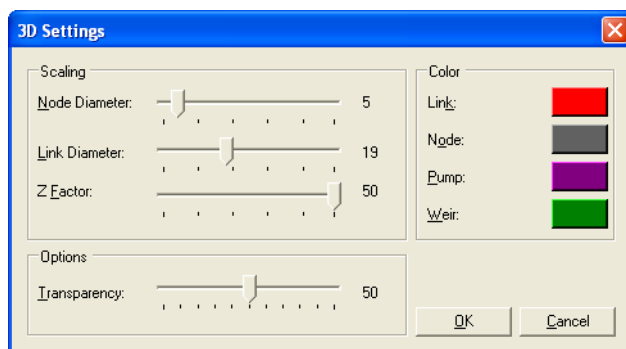


Figure 2.34 3D Setting for 3D Viewer

2.17 Overview Mode

The purpose of Overview Mode is to enable navigation in very large networks, where the time to update the map becomes significant. In Overview Mode, only pipes above a certain diameter are displayed on the map. This diameter is called the Limit Pipe Diameter.

The workflow when Overview Mode is enabled is to zoom to full extent, then use the large pipes shown in Overview Mode to navigate to an area of interest. Once zoomed into an area of interest, Overview Mode can quickly be disabled to show all pipes.

In order to use Overview Mode for quick navigation of large networks, the display of items that take significant time to draw should be disabled by unselecting them in the Table of Contents.

Note that the diameter (“Width” for Rectangular pipes and “Size” for the other pipes) is used for filtering of pipes in Overview Mode. CRS pipes and Natural Channels will always be displayed.

Overview Mode can only be enabled when Edit Mode is not active. This is done right-clicking on the map. A menu where "Overview Mode" can be selected will then pop up. When Overview Mode is enabled, the "Overview Mode" item in the menu is shown as enabled.

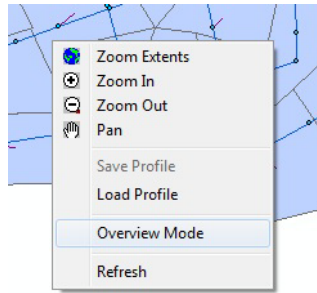


Figure 2.35 Enabling Overview Mode

When Overview Mode is enabled, it can be disabled by right-clicking on the map and by selecting the "Overview Mode" item.

The Overview Mode Limit Pipe Diameter can be configured in General Settings:

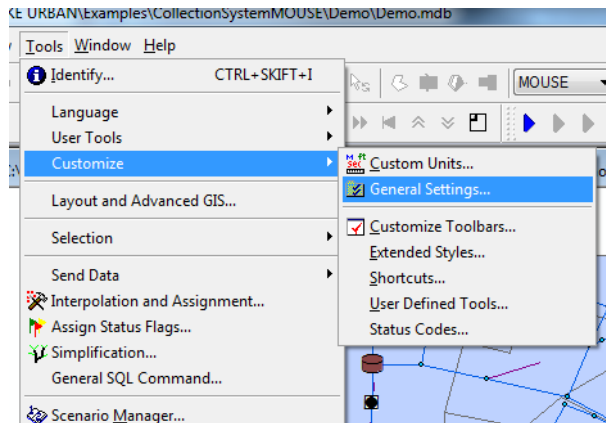


Figure 2.36 General Setting Menu

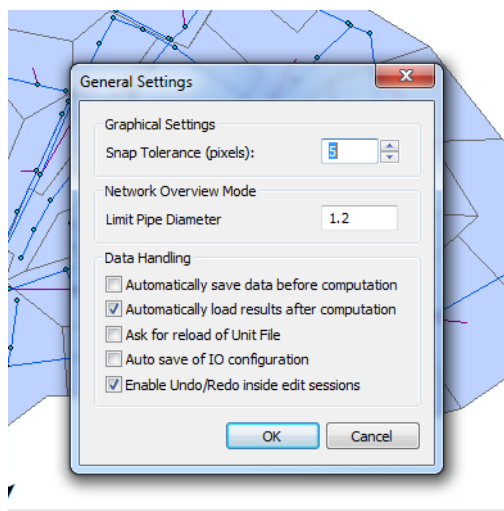


Figure 2.37 Configuring the Limit Pipe Diameter



3 Customizing MIKE URBAN

MIKE URBAN is highly customizable with respect to e.g.:

- Choice of unit system and default values
- Choice of language
- Set up of toolbars

3.1 Units, Default Values and Numeric Formats

MIKE URBAN is fully flexible concerning the applied units for numeric attributes, number of decimals and default values for any attribute in the database. The system allows for a selection between twelve predefined environments and provides a possibility for individual adjustments. By these means, any MIKE URBAN user can set up the current MIKE URBAN project exactly as it fits into established corporate and/or national standards and conventions, as well as his personal preferences.

The actual unit environment is valid for entire MIKE URBAN project, i.e. both for water distribution and collection system parts.

Indeed, in addition to maintain the data in the geo-database and presenting the computational results in wanted units, the system ensures proper unit conversions during import of the existing projects from the involved modeling systems (MOUSE, SWMM5) without any interference by the user. Equally, the system takes care that the project data, when submitted to the computation, are converted automatically into the unit system "understandable" for the respective computational engines.

3.1.1 Selecting an appropriate unit environment

The term "unit environment" stands for a pre-defined set of definitions for units, default values and display formats. There are twelve pre-defined unit environments, divided in two major groups: SI group (stands for SI unit system) and US group (stands for American unit system). Within each of the two groups, there is a number of unit environments which differ from each other by the applied units for flows and volumes.



Within the SI group, the following unit environments are available:

- **SI-m3_per_second:** SI environment, with flows in m³/s
- **SI-liter_per_second:** SI environment, with flows in L/s
- **SI-liter_per_min:** SI environment, with flows in L/min
- **SI-million_liter_per_day:** SI environment, with flows in ML/day
- **SI-m3_per_hour:** SI environment, with flows in m³/h
- **SI-m3_per_day:** SI environment, with flows in m³/day
- **SI-CUSTOM:** SI environment, where unit system resembles MOUSE's SI (flows in m³/s), SWMM5's CMS (flows in m³/s) and MIKE NET's LPS (flows in L/s).

Within the US group, the following unit environments are available:

- **US-feet3_per_second:** US environment, with flows in cfs
- **US-gallon_per_min:** US environment, with flows in gpm
- **US-million_gallon_per_day:** US environment, with flows in MGD
- **US-million_UKgallon_per_day:** US environment, with flows in IMGD (i.e. million imperial gallons per day)
- **US-acre-feet_per_day:** US environment, with flows in AFD

Additionally, this list may be extended with one additional user environment (reserved name "User_Defined") containing users own definitions.

User selects the wanted unit environment from the list, prior to the creation of a new database.

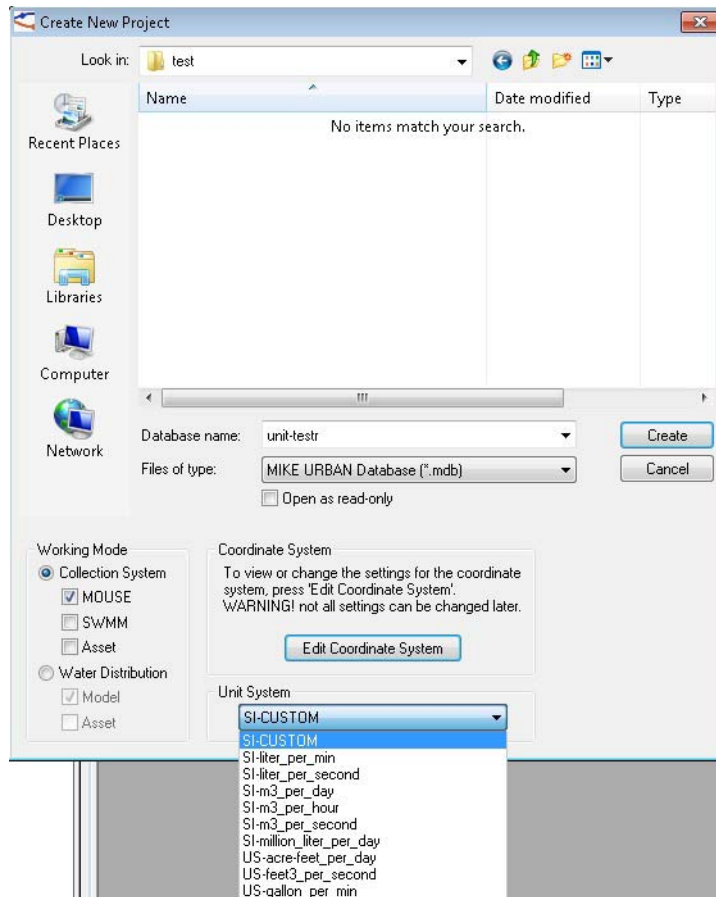


Figure 3.1 Selecting the unit environment

The unit environment becomes a part of the database initial configuration, but can be changed or modified at any stage in the project work through the available tools (Tools|Custom Units). However, the units can not be changed from a SI unit system to a US unit system or vice versa.

Changing an existing database from a SI unit system to a US unit system requires to create a new database with a US environment and then to import the data from the SI database.

3.1.2 Customizing unit environment

Modification of the unit environment configuration in the current project for an individual attribute, or changing the complete unit environment is possible through 'Tools|Custom Units'.

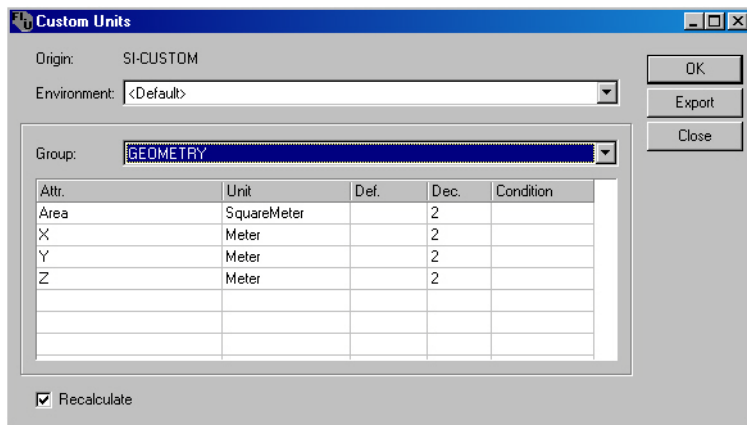


Figure 3.2 Custom Units dialog

On the top of the dialog, the 'origin' unit environment is displayed.

To adjust the configuration for individual attributes, the user has to select the table in which the attribute of concern is located (use 'Group' drop-down box). In the grid below, this displays a complete list of attributes from the selected table. Clicking in the "Unit" field next to the attribute of interest, opens a drop-down box, containing a choice of units available for this attribute, e.g. all length units (meter, millimeter, kilometer, feet, mile....) will be displayed for a pipe length attribute.

It is also possible to change the default value for input parameters as well as the number of decimals to display. When a new network component is created either graphically in the Map view or using one of the network component editors, the input variables for that network component will be assigned values that were defined as default. This can save a great deal of time when dealing with a large network system that has many components that have similar characteristics.

If the tick mark is set in 'Recalculate', the new value reflecting a change of unit will be computed.

The entire database can be recalculated from the current unit environment to another by picking up the wanted environment in the drop-down box 'Environment'.

3.1.3 Saving and Retrieving User Defined Unit Environment

If the user wants to apply the unit environment modifications in future projects, the complete current environment can be exported into the 'User_Defined' unit environment, simply by pressing the 'Export' button, i.e. the way to change your unit system in general is to do the changes, recalculate and save the database and then export to the muEnvUnit_UserDefined.xml file found in C:\Program Files\Common Files\DHICfg.



The exported unit environment is then available for use in a new project after restarting MIKE URBAN. It appears as a new choice ('User_Defined') on the list of unit environments.

3.2 Languages

You can switch language on the fly in your MIKE URBAN application by choosing 'Tools|Language' and then selecting the language of your choice.

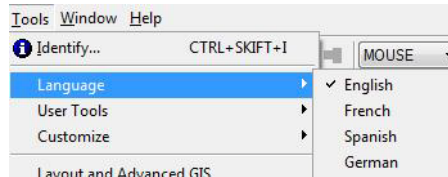


Figure 3.3 Switching language in MIKE URBAN

3.3 General Settings

Through the 'Tools|Customize|General Settings' dialog it is possible to set general properties for the daily work with MIKE URBAN.

- Snap Tolerance (pixels): Specifies the snapping tolerance used when doing graphical editing
- Automatically save data before computation: You need to save data before being able to launch a simulation. When this tick mark is set you can simply press the button to run the simulation and input data will be saved automatically. Caution must be exercised when toggling the tick mark on as unwanted data might be saved.
- Automatically load results after computation: With the tick mark set the simulation results just computed will be loaded automatically.
- Ask for reload of Unit File
- Auto save of IO configuration:

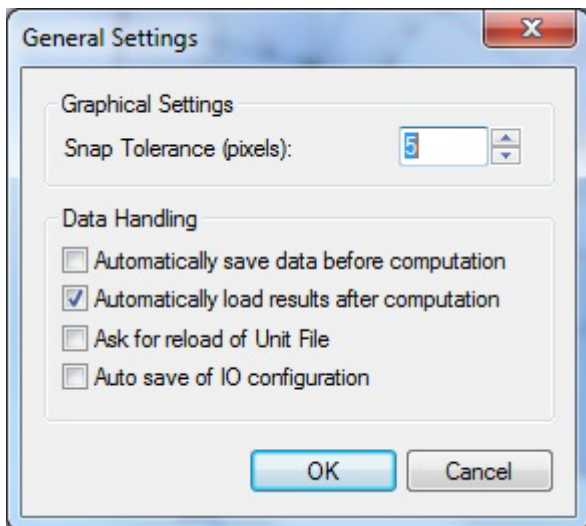


Figure 3.4 The general settings for MIKE URBAN

3.4 Toolbars

Individual toolbars can be switched on and off. MIKE URBAN saves the current toolbar combination for the next MIKE URBAN session. The displayed toolbars get automatically activated or de-activated (grayed-out) according to the presently active graphical window or dialog.

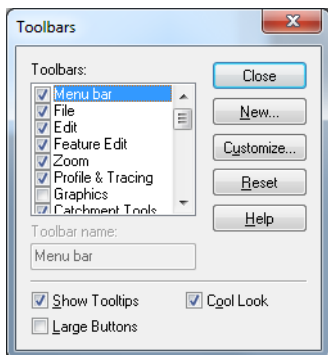


Figure 3.5 View|Toolbars

For customizing your toolbars go to 'Tools|Customize|Customize Toolbars', see Figure 3.6. The toolbars can be switched on and off, as well as the appearance of toolbars (e.g. 'Cool Look') and display of tooltips can be set.

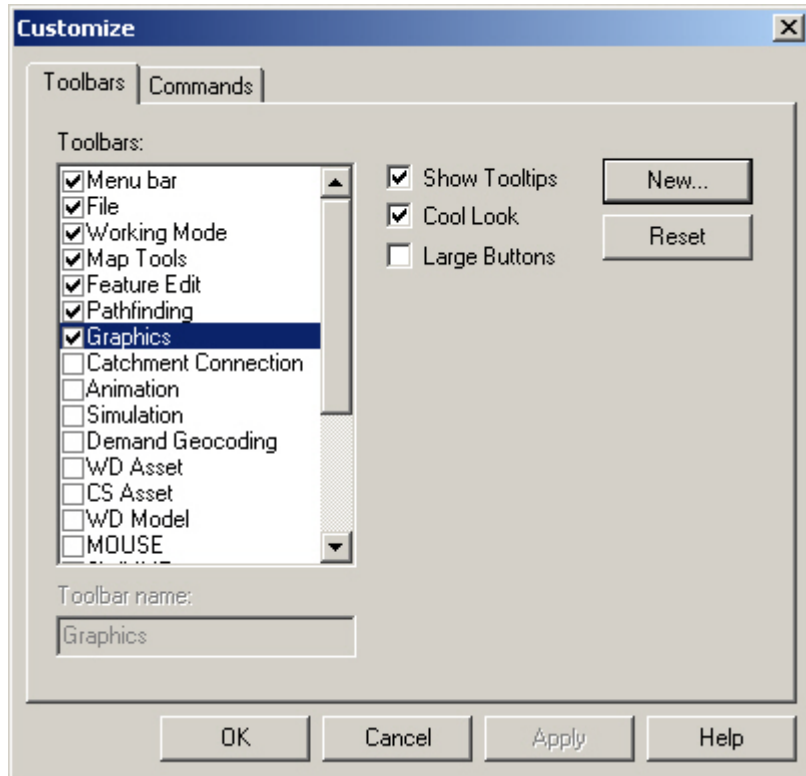


Figure 3.6 Customizing the toolbars

By pressing the ‘New ...’ button a new toolbar may be defined. The dialog shown in Figure 3.7 comes up and you type in the name of your new toolbar. Pressing ‘OK’ will add a new empty toolbar.

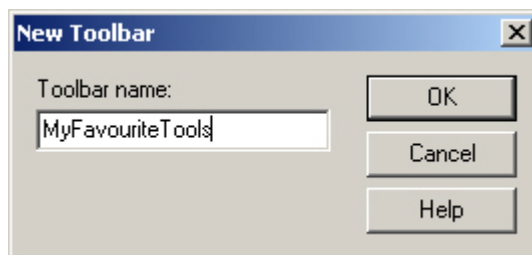


Figure 3.7 Entering name of a new tool bar

Now go to the ‘Commands’ tab page on the customize dialog, here you will find all the buttons available throughout MIKE URBAN, see Figure 3.8. By simply dragging and dropping the buttons from the various categories the can be added to either your own toolbars or any of the predefined toolbars. By dragging a button from the toolbar to the commands tabpage and back to



where the button original came from the button is removed again from the toolbar.

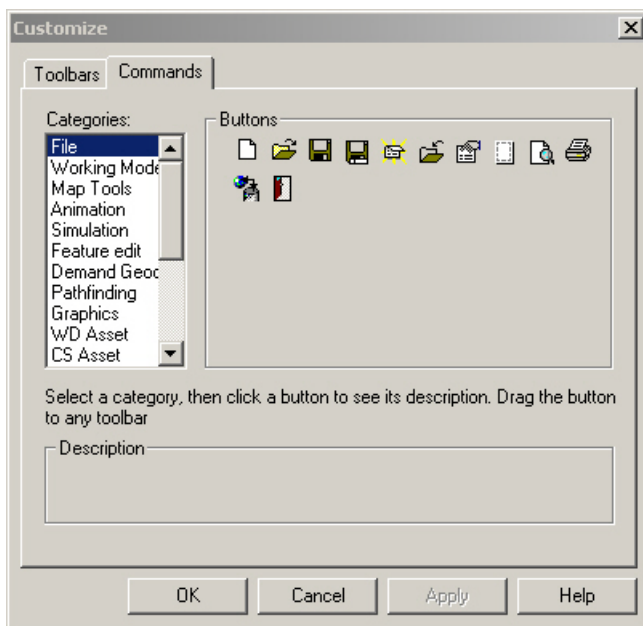


Figure 3.8 Available buttons for the toolbars

Under 'Tools|Customize|Extended Styles' you will find the ability to choose the style of the controls used in MIKE URBAN as well as e.g. determining if a splash screen should come up when launching MIKE URBAN.

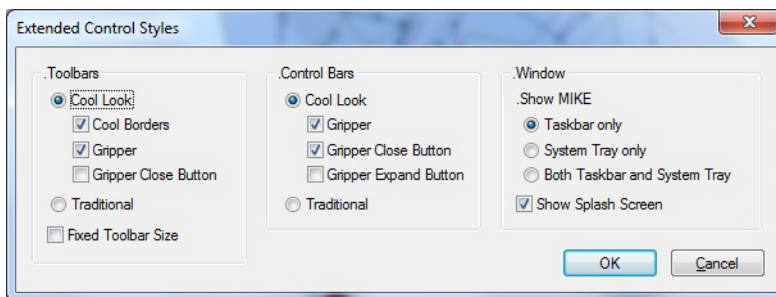


Figure 3.9 Extended Styles - allows you to choose the look and feel of controls and tools

3.5 User Defined Tools

With MIKE URBAN you can use tools developed by yourself (please refer to 3.8) and then add them to the menu under Tools|User Tools. You can also



create a shortcut for another application under the menu point. In below figure MIKE VIEW (free ware result viewer developed by DHI) is added to MIKE URBAN menu to allow for easy access.

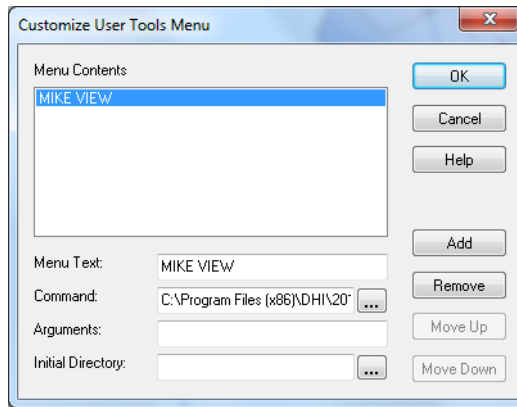


Figure 3.10 Under the Tools|User Tools menu you can create shortcuts to applications for easy access from MIKE URBAN

3.6 Menus

The menus in MIKE URBAN are dependent on the choice of work mode. When in the Collection System mode, you can choose to have the 'Asset', 'MOUSE' and/or 'SWMM5' menu turned on. When in Water Distribution mode you can choose to have the 'Asset' and/or 'EPANET' menu turned on.

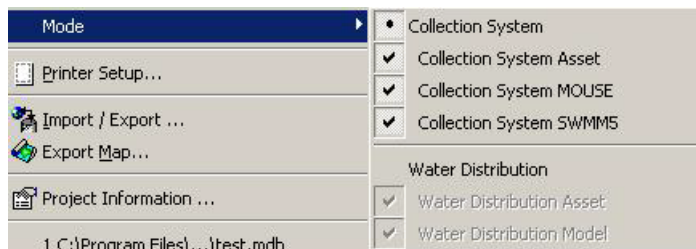


Figure 3.11 Choosing work mode

3.7 Symbols

MIKE URBAN comes with a set of pre-defined symbols for the different elements (nodes, basins, links, valves etc.). You can change these symbols so that MIKE URBAN will use your own preferred symbols. This is done through the ArcMap Style manager:

- Start ArcMap



- Select Tools|Styles|Style manager
- Click on the "Styles" button, pick "Add.."
- Choose the MU.style file - located in the cfg folder of the MIKE URBAN installation folder
- Now you can edit the symbols and save these for the next time you apply MIKE URBAN

3.8 User Written Extensions

An extension provides a mechanism for a developer to add a unit of additional functionality to a MIKE URBAN application.

MIKE URBAN provides an extension framework that developers can use for creating commands and tools for use within the application. This framework works in a very similar way as when creating an extension to an ArcGIS application - even core interfaces are identical.

Extending MIKE URBAN requires knowledge about ESRI's ArcGIS Engine. For more documentation on ArcObjects please go to: <http://arcobjectsonline.esri.com/> .

An extension can provide many different types of functionality such as deliver forms, perform event handling, store data, perform validation and much more.

An extension will be made available as a menu point under Tools | User Tools. The properties of the menu point will be controlled via the ICommand Interface. In MIKE URBAN any extension has to be invoked through a single menu point.

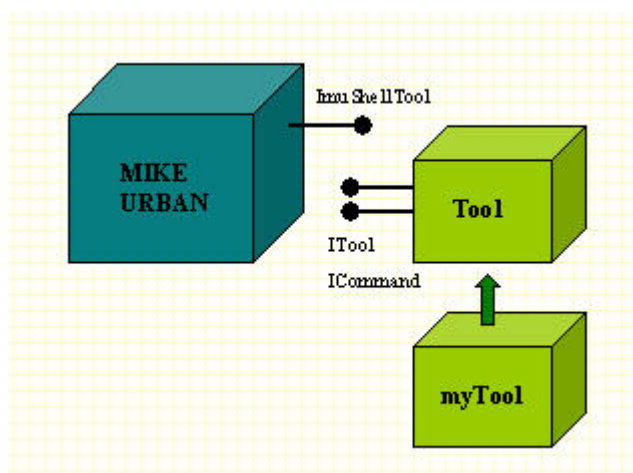


Figure 3.12 Creating extensions to MIKE URBAN



3.8.1 Developing extensions

Extensions are loaded when the MIKE URBAN application first starts up, and there is only ever one instance of an extension per running application

The main steps involved when implementing an extension are:

1. Create external dll in a COM compliant language.
2. Implement the following interfaces.
 - ICommand (provides hook into MIKEURBAN through ImuShellTool)
 - ITool (optional)
3. Register you extension with MIKE URBAN through the mu.xml file.
4. Invoke your extension within MIKE URBAN through Tools | User Tools | 'your ext'

3.8.2 Interfaces

ICommand

Provides access to members that define a COM command. Use the ICommand interface to create your own menu item. A simple action is performed when the menu item is clicked. The ICommand interface sets command properties such as caption, enabled state and obtains a hook into the application. It also defines what action happens when the command is clicked.

Table 3.1 ICommand properties

Property	Type	Supported	Description
Bitmap	int	No	
Caption	string	Yes	The Caption of the command (menu point)
Category	int	No	
Checked	bool	No	
Enabled	bool	Yes	Indicates if the menu point is enabled. A system event is periodically called to set the state of the commands, indicating which ones are enabled.
HelpContextID	int	No	
HelpFile	string	No	
Message	string	No	
Name	string	No	



Table 3.2 ICommand methods

Methods	Supported	Description
void OnClick ()	Yes	When implementing ICommand to create a custom command, write the code that performs the action when the command is clicked.
void OnCreate (Object Hook)	Yes	When implementing ICommand to create a custom command, use the OnCreate method to get a hook to the application.

ITool

Tools are similar to buttons but they also require interaction with the application's display. When you are creating a new COM tool, you need to implement both the ICommand interface and the ITool interface in your class code. With the ITool interface you can define what occurs on events such as mouse move, mouse button press/release, keyboard key press/release, double-click, and right click.

Table 3.3 ITool properties

Table 3.4

Property	Type	Supported	Description
Cursor	int	Yes	The mouse pointer for this tool. Once set, the cursor is constant through the lifetime of the tool. No system event is periodically called to check the state of the cursor.



Table 3.5 ITool methods

Table 3.6

Methods	Supported	Description
bool Deactivate ()	No	
bool OnContextMenu (int X, int Y)	Yes	Context menu event occurred at the given x,y location. X is the X coordinate, in device units, where the mouse button was pressed.Y is the Y coordinate, in device units, where the mouse button was pressed.
void OnDbClick ()	Yes	Occurs when a mouse button is double clicked when this tool is active.
void OnKeyDown (int key-code, int shift)	Yes	Occurs when a key on the keyboard is pressed when this tool is active. keyCode specifies which key on the keyboard was released. See further down.
void OnKeyUp (int keycode, int shift)	Yes	Occurs when a key on the keyboard is released when this tool is active.keyCode specifies which key on the keyboard was released. See further down.
Void OnMouseDown (int button, int shift, int x, int y)	Yes	Occurs when a mouse button is pressed when this tool is active. See further down.
Void OnMouseMove (int button, int shift, int x, int y)	Yes	Occurs when the mouse is moved when this tool is active. See further down.
Void OnMouseUp (int button, int shift, int x, int y)	Yes	Occurs when a mouse button is released when this tool is active. See further down.
Void Refresh (int hdc)	No	



Shift and Button, X and Y arguments:

Shift indicates whether the Shift key, Ctrl key, or Alt key is pressed when the mouse button is pressed.

The Shift argument is a bit field. Some, all, or none of the bits can be set indicating that some, all, or none of the keys are pressed. The values for Shift are as follows:

- 0 - No key pressed
- 1 - Shift key pressed
- 2 - Ctrl key pressed
- 3 - Shift and Ctrl keys pressed
- 4 - Alt key pressed
- 5 - Shift and Alt keys pressed
- 6 - Ctrl and Alt keys pressed
- 7 - Shift, Ctrl, and Alt keys pressed

Button specifies which mouse button is pressed.

The button argument is a bit field. Some, all, or none of the bits can be set indicating that some, all, or none of the mouse buttons are pressed. The values for button are as follows:

- 1 - Left button
- 2 - Right button
- 3 - Left and Right buttons
- 4 - Middle button
- 5 - Left and Middle buttons
- 6 - Right and Middle buttons
- 7 - All buttons

X is the X coordinate, in device units, where the mouse button was pressed. Y is the Y coordinate, in device units, where the mouse button was pressed.

ImuShellTool

The ImuShellTool interface provides a hook into the MIKEURBAN application. When you are creating a new COM extension, you need to cache this interface in order to access the main application interfaces such as IMap, IWorkspace and the network property.



Table 3.7 ImuShellTool properties

Table 3.8

Property	Type	Supported	Description
CurrentNetwork	String	Yes	<p>An indication of which network is currently the active in the application.</p> <p>mss_Net = SWMM network</p> <p>msm_Net = MOUSE network</p> <p>msa_Net = ASSET network</p> <p>mw_Net = WATER network</p> <p>mwa_Net = WATER ASSET network</p>
HWnd	int	Yes	A handle for the main window in the application
Map	IMap	Yes	<p>Provides access to members that control the map.</p> <p>The IMap interface is a starting point for many of the tasks one does with a Map. For example, use IMap to access map layers containing data from various sources including feature layers and graphics layers; access the various properties of a Map including the area of interest, the current map units, and the spatial reference; select features and access the Map's current selection.</p>
WorkSpace	IWorkspace	Yes	<p>Provides access to members that have information about the workspace.</p> <p>A Workspace is a container of spatial and non-spatial datasets such as feature classes, raster datasets and tables.</p> <p>The IWorkspace interface provides methods for accessing various properties of a workspace, such as its connection properties, and the collection of datasets it contains.</p>



3.8.3 Example

This example demonstrates the implementation of a simple zoom tool to MIKE URBAN.

The first task is to create a dll in a COM compliant language. In this example C# and the COM interoperability of the .Net framework is used. The class for the tool implements the ICommand and ITool interfaces. To create any type of command in C# for use in ArcGIS, you need to implement ICommand. Since we are creating a tool here, we also need to implement the ITool interface. Even if you don't need to write code for all the properties and events on these interfaces to meet the requirements of your command design, you must at least create stub code for all of properties and events.

Creating the COM library

The sample tool in this example replicates the existing Zoom In tool, where a user clicks on the display and drags a rectangle over the map display to define the area on which to zoom. To create the tool, create a new class library muExtension and implement both ICommand and ITool in a class myTool:



```

using System;
using System.Windows.Forms;
using System.Runtime.InteropServices;
using ESRI.ArcGIS.SystemUI;
using ESRI.ArcGIS.Carto;
using ESRI.ArcGIS.Display;
using ESRI.ArcGIS.Geometry;
using DHI.MIKEURBAN.mutShellShared;

namespace MUEExtension
{
    [ClassInterface(ClassInterfaceType.None)]
    [Guid("1D77A1D3-EFDE-4008-A28E-54D1BEB7CCED")]

    /// <summary>
    /// Summary description for Class1.
    /// </summary>
    public class myTool: ICommand, ITool
    {
        ImuShellTool _hook;

        public myTool()
        {
            //
            // TODO: Add constructor logic here
            //
        }
        ICommand Members
        ITool Members
    }
}

```

Please note that the assembly DHI.MIKEURBAN.mutShellShared.dll contains the type information about the ImuShellTool interface. This assembly is installed with MIKE URBAN and can be found in: C:\Program Files\DHI\MIKE URBAN\Bin.

Implementing the ICommand & ITool

You don't have to write code for all the properties and methods of the interfaces as long as the stub code is in place. In this example we will only implement the parts that are required for this particular tool.

In order for MIKE URBAN to be able to recognize the new tool, you have to implement the Caption property on the ICommand interface:



```
public string Caption
{
    get
    {
        return "myTool";
    }
}
```

The Enabled property is used to specify in what state the application should be in for the command to be enabled. It is important to minimize the amount of code that goes here because the system calls this property very often. We want our tool to be enabled all the time:

```
public bool Enabled
{
    get
    {
        return true;
    }
}
```

The OnCreate method occurs just after the command is instantiated and provides a hook to the application object that instantiated the command. Once you have this reference to the application object, you can access the other objects in the application. This reference is stored in a member variable:

```
public class myTool:ICommand, ITool
{
    ImuShellTool _hook;
```

The type of hook received by a tool may depend upon how the tool was instantiated, and the application which is hosting the command. In MIKE URBAN the hook will always be a ImuShellTool reference, but If you intend your command to be used not just in MIKE URBAN but also with the ArcGIS MapControl, PageLayoutControl, and ToolbarControl, you may wish to consider using an encapsulating object for storing your reference. Such an object will return references depending on what type of application your tool is instantiated from. In this case we just design the tool to work from MIKE URBAN:



```
public void OnCreate(object hook)
{
    _hook = hook as ImuShellTool;
}
--
```

The ITool interface differentiates a tool from a command, and allows a tool to interact with the application's display. Only one tool can be active in the application at a time. Using the members of ITool you can define what occurs on events such as mouse move, mouse button press and release, keyboard key press and release, double-click, and right-click, when your tool is active.

To add the zoom in functionality to your SampleTool, add code to the Mouse-Down method to allow the user to track a rectangle that defines the new extent of the view.

```
public void OnMouseDown(int button, int shift, int x, int y)
{
    if (button == 1)
    {
        IActiveView av = _hook.Map as IActiveView;
        IRubberBand rb = new RubberEnvelopeClass();

        av.Extent = (IEnvelope) rb.TrackNew (av.ScreenDisplay, null);
        av.Refresh();
    }
}
```

Registering the myTool with MIKE URBAN

In order to register the new tool, the C:\Program Files\DH\MIKE URBAN\Cfg\mu.xml file has to be updated with the name of the dll and the name of the class implementing the tool. The entry can be done either with class ID or component name:

```
<Group Name="MUTools">
    <Metadata>
        <Field Name="clsid" Type="Char(255)"/>
    </Metadata>
    <Rows>
        <R clsid="MUExtension.myTool"/>
    </Rows>
```

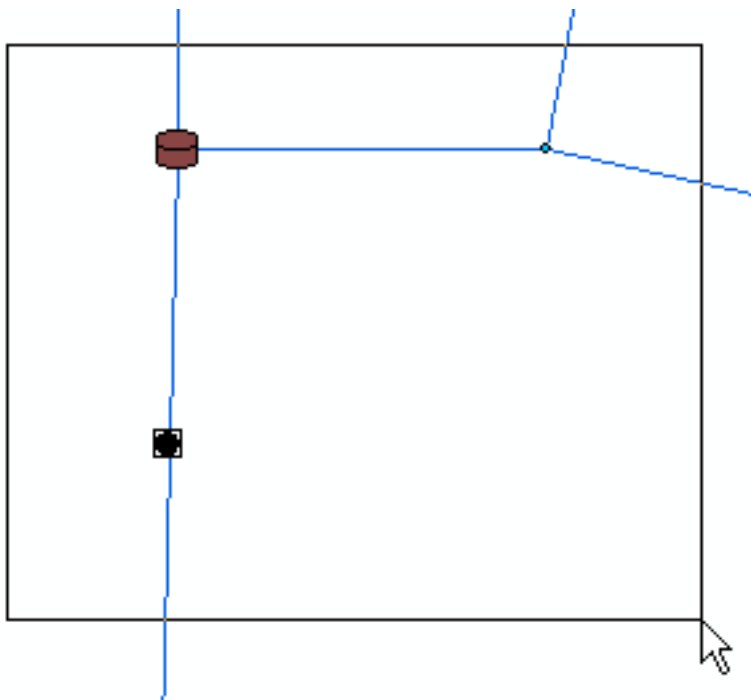


</Group>

Using the tool

When starting up, MIKE URBAN will create an instance of your tool and the new tool will appear under Tools|User Tools.

To activate the tool; select the tool from the menu point that you want to set as the current active tool. Use the 'User Tool' button on the feature edit toolbar to invoke the current active tool. The 'User Tool' button always refer the whatever tool is selected, only one Extension can be active at any time. To use the tool, press the 'User Tool' button and drag a rectangle in the map.





4 Internet Links and updates

You can easily access information on MIKE URBAN on the Internet. Two direct links are made available from within MIKE URBAN to the Internet.

[‘Help|MIKE URBAN on the Web ...’](#)

Provides access to information on MIKE URBAN on the web.

[‘Help|MIKE URBAN Tools and Utilities ...’](#)

Provides access to page where MIKE URBAN users can share their tools and utilities to be used from within MIKE URBAN.

[‘Help|MIKE URBAN updates on the Web ...’](#)

Provides direct access to the MIKE URBAN download area on the web.





5 Linking to ArcMap

ArcMap is the main desktop program from ESRI. Depending upon your license conditions you will have a number of possibilities of amending the functionality of MIKE URBAN with the more general functionality found in ArcMap.

MIKE URBAN operates on top of a personal geodatabase and stores all data in a designated data structure. This data structure is further documented in 'MIKE URBAN Tables' and in the overview diagrams.

As ArcMap is a general GIS desktop application it is important to realize that ArcMap does not support the data consistency checks and other protection mechanisms normally performed by the MIKE URBAN shell program.

The following precautions should be recognized:

Data Viewing

No restrictions. As long as ArcMap is not taken into 'edit mode' there is no danger of corrupting the database. Examples of tasks/objectives that can be accomplished in viewing mode are:

- advanced GIS analysis
- advanced presentations of model and saved data
- complete lay-out tasks for inclusion in final reporting.
- Ad-hoc analysis requiring the user to write scripts in VBA or similar.

Data Editing

Uncritical editing of data in the MIKE URBAN database may easily corrupt the database and make it unusable in MIKE URBAN. However, very powerful tools exist in ArcMap that may be utilized for editing. Provided that you take care to obey the constraints, editing from ArcMap can be very powerful.

In general the following rules apply:

Editing Geometry - In general the editing of geometry (pipe shapes, manhole placement, and catchments) can be done without danger of corruption to the data structure. This is probably also the area in which there is most benefit from working in ArcMap, as the editing tool here are more advanced than in MIKE URBAN.

Altering attributes - In general this can be done to most attributes without danger, EXCEPT when the attribute is an ID field used to maintain consistency of the database. To avoid this you may consult the datamodel appendix and as a general do not touch any attribute called something with 'ID'.

Inserting and deleting records should be avoided unless you have a very good knowledge about the MIKE URBAN data structure. Deleting or inserting in MIKE URBAN is often triggering a number of background operations that



updates other tables so the consequence of doing such operations outside MIKE URBAN may easily be a corrupt database.

5.1 Typical ArcMap Tasks

Typical tasks that may be done in ArcMap are:

- Complex editing and data analysis requiring joining and relating several tables from various data sources
- Tasks involving complex spatial analysis, spatial joining of data.
- Geoprocessing tasks like intersecting, overlaying etc. apart from the pre-defined tasks existing in MIKE URBAN.
- Working with lay-outs, plotting to scale and similar high-level reporting.
- CAD-style editing
- Further some organizations have developed in-house tools in ArcMap to perform certain functions. If these functions need to be performed on MIKE URBAN data, you will need to run the scripts from ArcMap.

5.1.1 Example

The following is an example on how to work on a MIKE URBAN database in ArcMap using the powerful join functionality in ArcMap.

The specific example will enable calculation of the impervious areas upstream of a node. The collection system model used is a MOUSE. For details on the steps that take place in MIKE URBAN, please refer to the relevant sections on tracing and showing connected catchments. In this example it is the assumption is that the setup uses Runoff Model A.

The necessary steps in MIKE URBAN:

1. Choose a node in MIKE URBAN and make a backward trace
2. Show the connected catchments
3. Go to Tools|Advanced GIS and Layout

This will open up ArcMap with the setup - including the selections made in MIKE URBAN. Now we would like to add a column to the catchment table that can be used for storing the impervious area. In order to add a column to the table only one application can access the database, so you will need to close MIKE URBAN after saving your edits before continuing.



Adding a column to the catchment table (ms_Catchment):

1. Open the attribute table of the catchments (right-click on the Catchments layer in the and choose 'Open Attribute Table')
2. Click the Table Options button in the table button
3. Click 'Add Field'
 1. Type the name of the field - e.g. CalcImpArea
 2. Click the 'Type' arrow and click the field type (choose Double)
 3. Click 'OK'

The column had now been added to the table, so you can close the attribute table. The next step is to join the catchment table (ms_Catchment) with the runoff model A table (msm_HModA). This is done because the percentage of the catchment that is specified as impervious is information found in the runoff model A table and not in the catchment table.

Joining the tables:

1. Using the join dialog (right click on the catchment layer in TOC and select Joins and Relates| Join) we now join the ms_catchment table to the msm_HModA using the Catchment ID as the common key. You will need to browse for the msm_HmodA

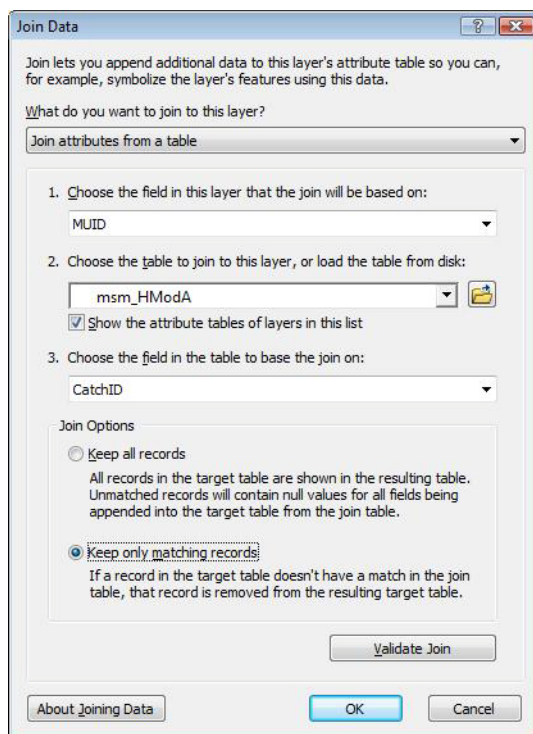


Figure 5.1 Joining data

The tables are now joined, so when you open the attribute table for the catchment layer it contains several new columns. In order to compute the impervious area you do the following:

1. Enter Edit mode (right-click on the Catchments layer and choose 'Edit Features' and 'Start Editing')
2. Open the attribute table of the catchments (right-click on the Catchments layer in the and choose 'Open Attribute Table')
3. Find the column you added (CalcImpArea) and right click in the column header, select the 'Field' calculator
4. Compute the impervious area as seen in Figure 5.2. The field calculator works on the selected rows.

In order to have the total impervious area of the selected catchments right-click on the column with the computed impervious areas and choose 'Statistics'. The Sum field will provide the total impervious area of the selected catchments.

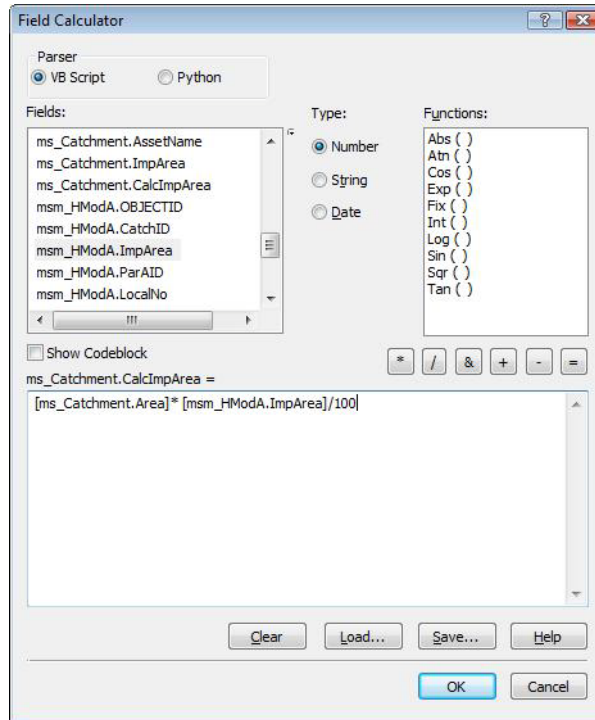


Figure 5.2 Using the field calculator for calculating the impervious area
 In similar way a number of operations can be made using ArcMap.





6 MIKE URBAN data model

6.1 MIKE URBAN Networks

MIKE URBAN Project database includes the data models for both water distribution and collection system networks. Moreover, inside the realm of these two distinct systems, there is the possibility to study the respective networks independently in several instances, each representing a fully-featured, independent network.

On the Water Distribution (WD) side, MIKE URBAN includes the ASSET network and the MODEL network. For Collection System (CS), the three available networks include ASSET network, MOUSE model network and SWMM5 model network. This makes a total of five networks.

ASSET networks are intended to contain the full topology (i.e. network layout) based on the actually available network asset information and a selection of attributes for the network elements. ASSET network should actually represent the best possible image of the actual network, in the environment much less rigorous than numerical models, with respect to missing information, model parameters, flexible element categorization, etc. ASSET network can be an image of either an external data source (e.g. GIS) or of the network itself. Actually, MIKE URBAN ASSET may serve as a primary data storage for urban water distribution and collection networks and a source of data for modeling.

A MODEL network represents a urban water network in a form as expected by the computational model engine. As such, it is subject to very strict model-specific data requirements, definite and fixed catalogue of element types (normally different than in the ASSET!), simplifications, etc. A MODEL network typically includes elements of significantly different character than in the ASSET and frequently, affective elements required for the model consistency, which inevitably leads to the modifications of the original network topology.

An example of a typical situation concerning the differences in data for the same segment of a collection network is illustrated in Figure 6.1. While in external sources (left) there is often no relation

between the network nodes and links (i.e. no actual network structure), after moving into MIKE URBAN CS Asset a network structure gets established. The same segment of the network, when transferred to the MOUSE model network (CS MOUSE) gets significantly modified topology and additional elements (e.g. pump function). In the continuation (not shown on the illustration), the model network can be simplified by removing the node "NODE B".

With the present architecture, MIKE URBAN handles this and similar situations through import routines and data processing within a single project database, with the possibility to overlay various "versions" of the network in the same project map.

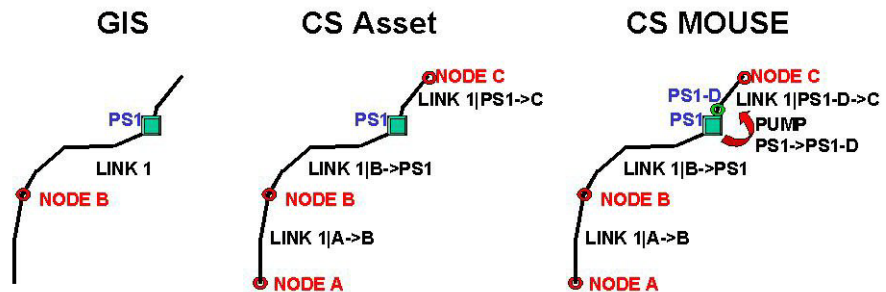


Figure 6.1 Typical modifications of collection network topology when moving from external GIS (left) into CS Asset (centre) and to CS MOUSE. Additional step is related to model network simplification

Tools are available for automated transfer of the data between the networks within WD and CS, respectively (Tools|Send Data). Initially, after the transfer, the data in the source and target networks are identical concerning the topology and plain attributes common to both networks. Where not directly comparable, the data transfer tools provide automatic re-interpretation of features and attributes.

The consistency of these three "versions" is maintained through a systematic subsequent application of the data transfer tools in "update" mode. However, it should be noted that the tools are capable of keeping the network topologies and the database attributes up-to-date, only as long as the individual elements identifiers are maintained identical throughout all the involved networks.

It should be understood that the utilization of up to three parallel networks in CS (and two in WD) is fully optional. I.e. the system does not "prescribe" any specific project organization: it is fully up to the user preference and up to the purpose of the actual MIKE URBAN project, which network combination will be used. The data from external sources can be imported into any of the net-



works and the data can be moved between any two networks in both direction at any time. Of course, it is not possible to transfer the network data between WD and CS.

Figure 6.2 illustrates a possible layout in a MIKE URBAN CS project. Actually, the same collection network is present in three versions: ASSET, MOUSE model and SWMM5 model. Each of the networks can be edited independently through a set of dedicated graphical editing tools and data editors. If required, i.e. if the performed editing is of general significance (e.g. inserting of missing data, corrections, etc.), the edits can be easily transferred between any two networks.

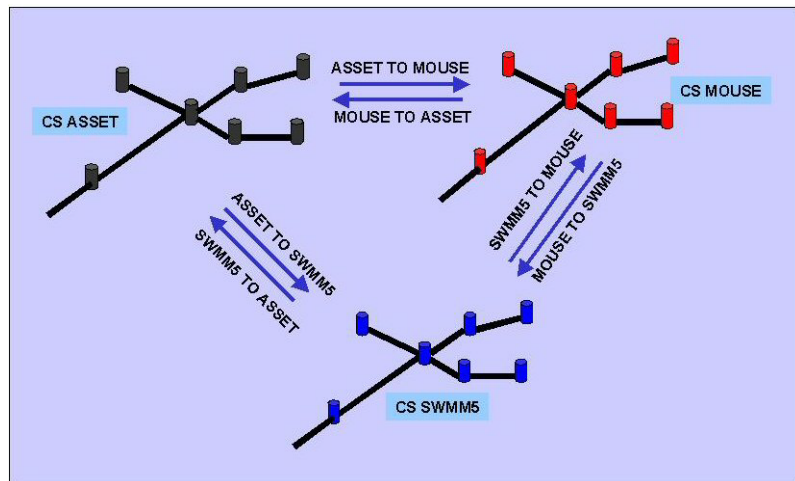


Figure 6.2 Schematic presentation of the three independent networks in MIKE URBAN CS with "Send Data" tools for easy transfer of data and consistency maintenance between any of the two networks

In conclusion, the MIKE URBAN system architecture with ASSET and MODEL networks allows for the MIKE URBAN system to be used for other purposes than numerical modeling, e.g. for urban water asset registration, development planning, presentations and, ultimately, for full urban water asset management. Even if "asset management" is not an issue, the MIKE URBAN architecture provides a reliable "bridge" between the existing databases/asset management systems and numerical models.

6.2 Data Model Structure

The MIKE URBAN data storage is divided into a number of data stores. The main storage for the model data is the Geodatabase. But a number of additional files and data stores define the MIKE URBAN functionality such as binary result files and configuration files.



6.2.1 Terminology

Geodatabase

This is a database with a GIS data structure. It is used to hold the model data such as a physical network description and other physical data, input data to the various numerical engines as well as general setup information. A Geodatabase may hold only one instance of a particular model (however one database may hold both water distribution and collection system data). The format of the Geodatabase is also the basic data format used by ESRI GIS systems (ArcGIS), it may thus be opened and operated upon using native GIS tools.

The project (MUP) file

This file holds all information about the current user setup. A project file will thus hold the individual settings for the user such as a symbology for the GIS, pointer to added background data etc.

Configuration and import bridges

These files are normally not touched by the user but may in certain circumstances be changed to fit individual user setups.

Time series and result data

Binary data in the form of input timeseries data as well as result data are stored externally in binary data files.

6.2.2 Geodatabase basics

The MIKE URBAN geodatabase is a so-called 'Personal Geodatabase', this means that it is based upon the Microsoft ACCESS database and that it normally resides on the PC where it is being used.

The Geodatabase is very much like a normal database consisting of tables having columns (or fields), what makes it special are:

- The geodatabase contains a predefined datastructure needed for GIS to operate correctly on the database
- Some tables are called 'feature classes' because they contain a special binary formatted column defining the spatial geometry of the object (row)

This means that you can operate with ACCESS/EXCEL or similar tools on the Geodatabase, but it also means that you must be very careful if you try to manipulate data with such tools, always have backup.



6.2.3 The Geodatabase and the Scenario Manager

The Geodatabase may contain a variety of scenarios of each model. These scenarios are managed by the MIKE URBAN scenario manager. Scenarios are in general stored as difference tables - the use and documentation of these are beyond the scope of this documentation.

It must be noted however that when opened with any tool the database will represent the current active MIKE URBAN scenario.

6.2.4 The MIKE URBAN database contents

The MIKE URBAN database will contain all model parameters for the active model(s). Data is stored in either

- Feature classes, these are database tables with a spatial contents (such as pipes, nodes etc.)
- 'standard' database tables i.e. tables without a spatial content.

All feature classes are collected in the feature dataset named mu_Geometry, the dataset ensures that all the MIKE URBAN feature classes share the same spatial reference.

Naming Convention

All feature classes and tables follow the same naming convention:

Table 6.1 Feature classes and tables - naming conventions

Table	Description
m_	means that the table is a general MIKE URBAN table covering all parts of MIKE URBAN (typical configuration information)
ms_	means that the table belong to the collection systems part of MIKE URBAN i.e. common to both MOUSE and SWMM models
mw_	are tables belonging to the water distribution part of MIKE URBAN
msm_	are tables specific to the MOUSE model of collection systems
mss_	are tables specific to the SWMM model of collection systems



Table 6.1 Feature classes and tables - naming conventions

Table	Description
msa_	are tables specific to the asset part of collection systems
mwa_	are tables attached to the asset part of water distribution

Information on the individual fields of the database may be found in the sections of the manual describing the individual parameters. Generally it is not recommended to use characters such as '?' and '"' in any unique names (MUID's). Database fieldnames are also shown as tooltip in the individual data editors when hovering over the field. The data models is further documented by means of overview diagrams and detailed table description in the 'MIKE URBAN Data Model' document.



7 Import and Export

7.1 Introduction to MIKE URBAN Import/Export

Import of various data from external systems into a MIKE URBAN project is of high importance when working with your project. Exporting the MIKE URBAN project into various external formats is equally important. As a variety of formats are commonly used for storage and management of urban water systems data, very flexible and versatile import/export tools are required.

MIKE URBAN comes with a number of standard routines for import and export to commonly used formats, which are essential for e.g. importing MOUSE and SWMM5 projects and/or for backing up the database. In addition to this it is possible to create and configure any number of custom import and export jobs.

This chapter provides detailed information on the technical background, the import/export wizard, the advanced mode import/export and the predefined and custom import/export jobs.

Any import/export session is initially carried out through the Import/Export Wizard. The wizard helps the user through the commonly used import/export jobs and guides the user when creating new import/export jobs. Optionally the advanced mode of import/export may be used where a wider range of options are available to the user.

- Section 7.2 provides the common technical background for the import including syntax for assignment, description of transfer modes etc.
- Section 7.3 describes the import/export wizard
- Section 7.4 describes the import/export functionality in the advanced mode.

For examples on setting up a typical import job, please refer to MIKE URBAN Tutorials manual.

7.2 Technical Background for Import and Export Functionality

The common word used for a import/export procedure is 'job'. A job may consist of one or more 'table configurations'. An import/export job will normally consist of several table configurations, making up a complete import/export. Each table configuration in the job relates to an individual table or feature class.

When importing data into a MIKE URBAN database from an external data source, the external data is referred to as the 'source' and the MIKE URBAN database is referred to as the 'target'. For the export jobs, the situation reverses - the MIKE URBAN database is then referred to as the 'source' and



the database or file that you wish to export the MIKE URBAN data to is referred to as the 'target'.

7.2.1 General Description

The procedure of importing/exporting data is done through a generic 'engine' as shown in Figure 7.1.

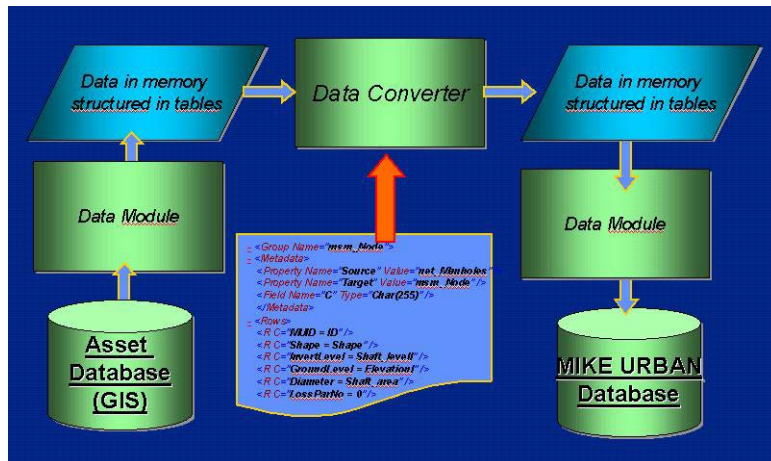


Figure 7.1 Generic presentation of Import/Export process

As the illustration suggests, import and export jobs are always executed following the same process scheme, involving storage drivers ('muStorages') - programs which read data from the source into the computer memory ('cache') and write data from the computer memory i.e. from 'cache' to the target, and a data converter ('muBridge') - a program which 'translates' the data from the source cache into the target cache.

Figure 7.2 illustrates the import/export process in more specific terms.

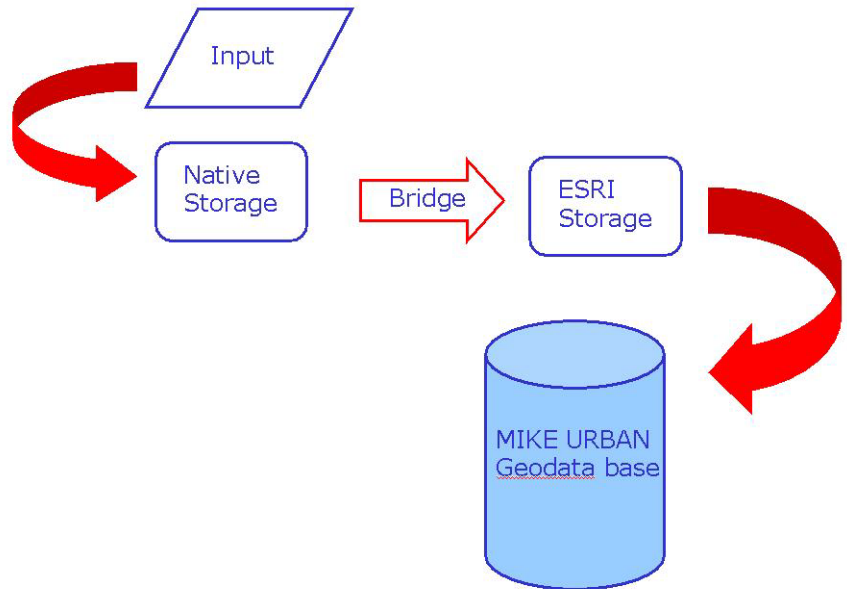


Figure 7.2 Import procedure - From an input source to a MIKE URBAN Geo-data-base

7.2.2 MIKE URBAN Storages

The 'Storage Drivers' or 'muStorages' may be regarded as data drivers that read/write a specific data source (ODBC, ASCII, Shape file etc.) to/from the computer memory. This means that it is possible to read/write only those data formats which are covered by an appropriate storage driver.

MIKE URBAN comes with the following storages drivers:

- EPANET 2.0 Storage: designed to read/write of EPANET 2.0 data files
- ESRI CAD Storage: designed to read/write of CAD data formats supported by ESRI technology, e.g. DXF.
- ESRI Geo-database Storage: designed to read/write of data from/to ESRI Geo-databases (i.e. also MIKE URBAN project databases)
- ESRI Shape Storage: designed to read/write of data from/to ESRI Shape Files
- MOUSE Storage: designed to read data from a MOUSE project (.MPR) and write data to a MIKE URBAN data file (.MEX).
- Simple PFS Storage: designed to read/write of data from/to DHI PFS Files
- Simple XML Storage: designed to read/write of data from/to MIKE URBAN XML Files



- ODBC Storage: designed to read/write of data from/to ODBC-compatible sources (EXCEL files, Access Files, dBase files....). Please note that only user-DSN should be used.
- SWMM5 Storage: designed to read/write of SWMM5 data files
- Oracle Spatial Storage: designed to read/write of data from/to Oracle Spatial databases (1D and 2D Oracle Spatial geometries only)
- The system design allows the implementation of new storages and their usage directly from the general Import/Export dialog. This requires programming skills. For details contact DHI.

7.2.3 MIKE URBAN Bridges

Conversions of data from one storage (format) to another storage is done through a data converter 'muBridgeDM' or by any .com component compliant with the guidelines for building bridges. In other words, the data converter transfers the data between the two memory structures (input/output cache).

MIKE URBAN comes with two data converters:

- Standard MU DM Bridge ('muBridgeDM'): a generalised converter for all types of data modules
- EPANET 2.0 Bridge: a special converter which in conjunction with Standard MU DM Bridge services import/export of EPANET 2.0 files

In the data transfer process several processes may take place including:

- mapping from one attribute to another;
- arithmetic conversion;
- filtering;
- lookup of values in other tables;
- conditional assignment.

The processing engine has access to both the source and the target cache - meaning that it can utilize data already in the database to control the import/export process.

The processing in the data converter is controlled by an XML configuration file. Normally this file is edited through the import/export dialog in MIKE URBAN, but it can be edited using any text editor to achieve special processing not directly supported by the user interface.

The default configuration file, containing the configuration for all predefined import/export jobs, is found in the 'MIKE URBAN\Cfg' folder under the name 'muIO.xml'. When making changes to this configuration file it is recommended to do so in a copy of the original file.



7.2.4 Assignments

The configuration file for the MIKE URBAN bridge contain the mapping between the source and target cache. The mapping is called assignments and is a script for how the target should be filled based on the available data from the source.

The assignment links the attribute names in the source to the attributes names in the target. The assignment can either be simple or more advanced.

Simple Assignments

If the values are imported without any modification it is called a simple assignment and the format is:

Target Attribute = Source Attribute

The target attribute can also be set to a predefined value where the source attribute is not available:

Target Attribute = Constant Value

Example

As an example available source data for manholes could be as shown in Table 7.1 where the “ID” represents the unique identifier for the manholes, “Size” is the diameter and “Shaft” is bottom level of the manholes.

Table 7.1 Example: data in source database

ID	Size	Shaft
MH20	0.2	13
MH21	0.3	10
MH22	0.2	11

The corresponding attribute names in MIKE URBAN are “MUID”, “Diameter” and “InvertLevel”. As all the values should be transferred without any modification a simple assignment is used. Since the table only includes manholes the “TypeNo” in MIKE URBAN (manhole, basin, outlet) can be set to 1 (manhole) for all imported elements.

MUID = ID

TypeNo = 1

Diameter = Size

InvertLevel = Shaft



Automatic assignment

If the source attribute names are the same as the target attribute names it is possible to use an automatic assignment. The automatic assignment will create simple assignments for all target attributes where an attribute exists with the same name.

The automatic assignment is specified as:

* = *

The command can be used with other commands, so any other command can be specified before or after this one.

Example

The source contains several of the same attributes names as MIKE URBAN and a simple assignment is needed. Instead of making a simple assignment for each attribute the automatic assignment can be used. For the attributes where the name differs the simple assignment is written explicitly.

MUID = ID

* = *

Table 7.2 Example: source data on left and resulting target data on the right

ID	Diameter	InvertLevel		MUID	Diameter	InvertLevel
MH20	0.2	13		MH20	0.2	13
MH21	0.3	10		MH21	0.3	10
MH22	0.2	11		MH22	0.2	11

Global Variables

Assignments are usually made to source attributes, but in some cases it is useful to save a value or result in a temporary variable for later use. These variables are called global variables and are defined for a specific job. The variables can be used in one or more table configurations, but it is recommended that they are set and used within the same table configuration.

The global variable is set under the bridge properties, see section 7.4.7 for more details.

When defined the global variable can be used as any other target attribute.

Advanced Assignments

If the assignment is more advanced the target can be set to a value based on a condition for the source value.

Target Attribute = Value1 WHERE (Source Attribute = Value2)



To create an advanced assignment there several types of condition and functions available.

Operators

When assigning a value it is possible to use the mathematical operators +, -, / and * for values that are numbers.

If + is used with text the result is a text where the two texts are concatenated. If one of the values is a number it will be converted to text.

Note that operators should always be 'padded' with spaces.

Examples

The source data stores the manholes without ID, but with a consecutive number and we will use this to make an ID. The diameter of the manholes are calculated based on the radius.

$MUID = "MH" + No$

$Diameter = 2 * radius$

Table 7.3 Example: source data on left and resulting target data on the right

No	radius		MUID	TypeNo
1	0.5		MH1	1.0
2	0.3		MH2	0.6
3	1.0		MH3	2.0

WHERE

The WHERE command can be placed after a simple assignment to indicate that the simple assignment should only be used in certain situations.

The syntax is:

Target Attribute = Value1 WHERE Condition

A condition can consist of one or more conditions. The syntax for a condition is:

(Source Attribute1 "relation" Value1) "logical operator" (Source Attribute2 "relation" Value2)

Where the "relation" can be:

- > : greater than
- < : less than
- = : equal



- <> : not equal

The “logical operator” can be:

- AND - both the condition before and after the logical operator should be true before the condition is evaluated as true.
- OR - just one of the conditions before and after the logical operator should be true before the condition is evaluated as true.

Example

The source data stores the information about a pipe type as text in the attribute “ptype” such as “Circular”, “Rectangular” and “Square”. MIKE URBAN stores the same information as numbers (1 = circular pipes, 3 = rectangular pipes).

The WHERE statement is used to transform the text into numbers. Since MIKE URBAN does not have a separate option for square pipes these are imported as rectangular pipes.

TypeNo = 1 WHERE ptype = “Circular”

TypeNo = 3 WHERE (ptype = “Rectangular”) OR (ptype = “Square”)

Table 7.4 Example: source data on left and resulting target data on the right

ID	ptype		MUID	TypeNo
P1	Circular		P1	1
P2	Square		P2	3
P3	Rectangular		P3	3

NULL

NULL represents missing and unknown data. NULL can be used in evaluations and assignment similar to other values.

Example

The source data stores the information about a pipe type as text in the attribute “ptype” such as “Circular” and “Rectangular”. MIKE URBAN stores the same information as numbers (1 = circular pipes, 3 = rectangular pipes). Any pipes where the type is undefined is assumed circular.

TypeNo = 1 WHERE (ptype = “Circular”) OR (ptype = NULL)

TypeNo = 3 WHERE (ptype = “Rectangular”)



Table 7.5 Example: source data on left and resulting target data on the right

ID	ptype		MUID	TypeNo
P1	Circular		P1	1
P2			P2	1
P3	Rectangular		P3	3

DeleteRecord

The DeleteRecord command deletes an element in the target table. This function should only be used with the use of a WHERE condition.

In some cases the Source Filter (see section 7.4.8) can be used as an alternative to DeleteRecord. In these cases it is recommended to use the Source Filter.

Note that the functionality of the DeleteRecord is that the imported element is first transferred to the target table and then deleted from here. Please note that no data is deleted in the source data.

The syntax is:

DeleteRecord WHERE Condition

Example

The source data includes information if a pipe is a service pipe or a mainpipe. Only pipes from the main line should be imported.

MUID = ID

DeleteRecord WHERE ptype = "Servicepipe"

Table 7.6 Example: source data on left and resulting target data on the right

ID	ptype		MUID
P1	Mainline		P1
P2	Servicepipe		P3
P3	Mainline		

LookUP

The LookUP command is used if values from a different table are used.

The syntax is:

LookUP("Table Name", "Attribute Name", Lookup Value, "Return Attribute")



The LookUP function takes 4 parameters:

1. Table Name - the name of the table where we look for data.
2. Attribute Name - the attribute that we look for in the table from parameter 1.
3. Lookup Value - the value of the attribute given in parameter 2. The parameter can either be an attribute in the current assignment source data, a constant, a global variable or another function.
4. Return Attribute - the attribute that should be returned for the record found.

If a value is not found the LookUP returns NULL.

LookUP can be used on all tables during an import/export job. This means that e.g. during an import it is possible to lookup values in a table from the source or from tables that have been imported in a previous table configuration.

Example

The source data does not include the diameter of the manholes. The diameter should be equivalent to the diameter of the connected pipes. A lookup is used to get the diameter from the table with pipes.

Table Configuration 1 (pipes):

MUID = ID

Diameter = Diameter

_FromNodeID = UpMH

_ToNodeID = DwMH

Table 7.7 Example: source data on left and resulting target data on the right

ID	Diameter	UpMH	DwMH		MUID	Diameter	_From-NodeID	_ToNodeID
P1	0.5	MH1	MH2		P1	0.5	MH1	MH2
P2	0.7	MH2	MH3		P2	0.7	MH2	MH3
P3	0.2	MH3	MH4		P3	0.2	MH3	MH4

Table Configuration 2 (nodes):

MUID = MHID



Diameter = LookUP("msm_Link", "_FromNodeID", MHID, "Diameter")

Table 7.8 Example: source data on left and resulting target data on the right

MHID		MUID	Diameter
MH1		MH1	0.5
MH2		MH2	0.7
MH3		MH3	0.2
MH4		MH4	

Example

The LookUP functionality can also be used to check for existence of a record in a different table.

The source data contain manholes that is not connected to the imported pipes. A lookup is used to exclude all manholes that are not connected to an imported pipe.

Table Configuration 1 (pipes):

MUID = ID

_FromNodeID = UpMH

_ToNodeID = DwMH

Table 7.9 Example: source data on left and resulting target data on the right

ID	UpMH	DwMH		MUID	_FromNodeID	_ToNodeID
P1	MH1	MH2		P1	MH1	MH2
P2	MH2	MH3		P2	MH2	MH3

Table Configuration 2 (nodes):

MUID = MHID



DeleteRecord WHERE (LookUP("msm_Link", "_FromNodeID", MHID, "MUID") = NULL) AND (LookUP("msm_Link", "_ToNodeID", MHID, "MUID") = NULL)

Table 7.10 Example: source data on left and resulting target data on the right

MHID		MUID
MH1		MH1
MH2		MH2
MH3		MH3
MH4		

Incremental assignment

The incremental assignment increases the target data with the number given in the simple assignment.

The incremental assignment is used if the last valid character on left side of assignment is "+". The syntax is:

Target Attribute+ = Number to increase with

Example

The source data does not contain data for the ground level, but data for both bottom level and depth. The ground level is calculated based on the bottom level and depth.

GroundLevel = Bottom

GroundLevel+ = Depth

Table 7.11 Example: source data on left and resulting target data on the right

ID	Bottom	Depth		MUID	GroundLevel
P1	12	4		P1	16
P2	10	3.5		P2	13.5
P3	9	4		P3	13

Unique

The Unique command creates a unique identifier. Since all tables in MIKE URBAN must have a unique identifier it can be useful to use this function if the ID in the source data is not unique.

The syntax is:

Target Attribute = Unique("Table Name", "Attribute Name", Current Value")



The function takes 3 parameters:

1. Table Name - the table in which the ID should be unique. In most cases this will be the table currently used as the target table.
2. Attribute Name - the name of the attribute that must contain the unique identifier. In most cases this will be MUID.
3. Current Value - the suggested ID. If the suggested ID is not already used the function will return this value. If the value already exists the function will return this value with a prefix ensuring that the value is unique. The parameter can either be an attribute in the current assignment source data, a constant, a global variable or another function.

Example

The source data has an identifier but it is not unique so before assigning this to the MUID the Unique function is used.

MUID = Unique("msm_Link", "MUID", ID)

Table 7.12 Example: source data on left and resulting target data on the right

ID		MUID
P1		P1
P2		P2
P2		P2_1

ChangeFileExt

The ChangeFileExt command changes the file extension of a parameter.

The syntax is:

Target Attribute = ChangeFileExt(File Name, "Extension")

The function takes 2 parameters:

1. File Name - the file name that should be changed. The parameter can either be an attribute in the current assignment source data, a constant, a global variable or another function.
2. Extension - the new extension

Example

The source contain information about the timeseries names for boundary conditions to the model. MIKE URBAN needs the extension of the timeseries file.



```
TSCConnection = ChangeFileExt(TimeSeries, ".dfs0")
```

Table 7.13 Example: source data on left and resulting target data on the right

TimeSeries		TSCConnection
Flow_P1		Flow_P1.dfs0
Flow_P2		Flow_P2.dfs0
Flow_P3		Flow_P3.dfs0

Log

The Log command writes to the MIKE URBAN log file. The log file can be found in the user folder under “\AppData\Local\DHI\MIKE URBAN version\MIKE URBAN\all-utf8.log”.

The Log command is used during the process of creating advanced assignments. When internal variables and lookups are used it can be useful to see variables to ensure that syntax is correct and data is correctly assigned. It is recommended to delete these commands again after the completion of the assignments.

The Log command can also be used to create error messages when something unexpected occurs during an import or export. It should be noted that the log file is not automatically displayed and it is up to the user to check the log file for any errors.

The syntax is:

```
Log(Value)
```

The command takes 1 parameter:

1. Value - the value that should be written to the log file. The parameter can either be an attribute in the current assignment source data, a constant, a global variable or another function.

Example

The source data is using the following enumeration for their pipe types - 1 = circular, 2 = user defined, 3 = rectangular, 4 = undefined. MIKE URBAN uses the same following enumeration for pipes types but type number 4 is an O-shape. The imported data should not contain undefined pipe types.

```
TypeNo = ptype
```




Log("The ptype was unknown for pipe: " & ID) WHERE ptype > 3

Table 7.14 Example: source data

ID	ptype
P1	1
P2	1
P3	4

The following message is written to the log file:

2010 Jul 01 00:00:00 8000 ROOT_CONTEXT: ("The ptype was unknown for pipe: " + ID) is The ptype was unknown for pipe: P3

UpperCase

The UpperCase command convert any value to an uppercase text.

The syntax is:

Target Attribute = UpperCase(Value)

The command takes 1 parameter:

1. Value - The value that should be converted to uppercase. The parameter can either be an attribute in the current assignment source data, a constant, a global variable or another function.

Example

The source data includes information if a pipe is a service pipe or a mainpipe. However there are no common syntax for how the data is written. In order to filter data in MIKE URBAN using e.g. select by attribute the syntax must be identical. All are made uppercased.

MUID = ID



Description = UpperCase(ptype)

Table 7.15 Example: source data on left and resulting target data on the right

ID	ptype		MUID	Description
P1	Mainline		P1	MAINLINE
P2	mainline		P2	MAINLINE
P3	MainLine		P3	MAINLINE

RemoveSpaces

The RemoveSpaces command removes all spaces from a text.

The syntax is:

Target Attribute = RemoveSpaces(Value)

The command takes 1 parameter:

1. Value - The value where spaces should be removed. The parameter can either be an attribute in the current assignment source data, a constant, a global variable or another function.

Example

The source data includes information if a pipe is a service pipe or a mainpipe. However there are no common syntax for how the data is written. In order to filter data in MIKE URBAN using e.g. select by attribute the syntax must be identical. All spaces are removed.

MUID = ID

Description = RemoveSpaces(ptype)

Table 7.16 Example: source data on left and resulting target data on the right

ID	ptype		MUID	Description
P1	Mainline		P1	Mainline
P2	Main line		P2	Mainline
P3	Service pipe		P3	Servicepipe

Sqrt

The Sqrt command takes the square root of a given parameter.

The syntax is:

Target Attribute = Sqrt(Value)



The command takes 1 parameter:

1. Value - The value of which the squareroot should be taken. The parameter can either be an attribute in the current assignment source data, a constant, a global variable or another function, but must be a number.

Example

The source contains information about the flow area in a pipe. To set the size of the pipe an equivalent diameter is calculated.

MUID = ID

Diameter = 2 * Sqrt(FlowArea / 3.1416)

Table 7.17 Example: source data on left and resulting target data on the right

ID	FlowArea		MUID	Diameter
P1	0.00785		P1	0.09997
P2	0.07069		P2	0.30000
P3	0.03142		P3	0.20001

NoOfRecords

The NoOfRecords command is used to find the number of records in a specific table.

The syntax is:

Target Attribute = NoOfRecords("Table Name")

The command takes 1 parameter:

1. "Table Name" - The name of the table where the number of records should be checked. The table listed must be available to the current job i.e. either as a source or target table.

Example

Simulation information should automatically be created for a runoff simulation and a pipeflow simulation. The source data contains both a table with catchments and one with simulation data. If the catchment table is empty only the pipeflow simulation should be created.

MUID = "CityX-" + SimType

ComputationTypeNo = 1 WHERE SimType = "RR"

ComputationTypeNo = 2 WHERE SimType = "HD"



DeleteRecord WHERE ((SimType = "RR") AND (NoOfRecords("ms_Catchment") = 0))

Table 7.18 Example: source data on the left and resulting target data on the right when no catchments are imported.

SimType		MUID	ComputationTypeNo
RR		CityX-HD	2
HD			

Additional Examples

Example using DeleteRecord and LookUP

This assignment will delete records in the target table if the PipeID is not found in the msm_Link table:

DeleteRecord WHERE LookUP("msm_Link", "MUID", PipeID, "MUID") = NULL

Example using DeleteRecord, LookUP and Global Variable

These assignments below will delete all items that when looked up are not type 1 or 2 in the specified table.

Var_Temp = LookUP("LEDNING", "ID", PipeID, "Type1")

DeleteRecord WHERE (Var_Temp < 1)

DeleteRecord WHERE (Var_Temp > 2)

Example using LookUP and Global Variable

Here is an example of how an assignment of a lookup-value to a temporary variable can be used to save processing time.

Imagine that you wish to import a list of pipe geometries from a table. The list is missing the starting point of the pipe. In order to resolve this we need to find the coordinates (X, Y) of the upstream node connected to the pipe.

As the pipe geometry table we are importing from only contains the pipe ID, and not the upstream node, we will need to make two lookups.

First we use the PipeID to find the upstream node ID in the Pipes table.

UpstreamNodeID = LookUP("PIPES", "ID", PipeID, "UpstreamNodeID")

Second we use the upstream node ID to find the coordinates in the nodes table.



```
X = LookUP( "NODES", "ID", UpstreamNodeID, "X")
```

```
Y = LookUP( "NODES", "ID", UpstreamNodeID, "Y")
```

These lookups can now be combined with or without use of a variable

A) Variable not used (slow)

The first lookup is simply placed inside the second lookup

```
X = LookUP( "NODES", "ID", LookUP("PIPES", "ID", PipeID, "UpstreamNodeID"), "X")
```

```
Y = LookUP( "NODES", "ID", LookUP("PIPES", "ID", PipeID, "UpstreamNodeID"), "Y")
```

B) Variable used (faster)

The upstream node ID is saved in a temporary variable list (Var_Temp) and this is used to make the second lookup

```
Var_Temp = LookUP("PIPES", "ID", PipeID, "UpstreamNodeID")
```

```
X = LookUP( "NODES", "ID", Var_Temp, "X")
```

```
Y = LookUP( "NODES", "ID", Var_Temp, "Y")
```

When comparing assignment (A) and assignment (B) it can be seen that time is saved using (B).

The reason for this is that the lookup of the UpstreamNodeID is only performed once.

7.2.5 MIKE URBAN Transfer Modes

When importing or exporting data 4 different modes are available for how to treat the new data compared to the existing data.

'Replace existing data'

This option clears any existing data in the actual target table cache and replaces them according to the assignment.

'Only append to existing data'

This option only appends new rows, but does not overwrite already existing attribute values in the existing data. In other words, no data already existing in the MIKE URBAN database gets changed.



The data are 'recognised' as new by the 'primary key', which is defined for each MIKE URBAN database table. In most cases the 'primary key' is a unique MIKE URBAN identifier (MUID), while in few cases it is some other identifier (where tables are related 1:1) or a unique combination of two attributes. The definition of the primary keys is available in the database documentation.

If the value of the source table attribute assigned to the MIKE URBAN table 'primary key' is not found in MIKE URBAN it is then defined as new.

This mode is usually used when the target table is composed of records from two or more source tables.

'Append and update existing data'

This option not only appends new data (i.e. adds new records), but it also updates (overwrites) the existing attribute values with new values if the ID is specified.

This mode is usually used when records in the target table include attributes from records from two or more source tables.

'Only update existing data'

This option only updates (overwrites) the existing attribute values with new values.

This mode is usually used when records in the target table include attributes from records from two or more source tables.

Using transfer modes

The four modes are available both for the job level and for the table configuration. When specified for table configuration the transfer mode relates to the action done between the source and target tables in memory. When specified for the job the transfer mode relates to the action done between the target table in memory and the saved target table. Referring back to Figure 7.1 the difference is shown in Figure 7.3.

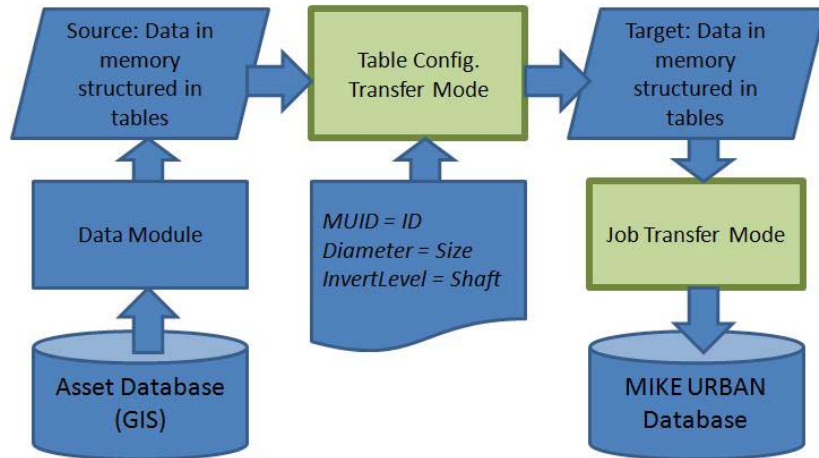


Figure 7.3 Difference between table config. and job transfer modes

The transfer modes for both job and table configuration should be considered depending on the purpose of the import/export. A list of common tasks and the corresponding transfer modes are shown in table

Table 7.19 List of transfer modes for common tasks

Task	Job Transfer Mode	Table Configuration Transfer Mode
Import data into a new database	Replace existing data	Replace existing data
Update data in an existing database. Include both new items and update existing.	Append and update existing data	Replace existing data
Import data into a new database. Data for e.g. nodes are located in two separate tables.	Replace existing data	Append and update existing data
The data already imported is the most updated. I just need to add a new area from a different source.	Only append to existing data	Replace existing data

Examples of different transfer modes for jobs

In most cases the transfer mode for the table configuration will be “Replace existing data” as an import will be configured from one table in the source database to one table in the MIKE URBAN database. This means that when



the import is initiated the target data in memory will be empty, so all data from the source data in memory should overwrite the target.

In the following a small example will be shown where data for a couple of manholes exists in both the source database and the MIKE URBAN database. The differences between the transfer modes will be explained.

Initially the two database contains information as shown in Table 7.20 and Table 7.21.

Table 7.20 Example: data in source database

ID	Size	Shaft
MH20	0.2	13
MH21	0.3	10
MH22	0.2	11

Table 7.21 Example: data in MIKE URBAN database

MUID	Diameter	InvertLevel
MH10	0.1	9
MH11	0.3	14
MH22	0.3	12

Following the procedure outlined in Figure 7.1 the first step is that the source data is loaded to the source cache. Note that the data structure is the same as in the database and that the target cache is empty.

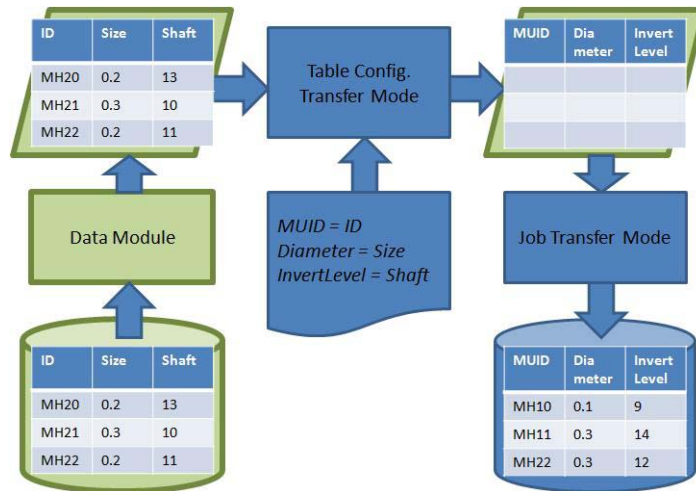


Figure 7.4 Loading of source data to memory structure

The second step is that the source data is converted to the format of the target structure (the MIKE URBAN database). The data converter is used and the table configuration transfer mode is set to “Replace existing data”.

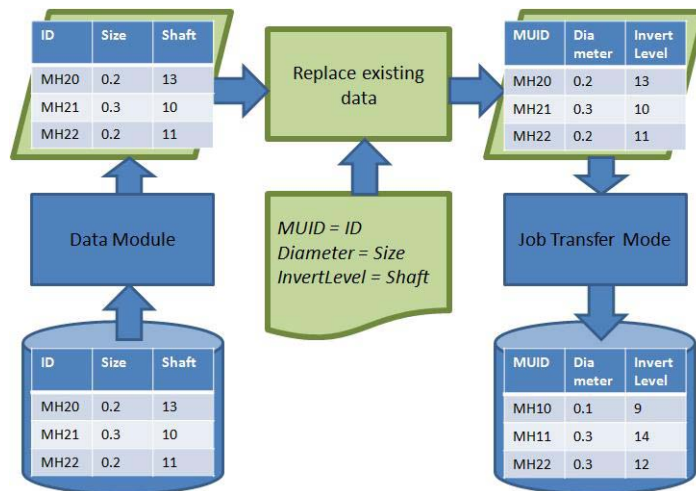


Figure 7.5 Converting source data structure to target data structure

The third step is to save the data stored in the target memory data structure to the MIKE URBAN database. Here the use of the different job transfer modes will give different results. Note that since the data is already in the same data format as the MIKE URBAN database it is already defined which attribute is the primary key (MUID).



Job Transfer Mode: Replace Existing Data

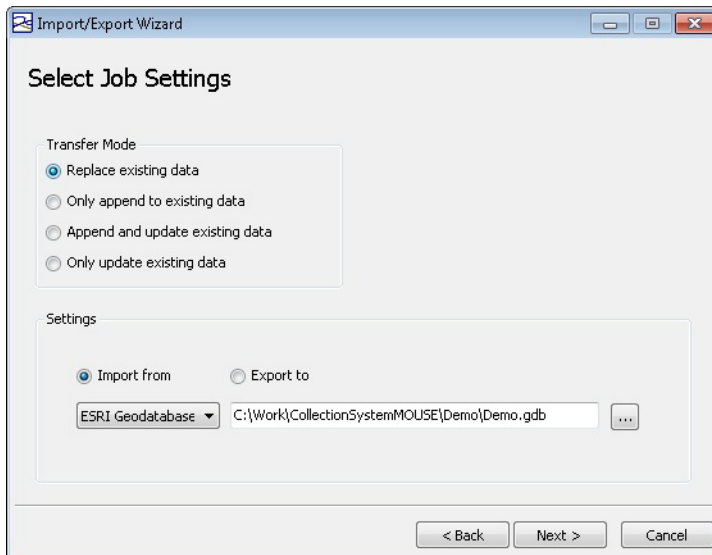


Figure 7.6 Job setting for “Replace existing data”

When using the “Replace existing data” the existing data in the MIKE URBAN database will first be deleted and then replaced with the data in the target memory data structure.

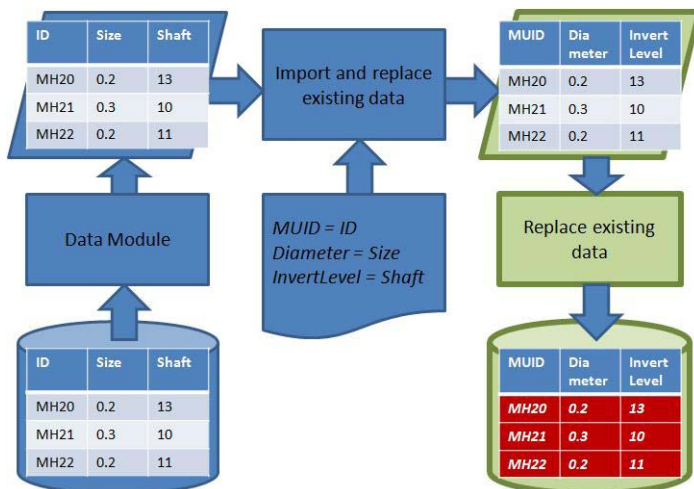


Figure 7.7 Saving target data structure to MIKE URBAN database using “Replace existing data”



Job Transfer Mode: Only append to existing data

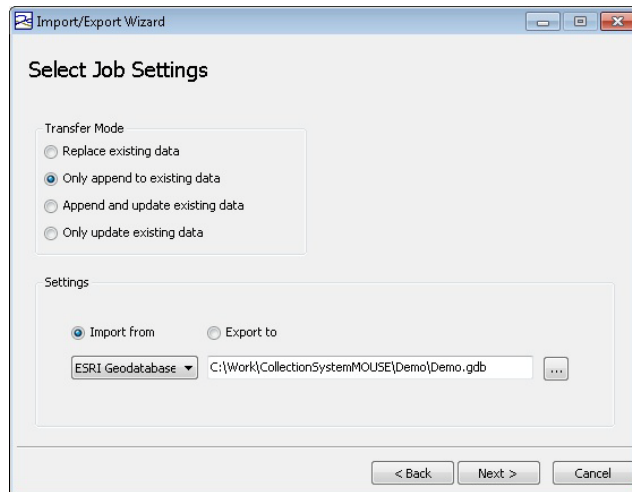


Figure 7.8 Job setting for “Only append to existing data”

When using the “Only append to existing data” the existing data in the MIKE URBAN database will remain unchanged. All new elements from the target memory data structure will be added e.g. in this example nodes MH20 and MH21.

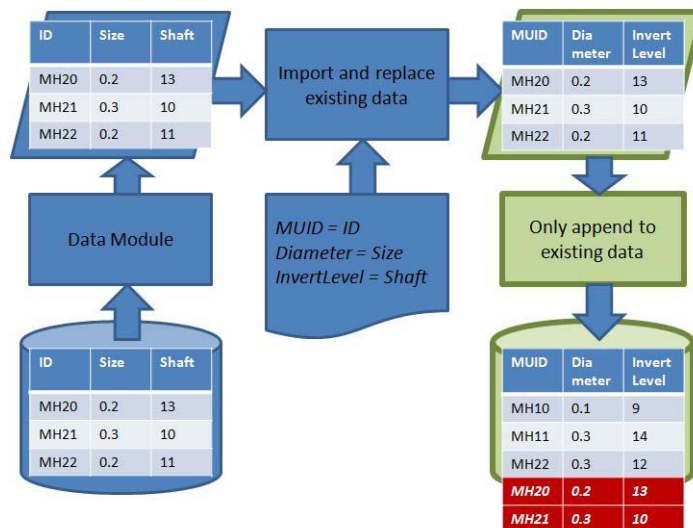


Figure 7.9 Saving target data structure to MIKE URBAN database using “Only append to existing data”



Job Transfer Mode: Only update existing data

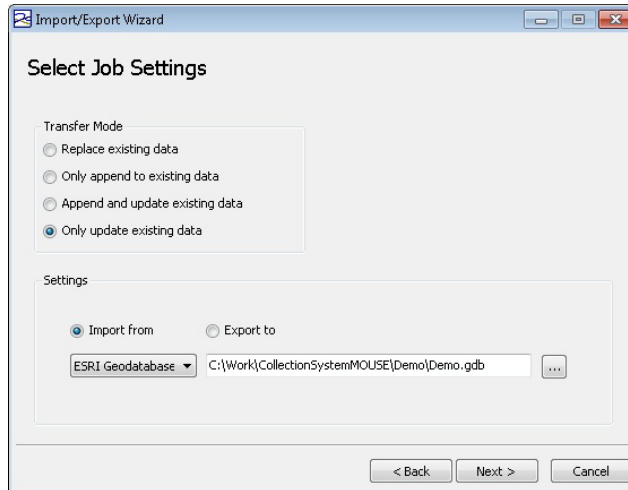


Figure 7.10 Job setting for “Only update existing data”

When using the “Only update existing data” no new elements will be added to the MIKE URBAN data. Only data already existing in the model will be updated where updates are available in the target memory data structure. In this example the node MH22 exists both MIKE URBAN and the memory data structure and this node will be updated.

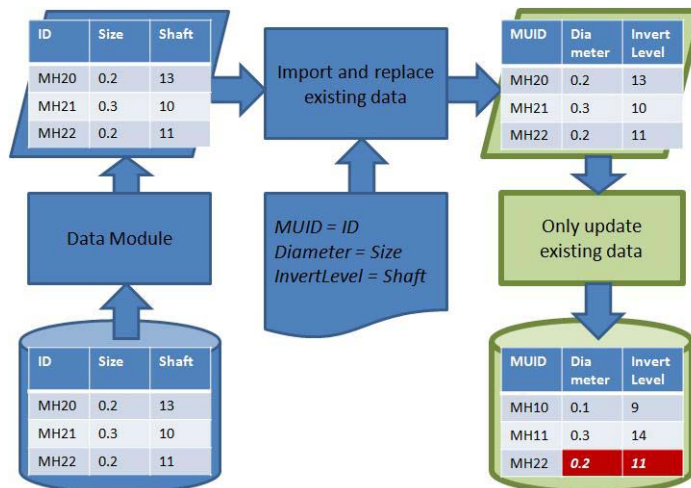


Figure 7.11 Saving target data structure to MIKE URBAN database using “Only update existing data”



Job Transfer Mode: Append and update existing data

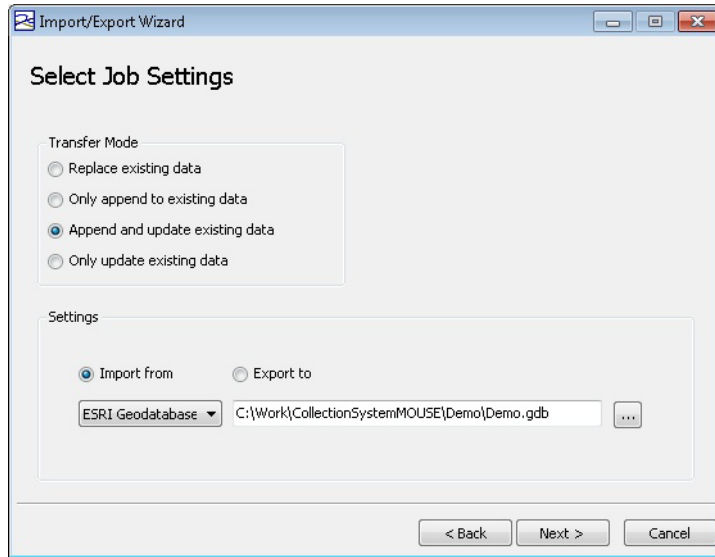


Figure 7.12 Job setting for “Append and update existing data”

When using the “Append and update existing data” new elements will be imported and existing updated.

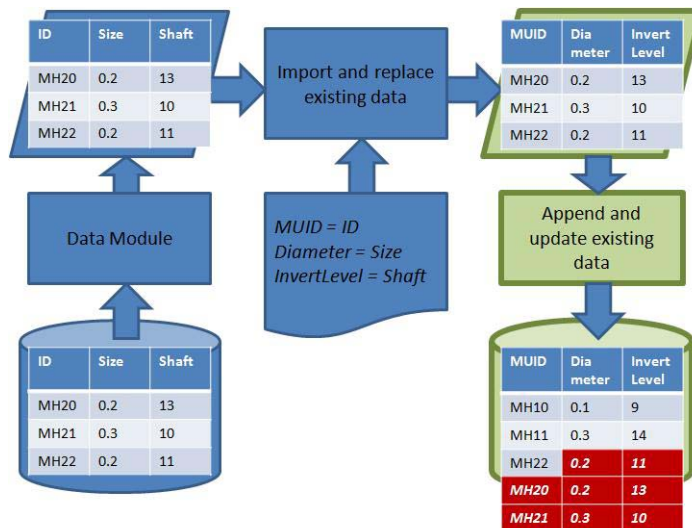


Figure 7.13 Saving target data structure to MIKE URBAN database using “Append and update existing data”



Examples of different transfer modes for table configurations

When data from the same table is imported from more than one table the use of different transfer modes for the table configurations can be used.

In the following a small example will be shown where data for a couple of manholes exists in two different tables in the source database. The differences between the transfer modes will be explained.

Initially the two tables contains information as shown in Table 7.22 and Table 7.23.

Table 7.22 Example: data in table 1 from source database

MHID	Size
MH10	0.2
MH21	0.3
MH22	0.2

Table 7.23 Example: data in table 2 from source database

ID	Shaft
MH20	13
MH21	10
MH22	11

Following the procedure outlined in Figure 7.1 the first step is that the source data is loaded to the source memory structure. Note that the data structure is the same as in the database and that the target memory structure is empty.

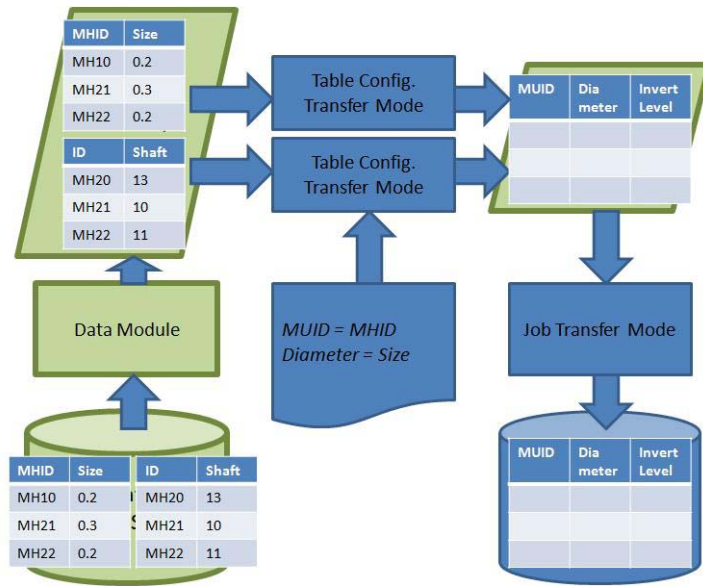


Figure 7.14 Loading of source data to memory structure

For each table a separate table configuration is needed with separate transfer modes. For the first table the target memory data structure is empty hence the transfer mode will always be “Replace existing data”.

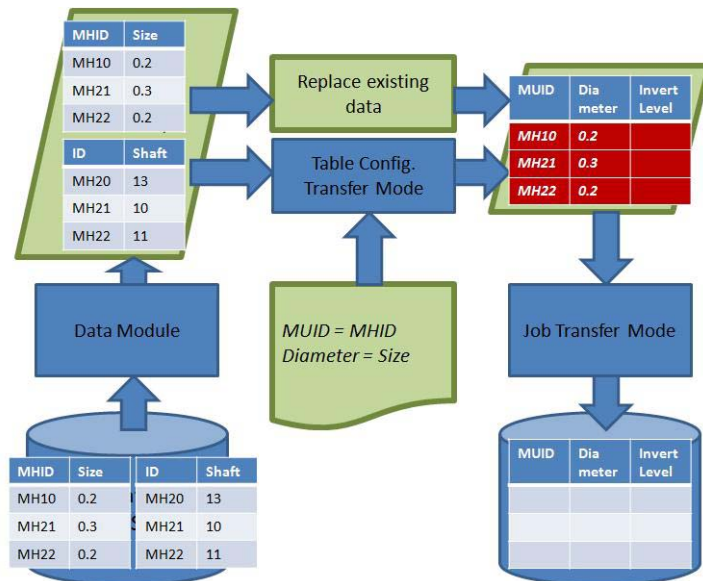


Figure 7.15 Converting the first table to target memory data structure



For the second table the target memory data structure is now filled with some data so here the use of the different table configuration transfer modes will give different results.

Table Configuration Transfer Mode: Replace existing data

The screenshot shows a dialog box titled "Import/Export Wizard" with the following settings:

- Select Source Table:** MHTABLE
- Select Target Table:** msm_Node
- Replicate Source Datastructure in Target
- Transfer Mode:**
 - Replace existing data
 - Only append to existing data
 - Append and update existing data
 - Only update existing data
- Source ID:** (empty dropdown)
- Target ID:** (empty dropdown)

Buttons at the bottom: < Back, Next >, Cancel

Figure 7.16 Table setting for “Replace existing data”

Using the “Replace existing data” will delete the data transferred from the first table and the target memory data structure will only be filled with data from the second table. This transfer mode should never be used when a previous table have been imported.

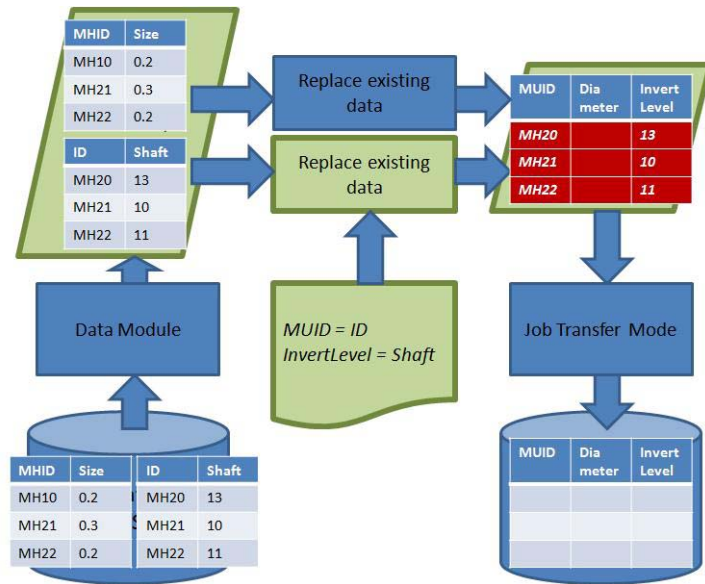


Figure 7.17 Converting the second table to target memory data structure using “Replace existing data”

Table Configuration Transfer Mode: Only append to existing data

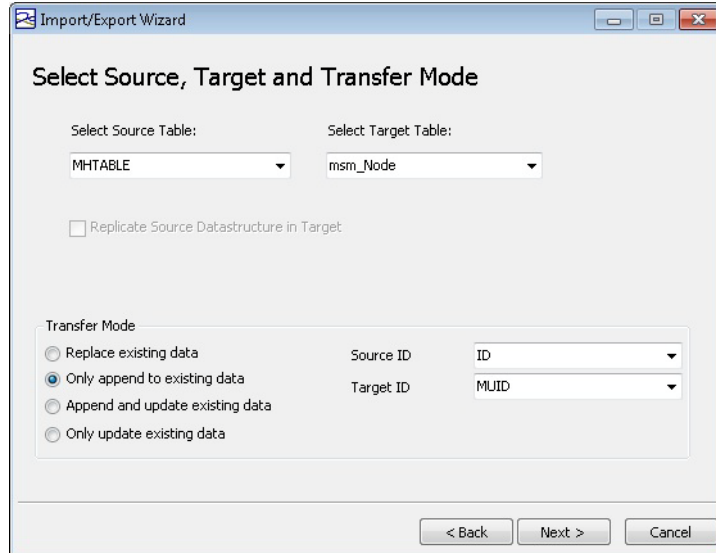


Figure 7.18 Table setting for “Only append to existing data”

Using “Only append to existing data” only new data will be included from the second table. So in this case only information for MH20 is included from the second table.



In order to update the target memory data structure from the source memory data structure it must be defined which unique ID should be used to match the two tables. These are specified as the “Source ID” and “Target ID”. In this example the “Source ID” is “ID” since this is the attribute in the source memory data structure that holds the identifier. Similar the “Target ID” is the “MUID”.

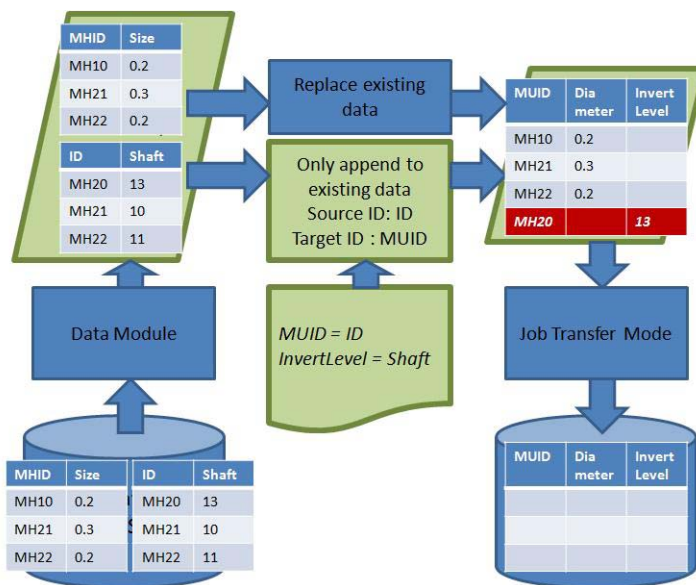


Figure 7.19 Converting the second table to target memory data structure using “Only append to existing data”



Table Configuration Transfer Mode: Only update existing data

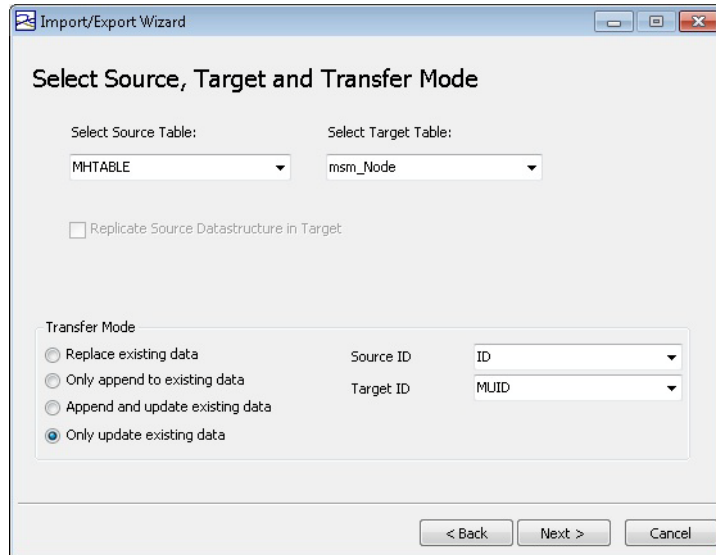


Figure 7.20 Table setting for “Only update existing data”

Using “Only update existing data” will only include data for elements already in the target memory data structure. So in this case the data for MH20 is ignored.

In order to update the target memory data structure from the source memory data structure it must be defined which unique ID should be used to match the two tables. These are specified as the “Source ID” and “Target ID”. In this example the “Source ID” is “ID” since this is the attribute in the source memory data structure that holds the identifier. Similar the “Target ID” is the “MUID”.

Note that only existing elements are updated so there is no need to set the MUID in the assignment.

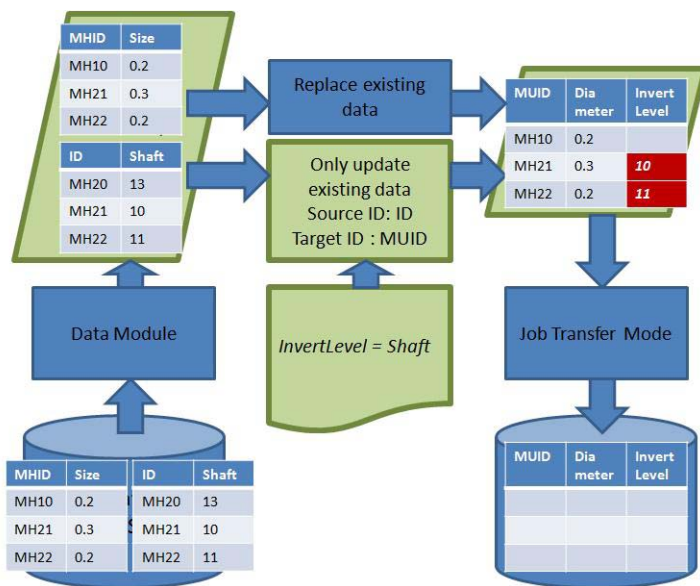


Figure 7.21 Converting the second table to target memory data structure using “Only update existing data”

Table Configuration Transfer Mode: Append and update existing data

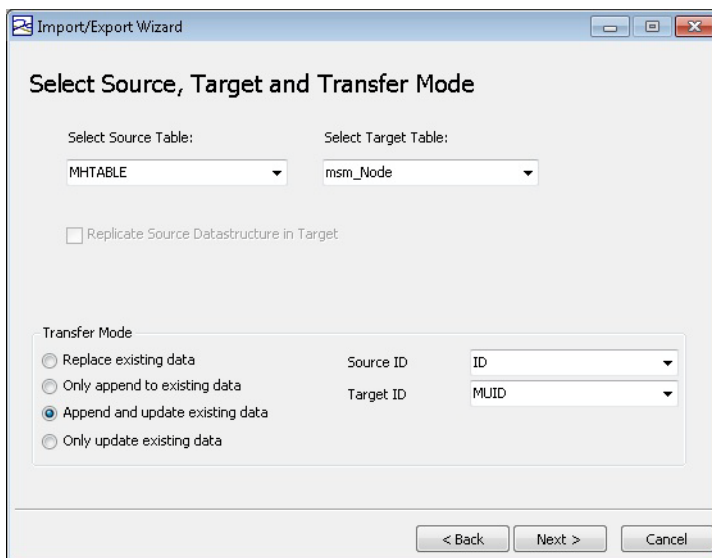


Figure 7.22 Table setting for “Append and update existing data”

Using “Append and update existing data” will both include new data and update data for elements already in the target memory data structure.



In order to update the target memory data structure from the source memory data structure it must be defined which unique ID should be used to match the two tables. These are specified as the “Source ID” and “Target ID”. In this example the “Source ID” is “ID” since this is the attribute in the source memory data structure that holds the identifier. Similar the “Target ID” is the “MUID”.

Note that since new elements are added it is important that the MUID is set in the assignment. This means that the MUID’s are overwritten for the existing elements. However, since the assignment is using the same attributes as the “Source ID” and “Target ID” it has no influence on the result.

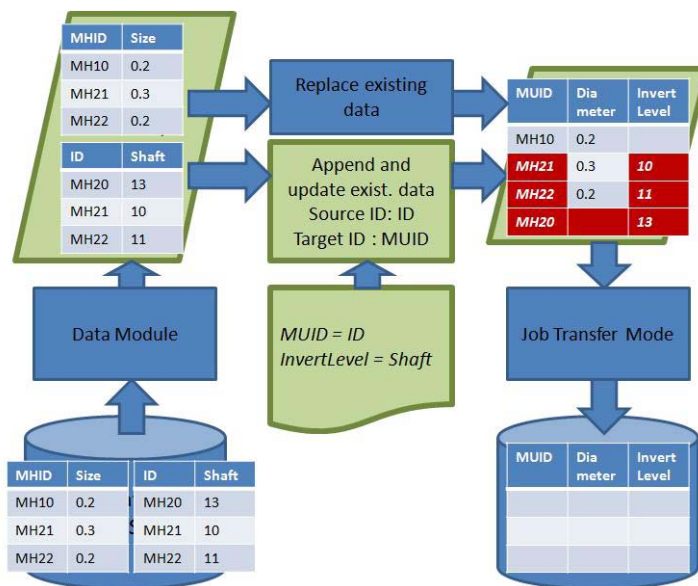


Figure 7.23 Converting the second table to target memory data structure using “Append and update existing data”

7.3 Import/Export - Wizard

7.3.1 Overall Description

The Import/Export functionality is accessed through “File | Import/Export”.

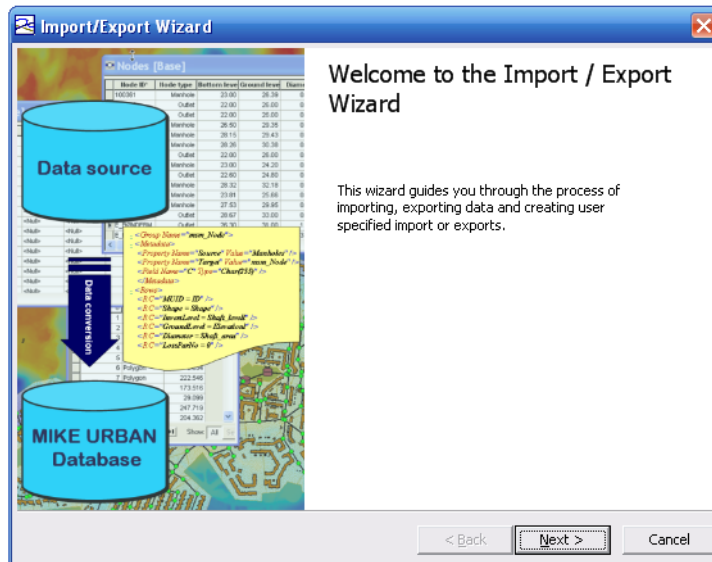


Figure 7.24 Welcome page of the Import/Export Wizard

There are in principle two ways of importing or exporting data - either through a wizard or through an advanced dialog. The intention of the wizard is to increase the user friendliness of standard import/export and guide the user when specifying a new import/export.

The wizard runs through a double loop, an outer loop for creating/editing/deleting import/export jobs and an inner loop for creating/editing/deleting table configurations.

Predefined import/export jobs exist for some data formats, e.g. import from MOUSE, SWMM 5 and EPANET. For these standard import/exports, the loops in the wizard are not reached.

For other import/export jobs (from e.g. ACCESS, EXCEL and shape files) the import/export must be specified by the user. The loops will then guide the user through the specification procedure for the various import/export jobs and table configurations.

The wizard driven import/export will not support all functionality. For those users that need more advanced options an advanced import/export dialog can be reached by selecting the Advanced option - see more in section 7.4.



7.3.2 “Use Wizard” option

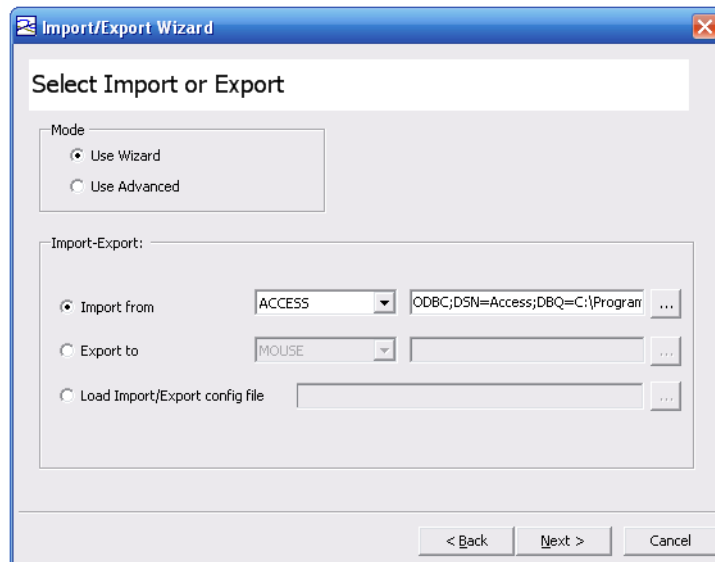


Figure 7.25 Selection of “Use Wizard” option

When the “Use Wizard” option is selected, there are different options available in the Import/Export group.

- Import from
- Export to
- Load Import / Export config file

“Import from”

Import from one of the following sources uses a predefined import job that is read from the muIO.xml file.

- MOUSE, SWMM5, EPANET, Restore From Simple XML.

The file from which data should be imported from must be specified using the file selection button. No further configuration is needed. When clicking <Next>, the wizard will prompt for edits to the import job, answering ‘no’ will activate the last page in the wizard and thereby skip the loops in the wizard.

Import from other sources like

- ESRI CAD, ESRI Geodatabase, ESRI OleDB, ESRI SDE, ESRI Shape, MapInfo, ODBC, Oracle Spatial, MU Simple XML, MU Simple PFS.

do not have a predefined import thus requiring the user to do the specifications of these import jobs through the loops.



“Export to”

Predefined Exports exist for following storages.

- MOUSE, SWMM5 and EPANET2, Restore From Simple XML.

Export to other storages like

- ESRI CAD, ESRI Geodatabase, ESRI OleDB, ESRI SDE, ESRI Shape, MapInfo, ODBC, Oracle Spatial, MU Simple XML, MU Simple PFS.

do not have a predefined exports thus requiring the user to do the specification of these export jobs through the loops.

“Load Import / Export config file”

This option is used for editing existing import/export jobs using the wizard.

7.3.3 Job loop (outer loop)

When the job loop is entered there are different options available to the user. Jobs available for editing are listed to the right in the wizard page.

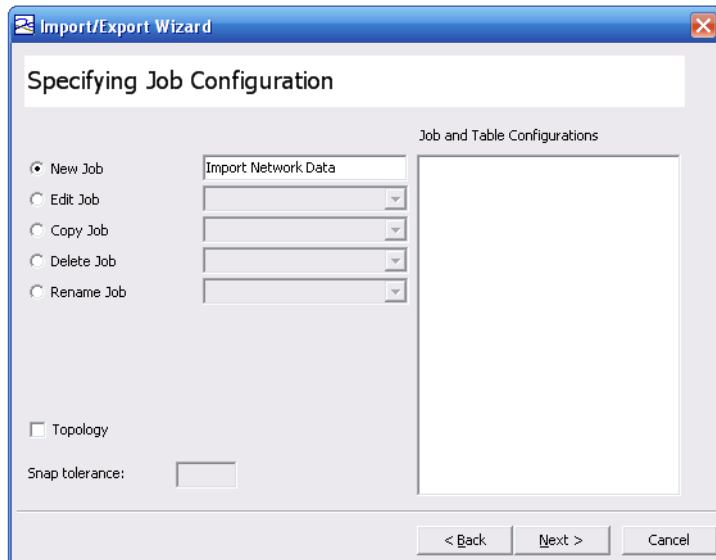


Figure 7.26 Specifying jobs (outer loop)

- New Job - used to specify name of new job
- Edit Job - selection of which job to be edited is done in this combo-box.
- Copy Job - selection of which job to be copied is done in this combo-box.



- Delete Job - selection of which job to be deleted is done in this combo-box.

For import jobs it is optional to activate the Topology-processing (by setting a tickmark next to 'Topology'). The topology processing can be used when importing e.g. from a shape file with pipes that do not contain information about the connectivity (upstream/downstream node). Then the topology processing tool can process the data and create a connected network and add nodes where these are missing. The snapping tolerance specifies if a pipe should be connected or not. The units for the snap tolerance equals the map unit. Other settings for the topology are set automatically by the wizard.

The transfer setting for the job is specified on the next page, see Figure 7.27. The following options are available:

- Replace existing data (will replace all existing data with new data)
- Only append to existing data (appends new data to existing data)
- Append and update existing data (updates existing data and appends new data)
- Only update existing data (only updates existing data)

For more information in transfer modes see section 7.2.5.

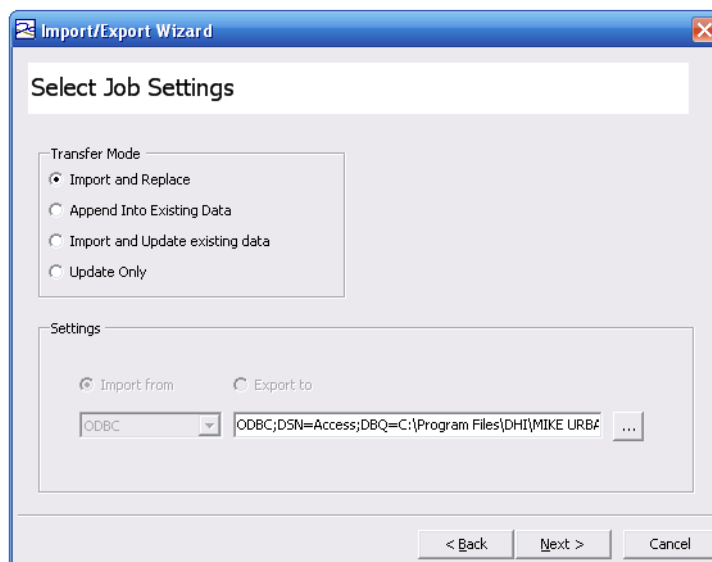


Figure 7.27 Transfer setting



7.3.4 Table configuration loop (inner loop)

When the table configuration loop is entered there are different options available to the user. Table configurations available for editing are listed to the right in the wizard page. The list is intended to give the user an overview but it is only a display list and cannot be used for activating the listed items.

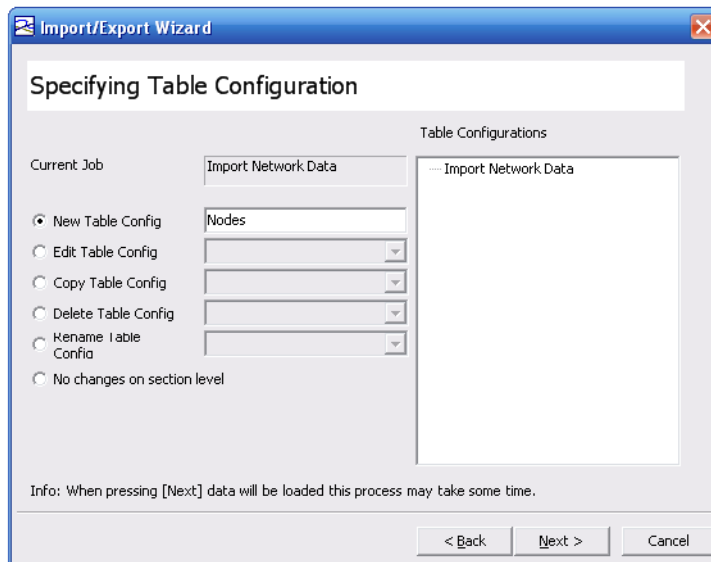


Figure 7.28 Specifying table configurations (inner loop)

- New Table Configuration - used to specify name of table configuration
- Edit Table Configuration - selection of which table configuration to edit is done in this combo-box.
- Copy Table Configuration - selection of which table configuration to copy is done in this combo-box.
- Delete Table Configuration - selection of which table configuration to delete is done in this combo-box.



Selection of source, target and transfer mode

Figure 7.29 Specification of Source, Target and Transfer mode

In the next wizard page the source table and the target table is selected from the two combo-boxes. Additionally the transfer setting between the source and target table is chosen. There are the following options available on for each table configuration:

- Replace existing data (will replace all existing data with new data)
- Only append to existing data (appends new data to existing data)
- Append and update existing data (updates existing data and appends new data)
- Only update existing data (only updates existing data)

For the “Only update existing data”, “Append and update existing data” and “Only append to existing data”, the Source ID and Target ID must be selected.

For further information about the transfer modes see section 7.2.5.



Assignment specification

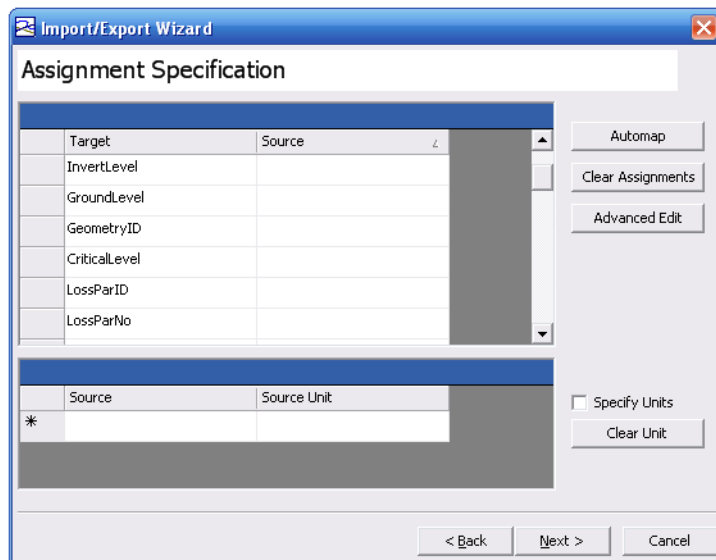


Figure 7.30 Assignment specification

In this wizard page the actual assignments also called mapping of data is carried out between the source and target table. The default setting of the wizard page is that the mapping is done in a grid. If more advanced mapping is necessary, then the “Advanced Edit” is changing the upper grid to text editor view in which the user can create more advanced assignments - like using WHERE statements etc. After entering the text editor view it is not possible to return back to the grid view without losing the changes done in the advanced mode.

For more information about setting up assignments see section 7.2.4.

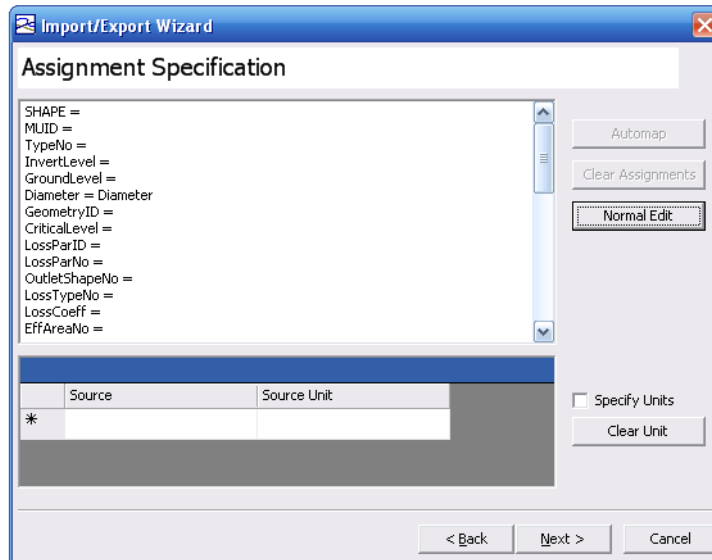


Figure 7.31 Advanced view for assignments

The “AutoMap” and “Clear Assignments” works on the upper grid in the wizard page. An “AutoMap” feature helps the user to do the assignments - so that identical variable names are assumed to be mapped to each other. The “Clear Assignments” clears all assignments.

The lower grid is activated if the “Specify Unit” is checked. Default setting is that it is not activated. In the lower grid, the unit for the source can be set. In MIKE URBAN there is a unit server which keeps track of the unit of each variable. Then by specifying the unit of a source variable, the source variable will be converted to the unit specified in MIKE URBAN. The “Clear Units” operates on the lower grid only and it clears all unit specifications.

Both the “Source” field grid cells and the “Unit” field grid cells act as combo-boxes filled with the available attributes - making it easier for the user to do the assignments and avoiding spelling/typing errors. The “Unit” field lists available units in MIKE URBAN.



7.3.5 Activation/Deactivation of jobs and table configurations

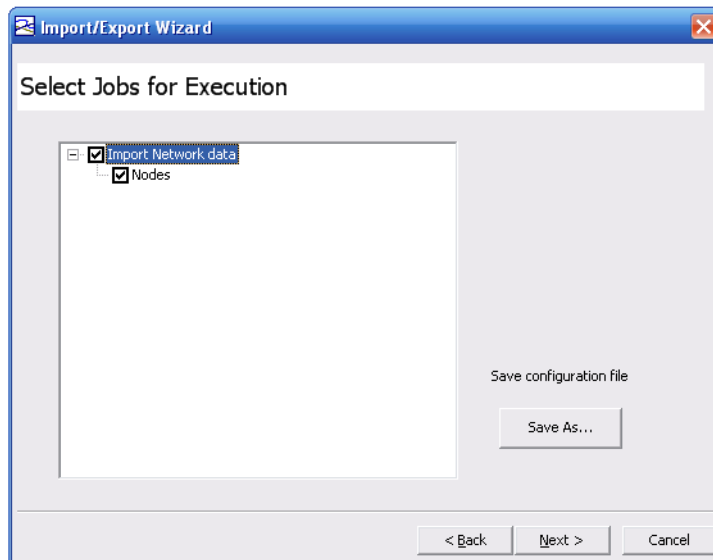


Figure 7.32 Activation/Deactivation of jobs and table configurations

Before the final page is reached in the wizard it is possible to activate or deactivate jobs and table configurations. Furthermore this page also offers the possibility to save a configuration file for later use.



7.3.6 Final wizard page

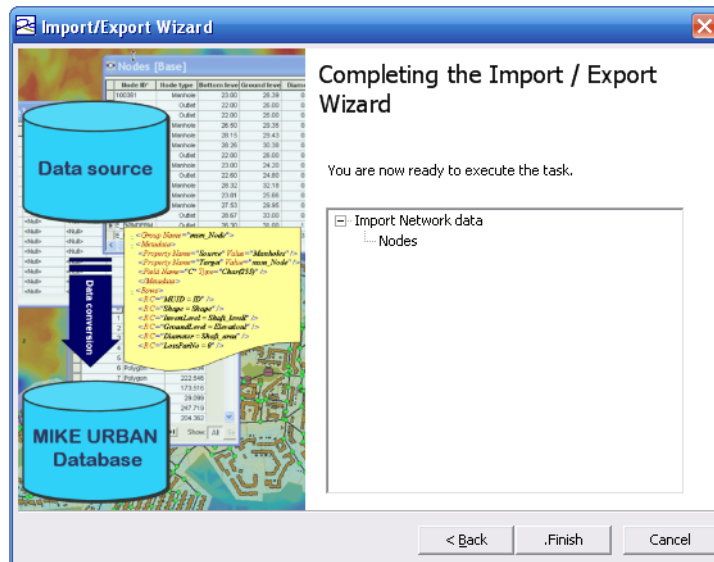


Figure 7.33 Final Wizard page

The final wizard page provides the user with an overview of jobs and table configurations that will be executed when clicking “Finish”.

7.4 Import/Export - Advanced dialog

7.4.1 Overall Description

When the Advanced mode in the Import/Export wizard is chosen the advanced ‘Import/Export’ dialog appears, see Figure 7.34. The left side of the dialog displays all the available import and export jobs. Initially, these are all predefined jobs, but the user-specified Import/Export jobs will also appear here.

The functionality of the predefined jobs is more or less self-explanatory. A more detailed information about each of them follows later in this chapter.

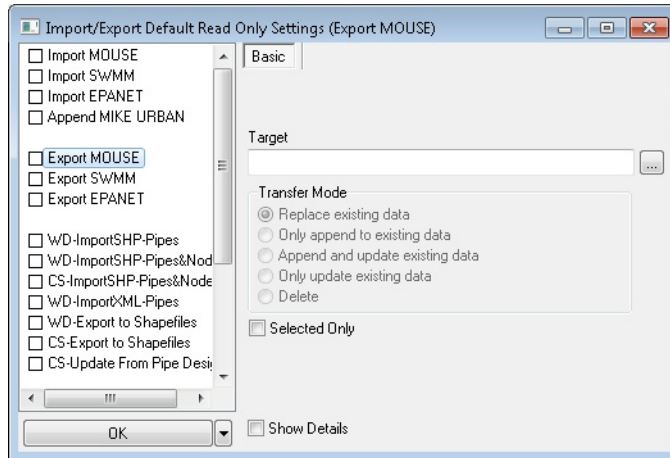


Figure 7.34 The Import/Export Dialog

The jobs can be expanded to show all table configurations for each job. The right side of the dialog is synchronized with the left and shows the settings for the current job or table configuration (highlighted). The check box 'Show Details' on the "Basic" tab page toggles between a simple and a detailed view. With 'Show Details' checked, several more tab pages can be accessed, containing various details on either the job or table configuration - either for better understanding and adjusting of the predefined configurations or for the definition of custom configurations.

Creation and configuration of custom import/export jobs requires a detailed understanding of the underlying principles described in the following table configurations.

7.4.2 The 'Action Button'

The 'action button' in the lower left corner operates on the dropdown principle. Clicking on the arrow opens the list of available actions.

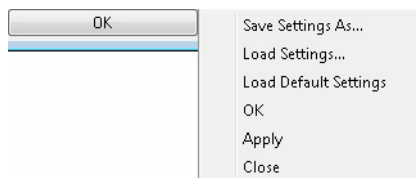


Figure 7.35 The 'Action Button' of the Import/Export Dialog

Save Setting As...

Saves all the jobs displayed on the list (both the predefined and user-specified) into the XML file under user specified name and destination. The default settings file 'muIO.XML' is per default read-only. However, user should not attempt to save the settings under the default name 'muIO.XML' into the



folder containing MIKE URBAN configuration files ('..MIKE URBAN\CFG'). This would make the subsequent restoration of the default settings impossible and potentially cause improper function of the predefined import/export jobs.

Per default any changes made in the import/export dialog is lost unless saved to a configuration file. However it is possible to specify that the configuration file should automatically be saved once the dialog is closed. This only applies if a user specified configuration name is given e.g. a configuration file has previously been loaded or the configuration have a been saved earlier. The setting is changed in the “General Settings” dialog found under the menu “Tools | Customize | General Settings”.

Load Settings...

Replaces the current settings with an earlier saved configuration file (XML).

Load Default Settings

Restores the default settings, i.e. replaces the current settings with 'muIO.XML'.

OK

Initiates the execution of all the selected import or export jobs and closes the dialog. Unsaved configuration edits are lost.

Apply

Initiates the execution of all the selected import or export jobs but leaves the dialog open.

Close

Closes the advanced Import/Export dialog.

7.4.3 Job Local Menu

For any import or export job available on the list, the right-click gets access to a local menu, see Figure 7.36.

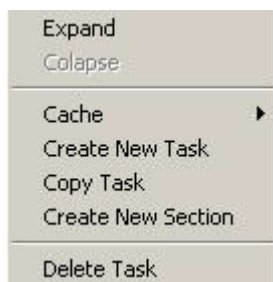


Figure 7.36 Local menu when right-clicking on a import/export job

Expand/Collapse

Expand and collapse toggles between displaying or hiding the individual table configurations.

Cache

The local options under Cache, see Figure 7.37, offer various useful functions. These functions can be very helpful when debugging import/export procedures, as they enable the step-wise import/export process.

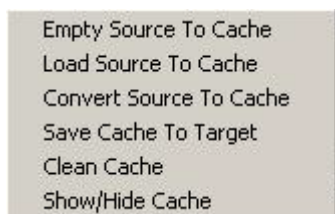


Figure 7.37 The options found under Cache

Empty Source To Cache

This loads an empty source to the cache, i.e. it makes it possible to see the various tables of the source, but no data is filled in.

Load Source To Cache

This actually loads the source data into the cache. The cache viewer will allow you to see the data that has been read from the source.

Convert Source to Cache

The source is converted to the target data module, as defined in the corresponding bridge. The converted data can now be studied before they are saved to the storage.

Save Cache

Saves the cache to the target data storage (file or database).



Clean Cache

Empties the cache.

Show/Hide Cache

Shows/hides the cache viewer.

Create New Job

Creates a new job (import or export) as defined on the tab pages to the right hand side of the dialog.

Copy Job

Copies an already existing job.

Create New Table Configuration

Creates a new table configuration within the selected job.

Delete Job

Deletes an already existing job.

This option, along with 'Create New Job', 'Copy Job' and the 'Save As' and 'Load' functions of the action button provides a complete toolset for composing and manipulating customized import/export configurations.

7.4.4 Table Configuration Local Menu

On the table configuration level, the local menu provides the possibility to create, copy and delete table configurations (Figure 7.38). The functionality of these options is self-explanatory.

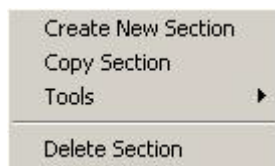


Figure 7.38 Local options on table configuration level

Table Configuration Tools

The 'Tools' option (Figure 7.39) contains a number of tools that support the efficient work with individual table configurations.

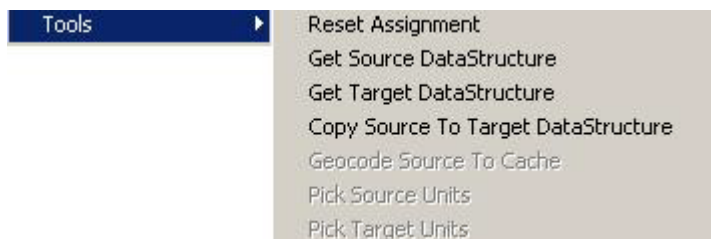


Figure 7.39 The table configuration tools

Reset Assignment

An assignment is a set of commands which control the data conversion from a source table to a target table (more details to be found further ahead in this chapter). Initially, the new assignment contains just the left side of the conversion commands, which is actually the list of all attributes in the target table. The right side of the assignment commands is empty, except when both the source and target table contain attributes with the same name. In that case, the system assumes that such attributes should be converted directly 1:1.

The 'Reset Assignment' removes any user-specified assignments for the current table configuration and restores the 'default' assignment.

Get Source Data Structure

The source data structure is not loaded automatically into the Import/Export workspace after activation of a job. If this is needed for any reason (e.g. review, modification or job design) the user can load and make the source data structure available here.

Get Target Data Structure

A similar functionality as with 'Get Source Data Structure', just operates on the target.

Copy Source To Target Data Structure

Creates for the current table configuration a data structure in the target, that is identical to the source. This is very useful when creating export jobs.

7.4.5 Transfer Modes

The following transfer modes can be applied:

- Replace existing data (will replace all existing data with new data)
- Only append to existing data (appends new data to existing data)
- Append and update existing data (updates existing data and appends new data)
- Only update existing data (only updates existing data)

For further information about transfer modes please see section 7.2.5.



7.4.6 Pre-defined Import and Export Jobs, Backup/Restore Jobs and Connectors

Pre-defined Import and Export Jobs

Default import and export jobs have been set up for MOUSE, SWMM5, and EPANET data, as well for some other widely used formats. More specifically, the predefined import and export jobs are:

- Import MOUSE
- Import SWMM5
- Import EPANET
- Export MOUSE
- Export SWMM5
- Export EPANET
- WD-ImportSHP-Pipes
- WD-ImportSHP-Pipes&Nodes (imports pipes and nodes from a shapefile and builds up the topology)
- CS-ImportSHP-Pipes&Nodes (imports pipes and nodes from a shapefile and builds up the topology)
- WD-ImportXML-Pipes (imports pipes from the xml file)
- WD-Export to Shapefiles (exports to shape file)
- CS-Export to Shapefiles (exports to shape file)

Some of these jobs are described in the following and a tutorial is found in the MIKE URBAN Tutorial manual.

Basically, each predefined import/export job requires a specification of the source/target (file name and path) only. Since the import/export jobs from/to special formats (the MOUSE, SWMM5, EPANET) have the default configurations available, the application is straightforward. On the other hand, import/export from/to generic format (i.e. where data are stored according to certain syntactic rule, but the actual structure of tables may vary), requires the definition of the attribute assignments before the actual import/export operation can be executed.

Please note that for large projects, import and export jobs may take substantial time.

Import from a MOUSE project

When importing from an existing MOUSE setup (MOUSE 2003 or newer format) do the following:

- Choose File|Import/Export.



- Click on 'Import MOUSE' to select it (if it is not selected already).
- Select the source file, i.e. the MPR-file, in the 'Source' field, on the right side of the dialog.
- Optionally, change the 'Transfer Mode' (see above for this).
- Press the 'OK' button, which will initiate the import process.
- After the completed import, the dialog will be closed automatically.

Import from SWMM/EPANET

When importing from SWMM do the following:

- Choose File|Import/Export
- Click on 'Import SWMM' or 'Import EPANET' to select the wanted job (if it is not selected already).
- Select the source file, i.e. the inp-file, in the 'Source' field, on the right side of the dialog.
- Optionally change the 'Transfer Mode' (see above for this).
- Press the 'OK' button.

After the completed import, the dialog will be closed automatically.

Pre-defined Backup and Restore Jobs

The 'Backup To Simple XML' and 'Restore From Simple XML' are predefined as a default backup/restore facility of MIKE URBAN.

The 'Backup To Simple XML' works on the entire database and produces a backup of it in XML format.

'Restore From Simple XML' restores the database again from the XML file respectively.

It should be noted that 'Backup To Simple XML' and 'Restore From Simple XML' will only apply to the active scenario and will not include other scenarios or alternatives.

A useful application of 'Backup...' and 'Restore...' may be e.g. related to moving the data between various database versions.

Connectors

Connectors are used for creating exports of your data in a simple way.

On the list of predefined import and export jobs some are named as 'connectors'. These actually do not convert between formats, since they are connected to the same type of data module ('muStorage') on both sides (Source and Target). This can easily be checked by looking at the job details, where



Job.Source and Job.Target are always the same storages. Implicitly, this means that no data conversion is done by a single connector.

The predefined connectors can be used alone or in a combination. A possible purpose of using a single connector is to load the source data into the cache ('Cache|Load Source to cache'), which opens the possibility to review the data.

Use of several connectors in the same operation may, e.g. include subsequent loading of data from various sources into the cache. This is achieved by repeating the action as described in the previous paragraph, just using an appropriate storages. Once all the wanted data are in the cache, the data can be reviewed and the entire content of the cache (or a sub-list) can be saved into any available target format. For example, the contents of several *.pfs files ('pfs' stands for 'parameter file system' and is a generic format for data storage developed by DHI and used by most of DHI software products) can be loaded into the cache and then saved into a single *.xml file.

Or, the entire MIKE URBAN project database can be exported into a *.pfs file. This is achieved by using the 'Connector To Project' and 'Connector To Simple PFS' in the following way:

- Right click on 'Connector To Project' and select 'Cache|Load Source to Cache' (this may take some time, due to the large number of data tables). This loads the complete content of the MIKE URBAN geo-database into the cache. The exact form of data depends on the Storage Properties on the 'Source' tab page. For normal usage there is no need to make any adjustment of the 'Properties'.
- Click on job 'Connector To Simple PFS' and select target file on 'Basic' tab page
- Right click on the job 'Connector To Simple PFS' and select 'Cache|Save Cache to Target'

The result of this operation is a complete backup of the MIKE URBAN data in a 'pfs' format.

Similarly, the MIKE URBAN geo-database can be backed up to XML.

If only a single table or a subset of tables should be loaded the "Modules Mask" can be set to specify which filter to use. Multiple tables are separated with ';' and '*' can be used as a wildcard. See section on 'Modules mask'.

7.4.7 Detailed Description of Import/Export Job Properties

Each job is described by some general settings/properties. For the currently selected job (highlighted by clicking), these settings are displayed on the right side of the dialog, on up to five tab pages. User can toggle between the detailed and basic view by check marking the 'Show Details' check box.



The tab pages contain the following information:

Basic

- **Source:** Defines the source of the data for the current job. For the import jobs the source is the name of the database, file or a directory containing the data to be imported. In export jobs, per default it is defined as 'Run-Time Workspace', which is a reference to the actual MIKE URBAN project. This can be replaced by any other MIKE URBAN project database file.
- **Target:** Defines the destination (target) of the data for the current job. For the predefined import jobs, the target is always the current MIKE URBAN project database (cannot be changed). For the export jobs, target is the name of the database, file or a directory where the data are to be exported to.
- **Transfer mode:** This is of relevance for import jobs only (see section 4 for the description).
- **Selected Only:** Exports only the data that are 'selected'. This is only available for export jobs of feature layers.

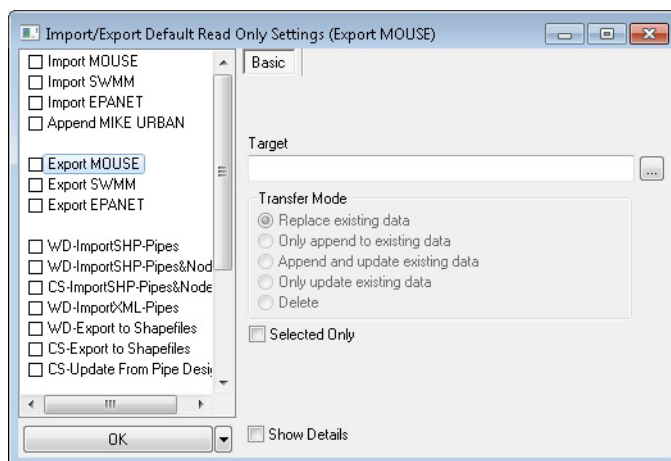
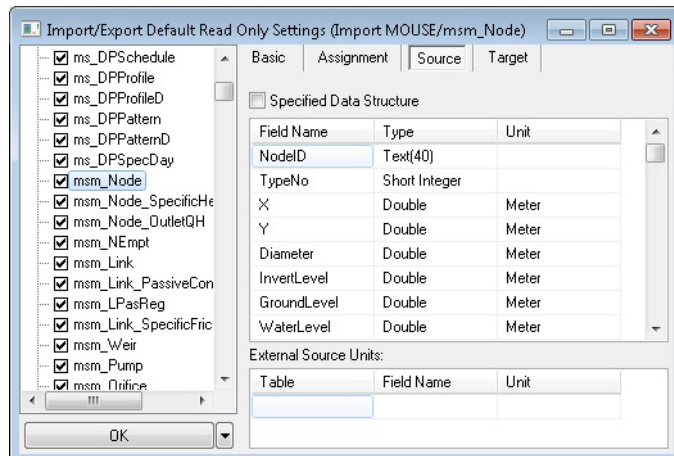


Figure 7.40 Import/Export dialog with 'Basic' job tab page displayed

Source

This tab page provides information related to the data source.



Import/Export dialog with 'Source' job tab page displayed

- Storage:** Defines the storage driver ('muStorage') that is to be used in association with the source. User can select only among the registered storages that appear on the list.
- Modules mask:** Controls which of the source tables are to participate in the execution of the job. The individual entries on the list are separated by semicolon. The full table names can be written, but regular expressions (e.g. wildcards '.*') can be used as a replacement of any other single character or a string of characters. An empty modules mask implies inclusion of all source tables into the job execution. The purpose of the modules mask is to limit the operations to the relevant tables only and by these means improve the operation efficiency. However, in the current implementation, the modules mask has effect only for the 'global jobs', i.e. the jobs without table configurations, and during manual 'Load Source to cache' (mainly used to speed up the debugging work during development of the complicated assignments. Modules mask has no effect on the straight execution of complex jobs including table configurations. Space is allowed between table names, when writing the modules mask (e.g. nodes; pumps)
- Storage Properties:** Storage properties are parameters that control the behaviour of the storage in various aspects. This table configuration of the dialog displays the list of relevant properties for the actual storage and the property values as they are set for the current job. When creating a new job and selecting one of the storages, the Storage Properties list is populated with all available properties and their default values. A new property can be added to the list, but if not recognised by the storage, it will be ignored. If a property is deleted from the list, its default value will be applied.



ESRI Geodatabase Storage

When importing to MIKE URBAN the target is a geodatabase. The storage properties for the ESRI Geodata, see Figure 7.41, are described in the following table.

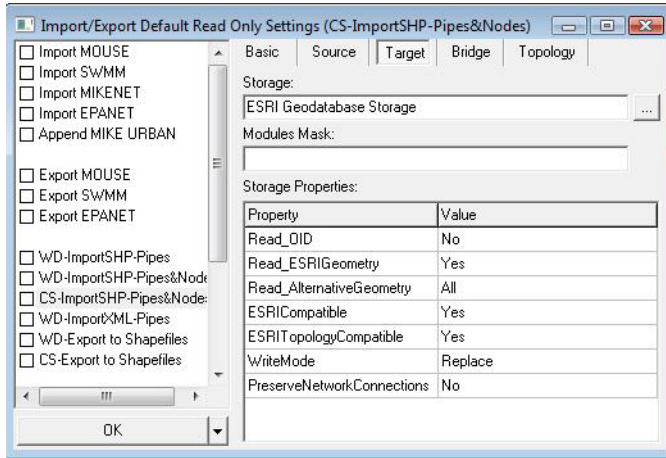


Figure 7.41 The Properties of the ESRI Geodatabase storage

Table 7.24 ESRI Geodatabase Storage

Property	Available settings	Description
Read_OID	Yes, No	OID means "Object Identifier". The OID is the unique name/key used to identify an object in a table in an ESRI database or shapefile. If used the OID will be transferred from source to target. If not used new OIDs will be generated.
Read_ESRIGeometry	Yes, No	Specifies if ESRI geometries (Shapes) should be read from the imported data.
Read_AlternativeGeometry	All, Points, Lines, Polygons	Intended for import of tabulated geometries. Format is: ID, SequenceNumber, X, Y This can be used to create shapes from a list of values.



Table 7.24 ESRI Geodatabase Storage

Property	Available settings	Description
Compute_FromToNode	Yes, No	Intended for import of e.g. MOUSE data to "msm_Link" and similar MIKE URBAN tables where "_ToNodeID" and "_FromNodeID" can be used. When set to yes, the bridge will autogenerate a shape (polyline) connecting the ToNode with the FromNode.
WriteMode	Replace, Append etc.	Reflects the transfermode. This should never be edited in the properties, but will be updated when the transfermode is changed.
PreserveNetworkConnections	Yes, No	If "No" (default value), an update of the location of nodes will not update connected elements e.g. links can be disconnected and others connected. If "Yes", an update preserves connected element e.g. link shapes are updated so that link is still connected to the moved node. PreserveNetworkConnection will not work when the transfer mode is "Replace existing data".
ESRITopologyCompatible	Yes, No	If "Yes" (default value), the 'Topology' tab will be available. If "No", then the 'Topology' tab is not displayed.

Target

This tab page provides information related to the job target.

- Storage: the same as on the 'Source' tab page.
- Modules mask: the same as on the 'Source' tab page.
- Storage Properties: the same as on the 'Source' tab page.

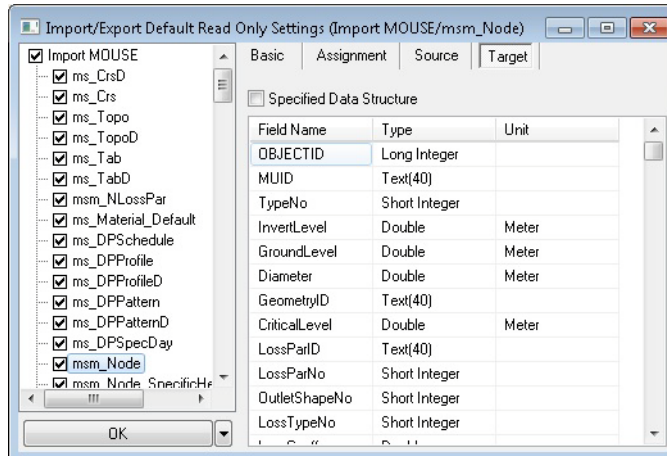


Figure 7.42 Import/Export dialog with 'Target' job tab page displayed

Bridge

This tab page provides essential information related to the bridge used in the current job.

- **Bridge:** Defines the data converter engine that is to be used in association with the job. User can select only the bridges available on the list. For most predefined jobs, the 'Standard MU DM Bridge' is used, or otherwise the 'EPANET 2.0 Bridge'. For user specified jobs the 'Standard MU DM Bridge' would be OK. In exceptional cases, a new type of bridge can be developed.
- **Bridge Properties:** Bridge properties are parameters that control the behaviour of the bridge in various aspects. This table configuration of the dialog displays the list of relevant properties for the actual bridge and the property values as they are set for the current job. When creating a new job and selecting one of the bridges, the Bridge Properties list is populated with all available properties and their default values. A new property can be added on the list, but if not recognised by the bridge, will be ignored. If a property is deleted from the list, its default value will be applied.



- Global Variables:** The bridge architecture allows access to certain attribute values from the tables constituting the current table configuration. By means of the 'LookUp' functions values from other tables can be accessed. However, this is frequently not sufficient to secure the wanted data conversion. The flexibility of the bridge in this respect may be improved by user-specified global variables. The value of a global variable is known throughout the entire job, and this value is maintained as long as it is not redefined in the sequential data conversion process. This part of the dialog displays all the global variables defined for the actual bridge and their type. A new global variable can be added on the list. Deletion of a global variable would cause problems in the function of the bridge.

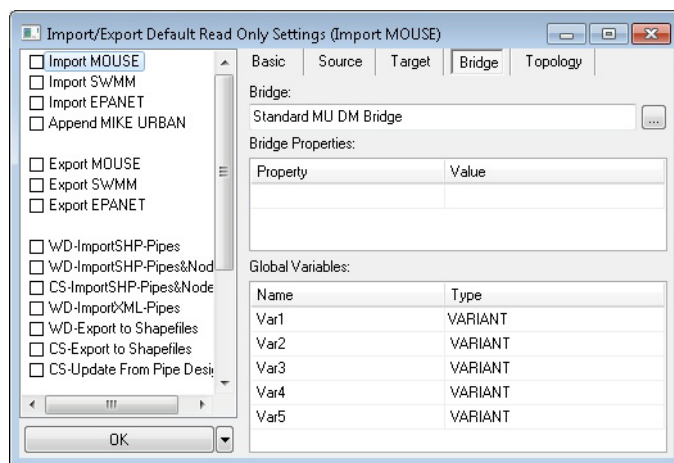


Figure 7.43 Import/Export dialog with 'Bridge' job tab page displayed

Topology

An important part of the network data import is the establishment of the network topology, i.e. relations between nodes and links. A MIKE URBAN network is a so called 'directed network', where each link emanates from one node and terminates at some other node. Frequently, such relations do not exist (or are incomplete) in the data source, and have to be established during the import process. A typical case are GIS files where nodes and pipes are stored in separate layers, without any explicit data relation, apart from the geographical position. The situations which have to be resolved by the import routines include e.g.: splitting of a single link into two or more links according to the overlaying nodes, association ('snapping') of the link edges with the nodes inside the 'snap tolerance', inserting missing nodes at the link edges, inserting missing nodes at link crossings and/or junctions, etc. In any import job, an automatic process for the establishment of the network topology may be activated and controlled through this tab page.



- **Postprocess Network Topology:** This check box activates or deactivates the topology post-processing. If not activated, the network will be imported exactly in the same shape as found in the source.
- **Settings:** Setting are parameters which control the behaviour of the network post-processing in various aspects. This table configuration of the dialog displays the list of relevant settings for the post-processing algorithm and the settings values as they are set for the current job. When creating a new job and activating the post-processing, the Settings list is populated with all available settings. The appropriate values need to be specified in order to make the post-processing algorithm functional.

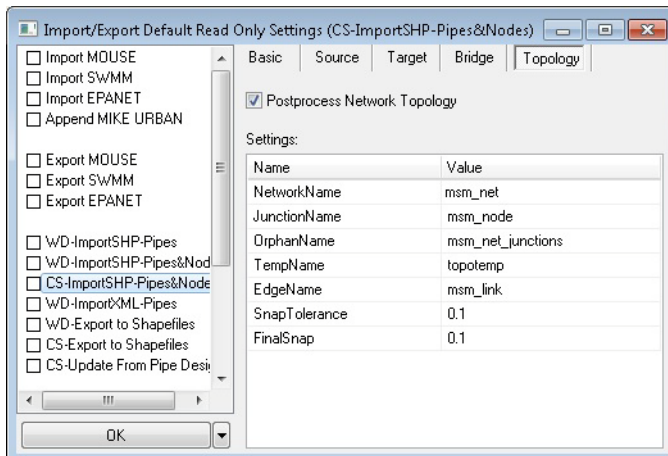


Figure 7.44 Import/Export dialog with 'Topology' job tab page displayed

The parameters for “NetworkName”, “JunctionName”, “OrphanName” and “EdgeName” will be different depending on the network type (MOUSE, SWMM or WD). The “Prefix” property can be omitted but can be used for adding a prefix to autogenerated MUID’s. This will later on make it easy to recognize which MUID’s have been autogenerated during import.

- “NetworkName” [String]: Name of data source (feature class) for the geometric network
- “JunctionName” [String]: Name of data source (feature class) for the geometric network’s junctions
- “OrphanName” [String]: Name of data source (feature class) for the geometric network’s orphan junctions.
- “TempName” [String]: Name of data source (feature class) for intermediate (during import session) junctions.
- “EdgeName” [String]: Name of data source (feature class) for the network’s edges.



- "Prefix" [String]: If specified, the MUID of auto-generated feature's (during import session) is prefixed with this string. If false or empty, no prefixing is done.
- "SnapTolerance" [Real Number]: Snap tolerance used during the import session for connecting junctions and edges. So during the import links and nodes are snapped if they are within SnapTolerance. After the import the dataset is recreated with FinalSnap snap tolerance and the geometric network will use this. This means that ArcGIS at a later time will 'see' geometries as connected if they in the map are within the tolerance the geometric network was created with.
- "FinalSnap" [Real Number]: After the import session the geometric network will have this snap tolerance.
- "RemoveTinyEdges" [Boolean]: If true, features (specified in property "EdgeName") smaller than "SnapTolerance", are removed

Please note: The snap tolerance specified under "Tools | Customize | General Settings" is used when creating new features in MIKE URBAN. MIKE URBAN will search within this tolerance for other features and make the cursor red if features that can be snapped to are found. This is necessary because the geometric network's snap tolerance is much too small for users to use when editing the network graphically.

Table 7.25 Topology settings for MOUSE, SWMM and EPANET

Parameter	MOUSE	SWMM	EPANET
JunctionName	msm_Node	mss_Node	mw_Junction
OrphanName	msm_Net_Junctions	mss_Net_Junctions	mw_Net_Junctions
TempName	topotemp	topotemp	topotemp
EdgeName	msm_Link	mss_Link	mw_Pipe
SnapTolerance	0.25	0.25	0.25
FinalSnap	0.25	0.25	0.25
Prefix			
RemoveTinyEdges	true	true	true

7.4.8 Detailed Description of Table Configurations Properties

Import/Export jobs may contain several table configurations. Each table configuration refers to a single target table Table Configuration that can be accessed (e.g. for review or editing) by expanding the currently selected job. When a table configuration is selected (highlighted by clicking), the right side of the dialog displays various information related to the table configuration, on up to five tab pages. User can toggle between the detailed and basic view by check marking the 'Show Details' check box on the 'Basic' tab page.



The tab pages contain the following information:

Basic

- **Source:** Defines the source table for the current table configuration. For the import jobs the source is the name of the file, a database table containing the data to be imported or whatever is relevant for associated Storage. In export jobs, the source is one of MIKE URBAN database tables.
- **Target:** Defines the destination (target) data table for the current table configuration. For the import jobs, the target is always one of the tables in the current MIKE URBAN project database. For the export jobs, target is the name of the file or a database table where the data are to be exported to (whatever is relevant for the associated Storage).
- **Table Configuration Mode:** This control the transfer mode when data are converted from the source to the target in the cache. The following transfer modes are available:
 - Replace existing data (will replace all existing data with new data)
 - Only append to existing data (appends new data to existing data)
 - Append and update existing data (updates existing data and appends new data)
 - Only update existing data (only updates existing data)

For further information on transfer modes please see section 7.2.5.

- **Source Filter:** Before actual conversion the records in the source data module can be filtered. Only those records, which match the filter definition, are converted into target module. For more on filter definitions see examples later in this section.
- **Source Sorting:** Before actual conversion the records in the source data module can be sorted.
- **Source Distinct:** This property defines, that only those records from source data module, which have unique value in the specified attribute are used for conversion. Value of this property is the name of required attribute. Based on a user specified column in the source table the Source Distinct function will filter records so that only one instance of a name/value is imported. This means that if multiple records have same name/value in the specified column only one will pass the filter. An example: Source Distinct can be employed when the user wants to create a list of all the Links that have a geometry defined in a table of link geometries. In the link geometry table each link will occur multiple times, each time with a different sequence number (and a coordinate set). If the column containing the "Link ID" is specified in the Source Distinct field, the import will generate a list of unique link IDs present in the link geometry table.



- **Load Target Data Before Conversion:** This option allows to populate target cache with current target data before actual conversion is triggered. It can be useful particularly in combination with Update Table Configuration Mode.
- **Save Target Immediately After Conversion:** This option allows you to save content of target cache directly after this table configuration conversion is done. Otherwise (by default) cache data will be saved on the end of job after all table configurations are converted.
- **Show Details:** Toggles between the simple and detailed view.

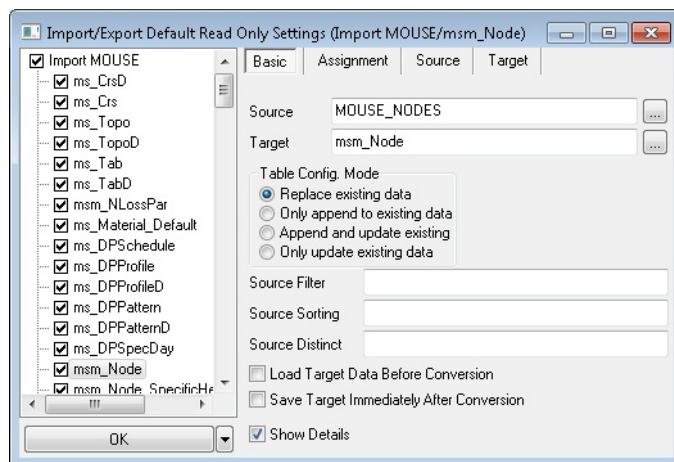


Figure 7.45 Import/Export dialog with 'Basic' table configuration tab page displayed in 'Detailed' mode

Examples of Source Filters

When setting up filters the operators AND and OR are useful.

- AND is used to narrow a filter, as items must pass both 1st requirement and 2nd requirement.
- OR is used to broaden a filter, as items only have to pass either 1st requirement or 2nd requirement.

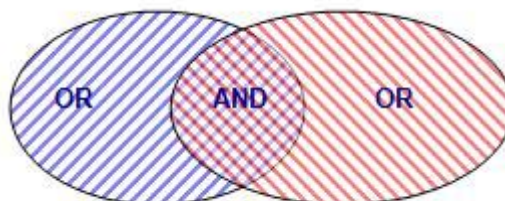


Figure 7.46 The difference between using AND and OR operators



Filters are specified as:

```
ColumnName1; ColumnName2:: ColumnName1 = value1 AND Column-  
Name2 IN (value2 , value3)
```

Example 1

Below will only import data when the column entitled 'type1' is either 1 or 2 as well as 'type2' will have values different from 15, 16 and 45.

```
type1;type2:: type1 IN (1,2) AND NOT type2 IN (15, 16, 45)
```

Example 2

Below will only import data when the column entitled 'type1' is either 1 or 2 and 'type2' will be equal to 4.

```
type1;type2:: type1 IN (1,2) AND type2 = 4
```

Assignment

The 'Assignment' tab page contains the list of commands for the current table configuration.

Each assignment command includes the target attribute on the left side of the equality sign, and a more or less complicated expression on the right side of the equality sign. The simplest expression is the source attribute name, which is used when a simple 1:1 data conversion applies.

Initially, the system provides the assignments for all target attributes. They are incomplete, and only those where source and target attributes are identical, are completed assuming a 1:1 conversion. User must complete the assignments by editing.

For more information about assignment see section 7.2.4.

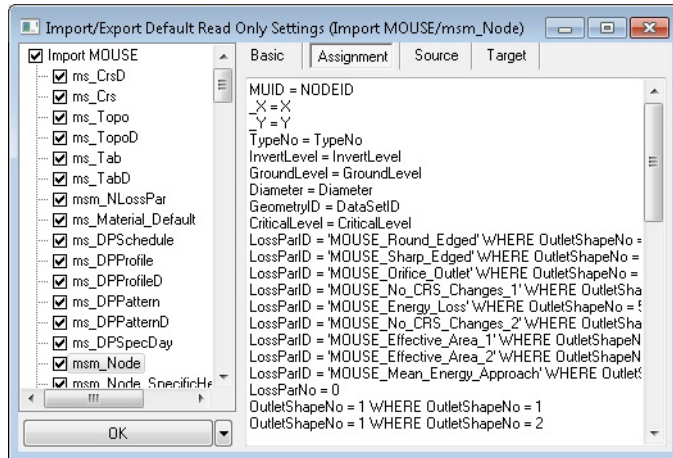


Figure 7.47 Import/Export dialog with 'Assignment' table configuration tab page displayed

Source

This tab page allows for displaying and editing of the source table data structure. The data will only be taken into account for an import job.

- **Specified Data Structure:** This checkbox toggles between the "read-only" and "edit" mode for the data structure grid. With the checkmarked box user can edit Type and Unit fields. This is particularly important for editing Unit field. With appropriate unit specified for numerical attributes, the system will automatically control the import process and perform the necessary re-computation of the values, according to the units actually specified for the current project. If the unit is not specified, the import will be carried out without any conversion, or according to an arithmetic expression specified in the assignment. Units must be specified by valid string identifiers, in order to be recognised by the system.
- **External Source Unit:** If the assignment involves data from some other table than this, units for these attributes can be specified here. This would allow that also these attributes are converted correctly.

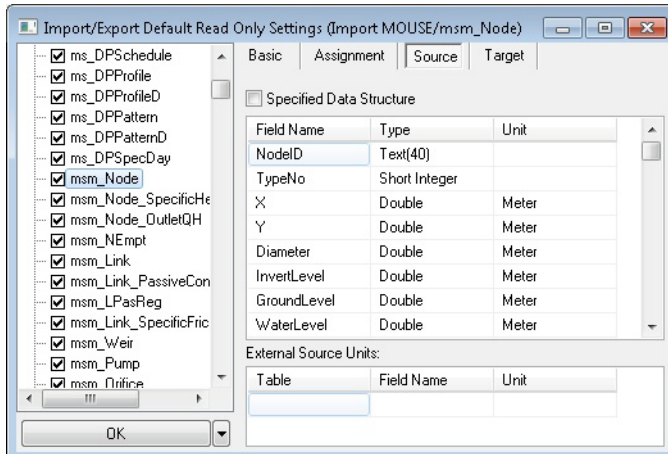


Figure 7.48 Import/Export dialog with 'Source' table configuration tab page displayed

Target

This tab page allows for displaying and editing of the target table data structure. The data will only be taken into account for an export job.

- Specified Data Structure: This provides exactly the same functionality and in the 'Source' tab page, but it operates on the 'Target' table. Here it will also be possible to specify the Field Name directly together with the Type and Unit fields.

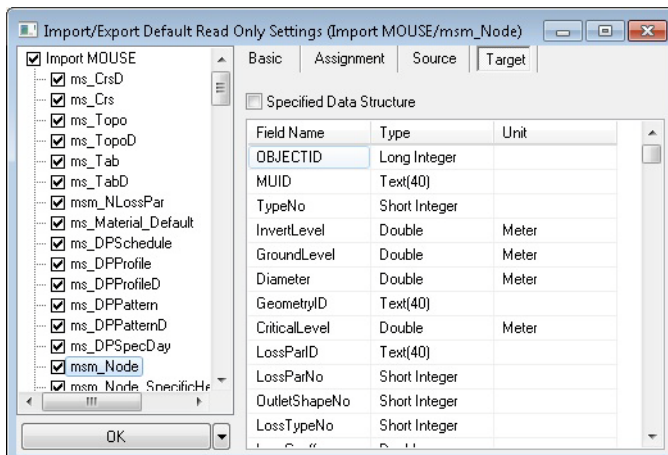


Figure 7.49 Import/Export dialog with 'target' table configuration tab page displayed



7.4.9 Defining Custom Import/Export Jobs

Defining custom import or export jobs requires a thorough understanding of the principles described in this document. It is recommended to start with some simpler jobs and gradually build up the confidence and experience towards more complex jobs. Here it can be a good idea to start by defining the configuration in the wizard and then load the configuration in the advance mode for further modifications. It is in this area where DHI can provide assistance upon request.

Creating a New Job

The following actions are required in the process of creating a new import or export job:

1. Right click on Job List and select "Create New Job". This inserts a new job on the list. It is possible to rename the new job to an appropriate name.
2. Define the job source storage (on the "Source" tab page to the right)
3. Define job target storage (on "Target" tab page)
The Modules Mask is not important for conversions but can be useful for debugging.
4. Define Bridge (on "Bridge" tab page)
5. Define source and target on the "Basic" tab page

Optionally, (under some circumstances where target is MIKE URBAN project), Topology settings can be specified on "Topology" tab page

There is a number of tools (right click on job and select "Cache" menu) which can help you to go through this process.

The actual data conversion is driven by list of conversion table configurations under the job.

Creating New Table Configurations

To create a new table configuration you should:

1. Right click on Job and select "Create New table configuration". Here you should specify at least Source and Target on "Basic" tab page and conversions itself on "Assignment" tab page.
2. There is a number of tools (right click on table configuration and select "tools" menu) which can help you to go through this process.





8 Flagging

8.1 Introduction to the MIKE URBAN Flagging System

MIKE URBAN provides the ability to track your data in a very flexible way, the tracking is done by assigning user defined 'flags' to the data. As the flags are defined by the user and as the user is in complete control of setting the flags they may be used for a variety of purposes.

8.1.1 What are flags?

Flags are additional data attributes in the database that can be set to an integer value. The valid values and the meaning of the integer value is controlled by 'Coded Value Domains'. A coded value domain is a list of integers and a corresponding value, typically the code will be automatically translated to the corresponding value when shown in the user interface or in the map.

8.1.2 What can be flagged?

Flagging can be attached to all tables representing the physical model (i.e. pipes, manholes, nodes, catchments etc.) and flags exist in two levels:

- **Record Flagging:** This flag is attached to the whole record, a typical use could be to keep track of the data source. The corresponding database attribute is always called 'Element_S'.
- **Attribute Flagging:** The major geometrical attributes of the model elements such as Diameters, sizes, levels have their individual flags on the attribute level. These flags are typically useful for tracking data manipulation and editing operation. The database attribute to hold these flags are called XX_S, where the XX part is replaced by the name of the attribute i flags i.e. like Diameter_S.

8.2 Defining Flag Values

MIKE URBAN will have a few values predefined in the flag domains, but usually the first task will be to define and implement the flagging system that is most relevant to your workflow. You will need to define a list of codes and corresponding values like:

- 1 - Data OK
- 2 - Needs Verification
- 3- Measured
- 4 - Interpolated
- etc.



As MIKE URBAN uses the value and not the code in the GUI it is not important to keep a system in the code values (they can both be consecutive or follow a system). Just remember that if you access from outside GIS and MIKE URBAN you will see the codes not the values.

You can edit the code values in the editor found in the menu under Tools|Customize|Status Codes, see Figure 8.1.

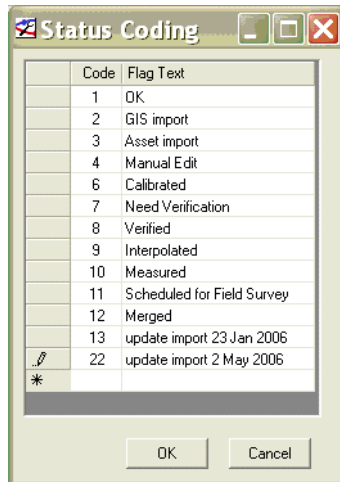


Figure 8.1 Defining the status codes

The value of codes can be edited on-the-fly at any time. If codes are deleted the status fields will retain their numerical value but the code will not be 'translated' in the GUI.

8.3 Setting the Flags

The default value for all flags is 'null'. The value for a flag may be set by one of several methods:

During Import

Simply setting the flag value during the import is the most easy way to track the source of data in case you are getting data from several sources or in batch. The assignment is done in the assignment specification as seen in Figure 8.2.

Element_S = 1

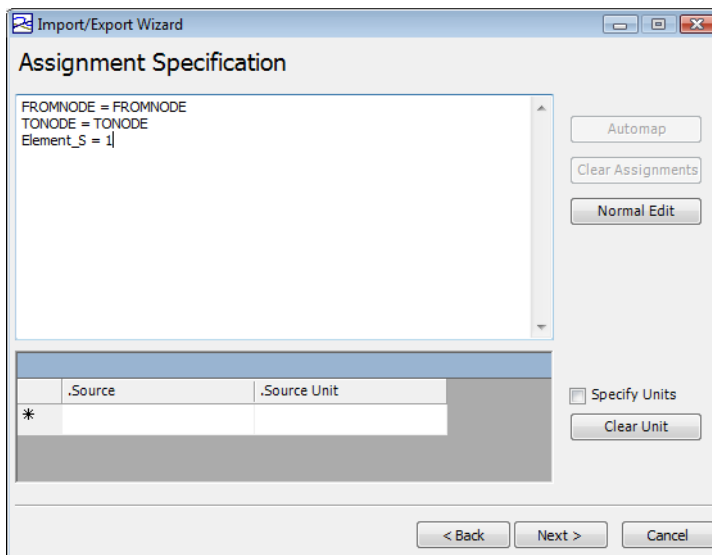


Figure 8.2 Setting the flags when importing data

Notice that you need to use the code value in the assignment specification. If your external system has a tracking/flagging system this can of course be utilized to keep the two data bases synchronised. In that case you should assign the flag values from an external field like:

Diameter_S = QC_Status

Using the Flag Tool

A special tool for setting status flags based upon selections is included in the 'Tools' menu. This tool will mark all selected records with the status code chosen by the user. Both record level and attribute flags may be assigned using this tool, see Figure 8.3.

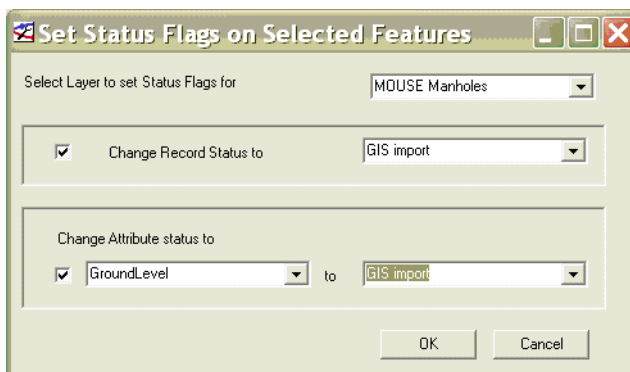


Figure 8.3 Setting status flags on selected features



Note that this tool will not set flags for all records if nothing is selected; this behaviour is chosen to avoid mistakes.

The tool is typically used following a GIS analysis process involving various selection tools. Once records have been identified (selected) the flags can be set to flag the action to be taken on those records. An example could be to compare ground elevations with invert levels to identify all manholes with unrealistic depth, mark them and then schedule a survey.

Assigning Flags with bulk Editing Tools

Several of the tools in MIKE URBAN that change data will offer to flag the affected records/attributes. By selecting the desired flagging when prompted by the wizard the data calculated in a particular session with a particular tool can be flagged for further processing/check etc.

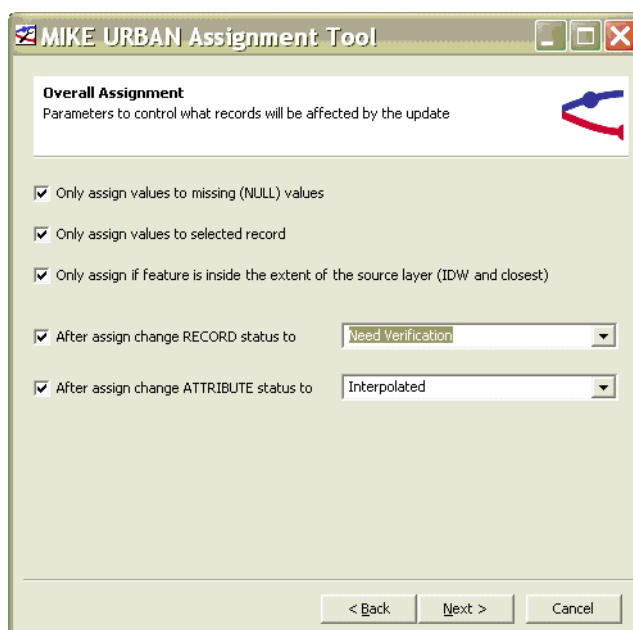


Figure 8.4 Using the assignment tool for setting the flags

Figure 8.4 shows an example from the assignment tool where the tickmarks in the two lower boxes force the tool to assign flag values of 'need Verification' and 'Interpolated' to the status fields.

Other means of setting the flags

Flag values can also be assigned by manual editing in the data editors (in the grid section only) or using the Field Calculator from the editors.



8.4 Using the Flags

The two primary functions of the flags are:

- Select records for further processing.
- Use the flag values to highlight map feature to e.g. provide an overview of data quality.

Selection is most easily done by using the select by attribute tool, see Figure 8.5.

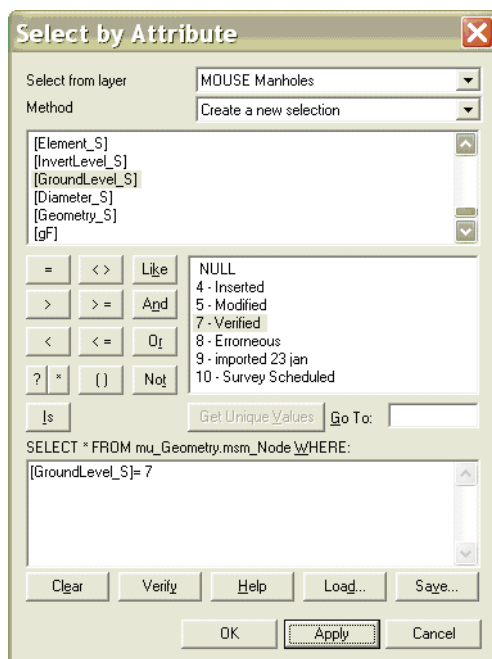


Figure 8.5 Selection by attribute

This is done by selecting depending on the status fields (_S) directly. Pressing the button 'Get Unique Values' will show you the coded values, and which ones are in use.

The flag values are ideally suited for the symbology method called 'Categories - Unique Values'. Here again you can let the system show all the values in use and then you can choose a simple color scale, or as shown in Figure 8.6 you can assign the individual flags their own symbol and color. In this way it is easy to provide both an overview and at the same time highlight important features of a certain category.

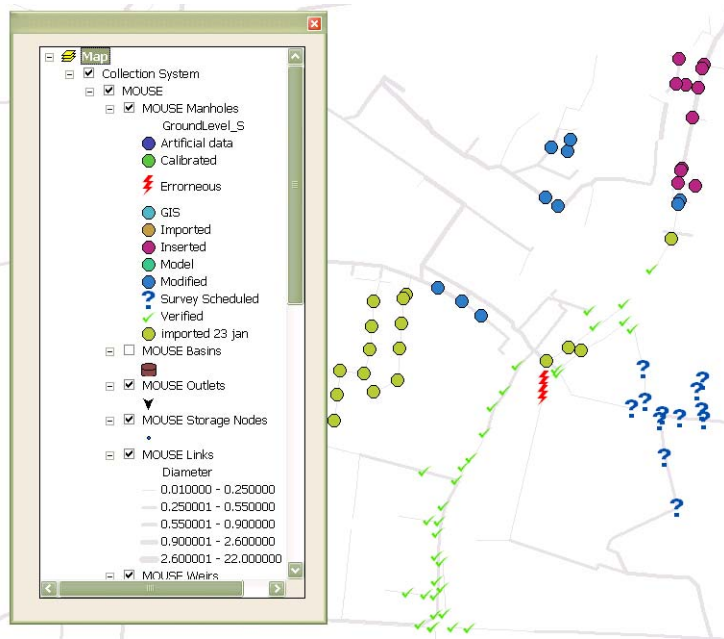


Figure 8.6 Utilizing the symbology for displaying flags on the horizontal plot.



9 Editing Tools

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

If a map of the water or wastewater system is available, MIKE URBAN can import this map and display it as a background image—allowing the user to graphically construct and layout the pipe network system using the Map window. Network components selected from the Components (floating) toolbar can be graphically placed on the screen at the precise location of each component.

9.1 Overview

Editing tools are available for editing network model components (such as junction nodes, pipes, sewer manholes, storage tanks, pumps, valves), network asset components (such as service pipes, hydrants, section valves), demand points (consumption points), or catchments (runoff catchments).

All editing takes place within an edit session. The alterations/changes you make are immediately visible on the map but are not saved to the database until you choose to do so.

- Start editing: Click the Edit menu and click Start Editing. The Editor toolbar is now active.
- Save edits: Click the Edit menu and click Save Edits. Any edits you have made are saved to the database
- Stop editing: Click the Edit menu and click Stop Editing. Click Yes to save changes. Click No to quit without saving.

9.2 Graphical Editing

The easiest way of defining the network data is to graphically define them in the Map window. To display the Map window, select View | Horizontal Plan. Because there is no data defined for this project, the Horizontal Plan window is opened using the default coordinate extension. The default coordinate extents can be defined in the Create New Project dialog box. In this example we will use the default coordinate extents for defining the water and wastewater distribution network.

We will use the Map window to interactively layout a schematic representation of the water and wastewater distribution network. Once the network has been graphically laid out, it is possible to further define the actual junction node, pipe, and other related data using the network component editors.



Figure 9.1 Use the Add Node tool for interactively laying out the network manhole nodes in the CS MOUSE mode

In order to quickly draw a network layout, click on the Create Feature tool from the Feature Edit floating toolbar as shown in Figure 9.1. The icon will become active (it will appear pressed down) and you can point and click within the Map window to graphically add manhole node locations. When finished, click the Add Nodes tool again (it will pop back up) to end the insertion of manhole nodes.

To layout interactively the sewer network components, activate the CS mode by selecting File | Mode | Collection System from the main program menu. In order to draw links, select the Links tool from the Feature Edit toolbar and click within the Map window to graphically add links. Note that the cursor colour changes when snapping onto the existing nodes.

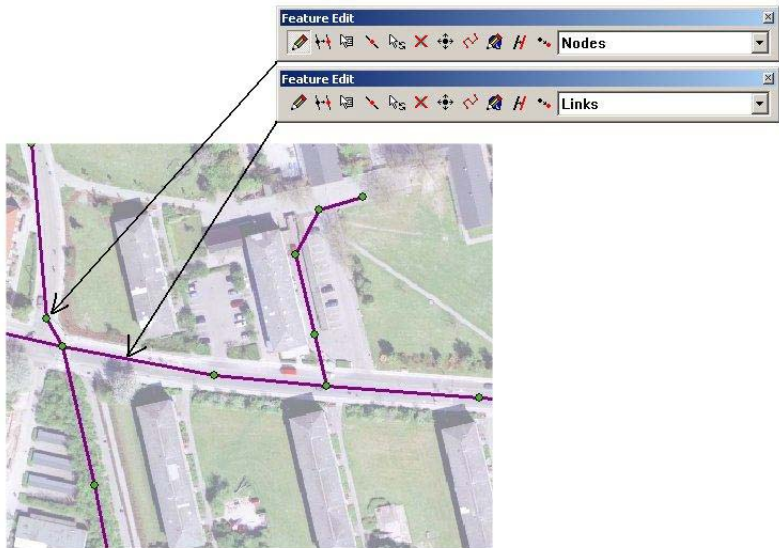


Figure 9.2 Use the Feature Edit toolbar for interactively laying out the network components in the CS mode

To layout interactively the water network components, activate the CS mode by selecting File | Mode | Water Distribution from the main program menu. In order to draw pipes, select Pipes tool from the Feature Edit toolbar and click within the Map window to graphically add pipes. Note that the cursor colour changes when snapping onto the existing nodes.

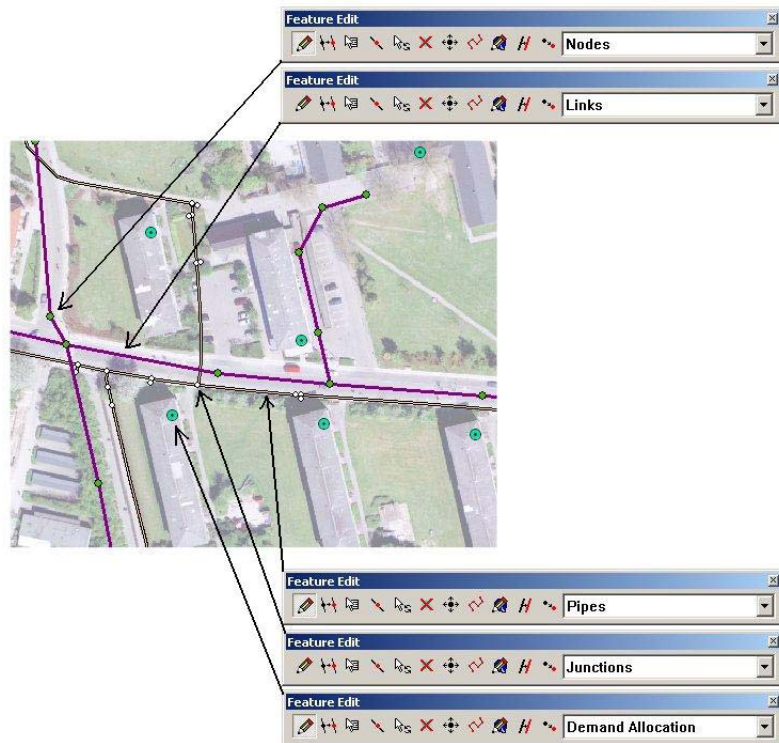


Figure 9.3 Use the Feature Edit toolbar for interactively laying out the network components in the WD mode

9.2.1 Geometric Network Selection

The MIKE URBAN data model consists of several geometric network definitions. They are used to store model and asset data in both Collection System (CS) and Water Distribution (WD) mode. Feature classes representing model elements, such as sewer manholes, sewer pipes are all a part of the collection system network “msm_net”, while feature classes such as water junctions, water pipes are all part of the water distribution system network “wd_net”.

The correct choice of a geometric network is required for graphical editing as well as for network tracing and other network based operations. The following geometric networks are available in MIKE URBAN:

- MOUSE: collection system MOUSE geometric network
- CS Asset: collection system ASSET geometric network
- SWMM: collection system SWMM geometric network
- WD Model: water distribution MODEL geometric network

- WD Asset: water distribution ASSET geometric network

9.2.2 Toolbars

The Feature Edit toolbars are used for interactively laying out the network components of the collection system or water distribution system depending on the selected mode.

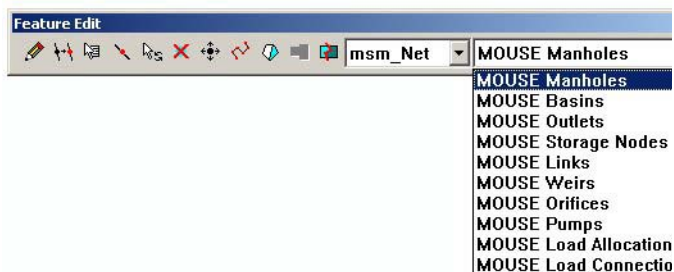


Figure 9.4 Collection System MOUSE editing toolbars

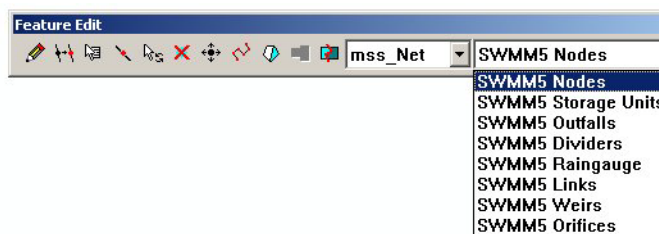


Figure 9.5 Collection System SWMM5 editing toolbars



Figure 9.6 Collection System editing toolbars for ASSET

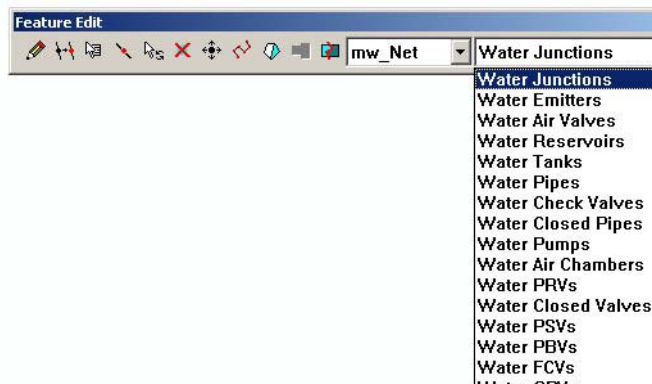


Figure 9.7 Water Distribution editing toolbars for MODEL



Figure 9.8 Water Distribution editing toolbars for ASSET

Below is a list of the Feature Edit tools with a short description given for each tool.



Create new feature

This tool is used graphically add a component by clicking within the Map window.



Change feature type

This tool is used to replace an existing component with a corresponding component by clicking within the Map window.



Move selected features

This tool is used to move selected components graphically within the Map window. Note that all adjacent nodes and links are moved along with the selected components. To select components graphically, click within the Map window with the Select Tool. In WD mode it is also possible to split a pipe A by moving pipe B end over a pipe A. In order to do so, start editing and select an outer node of a pipe you would like to move i.e. connect to a segment of another pipe and move it over the top of that pipe using "Move" tool. MIKE URBAN will display a message box "Do you want to split the nearest pipe?" If you select "Yes", pipes will be split and merged.



Deleted selected features

This tool is used to delete graphically selected components within the Map window. To select components graphically, click within the Map window using the Select Tool.



Edit polyline

This tool is used to alter the geometry of a corresponding pipe within the Map window. In order to insert a point on the existing pipe, select the pipe and select Insert Point from the right-mouse click pop-up menu. Click the pipe to insert a new pipe vertex in the cursor position.



Split line feature

This tool is used to split a pipe graphically by clicking the existing pipe within the Map window.



Reverse orientation of a line

This tool is used to swap the pipe orientation (start and end nodes) by clicking the pipe within the Map window.



Edit polygon

This tool is used to insert a new catchment.



Split polygon

This tool is used to split an existing catchment into two smaller catchments. The split is performed along a line drawn across the catchment (drawing must start and end outside the catchment's contours). The tool automatically deletes the existing catchment record and inserts two new catchment records.



Append polygon

This tool is used to insert a new catchment graphically by appending it to external contours of already existing catchment area. Drawing of a new catchment must start and end within an existing catchment area.



Merge selected polygons

This tool is used to merge two, or more, already existing catchments (according to selection) into one larger catchment. The tool automatically deletes the records of the catchments to be merged and inserts a new catchment record.



Open Editor

This tool is used to open a corresponding component editor by clicking the network components within the Map window.

9.2.3 Keyboard Shortcuts

You can use shortcut keys to perform certain functions from within the environment of MIKE URBAN. Shortcut keys are available for creating a new pro-



ject, saving a project, opening a new project, accessing model editors, selected graphical tools and similar. For example Ctrl+S is the keyboard shortcut key to save a project. The shortcuts are displayed next to the menu item. Tools|Customize|Shortcuts allows you to define your own shortcuts in the application.

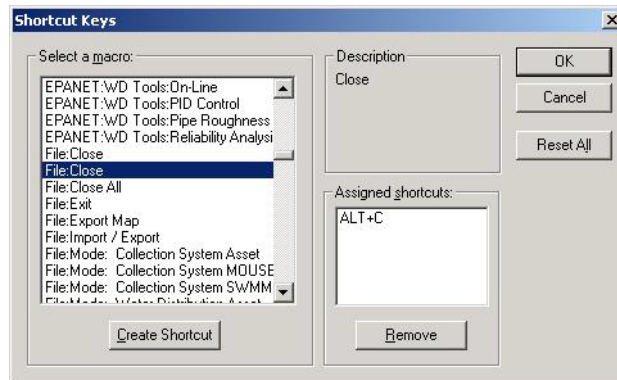


Figure 9.9 Defining your shortcuts in MIKE URBAN





10 Modeling Collection Systems

With MIKE URBAN collection system you can model your wastewater and storm water system. You can enter model data suited for your choice of numerical engine MOUSE (developed by DHI) or SWMM5 (public domain). You can also enter 'Asset' data suited for a collection system. The SWMM 5 engine can be run from the Model Manager, while running MOUSE requires you to have license for the MIKE URBAN Collection System Pipeflow or Collection System Rainfall-Runoff module.

10.1 Getting Started on a Collection System Project

In order to start modeling your collection system, please do the following:

1. Start up MIKE URBAN (please refer to 2.1 for further)
2. On the dialog that comes up, see Figure 10.1, choose location and name of your database
3. Choose Working Mode to be Collection System
4. Then choose the menus to display initially. If you plan to enter data into the MOUSE model toggle 'MOUSE' on, if you wish to enter data for SWMM5 toggle 'SWMM' on. 'Asset' will activate the Asset menu for collection systems - relevant for collection systems in general. You can always change you initial choice with respect to which menus to activate/de-activate afterwards from within MIKE URBAN.
5. Choose a coordinate system among the list of available coordinate systems, see 2.9. This choice cannot be changed later on for the project in question.
6. Choose a unit system to work in among the list of available unit systems
7. Go to File|New and in the dialog that comes up. Please refer to Figure 10.1.
8. You also need to choose the unit system to work in. This overall unit choice cannot be changed later on for the project in question. You can, however, choose to use custom units for specific attributes later on. Please refer to 3.1 for this.

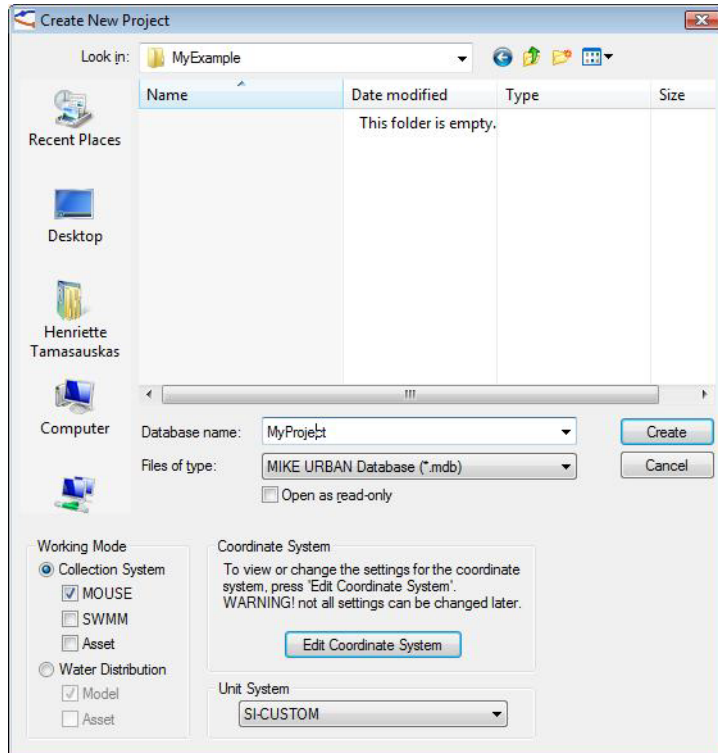


Figure 10.1 Creating a new collection system project

10.1.1 Entering data, Edit mode

You are now ready to start entering data into your model. This can be done by typing data in manually, importing data by connecting to an external data storage or by graphically entering data. Or a mixture of all three methods.

Before you can start to type in data or graphically add data you need to be in Edit mode (importing will automatically turn MIKE URBAN into edit mode during import). You enter Edit mode by:

- Go to 'Edit|Start Editing' - see Figure 10.2

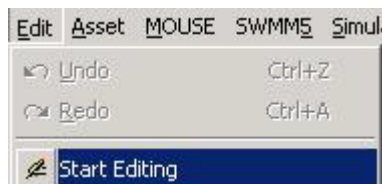


Figure 10.2 Start editing



Within an Edit mode session you can enter data and get access to an unlimited number of 'undo' and 'redo'. This is done by accessing 'Edit|Undo' to undo changes. The changes will be 'undone' in the order they were entered. 'Edit|Redo' will redo the changes in the order they were 'undone'. 'Edit|Stop Editing' will prompt to save changes done within the edit session or not.

10.1.2 Activating menus to display

Within the collection system work mode you can choose to have the following menus activated:

- Asset
- MOUSE
- SWMM5

Activating/de-activating the menus is done from the File menu as seen in Figure 10.3.

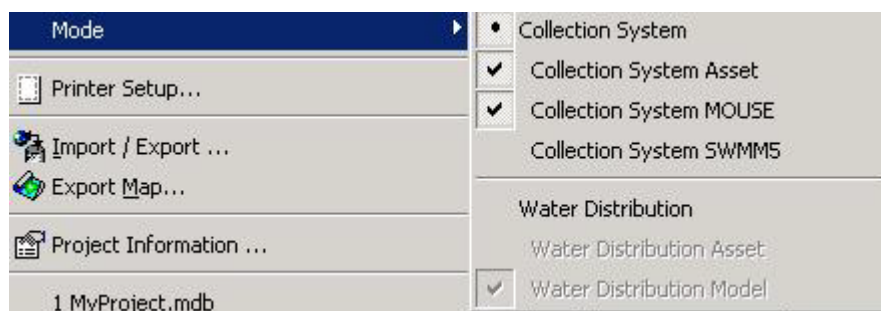


Figure 10.3 Work mode and activating menus

10.1.3 Project information

The project information will be provided for the menus that you currently have active. E.g. if you have Asset, MOUSE, SWMM 5 active the project information dialog, see Figure 10.4, will be display three tabs with information on each of these, when having e.g. only the 'MOUSE' menu active the project info will only display the MOUSE tab.

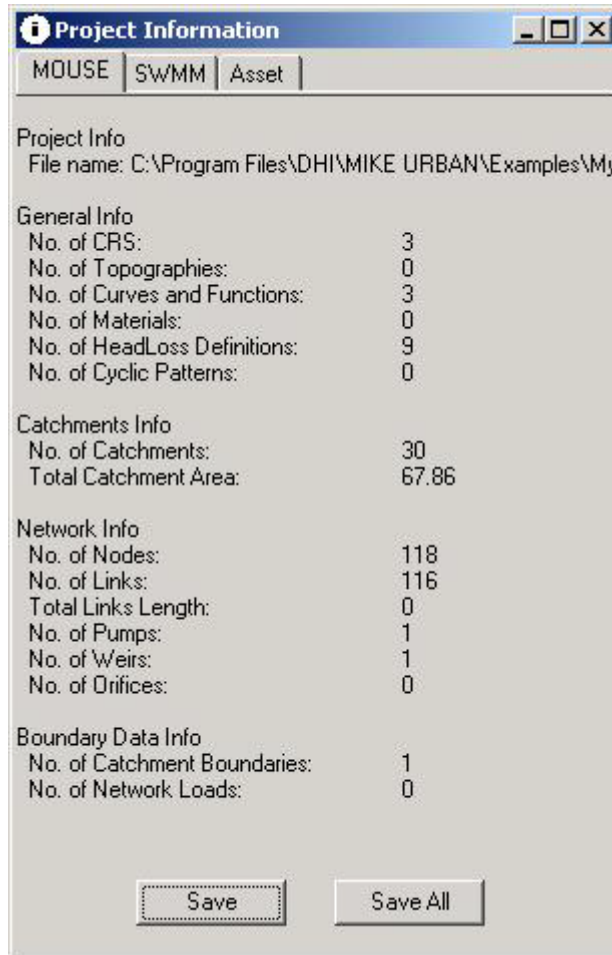


Figure 10.4 Example of the project info dialog

Using 'Save' on the dialog will save the information on the active tab page to a text file with a user specified name. Using 'Save All' will save the information found on all the visible tabs to a text file with a user specified name.

10.2 Collection System (CS) Asset Network

The editors for CS asset network component editors are all available from the Asset CS Menu, as shown in Figure 10.5.

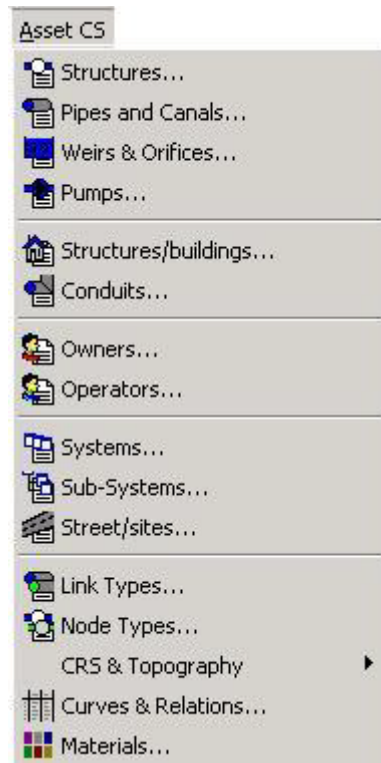


Figure 10.5 The Asset Menu allow access to the asset component editors

Note that a number of Asset network editors (Structures/Buildings, Conduits, Owners, Operators, Systems, Sub-Systems, Streets/Sites) are common for Collection and Distribution systems. Non of these is organically related to the network structure. These are described in a separate Chapter.

10.2.1 CS Asset Structures Editor

The CS Asset structures represent the collection system network nodes - manholes, inspection shafts, junctions, basins, gate chambers, etc. I.e. both the actual asset objects and the points of relevance for the network topology (junctions, dead-ends, natural ponds, etc.) without meaning in asset terms.

CS ASSET Structures in MIKE URBAN are represented as point features.

The CS Asset Structures Editor allows for the insertion of new "Structure" records, the specification of the structure type, geographical location, the basic physical properties, as well as essential information concerning the structure's administrative status in terms of ownership and position in the asset management system. The CS Structures Editor is reached by selecting **Asset CS | Structures**.

Node ID*	Asset ID	Data source	Type*	Bottom level	Ground level	Critic

Figure 10.6 The CS Asset Structures editor

CS ASSET Structures can also be inserted interactively on the Map window using the Add tool of the Feature Edit toolbar. Other graphical tools, like "Move Selected" can also be used.

10.2.2 Asset Pipes and Canals Editor

CS ASSET Pipes and Canals include all types of collection system network links (conduits) - sewer pipes, pressure mains, canals, natural streams, etc. - i.e. all linear water bodies of relevance for storm- or wastewater collection and conveyance.

CS ASSET Pipes and Canals in MIKE URBAN are represented as polyline features. The network structure is ensured through the relation to the network nodes, i.e. the definition of upstream ("From") and downstream ("To") nodes for each link.

The CS Asset Pipes and Canals Editor allows for the specification of the link type, the basic physical properties, as well as essential information concerning the structure's administrative status in terms of ownership and position in the asset management system. The CS Pipes and Canals Editor is reached by selecting **Asset CS | Pipes and Canals**.



Link ID*	Asset ID	Data source	Type*	UpLevel	DwLevel	UpLevel_C	DwLevel_C
▶ Link_3	<Null>	<Null>	<Null>	<Null>	<Null>	<Null>	<Null>

Figure 10.7 The CS Asset Pipes and Canals editor

10.2.3 CS Asset Weirs and Orifices Editor

CS ASSET Weirs & Orifices contains the information about individual functional elements in the collection system networks, like overflow weirs, orifices, gates, and similar objects. CS ASSET Weirs & orifices in MIKE URBAN are represented as point features, but unlike CS ASSET Structures, the CS Weirs & Orifices are not constitutive elements of the ASSET network. Each element of this type is associated with one CS ASSET node (Structure), which automatically determines its geographical position.

The CS Asset Weirs & Orifices Editor allows for the specification of the device type, its location in the network, the basic physical and functional properties, as well as essential information concerning the device's administrative status in terms of ownership and position in the asset management system. The CS Weirs and Orifices Editor is reached by selecting **Asset CS | Weirs and Orifices**.

Weir/orifice	Asset ID	Data source	Type	Flap	Orientation	Level

Figure 10.8 The CS Asset Weirs and Orifices editor

CS ASSET Weirs & orifices can also be inserted interactively on the Map window using the Add tool of the Feature Edit toolbar. Other graphical tools, like "Move Selected" can also be used.

10.2.4 CS Asset Pumps Editor

CS ASSET Pumps contains the information about individual pump units in the collection system. CS ASSET Pumps in MIKE URBAN are represented as point features, but unlike CS ASSET Structures, the Pumps are not constitutive elements of the ASSET network. Each pump is associated with one CS ASSET node (Structure), which automatically determines its geographical position.

The CS Asset Pumps Editor allows for the specification of the pumps type, its location in the network, the basic physical and functional properties, as well as essential information concerning the pump's administrative status in terms of ownership and position in the asset management system.

Pumps (or pumping stations) are represented by "points" in the MIKE URBAN network model. Pumps are either defined interactively on the Map window using the Add tool, or by manual data entry using the Pumps Editor as shown in Figure 13.7. The Pumping Stations Editor allows you to define the pump's ID, description, and other attributes.



The CS Asset Pumps Editor is reached by selecting **Asset CS | Pumps**.

Link ID*	Asset ID	Data source	Manufactur	Product cod	Pump type	Nominal c

Figure 10.9 The CS Asset Pumps editor

CS ASSET Pumps can also be inserted interactively on the Map window using the Add tool of the Feature Edit toolbar. Other graphical tools, like "Move Selected" can also be used.

10.2.5 CS Asset Link types Editor

The CS Link types Editor allows the definition of asset link types according to the actual classification used in the current system, e.g. drainage canal, tunnel, main interceptor, etc. The specified types include a reference to an appropriate alias (pressure pipe, free-flowing pipe or open channel), which supports a more intuitive function of the CS ASSET Pipes and Canals editor and a smoother data transfer to the Collection System models MOUSE and SWMM5. The CS Asset Link Types Editor is reached by selecting Asset CS | Link types.

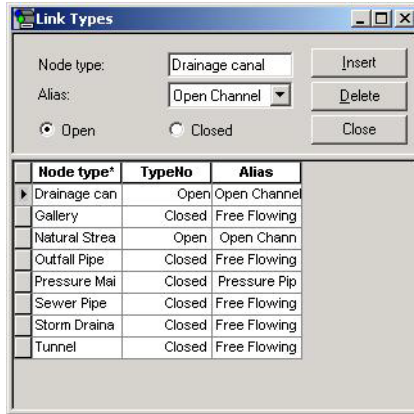


Figure 10.10 The CS Asset Link type editor

10.2.6 CS Asset Node types Editor

The CS Asset Node types Editor allows the definition of asset node types according to the actual classification used in the current system, e.g. man-hole, gate chamber, junction, detention pond, etc. The specified types include a reference to an appropriate alias (junction, basin, manhole, outlet), which supports a more intuitive function of the CS ASSET Structures dialog and a smoother data transfer to the Collection System models MOUSE and SWMM5. The CS Asset Node Types Editor is reached by selecting Asset CS | Node types.

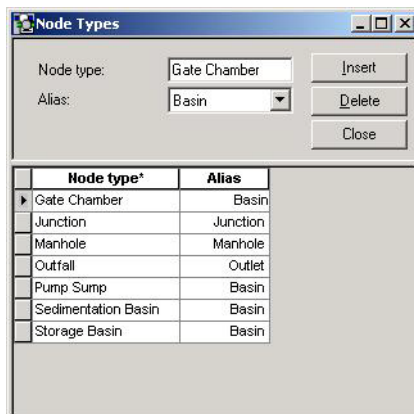


Figure 10.11 The CS Asset Node types editor



10.2.7 CS Asset CRS & Topography

The CRS and Topography editors allow the definition of the conduit cross-sections and natural streams topographies, according to you to define the following:

- CRS editor: Open and closed Cross sections. The X, Z types are appropriate for irregular cross sections, while H, W are best for symmetric cross sections.
- Topography: The "Channel Topography" dialog is an editor for MOUSE topography data. A topography is made up from a number of cross sections, specified in the cross section editor.

The CRS and Topography Editors are reached by selecting **Asset CS | CRS & Topography**.

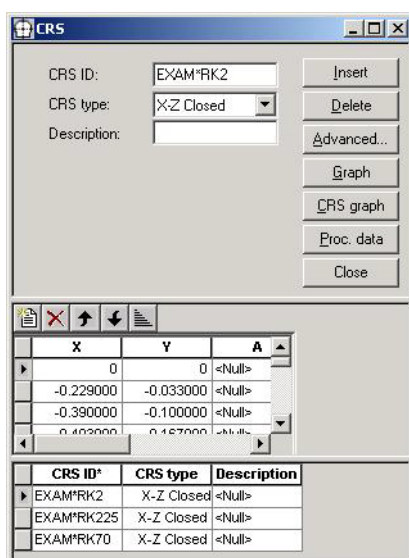


Figure 10.12 The CRS editor

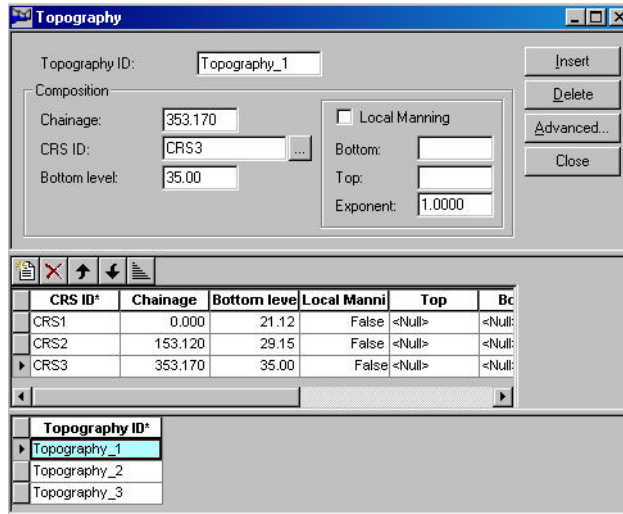


Figure 10.13 The Topography Editor

10.2.8 CS Asset Curves & Relations

For the specification of basin geometry in the ASSET network, the same editor is used as for the MOUSE model network. This implies that a number of curve types are presented, which have no relevance for CS ASSET network. User has simply to ignore the curve types obsolete for the CS ASSET network.

The basin geometry is specified with at least two sets of three values:

- H = elevation
- Ac = Wetted Cross-Section area, perpendicular to the main flow direction
- As = Surface area

The table is correctly specified if the values of H and Ac are monotonously increasing

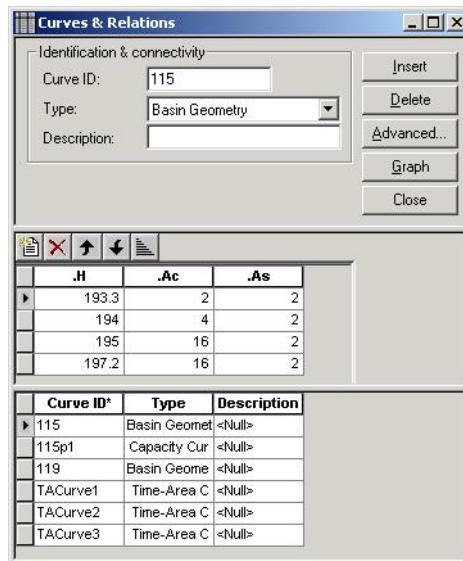


Figure 10.14 The Curves & Relations editor

10.2.9 CS Asset Materials

Each CS ASSET Link can be attributed with a material. MIKE URBAN applies the same list of materials for the CS ASSET and MOUSE networks. MIKE URBAN comes with a default list of materials, which can be modified/expanded according to the actual needs.

Each material is characterised by Material ID, and the surface properties: the value of Manning number and equivalent roughness. Usually, the surface properties are of no relevance for asset network, but may be relevant if e.g. an appropriate information from the pipe vendor catalogue is available.

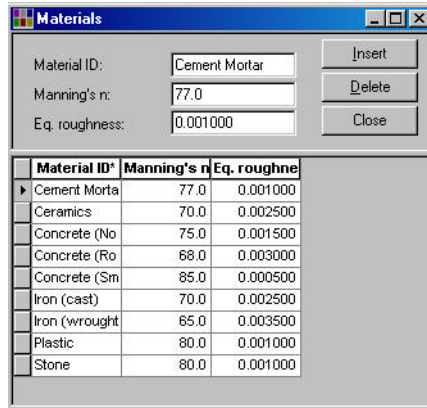


Figure 10.15 The Materials Editor

10.3 Transferring data between 'CS Asset', 'MOUSE' and 'SWMM5'

It is possible to transfer data between data entered Asset, MOUSE and SWMM5. This is done from 'Tools|Send Data', see Figure 10.16

This means that if you e.g. start your model by importing data into the Asset layer from e.g. your GIS system, you can then easily start the modeling in e.g. MOUSE or SWMM 5 by transferring those data that are relevant for your modeling. Equally you can e.g. receive a SWMM 5 model and then transfer the data that may be transferred to MOUSE in order to continue the modeling in MOUSE instead.

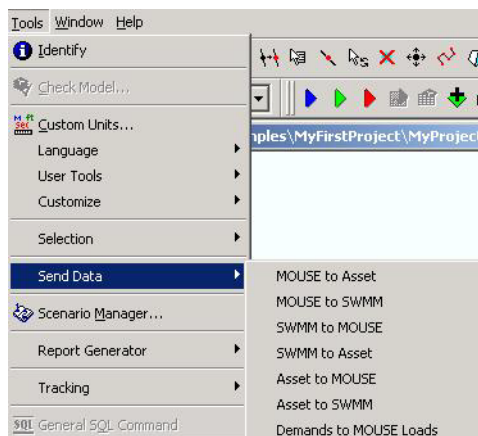


Figure 10.16 Transferring data between the Asset, MOUSE and SWMM 5

You can transfer data in three different modes, see e.g. Figure 10.17. The unique ID for elements will be used for identifying if a record is new or exist-



ing. If the 'Selected Only' is toggled on in the dialog only selected records will be transferred. The three transfer modes:

- Import and replace existing data: This will add new records in the data structure where data is being transferred to as well as replace existing records.
- Append into existing data: This will add new records into the data structure where data is being transferred to. No records will be updated or replaced.
- Import and update existing data: This will add new records in the data structure where data is being transferred to as well as update existing records.

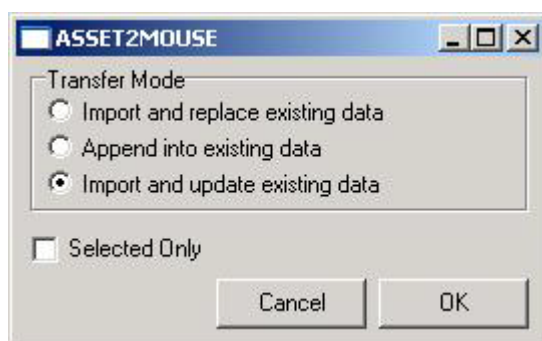


Figure 10.17 The three different transfer modes

10.4 SWMM 5

It is possible to execute the SWMM 5 engine from the MIKE URBAN Model Manager. Input data for the SWMM 5 model is entered from the SWMM 5 menu, see Figure 10.18.

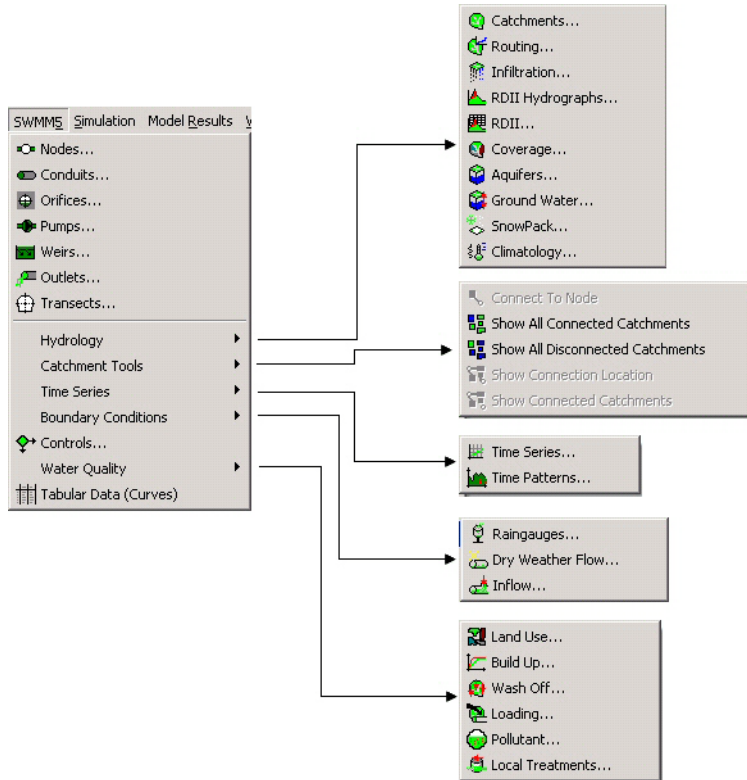


Figure 10.18 The SWMM 5 menu

Once input data is entered SWMM 5 may be run from 'Simulation|Run SWMM5', see Figure 10.19.

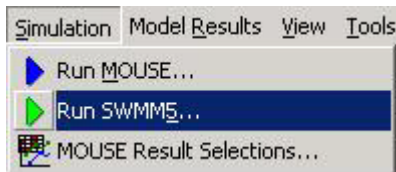


Figure 10.19 Running the SWMM5 engine

The dialog that comes up, Figure 10.20, will allow you to set the computational settings as well as performing a simulation.



The screenshot shows the 'Project' dialog box with the following settings:

- General** | Runoff | Network | Summary
- Identification & connectivity:**
 - Simulation ID: 3
 - Scenario ID: Base
 - Description: Example 1
- Simulation period:**
 - Start analysis on: 01-01-1998 00:00:00
 - End analysis on: 02-01-1998 12:00:00
 - Start sweeping: 01-01-1899 00:00:00
 - End sweeping: 31-12-1899 00:00:00
 - Start reporting: 01-01-1998 00:00:00
 - Reporting time step: 01:00:00
- Start simulation:** Single run
- Units selection:**
 - CFS (cubic feet per second)
 - CMS (cubic meters per second)
 - GPM (gallons per minute)
 - LPS (liters per second)
 - MGD (millions gallons per day)
 - MLD (million liters per day)

Figure 10.20 Setting the computational parameters for SWMM5

For more detailed information on entering input for SWMM 5, please refer to the documentation on modeling Collection Systems with SWMM 5.





11 Modeling Water Distribution Systems

With MIKE URBAN Water Distribution system you can model your water supply and water distribution system. 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/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' 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.

11.1 Getting Started on a Water Distribution System

In order to start modeling your collection system, please do the following:

1. Start up MIKE URBAN.
2. On the dialog that comes up, see Figure 11.1, choose location and name of your database

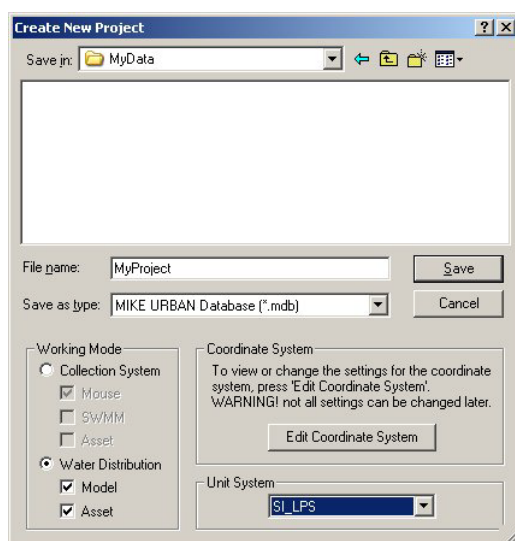


Figure 11.1 Creating a new water distribution system project

3. Choose Working Mode to be Water Distribution System
4. Then choose the menus to display initially. If you plan to enter data for the water distribution model toggle 'Model'. 'Asset' will activate the Asset menu for water distribution systems - relevant for water distribution sys-

tems in general. You can always change you initial choice with respect to which menus to activate/de-activate afterwards from within MIKE URBAN.

5. Choose a coordinate system among the list of available coordinate systems, see 2.9. This choice cannot be changed later on for the project in question.
6. Choose a unit system to work in among the list of available unit systems. This overall unit choice cannot be changed later on for the project in question. You can, however, choose to use custom units for specific attributes later on.

11.1.1 Entering data, Edit mode

You are now ready to start entering data into your model. This can done by typing data in manually, importing data by connecting to an external data storage or by graphically entering data. Or a mixture of all three methods.

Before you can start to type in data or graphically add data you need to be in Edit mode (importing will automatically turn MIKE URBAN into edit mode during import). You enter Edit mode by:

- Select Edit | Start Editing - see Figure 11.2

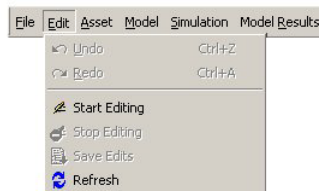


Figure 11.2 Start editing

Within an Edit mode session you can enter data and get access to an unlimited number of 'undo' and 'redo'. This is done by accessing Edit | Undo to undo changes. The changes will be 'undone' in the same order they were entered. 'Edit | Redo' will redo the changes in the order they were 'undone'. "Edit | Stop Editing" will prompt to save changes done within the edit session.

11.1.2 Activating menus to display

Within the water distribution system work mode you can choose to have the following menus activated:

- Asset
- Model (EPANET engine, WD-Tools, WD-WH water hammer module)



Activating/de-activating the menus is done from the File menu as seen in Figure 11.3.

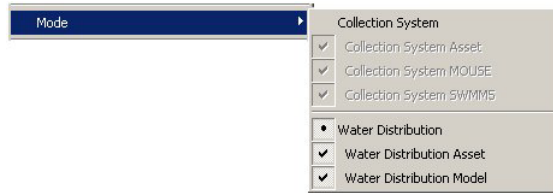


Figure 11.3 Work mode and activating menus

11.1.3 Project information

The project information provides an overview of the active setup, see Figure 11.4. The information is organized on several tabbed pages.

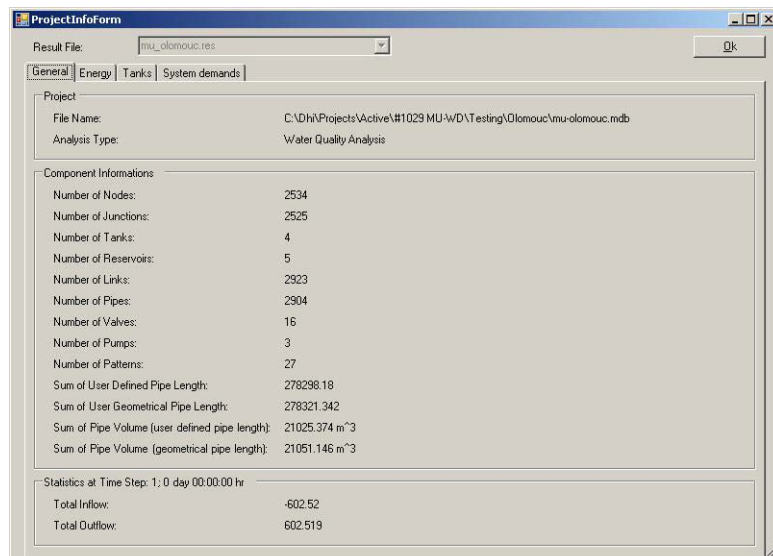


Figure 11.4 Project information dialog

11.2 Water Distribution (WD) Asset

The following sections describe the various asset (GIS) component editors (i.e., pipes, service pipes, pumps, reservoirs, tanks, and others). These asset component editors allow you to define each component individually using a spearheads like data entry. 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 asset component editors are all available from the **Asset WD** Menu, as shown in Figure 11.5.



Figure 11.5 The Asset Menu allow access to the asset component editors

11.2.1 WD Asset 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 tool, or by manual data entry using the Pipe Editor as shown in Figure 11.6. The Pipe Editor allows you to define the pipe's ID, diameter, description, and other attributes. The Pipe Editor is reached by selecting **WD Asset | Pipes**.



Link ID	Asset ID	Data source	Description	Length	Diameter	Population	Material

Figure 11.6 The Pipe Editor allows you to define the pipes that make up the water distribution network

A list of the Pipe Editor data entries for Figure 11.6 follows, with a short description given for each entry.

Asset ID

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

This data entry is used to specify an ID which uniquely identifies the pipe link. A new pipe ID is automatically suggested by MIKE URBAN whenever a new asset 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.

Description

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

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.

Length

This data entry defines the pipe length, in units of ft. or m, for example.

Diameter

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



Material

This option allows the user to define the material of pipe construction.

Demand coefficient 1,2

These two data entries define the demand coefficients, k_{1i} and k_{2i} which can be used in the demand management tools.

Street name

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

Population

This data entry defines the population size served by the pipe.

Details

Note that you can enter more data by selecting More | Details from within the Asset Pipe Editor.

Location	Locality	Joint type	Surface typ	Assesmer
----------	----------	------------	-------------	----------

Figure 11.7 The Details editor allows you to define more parameters defining the asset pipe

Location

This data entry defines the location of the asset pipe. Note that you can select the location name from the list.

Locality

This data entry defines the locality name of the pipe. Note that you can select the locality name from the list.

Joint type

This data entry defines the joint type such as CM, FL, MECH, and others. Note that you can select the join type from the list.



Surface type

This data entry defines the surface type. Note that you can select the surface type from the list.

Assessment depth

This data entry defines the assessment depth.

Assessment type

This data entry defines the assessment type, such as “in ground“, “protection pipe“, “collector“, and others. Note that you can select the assessment type from the list.

Wall thickness

This data entry defines the pipe wall thickness of the pipe.

Pressure normal

This data entry defines asset pipe normal operating pressure.

Pipe type

This data entry defines the asset pipe type. Note that you can select the pipe type from the list.

Pipe state

This data entry defines the pipe state, such as “good“, “bad“, “break-down“, “critical“, and others. Note that you can select the pipe state from the list.

Details

Note that you can enter more data by selecting More | Details from within the Asset Pipe Editor.



Municipality	Municipality	Street nam	Cadastre	Owner

Figure 11.8 The Asset editor allows you to define more asset parameters defining the asset pipe

Municipality

This data entry defines the municipality name served by the pipe.

Municipality part

This data entry defines the municipality part name served by the pipe.

Street name

This data entry defines the street name of the pipe. Note that you can select the street name from the list.

Cadastre

This data entry defines the cadastre name of the pipe.

Owner

This data entry defines the owner name of the pipe. Note that you can select the owner name from the list.

Operator

This data entry defines the operator name of the pipe. Note that you can select the operator name from the list.

Property

This data entry defines the property name of the pipe.

System

This data entry defines the system name of the pipe. Note that you can select the system name from the list.



Sub-system

This data entry defines the sub-system name of the pipe. Note that you can select the sub-system name from the list.

Construction name

This data entry defines the construction name of the pipe.

Water office permit

This data entry defines the water office permit name of the pipe.

Inspection data

This data entry defines the inspection date of the pipe.

Acceptance date

This data entry defines the acceptance date of the pipe.

Documentation

Note that you can enter more data by selecting More | Documentation from within the Asset Pipe Editor.

Figure 11.9 The Documentation editor allows you to define more documentation parameters defining the asset pipe

Documentation type

This data entry defines the documentation type, such as “photo of pipe”, “photo of pipe failure”, “documentation”, and other. Note that you can select the documentation type from the list.

Author

This data entry defines the documentation author name.

Date

This data entry defines the documentation date.

Print to passport

This data entry defines the documentation print to passport option.



File name

This data entry defines the documentation file name.

Description

This data entry defines the optional documentation description.

Costs

The supply and construction cost of the corresponding asset component can be defined using the supply and construction cost per item and/or per unit length. The total cost of the selected part of the asset system or the entire asset system can be calculated using the supply and construction costs.

Supply cost per unit length

This data entry defines the supply cost of the corresponding element per the unit length.

Construction cost per unit length

This data entry defines the construction cost of the corresponding element per the unit length.

Supply cost per item

This data entry defines the supply cost of the corresponding element per item.

Construction cost per item

This data entry defines the construction cost of the corresponding element per item.

11.2.2 WD Asset Service Pipe Editor

Service pipes represent the house connections and they are not typically included in the hydraulic model. Service pipes do not require the main pipe to be split i.e. they are represented by “*complex edges*” in the MIKE URBAN network model

Service pipes are either defined interactively on the Map window using the Add tool, or by manual data entry using the Service Pipe Editor as shown in Figure 11.10. The Service Pipe Editor allows you to define the pipe’s ID, diameter, description, and other attributes. The Pipe Editor is reached by selecting **Asset WD | Service Pipes**.



Pipe ID	Asset ID	Data source	Description	Length	Diameter	Population	Material

Figure 11.10 The Service Pipe Editor allows you to define the pipes that make up the house connections

A list of the Service Pipe Editor data entries for Figure 11.10 follows, with a short description given for each entry.

Asset ID

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

This data entry is used to specify an ID which uniquely identifies the pipe link. A new pipe ID is automatically suggested by MIKE URBAN whenever a new asset 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.

Description

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

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.

Length

This data entry defines the pipe length, in units of ft. or m, for example.



Diameter

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

Material

This option allows the user to define the material of pipe construction.

Demand coefficient 1,2

These two data entries define the demand coefficients, k_{1i} and k_{2i} which can be used in the demand management tools.

Street name

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

Population

This data entry defines the population size served by the pipe.

Details

Note that you can enter more data by selecting More | Details from within the Asset Service Pipe Editor.

Documentation

Note that you can enter more data by selecting More | Documentation from within the Asset Service Pipe Editor.

Costs

Note that you can enter the supply and construction cost of the corresponding asset element.

11.2.3 WD Asset Hydrants Editor

Hydrants are represented by “*points*” in the MIKE URBAN network model. Hydrants are either defined interactively on the Map window using the Add tool, or by manual data entry using the Hydrants Editor as shown in Figure 11.11. The Hydrants Editor allows you to define the hydrant’s ID, diameter, description, and other attributes. The Pipe Editor is reached by selecting **Asset WD | Hydrants**.



Hydrant ID*	Asset ID	Pipe ID*	Data source	Description	Hydrant ele	Minimum pr	Required pr

Figure 11.11 The Hydrants Editor allows you to define the hydrants that make up the water distribution system

A list of the Hydrants Editor data entries for Figure 11.11 follows, with a short description given for each entry.

Asset ID

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

Hydrant ID

This data entry is used to specify an ID which uniquely identifies the hydrant. A new hydrant ID is automatically suggested by MIKE URBAN whenever a new hydrant pipe is placed into the list by pressing «Insert». When defining the hydrants graphically on the Map window using the Add tool, the hydrant ID is automatically defined.

Pipe ID

This data entry is used to specify a pipe ID to which the hydrant is connected.

Description

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

Data source

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



Hydrant elevation

This data entry defines the hydrant elevation, in units of ft. or m, for example.

Minimum pressure

This data entry defines the minimum acceptable pressure

Required pressure

This data entry defines the required operation pressure, in units of psi or m, for example.

Required flow

This data entry defines the required fire flow, in units of cfs, gpm, or l/s, for example.

Details

Note that you can enter more data by selecting More | Details from within the Asset Hydrants Editor.

Documentation

Note that you can enter more data by selecting More | Documentation from within the Asset Hydrants Editor.

Costs

Note that you can enter the supply and construction cost of the corresponding asset element.

11.2.4 WD Asset Valves Editor

Valves are represented by “*points*” in the MIKE URBAN network model. Valves are either defined interactively on the Map window using the Add tool, or by manual data entry using the Valves Editor as shown in Figure 11.12. The Valve Editor allows you to define the valve’s ID, diameter, description, and other attributes. The Valve Editor is reached by selecting **Asset WD | Valves**.



Valve ID	Asset ID	Data source	Description	Valve elevat

Figure 11.12 The Valves Editor allows you to define the valves that make up the water distribution system

A list of the Valves Editor data entries for Figure 11.12 follows, with a short description given for each entry.

Asset ID

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

This data entry is used to specify an ID which uniquely identifies the valve. A new valve ID is automatically suggested by MIKE URBAN whenever a new valve pipe is placed into the list by pressing «Insert». When defining the valve graphically on the Map window using the Add tool, the valve ID is automatically defined.

Pipe ID

This data entry is used to specify an pipe ID on which the valve is installed.

Description

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

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.

Valve elevation

This data entry defines the valve elevation



Details

Note that you can enter more data by selecting More | Details from within the Asset Valves Editor.

Asset

Note that you can enter more data by selecting More | Details from within the Asset Valves Editor.

Documentation

Note that you can enter more data by selecting More | Documentation from within the Asset Valves Editor.

Costs

Note that you can enter the supply and construction cost of the corresponding asset element.

11.2.5 WD Asset Pumping Stations Editor

Pumping stations (or pumps) are represented by “*points*” in the MIKE URBAN network model. Pumps are either defined interactively on the Map window using the Add tool, or by manual data entry using the Pumps Editor as shown in Figure 11.13. The Pumping Stations Editor allows you to define the pump’s ID, description, and other attributes. The Pumping Stations Editor is reached by selecting ‘Asset WD | Pumping Stations’.

Pump ID*	Asset ID	Node ID*	Data source	Description

Figure 11.13 The Pumping Stations Editor allows you to define the pumps that make up the water distribution system

A list of the Pumping Stations Editor data entries for Figure 11.13 follows, with a short description given for each entry.

Asset ID

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



Pump ID

This data entry is used to specify an ID which uniquely identifies the pump. A new pump ID is automatically suggested by MIKE URBAN whenever a new valve pump is placed into the list by pressing «Insert». When defining the pump graphically on the Map window using the Add tool, the pump ID is automatically defined.

Node ID

This data entry is used to specify an model node ID to which the pump is connected.

Description

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

Data source

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

Details

Note that you can enter more data by selecting More | Details from within the Asset Pumping Stations Editor.

Asset

Note that you can enter more data by selecting More | Details from within the Asset Pumping Stations Editor.

Documentation

Note that you can enter more data by selecting More | Documentation from within the Asset Pumping Stations Editor.

Costs

Note that you can enter the supply and construction cost of the corresponding asset element.

11.2.6 WD Asset Water Tanks Editor

Water tanks are represented by “*points*” in the MIKE URBAN network model. Water tanks are either defined interactively on the Map window using the Add tool, or by manual data entry using the Pumps Editor as shown in Figure 11.14. The Water Tanks Editor allows you to define the tank’s ID, description, and other attributes. The Water Tanks Editor is reached by selecting **Asset WD | Water Tanks**.



Hydrant ID*	Asset ID	Node ID*	Data source	Description
-------------	----------	----------	-------------	-------------

Figure 11.14 The Water Tanks Editor allows you to define the tanks that make up the water distribution system

A list of the Water Tanks Editor data entries for Figure 11.14 follows, with a short description given for each entry.

Asset ID

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

Tank ID

This data entry is used to specify an ID which uniquely identifies the water tank. A new water tank ID is automatically suggested by MIKE URBAN whenever a new valve water tank is placed into the list by pressing «Insert». When defining the water tank graphically on the Map window using the Add tool, the water tank ID is automatically defined.

Node ID

This data entry is used to specify an model node ID to which the water tank is connected.

Description

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

Data source

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

Details

Note that you can enter more data by selecting More | Details from within the Asset Water Tanks Editor.



Asset

Note that you can enter more data by selecting More | Details from within the Asset Water Tanks Editor.

Documentation

Note that you can enter more data by selecting More | Documentation from within the Asset Water Tanks Editor.

Costs

Note that you can enter the supply and construction cost of the corresponding asset element.

11.3 Transferring data between 'Asset' and 'Model'

It is possible to transfer data between Asset and Model data modes. This is done using 'Tools | Send Data', see Figure 11.15.

This means that if you e.g. start your model by importing data into the Asset layer from your GIS system, you can then easily start the modeling by transferring the corresponding data to the model database. This will allow you to edit and maintain both original (GIS) data and model data in the same MIKE URBAN project. Note that you can import your GIS data to the model data directly if you do not wish to maintain the original GIS data in MIKE URBAN.



Figure 11.15 Transferring data between the Asset and Model

You can transfer data in three different modes, see e.g. Figure 11.16. The unique ID for elements will be used for identifying if a record is new or existing. If the 'Selected Only' is toggled on in the dialog only selected records will be transferred. The three transfer modes:

- Import and replace existing data: This will add new records in the data structure where data is being transferred to as well as replace existing records.
- Append into existing data: This will add new records into the data structure where data is being transferred to. No records will be updated or replaced.
- Import and update existing data: This will add new records in the data structure where data is being transferred to as well as update existing records.

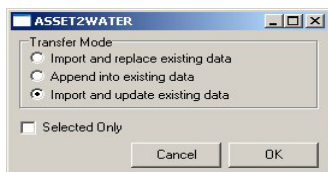


Figure 11.16 The three different transfer modes

The transfer of Asset data to the Model data and vice versa can be used for creating model pipes from the asset (GIS) pipes, for example.



12 Common Editors for CS and WD Asset Data

Some editors are the same on the CS Asset and WD Asset menu. Data is stored in different tables belonging to CS and WD respectively but the input fields are the same. These editors that are similar are described in this section.

12.1 Asset Structures and Buildings Editor

ASSET Buildings, represented as polygon features, serve mainly the purpose of proper visualisation of various objects in the analysed systems. While, in terms of network topology, structures take the form of non-dimensional nodes-points, Asset Buildings are supposed to display the actual geographical extent of various structures in polygon shapes. A building polygon may be related to a single node (e.g. detention pond), while in some other case it may encapsulate smaller or larger parts of the network. E.g. an asset building identified as a pumping station would embrace all the nodes and links within the perimeter of the actual pumping station building.

The buildings are categorised in types according to the configurable domain.

ASSET Buildings are independent features, i.e. they do not have any "organic" relation to the collection network. The relation exists on the attribute level, so that each asset network element can (optionally) be associated with one building.

The ASSET Buildings are inserted by means of "Add" feature edit tool. Other graphical tools for polygon editing are also available.

The Asset Structures Editor allows for the specification of the building type and some essential information concerning the building's administrative status in terms of ownership and position in the asset management system. The ASSET Buildings Editor is reached by selecting **Asset CS | Structures/Buildings** (when in CS mode) or from **Asset WD | Structures/Buildings** (when in WD mode).



Structure ID	Structure ca	General info	System*	Sub-s
▶ Detention Pon	Storage Basin	Storm water	Mouseville Se	Easter

Figure 12.1 The Structures and Buildings Editor - allows to develop a layer with various buildings in the analysed system.

A list of the Asset Streets Editor data entries for Figure 12.1 follows, with a short description given for each entry.

Structure ID

This data entry is used to specify a name of the corresponding structure.

Structure type

This data entry is used to specify a type of the corresponding structure.

System

This data entry is used to specify a name of the corresponding system. Note that you can select the system name from the list.

Sub-systems

This data entry is used to specify a name of the corresponding sub-system. Note that you can select the system name from the list.

Street name

This data entry is used to specify a street name of the corresponding structure. Note that you can select the system name from the list.



General information

This data entry is used to provide optional information.

Owner

This data entry is used to specify a name of the owner of the corresponding structure. Note that you can select the system name from the list.

Operator

This data entry is used to specify a name of the operator of the corresponding structure. Note that you can select the system name from the list.

12.2 Asset Conduits Editor

ASSET Conduits, represented as polyline features, serve mainly the purpose of a convenient visualisation of various conduits in the collection systems. While, in terms of network topology, the Links play an essential role, extending between two nodes ("From" and "To"), Asset Conduits are supposed to display the actual geographical extent of various conduits independently of the network nodes. A conduit polyline may extend along a single link (e.g. natural stream), while in some other case it would cover a long segment of certain pipeline, across many nodes. E.g. an asset conduit identified as an interceptor sewer would embrace all the nodes and links belonging to this particular pipeline.

The conduits are categorised in types according to the configurable domain.

ASSET Conduits are independent features, i.e. they do not have any "organic" relation to the collection network. The relation exists on the attribute level, so that each asset network element can (optionally) be associated with one conduit.

The ASSET Conduits are inserted by means of "Add" feature edit tool. Other graphical tools for polyline editing are also available.

The Asset Conduits Editor allows for the specification of the conduit type and some essential information concerning the conduit's administrative status in terms of ownership and position in the asset management system. The ASSET Conduits Editor is reached by



selecting **Asset CS |Conduits** (if in the CS mode) or by selecting **Asset WD |Conduits** (if in the WD mode).

Conduit ID*	Conduit cat	General info	System*	Sub-syst
Eastern Interc	Interceptor	Diameter 1.80	Mouseville Se	Eastern S

Figure 12.2 The Conduits Editor

A list of the Asset Streets Editor data entries for Figure 12.2 follows, with a short description given for each entry.

Conduit ID

This data entry is used to specify a name of the corresponding conduit.

Conduit type

This data entry is used to specify a type of the corresponding conduit.

System

This data entry is used to specify a name of the corresponding system. Note that you can select the system name from the list.

Sub-systems

This data entry is used to specify a name of the corresponding sub-system. Note that you can select the system name from the list.



Street name

This data entry is used to specify a street name of the corresponding conduit. Note that you can select the system name from the list.

General information

This data entry is used to provide optional information.

Owner

This data entry is used to specify a name of the owner of the corresponding conduit. Note that you can select the system name from the list.

Operator

This data entry is used to specify a name of the operator of the corresponding conduit. Note that you can select the system name from the list.

12.3 Asset Owners and Operators Editors

Each of the ASSET elements can optionally be attributed to an "Owner" and "Operator".

"Owner" and "Operator" are references to the agencies involved.

Within the present system, these attributes can be utilised e.g. for the preparation of thematic maps based on ownership and/or operational conditions for the system treated by the project.

Both Owners and operators can be typed as a free text, but can also be selected from previously prepared or imported lists.

The "Owners" and "Operators" list can be edited through the respective editors accessible under **Asset CS | Owners** and **Asset CS |**



Operators in CS mode and **Asset WD | Owners** and **Asset WD | Operators** in WD mode, respectively.

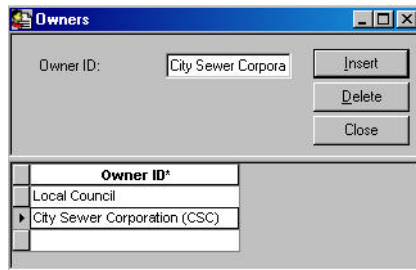


Figure 12.3 The Owners Editor allows you to develop the list of owners of asset components

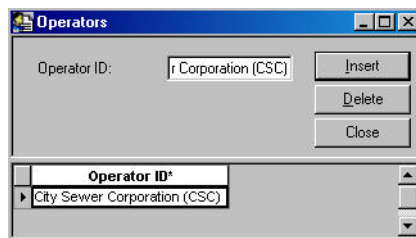


Figure 12.4 The Operators Editor allows you to develop the list of operators of asset components

Owner ID

This data entry is used to specify a name of the corresponding asset owner company/agency.

Operator ID

This data entry is used to specify a name of the corresponding asset operator company/agency.

12.4 Asset Systems, Sub-Systems and Streets/Sites

Each of the ASSET elements can optionally be attributed to a "System" and "Sub-System". Furthermore, a micro-location (address, street name, site...) can be specified.

"System" and "Sub-System" are references to the administrative or technical division of the analysed network into smaller sections. Within the present system, these attributes can be utilised e.g. for the preparation of thematic maps based on administrative or functional conditions for the system treated by the project.



"Street/Site" relates an asset element to a certain micro location.

All three attributes can be typed as a free text, but can also be selected from previously prepared or imported lists.

The "System", "Sub-System" and "Street/Site" lists can be edited through the respective editors accessible under **Asset CS | Systems**, **Asset CS | Sub-Systems** and **Asset CS | Street/Site** (for CS mode), and **Asset WD | Systems**, **Asset WD | Sub-Systems** and **Asset WD | Street/Site** (for CS mode), respectively.

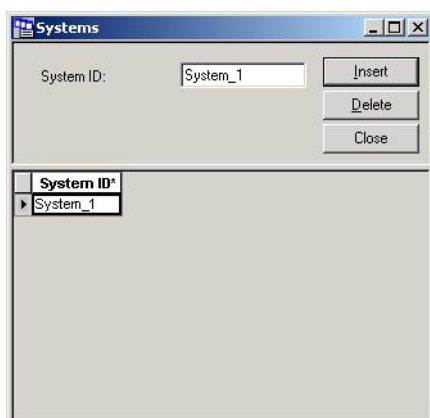


Figure 12.5 The Systems Editor allows the definition of the list of systems for the involved networks

A list of the Asset Systems Editor data entries for Figure 12.5 follows, with a short description given for each entry.

System ID

This data entry is used to specify a name of the corresponding system.

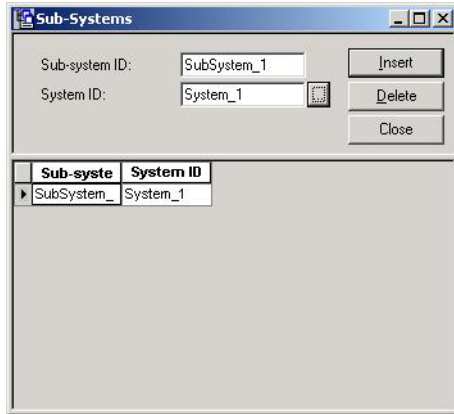


Figure 12.6 The Sub-Systems Editor allows the definition of the list of sub-systems for the involved networks

A list of the Asset Sub-Systems Editor data entries for Figure 12.6 follows, with a short description given for each entry.

Sub-system ID

This data entry is used to specify a name of the corresponding asset sub-systems.

System ID

This data entry is used to specify a name of the corresponding parent asset system.



Figure 12.7 The Streets Editor allows you to develop the list of streets or sites

A list of the Asset Streets Editor data entries for Figure 12.7 follows, with a short description given for each entry.



Street name

This data entry is used to specify a name of the corresponding street or site.





13 Catchments and Catchment Tools

13.1 MIKE URBAN Catchments

MIKE URBAN catchments are geographical polygon features which represent hydrological urban catchments or wastewater drainage areas. As such, MIKE URBAN catchments may be used for hydrological modeling or as wastewater sources for both models available in MIKE URBAN Collection System (MOUSE and SWMM5). Furthermore, MIKE URBAN catchments can serve for various analyses, independently of any computational model.

MIKE URBAN Catchments are featured as a single layer, independent of any network. It is only after the definition of the catchments' connections to a model network, that the catchments become a source of loads for a network model.

13.2 Management of MIKE URBAN Catchments

MIKE URBAN Catchments can be managed both graphically and through the "Catchments" editor. The two modes complement each other. Only through a joint application of both modes an efficient and complete management of catchment data can be achieved.

The graphical mode ("Feature Edit") allows for digitalization of catchment extent by tools like "Insert", "Edit", "Move", "Append", "Split" and "Merge". Indeed, the graphical mode comes short in specifying various catchment attributes.

The "Catchment" editor is used for

- Editing the catchment attributes. It is possible to insert catchments through the editor; these are given a schematized quadratic shape (see 13.3).
- Editing connections to model networks and hydrological data for Rainfall-Runoff models (see 13.4, 13.5 and 13.6).

Finally there is a collection of tools for automated delineation, connection and hydrological parameter estimation storm water catchments. These tools includes fast ways to generate reasonably good input to the models which can then subsequently be edited by the "normal" editing tools.

13.3 The Catchment Editor

The "Catchments" editor can be accessed through the menu (MOUSE|Catchments and catchment parameters or SWMM5|Hydrology|Catchments). Also, several toolbars include a tool for direct access to the "Catchments" editor.



MUID *	Drainage ar	User X coord	User Y coord	Total area	X coordinat	Y coordinat	Max. level	f
10_22_22	4,200	-687562,200	-1056577,00	1,477	-687531,194	-1056589,63	<Null>	<f
100_16_16	2,280	-687887,600	-1056368,90	0,515	-687904,138	-1056368,74	<Null>	<f
105_1_1	4,000	-687516,600	-1056363,80	0,887	-687502,017	-1056379,07	<Null>	<f
113_21_21	4,000	-687369,600	-1056466,40	1,841	-687372,867	-1056427,94	<Null>	<f
118_30_30	3,300	-688020,800	-1056332,10	4,267	-688009,184	-1056183,43	<Null>	<f
119_32_32	0,814	-687939,827	-1056574,99	1,942	-687939,827	-1056574,99	<Null>	<f
14_20_20	2,470	-687968,400	-1056513,00	0,310	-687948,719	-1056479,84	<Null>	<f
20_2_2	2,500	-687471,900	-1056525,60	1,688	-687428,886	-1056529,11	<Null>	<f
22_8_8	2,560	-687916,500	-1056494,60	0,970	-687829,632	-1056493,66	<Null>	<f
25_26_26	3,800	-687763,100	-1056440,10	1,091	-687746,224	-1056425,07	<Null>	<f
28_6_6	0,350	-687663,600	-1056404,80	0,277	-687666,628	-1056396,89	<Null>	<f

Figure 13.1 The 'Catchments' Editor

13.3.1 The Editor Fields

Table 13.1 Overview of the Editable Catchments Attributes (Table 'ms_Catchment')

Edit field	Description	Usage	Attribute Table Field
Catchment ID	Unique catchment identifier	Yes	MUID
Data Status	Used for the categorization of data status within a MIKE URBAN Project	Optional	Element_S



Table 13.1 Overview of the Editable Catchments Attributes (Table 'ms_Catchment')

Edit field	Description	Usage	Attribute Table Field
System	Relates the catchment to a pre-defined asset system, according to administrative or physical setting	Optional	SystemID
Sub-system	Relates the catchment to a pre-defined sub-system according to administrative or physical setting	Optional	SubSystemID
X co-ordinate	X co-ordinate of the catchment centerpoint	Optional/Calculated (read-only)	X(X_C)
Y co-ordinate	Y co-ordinate of the catchment centerpoint	Optional/Calculated (read-only)	Y(Y_C)
Total Area	The actual geographical extent of the catchment polygon	Calculated (read-only)	Area_C
Drainage Area	The area of relevance for the project (<= Total Area)	Optional	Area
Max. Level	The highest elevation within the catchment perimeter, based on DEM	Calculated (not implemented, read-only)	MaxLevel
Min. Level	The lowest elevation within the catchment perimeter, based on DEM	Calculated (not implemented, read-only)	MinLevel
Description	Free text description	Optional	Description
Person Equivalents (PEs) - checkbox	Allows for the specification of PE's attached to the catchment	Optional	PEsNo
Person Equivalents (PEs) - field	Number of PEs attached to the catchment	Optional, used as one method for WW load definition	Persons
Type	Assigns the catchment to one of predefined categories (Storm, Wastewater, Combined). Affects the possibilities of connecting to the network and load assignments	Optional	NetTypeNo

13.3.2 Calculated vs. User Specified Values

For a number of geographical properties (centerpoint co-ordinates, total area) the system provides the values automatically (i.e. upon request "Advanced|Recompute").

Optionally, some of these values (centerpoint co-ordinates, drainage area) may be supplemented by user-specified values. If present, the user-specified values will replace the computed values when used in modeling. The user-specified "Drainage Area" (<>nil) will be used in hydrological calculations, and NOT the "Total Area". The user-specified co-ordinates (<>nil) will be used in simulations of spatially distributed rain, and NOT the computed co-ordinates.

13.3.3 Tools for Graphical Catchment Editing

The various tools for graphical catchment editing can be accessed through the menu (EDIT|Feature Edit) or through the default toolbar "Feature Edit". Furthermore, some of the tools are available on the map's local menu (right click), as long as the application is in "catchment edit mode".

The "Feature Edit" toolbar is put in the "catchment" mode by selecting the catchment layer to be "Active Layer".



Figure 13.2 The 'Feature Edit' toolbar ready for editing catchments

All tools for graphical editing are fully supported by "Undo" function (Edit|Undo).

13.3.4 Create Catchment Feature

This tool allows for digitalization of perimeter of detached or overlapping catchment polygons. The precisely adjacent catchments can be digitalized by the "Append catchment" tool, or by splitting an existing catchment into two parts.

When this tool is active, a small polygon icon is displayed besides the cursor. The polygon is digitalized by clicking on the cursor along the wanted catchment perimeter. A double-click completes the current digitalization process and the system is immediately ready for the next catchment.

If <Shift> is pressed while clicking and the cursor is within snapping tolerance of an existing point in another catchment the inserted point will be created at exact same location as the existing point.



The tool is deactivated by selecting some other tool.

Each new catchment polygon is added as a new record in the catchments attribute table. Per default, a name (i.e. identifier) is given as "Catchment_n", where "n" stands for internal catchment index. The default identifier would normally be changed into some meaningful name.

13.3.5 Edit Catchment Feature

This tool allows for editing the already existing catchment polygons. When activated, the tool brings a catchment into the editing mode on a mouse click inside the catchment. The catchment turns pink and the polygon vertices are highlighted.

Individual vertices can then be "glued" to the cursor (the icon changes to a "+" icon) and dragged to a wanted position. The existing vertices can be deleted or new vertices can be inserted, as needed. The tools for vertex delete/insert are accessible on the local menu (right mouse click), while in "Edit" mode. The available tool toggles according to the current cursor position: if the cursor is placed onto or very close to the vertex, the "Delete Point" tool is active. Otherwise, the "Insert Point" is active.

The "Delete Point" remove a vertex and connects the two closest adjacent vertices along the catchment perimeter by a straight line. The "Insert Point" inserts a vertex at the closest point on the catchment perimeter. The new vertex can subsequently be dragged to the wanted position.

If <Shift> is pressed while moving a vertex and the vertex is identical to vertex in one or more other catchments all the relevant catchments will be edited at same time. When moving vertices belonging to the boundary of 2 or more catchments, the cursor changes into a polygon symbol with a red dot on top. When moving within snapping distance to another vertex the cursor will change symbol to allow snapping to the other vertex.

By clicking the mouse outside the catchment, the editing session is completed (the catchment turns back to original color), and the system is ready for editing another catchment.

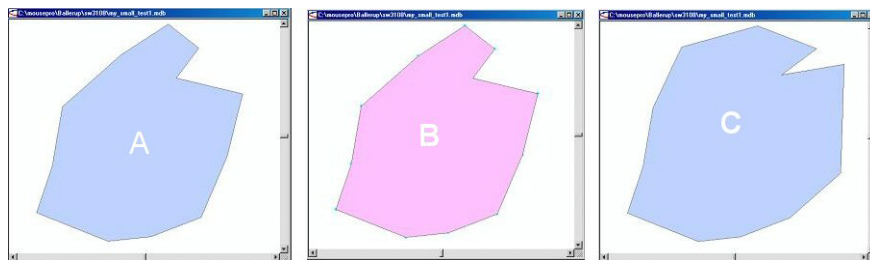


Figure 13.3 The original catchment polygon (A); Edited polygon (pink) with highlighted vertices; Catchment after completed editing (C).

13.3.6 Move Catchment

An individual catchment polygon or a group of catchments can be moved (translated) to a new position.

The "Move" tool operates on the selected set of catchments. The Move tool remains active as long as some other tool is activated.

13.3.7 Split Catchment

An existing catchment polygon can be split into two adjacent catchment polygons. The digitalization of the split line is started by a mouse click. The first click **MUST** be outside the polygon to be split. The line is drawn with subsequent clicks, as long as the catchment perimeter is reached. A double-click outside the polygon (typically on the opposite side) ends up the splitting process. Upon completed digitalization of the split line, the system prompts the user to confirm the split. Care is required here, because the system deletes the original catchment record and loss of valuable information is possible. Also, the network connection (if any) for the original catchment is deleted, i.e. the new catchments must be re-connected.

If the user confirms the split, the system deletes the original catchment and inserts two new catchments. The automatically provided identifiers of the new catchments would normally be changed into some meaningful catchment names.

The imperviousness for the new catchments is copied from the original catchment, while e.g. area, drainage area persons equivalents from the original catchment are split into two based on the areas of the splitted catchments.

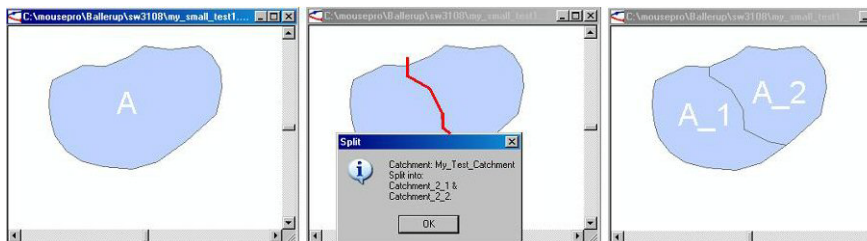


Figure 13.4 The splitting process starts with an existing catchment (A); The split line is drawn along the wanted path. The split action is initiated by a double click. User must confirm the split action; The original catchment is split into two adjacent catchments.

13.3.8 Append Catchment

A new catchment can be appended to already existing catchment(s). The result of this action is identical as with "Create catchment" tool, except that part of the catchment perimeter coincides with the perimeter of the adjacent catchment(s).

The digitalization of the new catchment is started by a mouse click. The first click **MUST** be inside the existing polygon. The catchment perimeter is drawn with subsequent clicks. A double-click inside any of the existing polygons ends up the append process.

Note that the new catchment will be created based on the digitized perimeter and between the first and the last mouse click. Hence if the edge (where the catchment is appended) is jagged, then care should be taken to digitize the new catchment so it extends the whole area to avoid any gap between the new catchment and the existing ones.

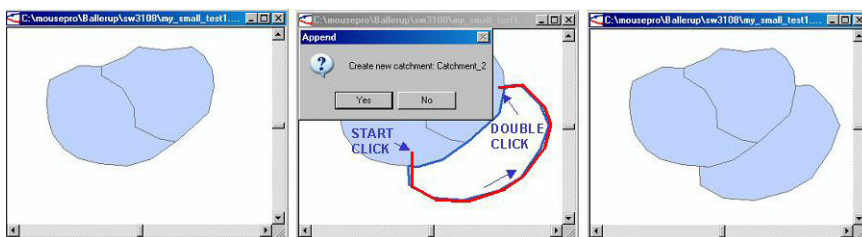


Figure 13.5 Existing catchment polygons (left); Digitalization starts and ends up **INSIDE** an existing polygon (middle); The new catchment has been appended (right).

The automatically provided identifier would normally be changed into some meaningful catchment name.

13.3.9 Merge Catchments

This tool merges two or more existing catchments into one new catchment.

The catchments to be merged must be selected. Upon activation of the tool, the system prompts the user to confirm the merger. Care is required here, because the system deletes the original catchment records for the involved catchments and loss of valuable information is possible. Also, the network connections (if any) for the original catchments are deleted, i.e. the new catchment resulting from the merger must be re-connected.

The tool supports merger of any catchments, i.e. even detached catchments can be merged into one catchment. Also, overlapping catchments can be merged. In this case the area of the new catchment is equal to the sum of the involved areas, reduced by the overlap area.

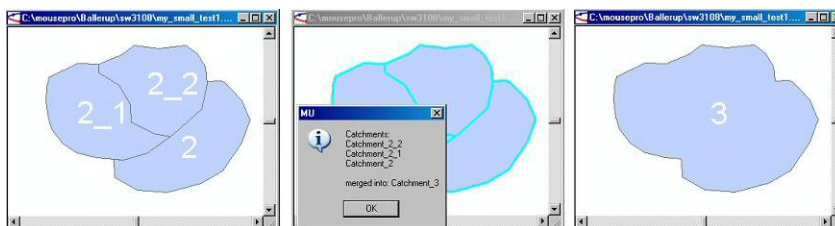


Figure 13.6 The merger operation starts from three original catchments (left); Upon selection of the wanted catchments and activation of the tool, the system prompts the user to confirm the merger (middle); The merged single catchment (right).

13.4 Connecting catchments to the drainage/wastewater collection network

In order to utilize the MIKE URBAN catchments in the context of network modeling, they have to be connected to the network.

Within the frame of MIKE URBAN Collection System, the catchments can be independently connected to the MOUSE and SWMM5 networks. Connections to the ASSET network are not supported.

MIKE URBAN supports N:1 connections, i.e. multiple catchments at a single network location (node).

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



13.4.1 MOUSE/SWMM Catchment Connection Editor

The connection of catchments to a MOUSE network can be accessed through the catchment dialog on the panel for "hydrology - MOUSE". An overview of all catchment connections is found under MOUSE|Catchment and catchment parameters|Catchment Connections (see also below table for fields found on this dialog).

The connection of catchments to a SWMM network can be accessed through SWMM|Hydrology|Routing.

Table 13.2 Provides an overview of the editor fields for connecting nodes to a MOUSE network

Edit field	Description	Usage	Attribute Table Field
Catchment ID	Unique catchment identifier	Yes	CatchmentID
Connection Type	Specifies the connection type. Only 'None' and 'Single Node' are included	Yes	TypeNo
Connection to Node	Identifier of a connection node	Yes, if 'Single Node' connection type	NodeID

In SWMM5, the surface runoff from a catchment can either be connected to a network load point (similarly as in MOUSE) or to another catchment. In the former case only the runoff for impervious area is considered, while in a connection to another catchment the surface runoff from either the impervious or pervious area of a catchment can be chosen.

The type of connection is controlled by the setting of "Subarea routing". If the "Subarea routing" is set to 'OUTLET' (Default), then the runoff flow from the catchment will be connected to a network load point. Otherwise, if the "Sub-area routing" is set to 'PERV' or 'IMPERV', the runoff from the catchment will be connected to the specified catchment.



13.5 Graphical Tools for Connecting Catchments to MOUSE and SWMM5 Networks

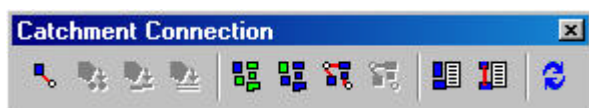


Figure 13.7 The "Catchment Connection" Toolbar

A set of graphical tools supports the process of connecting catchments to MOUSE and SWMM5 networks. These tools can be accessed through the menu (MOUSE|Catchment Tools or SWMM5|Catchment Tools). A dedicated "Catchment Connection" toolbar is available, including all relevant tools and direct access to relevant editors. Furthermore, some of the tools are available on the map's local menu (right click), as long as the application is in "catchment edit" mode.

The tools support only the option of connecting a catchment to a network node. I.e. connections between catchments as supported by SWMM5 cannot be handled graphically. If connection of a catchment runoff to another catchment is wanted as a part of a SWMM5 model setup, then it must be done through the SWMM5 Routing editor.

13.5.1 Catchment Dialog



This tool opens the catchment dialog.

13.5.2 Find Catchment Overlays/Gaps



This tool highlights catchment overlays - that is all areas covered by 2 or more catchments.

The highlighted graphics can be removed by pressing the "Find Catchment Gaps" tool again.



This tool highlights catchment gaps - that is all areas which is not covered by any catchment, but completely surrounded by catchment polygons.

The highlighted graphics can be removed by pressing the "Find Catchment Overlaps" tool.



13.5.3 Connect to Node



This tool allows for connecting one or more catchments to a single network node. The tool is accessible if one or more catchments are selected.

After picking the tool and clicking the mouse close to the node which is wanted as a point of connection, the system highlights the selected catchments and the connection node, draws the connection symbol and prompts the user to confirm the connection.

Execution of this action creates a new record in the catchment connection table "msm_CatchCon" (when connecting to the MOUSE network) or in the routing table "mss_Routing" (when connecting to the SWMM5 network), provided that the actual catchment has not already been connected to the network. If the selected catchment has already been connected, it would be re-connected to the currently selected node.

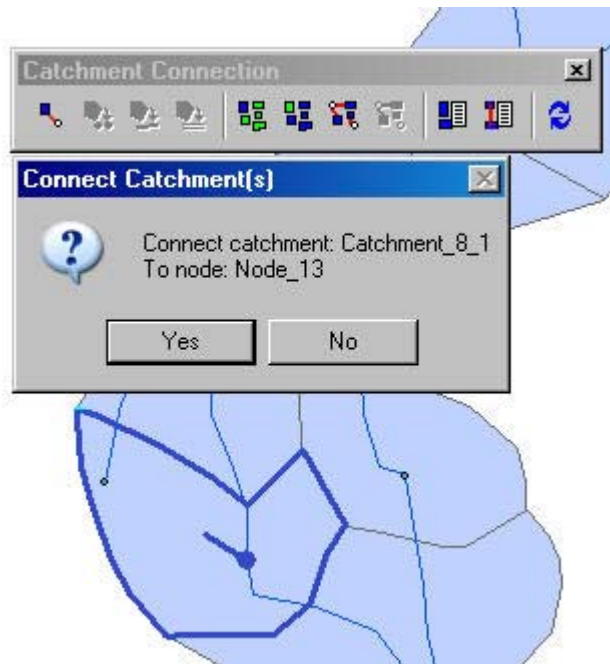


Figure 13.8 Graphical connection of a catchment to a network node - An example

13.5.4 Show All Connected Catchments



This tool selects all the catchments connected to the currently active network.

The highlight graphics can be removed by the "Refresh" tool.



13.5.5 Show All Disconnected Catchments



This tool selects all the catchments without connections to the currently active network.

The highlight graphics can be removed by the "Refresh" tool.

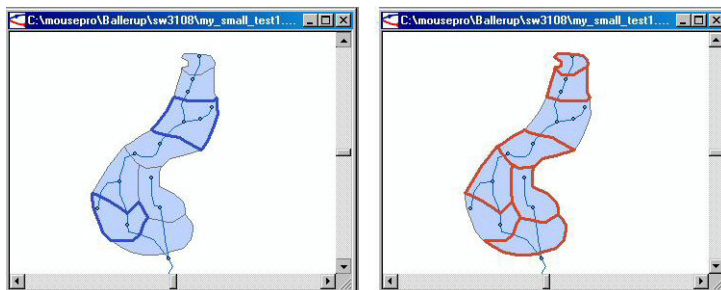


Figure 13.9 Highlighted connected catchments (left) and disconnected catchments (right)

13.5.6 Show Nodes Connected To Catchments/Catchments Connected To Nodes



This tool highlights the network node(s) of the currently active network connected to the selected catchment(s) and draws the connection symbol.

The tool is active if one or more catchments are selected.

The highlight graphics can be removed by the "Refresh" tool.



This tool highlights the catchment(s) connected to the selected network node and draws the connection symbol.

The tool is active only if ONE node of the active network is selected.

The highlight graphics can be removed by the "Refresh" tool.

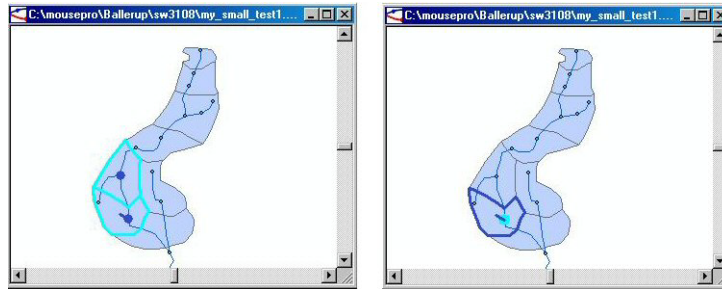


Figure 13.10 The highlighted nodes connected to the selected catchments (left) and a highlighted catchment connected to the selected node

13.6 Automated Catchment Tools

The catchment toolbox is a collection of tools that makes delineation of catchments for storm water networks extremely easy and fast.

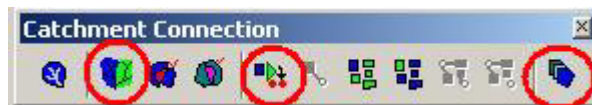


Figure 13.11 The Catchment Tools toolbar

The toolbox includes 4 automated tools:



The catchment delineation wizard helps you delineate catchment polygons in an automated and reproducible way. The catchments can be automatically created as Thiessen polygons derived from a layer of points or lines or as polygons derived from a digital elevation model (DEM).



The (catchment) connection wizard automatically connects all selected catchments to manholes based on a number of user specified principles, e.g. to the nearest manhole. For those places where you want the catchments to be connected differently the connections can easily be moved by use of the graphical editing tools.



The catchment parameter processing wizard is an automated and reproducible way to calculate imperviousness, time of concentration and other hydrological parameters for your hydrological models - traditionally a very time consuming task with big risk of making errors and inconsistencies. The hydrological parameters can be calculated for MOUSE rainfall-runoff model A and SWMM rainfall-runoff.



The catchment slope and length tool is an automated way to calculate the slope and length of a catchment based on a DEM. These parameters are used for MOUSE rainfall-runoff Kinematic Wave (B) model and for the SWMM rainfall-runoff model.

13.6.1 The catchment delineation wizard

The catchment delineation wizard helps you delineate catchment polygons in an automated and reproducible way. The catchments can be automatically created as Thiessen polygons derived from a layer of points or lines or as polygons derived from a digital elevation model (DEM)

The wizard guides you through the steps of the delineation process - starting with a short introduction dialog, see Figure 13.12.

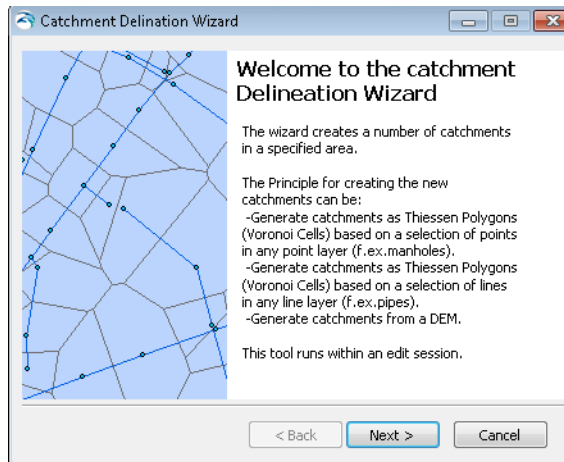


Figure 13.12 The start-up dialog of the catchment delineation wizard

First step in the delineation wizard is selection of the delineation principle, see Figure 13.13.

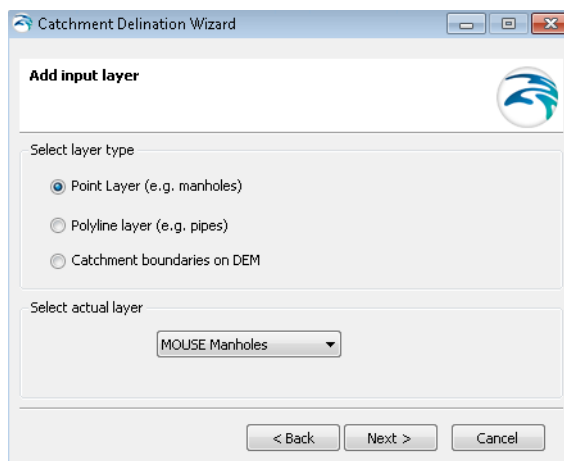


Figure 13.13 Selection of the delineation principle



There are the following possibilities:

- Generate polygons with Thiessen algorithm based on points (select point layer)
- Generate polygons with Thiessen algorithm based on lines (select polyline layer)
- Generate catchment boundaries from DEM (select raster layer). The catchments will describe the real directions of the surface flow. Please note that this option required to have the Catchment Analysis module in the license.

The selection of the actual layer allows you to select any layer of the relevant type which is represented in the Table of Contents.

Next step is selection of the outer boundary for the delineation, see Figure 13.14.

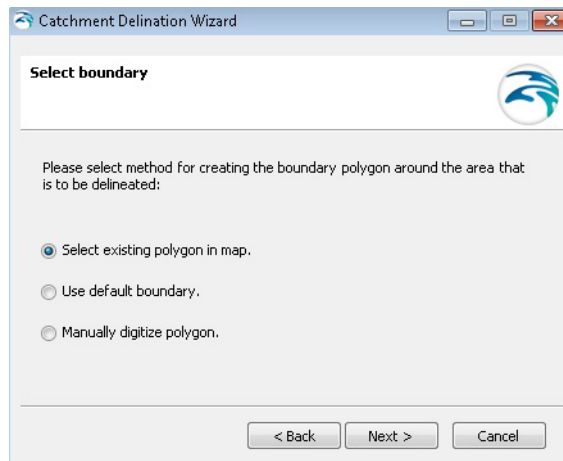


Figure 13.14 Selection of the outer boundary for the delineation

There are the following possibilities:

- Select an existing polygon from any polygon layer represented in the Table of Contents. If this method is selected the next step in the wizard will be to select the layer and (if this is not already done). Select one polygon in the layer with the normal selection tool in MIKE URBAN. See Figure 13.15.
- Use a default Boundary - which is a rectangle with round corners covering the complete network. There is no additional dialog for this selection.
- Digitize the boundary manually directly in MIKE URBAN. If this method is selected the next step in the wizard will be to draw the boundary directly in the horizontal plan - end the polygon by double clicking. See Figure 13.16.

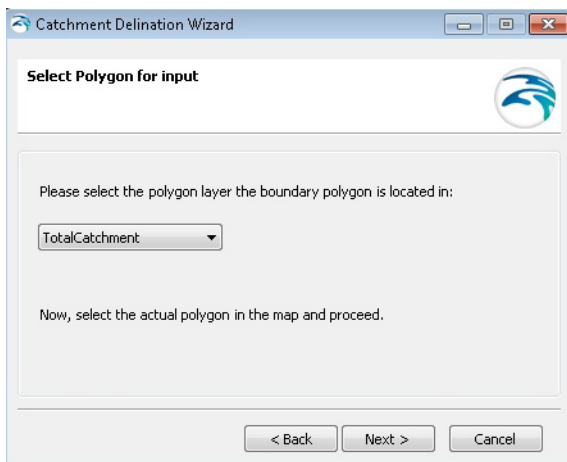


Figure 13.15 Selecting a polygon as a boundary for the delineation

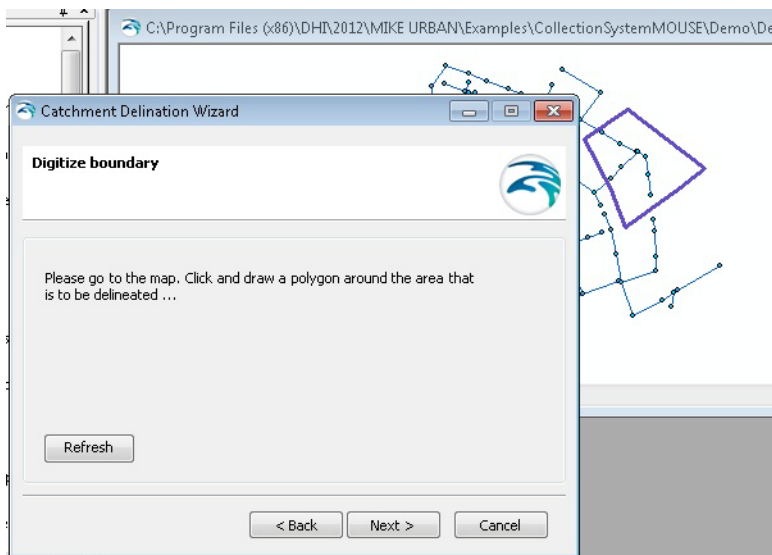


Figure 13.16 Digitizing the boundary to be used

The final step before the actual delineation is a short summary of the selections you have made. Press <finish> and wait for the delineation process to be finalized.



Figure 13.17 Input summary for the delineation procedure

Catchments generated as Thiessen polygons

When the catchments are created as Thiessen polygons (or Voronoi cells) you start out by specifying a selection of points or lines to use as an input layer. Typically either all/selected manholes or all/selected links.

The tool will proportionally divide and distribute a point coverage into the polygons known as Thiessen polygons. When a line layer is used as input the points used are the midpoints of the lines. Each polygon contains only one input features point. Each polygon has the unique property that any location within the polygon is closer to the polygon's point than to the point of any other polygon.

The Thiessen polygons (or Voronoi cells) are constructed as follows:

- All points are triangulated into a triangulated irregular network (TIN) that meets the Delaunay criterion.
- The perpendicular bisectors for each triangle edge are generated, forming the edges of the Thiessen polygons. The location at which the bisectors intersect determine the locations of the Thiessen polygon vertices.

The outside boundary of the Thiessen polygons needs to be specified. Either as a user chosen polygon (created on the fly or loaded in as a back file) or as extent of the points used plus an additional area.

Please note that the underlying Delaunay triangulation method used works best with data in a projected coordinate system.

Catchments boundaries based on DEM method

This method is using geometrical network together with DEM data when generating catchment.

To use this option you have first to specify the DEM you want to use and the 'Network inlets' which are the nodes where catchment runoff can enter the network.

The tool will then use Spatial Analyst to preprocess the data and generate two additional raster grids:

- A DEM with holes which is a raster with no-data holes where the nodes are located.
- A Flow direction grid which is generated based on the calculation of the flow direction. This calculation is done by processing the DEM with holes and by filling local minima in order to ensure that the surface lead downhill from all the areas to the no-data holes or edge of the DEM with holes.

Those two additional rasters can be added as background layers and can be re-used by the delineation wizard (Figure 13.18)

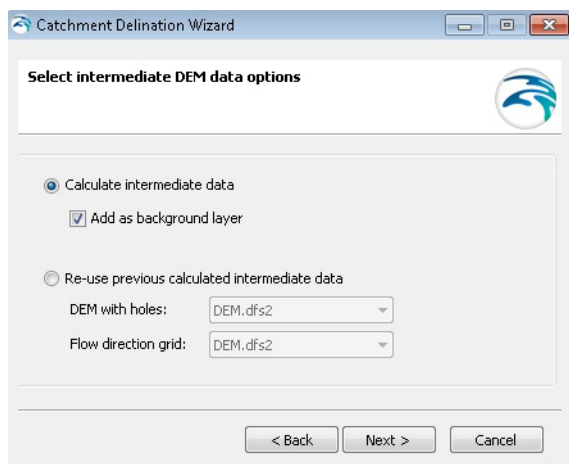


Figure 13.18 Selection of the preprocessed rasters

The tool will then use the Flow direction grid to delineate the catchments

13.6.2 The catchment connection wizard

The (catchment) connection wizard is a generic wizard which can be used to connect catchments, load points and demand points.

The wizard automatically connects all selected catchments to manholes based on a number of principles - f.ex. to the nearest node. For those places where you want the catchments to be connected differently the connections can easily be moved by use of the graphical editing tools.

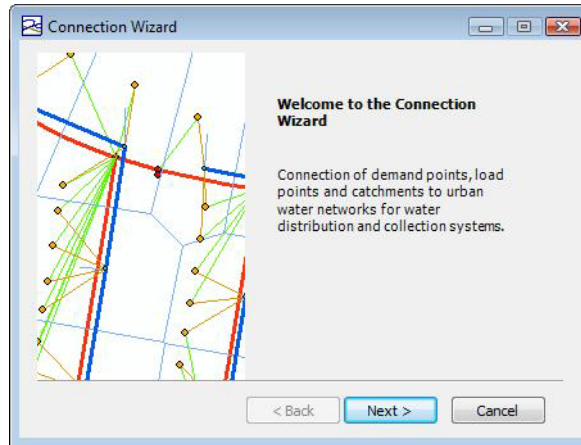


Figure 13.19 The start-up dialog of the (catchment) connection wizard

The connection wizard basically consist of 3 steps

1. Selection of items that should be connected (catchments, load points or demand points)
2. Selection of target to which the items should be connected (MOUSE, SWMM or EPANET network)
3. Selection of method for the connection. The available methods are different depending on the choices above. Setting the maximum search distance to -1.00 means that the search will be indefinitely.

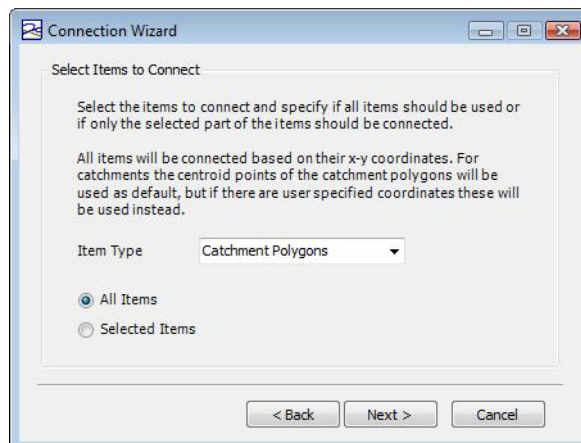


Figure 13.20 Selection of items that should be connected

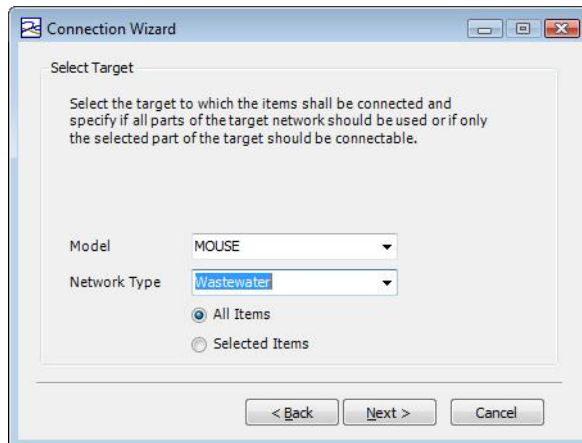


Figure 13.21 Selection of target to which the items should be connected

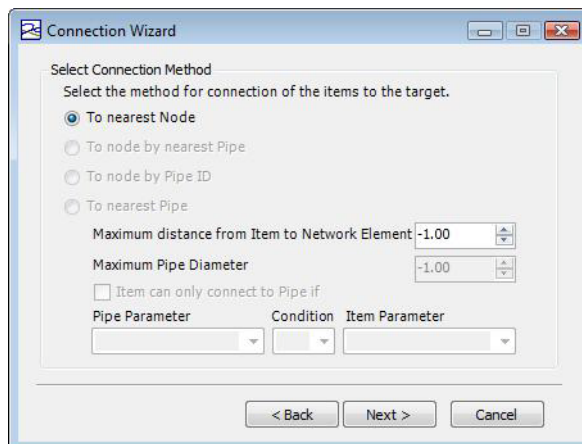


Figure 13.22 Selection of method for the connection

Final step before the actual connection is a short summary of the selections you have made. Press <finish> and wait for the connection process to be finalized.

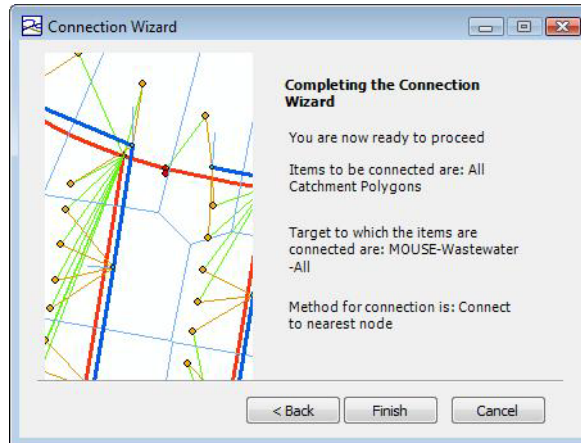


Figure 13.23 Input summary for the connection procedure

13.6.3 The catchment parameter processing wizard

The catchment parameter processing wizard is an automated and reproducible way to calculate imperviousness, time of concentration and other hydrological parameters for your hydrological models.

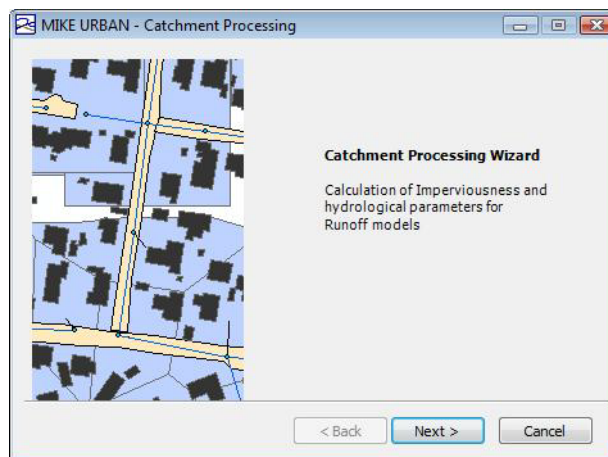


Figure 13.24 The start-up dialog of the (catchment) connection wizard

First step in the catchment processing is selection of which parameters to calculate. The wizard can be used for calculation of hydrological parameters for MOUSE model A or SWMM routing as well as calculation of concentrations for the SWQ module.

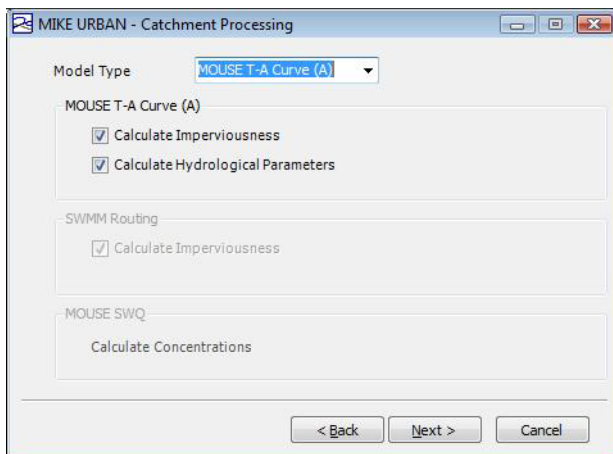


Figure 13.25 Selection of model type and parameters to calculate

Imperviousness for MOUSE or SWMM can be calculated as a constant value or as a weighted average of imperviousness in any number of layers. The layers have to be present in the TOC in order to be selectable in the wizard

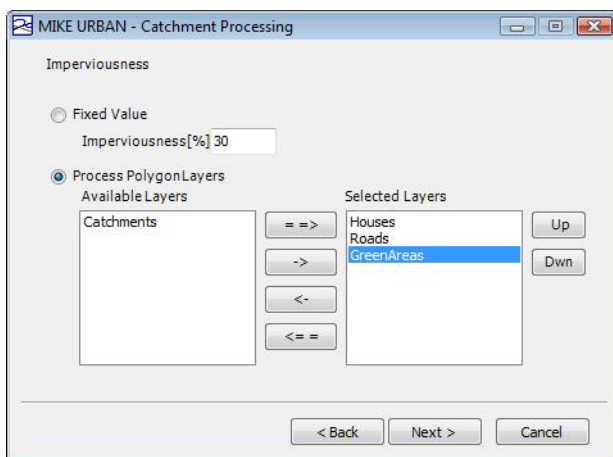


Figure 13.26 Selection of principle for calculation of imperviousness

Second step in calculation of imperviousness is specification of the imperviousness of each of the selected layers. Please note that the order of the layers is important - if some of the polygons are overlaying the value from the first specified layer (in top of the list) is preferred.



MIKE URBAN - Catchment Processing

MOUSE T-A Curve (A)

Set values for each layer

Layer	Imperviousness [%]
Houses	90
Roads	75
GreenAreas	10

< Back Next > Cancel

Figure 13.27 Specification of parameters for calculation of imperviousness

For MOUSE model A a number of hydrological parameters can be calculated and the specifications is therefore split in two pages of the wizard. The principles are explained directly in the wizard.

MIKE URBAN - Catchment Processing

HydrologicalParameters1(2)

Time of Concentration

Mean Surface Velocity [m/s]

Length will be calculated as maximum distance from connected node to the border of the catchment.

The calculated Time of Concentration is rounded to whole minutes.

Fixed Values

Hydrological Reduction Factor

Initial Loss [mm]

< Back Next > Cancel

Figure 13.28 Specification of hydrological parameters for MOUSE 1(2)

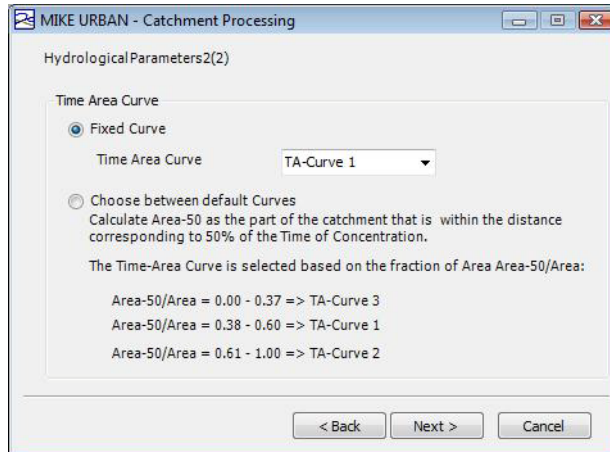


Figure 13.29 Specification of hydrological parameters for MOUSE 2(2)

Final step before the actual catchment processing is a short summary of the selections you have made. Press <finish> and wait for the catchment process to be finalized.

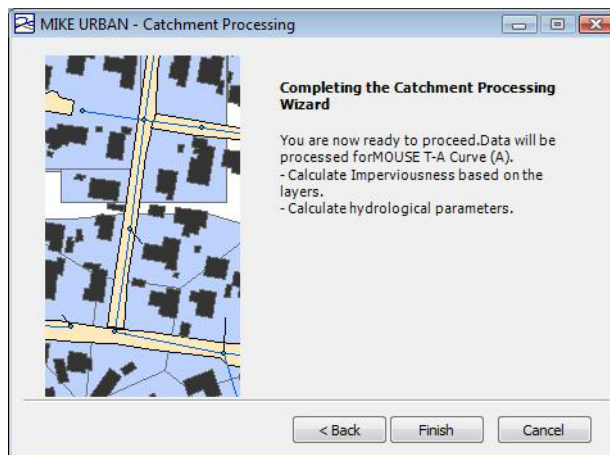


Figure 13.30 Input summary for the connection procedure.

13.6.4 Catchment Slope and Length Tool

As part of the hydrological modeling, the catchment slope and length must be estimated for some rainfall-runoff models.

Based on the delineated catchments, a DEM and lines for the flow path inside a catchment, the slope and length can be automatically estimated for each catchment using the Catchment Slope and Length tool. The tool performs automatic estimation of hydrological parameters for each catchment in a consistent, documented and reproducible way.



The tool is initiated from the Catchment Tools menu or toolbar.

To calculate the slope and length the typical flow path within the catchment must be digitized (the slope lines). These can be drawn from the load point or towards the load point but a consistent methodology should be used in a project. A multiple number of slope lines can be defined for each catchment. The slope lines must be a line feature in MIKE URBAN either from a background layer or an existing (unused) layer in the database.

The slope and length are calculated as an average slope and length of the lines that are completely contained within the catchment.

An example of slope lines are shown in Figure 13.31.



Figure 13.31 Example where two slope lines are drawn from the loadpoint in the network up into the catchment along the normal flow path

Please note that the tool will work on the active mode regardless if the tool is started from the MOUSE or SWMM menu.

When the tool is opened the slope line layer must be specified together with the direction the lines were digitized to obtain the correct sign of the slope. A minimum slope is also specified that will be assigned to all catchments with smaller slopes. The DEM must be added as a background layer in MIKE URBAN to be available in the tool, see Figure 13.32.

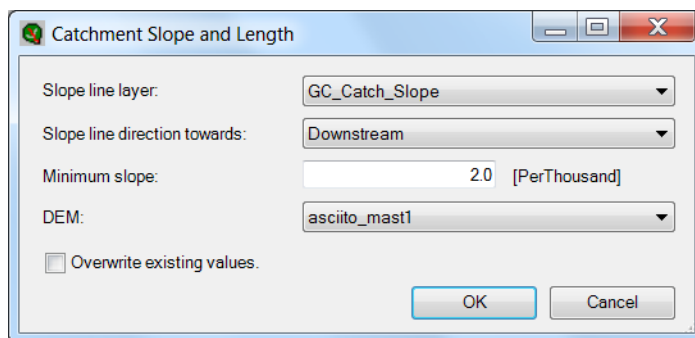


Figure 13.32 Catchment Slope and Length Tool

The tool will calculate the length and slope for all selected catchments. If no catchments are selected, the length and slope will be calculated for all catchments containing slope lines.



14 Load allocation through geo-coding

MIKE URBAN supports allocation of geographically determined load points to the nodes of collection system model network (only MOUSE in the present release) through several geo-coding methods. The allocated loads may then constitute a component of the overall network load definition for a collection system hydraulic model.

The load points are geographical point features, typically representing water, pollution and/or sediment sources (households, factories, etc.). Concerning the source type, each point can be attributed by one of the available source categories: Domestic Wastewater, Industrial Wastewater, etc. To be used in MIKE URBAN, each point must be attributed by the load size (volume or mass per unit time).

The present MIKE URBAN release supports the water point loads only, i.e. the water quality properties for the point loads (if available) cannot be utilized.

Typical origin for the useful sets of point loads would be water consumption records, normally available in GIS applications related to management of urban water utilities. A specific source of point loads is the demands allocation table found in each MIKE URBAN Water Distribution project. I.e., the water demand allocations can be directly imported into the Collection System project, to be used as the MOUSE network load.

14.1 Management of Point Loads

The management of Point Loads for collection system hydraulic models consists of the following distinct steps:

- Inserting/Editing/Importing of load points;
- Allocation (geo-coding) of load points to the model network;
- Aggregation of load allocations.

Furthermore, in order to include the allocated load points into the hydraulic model computations, a proper definition of the load as boundary condition must be given.

On the general level, the load points are managed either through customized import from MIKE URBAN Water Distribution or through user-defined import from various external sources.

On the detailed level, the load points can be managed both graphically and through the "Load Allocation" editor. The two modes complement each other. Only through a joint application of both modes an efficient and complete management of point load allocation can be achieved.



The graphical edit mode ("Feature Edit") allows for digitalization of load points by tools "Insert", "Move" and "Delete". A special graphical tool is available for the load allocations (connections) to the network nodes.

The "Load allocation" editor is primarily used for reviewing and editing the load points attributes, deleting obsolete load points and for access to the vital related tools for geo-coding and aggregation.

14.2 The Load Allocation Editor

The "Load Allocation" editor can be accessed through the menu (MOUSE|Load Allocation). Also, the "Load allocation" toolbar includes a tool for direct access to the "Load Allocation" editor.

MUID*	Load ID	Data source	Node I	LinkID*	Load me	Water	X coordinat	Y
334	WaterUse2004_11	mw_DemAllo	94	<Null>	Loads	1.111000	512416	
335	WaterUse2004_12	mw_DemAllo	95	<Null>	Loads	0.735	<Null>	
336	WaterUse2004_13	mw_DemAllo	105	<Null>	Loads	1.111000	<Null>	
337	WaterUse2004_14	mw_DemAllo	105	<Null>	Loads	1.111000	<Null>	

Figure 14.1 The "Load Allocation" editor



14.2.1 The Editor Fields

Table 14.1 Overview of the editable load allocation attributes (Table 'ms_LALoad-Alloc')

Edit field	Description	Usage	Attribute Table Field
Load ID	Reference to the load point identifier in the original source.	Optional	ReferenceName
Data Source	Used for identification of the data source (file & pathname)	Optional	DataSource
Description	Free text description	Optional	Description
X co-ordinate	X co-ordinate of the load point. Provided automatically by the system	Calculated (read-only)	X
Y co-ordinate	Y co-ordinate of the load point. Provided automatically by the system	Calculated (read-only)	Y
Category	Classifies the load point into one of the available load categories. Relevant when several categories of point loads are to be distinguished in the project.	Optional	LoadCategoryNo
Owner	Identifies the "owner" (e.g. water consumer) of the load point.	Optional	LoadOwner
Location	Identifies the site (e.g. mailing address) associated with the load point.	Optional	LoadLocation
Date	Specifies the date of entry or load validity period.	Optional	LoadDate



Table 14.1 Overview of the editable load allocation attributes (Table 'ms_LALoad-Alloc')

Edit field	Description	Usage	Attribute Table Field
Load Method	Defines the calculation method as (1) direct load or (2) load units, e.g. person equivalents. Only "Load" option is implemented	Yes	MethodNo
Water	Defines the water load amount as volume/time	Yes	WaterLoad
MOUSE Node ID	Allocates the point load to the specified MOUSE node. Can be filled in manually, by graphical tools or by automatic load allocation tool.	Optional, in alternation with MOUSELinkID, when geo-coding to MOUSE network	MOUSENodeID

14.3 Importing Load Points

14.3.1 Importing Load Points From MIKE URBAN Water Distribution

In MIKE URBAN projects which include both water distribution and collection systems, the point loads for the collection system would typically be imported from the water distribution part of the project. Namely, the water demands are specified for the water distribution network as demand points, equivalently to the collection system's load points. I.e. water demands are turned into the collection system loads.

MIKE URBAN supports this transfer through a custom tool for internal data transfer (TOOLS/Send Data/WD Demands to CS Loads). If the demands point data exist in Water Distribution, the tool copies the point features and relevant attributes from the Water Distribution layer "Water Demand Allocation" (database table mw_DemAlloc) into the Collection System layer "CS Load Points" (database table ms_LALoadAlloc).

Upon completed data transfer, the point loads data are ready for further processing in the Collection System project.



14.3.2 Importing Load Points From External Sources

The load points are typically found as a layer in a GIS application or as tabulated data in database tables, spread sheets or ASCII files. They usually represent water consumption records or wastewater and/or pollution emissions according to discharge permissions.

If load points data is part of GIS, then the geographical information is intrinsically present. When stored in any other tabular format, the table must include the columns with X and Y co-ordinates. Indeed in both cases, for correct overlay of the network data and the load points, it is essential that the co-ordinate systems of the current MIKE URBAN project and the external GIS are identical.

Concerning external GIS as a source of data, MIKE URBAN supports import for shape file (*.shp). All major GIS systems can export the data into shape file.

14.4 Graphical Editing of Load Points

The tools for graphical editing of load points can be accessed through the menu (EDIT|Feature Edit) or through the default toolbar "Feature Edit".

The "Feature Edit" toolbar is put in the "Load points" mode by selecting the load points layer to be "active layer".



Figure 14.2 The "Feature Edit" toolbar ready for editing load point

All tools for graphical editing are fully supported by "Undo" function (Edit|Undo).

14.4.1 Create a Load Point

This tool allows for digitalization of load points.

When this tool is active, a small icon is displayed besides the cursor. The load points are digitized by clicking on the cursor at the desired locations on the map.

The tool is deactivated by selecting some other tool.

Each new load point is added as a new record in the load allocation attribute table (ms_LALoadAlloc). Per default, a name (i.e. identifier) is given as "Load_Point_n", where "n" stands for internal load point index. If required, the default identifier would normally be changed into some meaningful name.



14.4.2 Move Load Point

An individual load point or a group of load points can be moved (translated) to a new position.

The "Move" tool operates on the selected set of load points. The Move tool remains active as long as some other tool is activated.

14.4.3 Delete Selected Load Point

When clicked by mouse, this tool deletes the selected loading points.

14.5 Allocating the load points to the Model Network

Three load allocation methods are available:

1. Manual load allocation
2. Graphical Load Allocation
3. Automatic allocation by GIS geo-coding.

14.5.1 Manual Load Point Allocation

Individual loading points may be allocated to the collection system nodes through the Load Allocation editor. This method is appropriate for individual corrections and/or for the smaller sets of load points, see Figure 14.3.

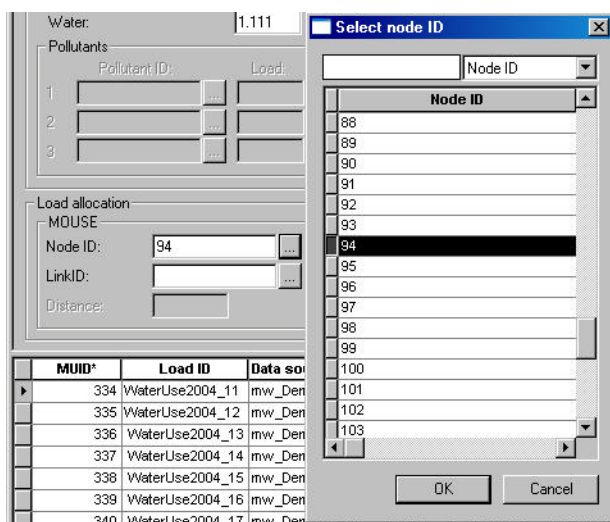


Figure 14.3 "Manual" allocation of the load point to a MOUSE node - Example



14.5.2 Graphical Load Point Allocation

Individual loading points may be allocated to the collection system nodes through the "Place To Node" graphical tool. This can be activated by the "Place" button on the "Load Allocation" editor or by the "Place To Node" button on the "Load Allocation" toolbar.



Figure 14.4 The Load Allocation Toolbar

The work process (for MOUSE network) is as follows:

1. Make the "CS Load Points" layer to be selectable layer
2. Select one load point
3. Click on the "Place" button or the "Place To Node" tool.
4. Point the desired MOUSE node and click.
5. Confirm or reject the specified allocation.

Upon user confirmation, the system fills in the MOUSENodeID field in the ms_LALoadAlloc table for the current load point and plots the connection line between the load point and the selected node.

If the current load point has already been allocated to some other node, the confirmation of the allocation action would re-connect the load to the currently pointed node.

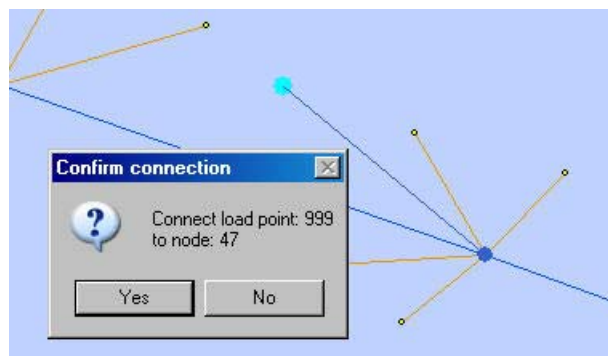


Figure 14.5 The Load point "999" is about to be allocated to the MOUSE node "47". The system draws the connection and prompts the user to confirm or reject the allocation action.

This method is appropriate for individual corrections and/or for the smaller sets of load points.

14.5.3 Automatic Load Points Allocations by GIS Geo-Coding

MIKE URBAN supports automatic allocation of load point to collection system nodes through GIS geo-coding process. The geo-coding process is initiated and controlled through the connection wizard (Figure 14.6). This dialog opens by the "Geocode" button on the "Load Allocation" editor.

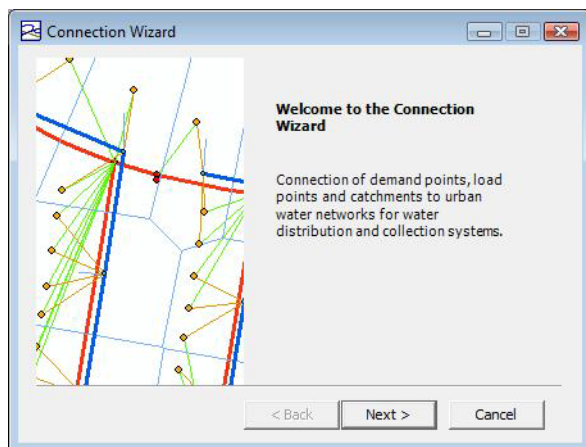


Figure 14.6 The connection wizard

The following parameters affect the geo-coding process:

Model: At present "MOUSE" is the only choice available.

Network Type: The load points are allocated only to the network elements (nodes and pipes) of the specified network type. Thus, it is avoided that wastewater loads are allocated to the storm drainage network present in the same project.

Load category: A load point may be classified according to the available load types. Sometimes (e.g. in case when pollution associated with each category defines separately), it might be necessary to maintain the various load categories separated, so that the model boundary conditions can be defined properly. Selecting some records (e.g. based on the category) the connection wizard then allows to perform the geo-coding only on the selected load points (see Figure 14.7).

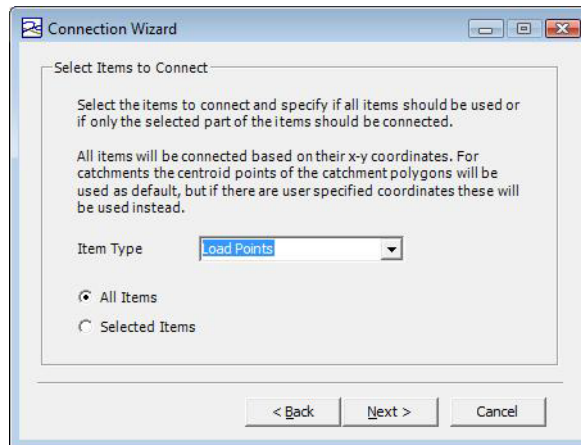


Figure 14.7 Connecting load points - choose between the selected or all

Geo-code method: There are two geo-coding methods available for connecting load points to the MOUSE model (see Figure 14.8):

- Geo-coding to Nodes: The load point is allocated to the MOUSE node (manhole or basin) which is geographically closest to the load point
- Geo-coding to Nodes (by nearest pipe): The load point is allocated to the downstream node of a link which is geographically closest to the load point. There are two additional parameters which affect the geo-coding process in this method:
 - Maximum distance from Item to Network Element (snap tolerance radius): The specified value (in map units) determines the largest distance for which is the geo-coding performed. All load point which are not within the specified snapping distance to any pipe will remain non-allocated.
 - Maximum Pipe diameter: The specified value (in units for pipe diameter) limits the largest circular pipe which is eligible for geo-coding. I.e. all larger the pipes (presumably trunk sewers) are assumed not to receive any direct loads.

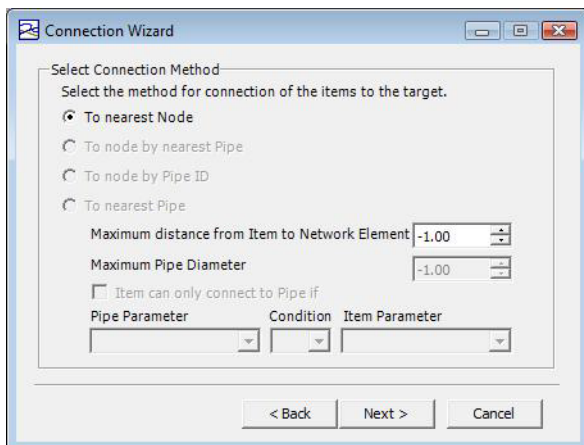


Figure 14.8 The geo-code methods, the upper two options are available for geo-coding load points to a MOUSE model

And please note, the geo-coding process works on the selected set of load points or on the entire set.

The user should be aware that geo-coding of a large set of load points is a computationally intensive process and may take some time.

If the geo-coding is attempted for already allocated points, the existing allocations will be cleared and replaced by the new.

14.6 Transfer of Data to Model and Load Aggregation

The load allocations as described in the previous paragraphs are not used directly as model loads. Prior to the use by a model, the load allocation data must be transferred into the respective model (at present only to MOUSE database table `msm_LAAggrLoad`). The content of this table is accessible for viewing and as a grid attached to the dialog data transfer & aggregation dialog (see further below) An overview of the table is given in Table 14.2.

Table 14.2 Overview of the table 'msm_LAAggrLoad'

Edit field	Description	Usage	Attribute Table Field
N/A	Internal identifier	Yes	MUID
N/A	Defines the load target as a Node or a Link, (always set to "Node" in current release)	Yes	TargetTypeNo
N/A	Load connection node	Optional, in alternation with LinkID	NodeID



Table 14.2 Overview of the table 'msm_LAAggrLoad'

Edit field	Description	Usage	Attribute Table Field
N/A	Classifies the load point into one of the available load categories. Relevant when several categories of point loads are to be distinguished in the project.	Yes	LoadCategoryNo
N/A	Actual value of aggregated load	Yes	AggrLoadValue

Optionally, simultaneously with the transfer, the load allocations get aggregated as single loads per node. I.e. multiple loads allocated to a single node get summed up and replaced by one load. This usually reduces the amount of information which must be handled by the model significantly, without any loss of accuracy.

The data transfer and (optional) aggregation process is initiated by the "Transfer to MOUSE" button on the Load Allocation editor and controlled by the parameters on the "Transfer to MOUSE" dialog (Figure 1-7).

The aggregation and transfer process works on the entire set of load points.

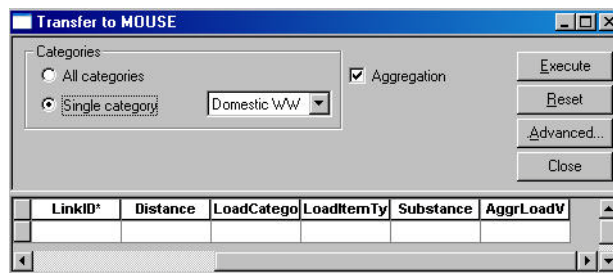


Figure 14.9 The "Transfer To MOUSE" dialog, with the grid display of the table msm_LAAggrLoad

The aggregation can be switched ON (default) and OFF. If the aggregation process is OFF, the records are transferred to the database table msm_LAAggrLoad as 1:1.

The transfer and aggregation processes can be executed for all load categories simultaneously or for individual categories. This results in the following combinations:



1. "All categories" AND Aggregation=ON: all the loads allocated to a single node will be aggregated (one value per category) and transferred to MOUSE.
2. "All categories" AND Aggregation=OFF: all the load allocation records will be transferred to MOUSE as 1:1.
3. "Single Category" AND Aggregation=ON: all the loads of the specified category allocated to a single node will be aggregated into a single value and transferred to MOUSE.
4. "Single category" AND Aggregation=OFF: all the load allocation records of the specified category will be transferred to MOUSE as 1:1.

The transfer and aggregation processes is executed by the "Compute" button. In consecutive "Compute" actions involving the same load category and the same node, all previously existing records related to that load category and that node are cleared prior to the computation of a new load. This prevents duplication, i.e. ensures that certain load point can be used once and only once as a network load.

The "Reset" button reverses the "Compute" action, i.e. clears the `msm_LAA-grLoad` table of all the records associated with the specified load category. Indeed, if "All categories" is selected, the table is completely cleared.



15 Interpolation and Assignment Tool

15.1 Introduction

The field assignment and interpolation tool is a wizard controlled tool that will assign values to any field in the MIKE URBAN database either by taking the attribute value directly from another feature or by interpolating between and number of other features.

The source of the data (i.e. the features where data is taken from) may be any layer in the MIKE URBAN map view, including layers you have added yourself as background layers or layers belonging to another model (e.g. assigning values to the MOUSE model from the water distribution model).

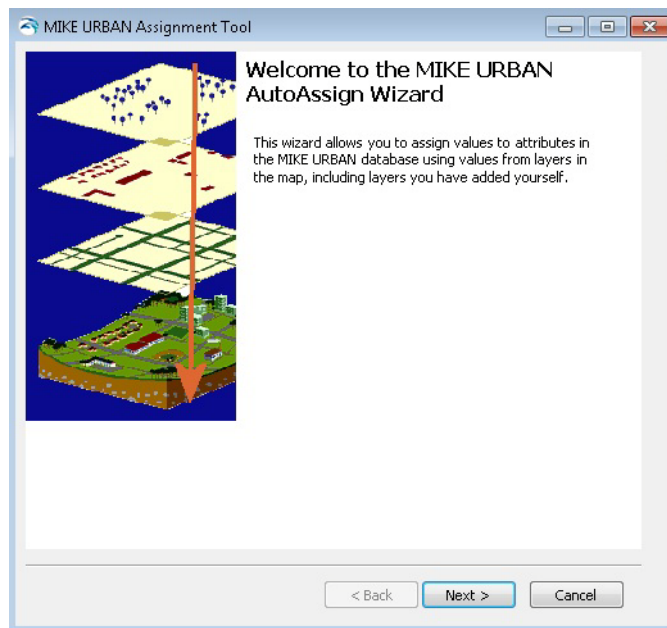


Figure 15.1 The start-up dialog of the Interpolation and Assignment tool

Examples of the tasks that may be performed with this tool are:

- Assign values to 'GroundElevation' from a raster layer representing the DEM.
- Assign the Diameter of manholes to be equal to the largest pipe entering the manhole.
- Calculate missing values for GroundElevation from a point theme using Inverse Distance weighted spatial interpolation



- Calculate pipe levels by interpolating values following the network (pipes).
- Assign a value to construction year and or contractor based upon a polygon theme giving city areas

The tool is completely generic in the sense that it can assign any compatible data value to almost any field in the database. This also means that it should be used with some care as it obviously also can make completely non-sense assignment if the wrong fields or names are specified.

The tool is accessed through Tools|Interpolation and Assignment. The first dialog that comes up is shown in Figure 15.1. The tool is built up using a wizard approach.

In the following sections the various wizard screens are shown with explanations of the various input options.

15.2 Select Target for the Assignment

After pressing 'Next' on the first dialog the dialog displayed in Figure 15.2 is shown. On this page you select the attribute you wish to assign values to. This is done by selecting first the map layer and then in the attribute box, the actual attribute you wish to update/assign. In the shown dialog the MOUSE manholes are selected and the ground level is chosen as the attribute.

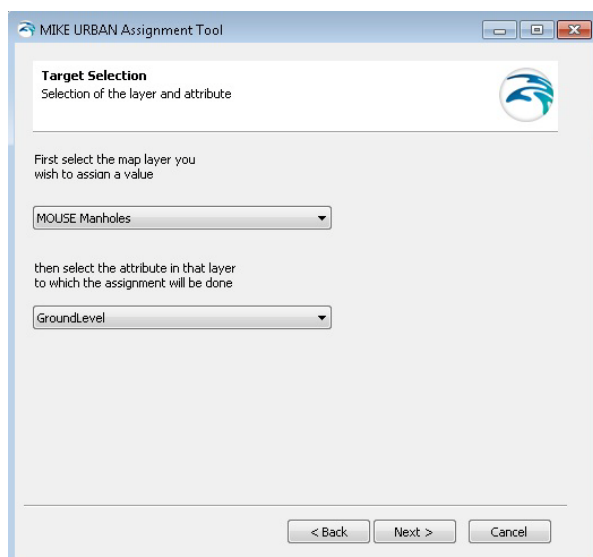


Figure 15.2 Target Selection dialog



15.3 Selection of Method and Data Source

After pressing 'Next' the next wizard page, Figure 15.3, lets you select the method you wish to use for the update and the data source you wish to use for updating the attribute.

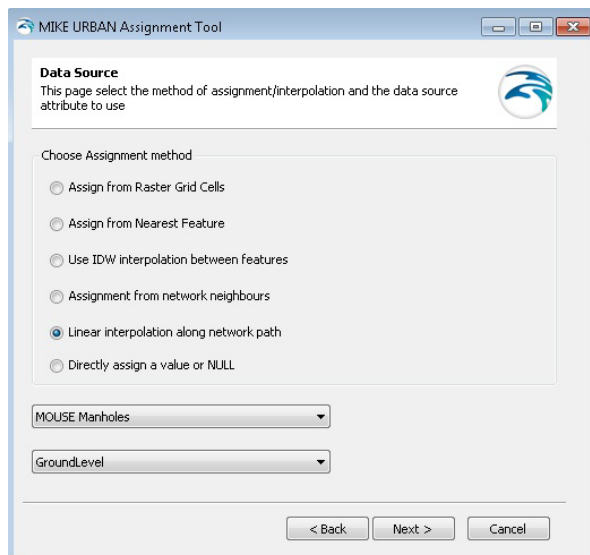


Figure 15.3 Data Source dialog

First you must select the actual method as this will influence the valid choices for data source. The choices are:

- Assign from raster cells - this will assign the value from the raster cell in which a point is placed to the target attribute. If the target is lines or polygons the tool will use the centroid position to determine the correct cell. No interpolation is done.
- Assign From Nearest Feature - In this case the tool will locate the feature from the source layer that is closest to each feature in the target layer. If lines or polygons are used the centroid position is used for calculating distances.
- Use IDW interpolation between features - this option will make an Inverse Distance Weighted (IDW) interpolation between features in the source layer to determine the value for each affected target feature. The IDW parameters are fixed to the following: max number of points is 12 and the max distance away from the target feature is 300 (map units)



- Assign from Network Neighbours - This option will take the source value from a network neighbour to the feature being updated. This obviously requires both the target and the source to be included in the same network like manholes and pipes in a sewer network. The following wizard page will let you choose between options to determine exactly which neighbour that will be used. Assignment will only be done if the immediate neighbour has the requested value i.e. the network will not be traced.
- Interpolation along Network path - This option will do a distance weighted interpolation along the path of the network. If the direct neighbours does not contain values (null) the network is traced until a value is reached or the number of 'hops' (number of network nodes traced though) exceed a given maximum.
- Directly assign a value or NULL - This option allows to assign a specific value or to delete the content of an attribute (by assigning the NULL value).

Depending upon the choice above the two selection boxes for the source data will be filled with layers/attributes compatible with the choice of method (i.e. only raster layers will be shown for raster assignment) or greyed out in the case of the last option.

After selecting the source fields you may press next and get to one of the following pages depending upon your choices.

15.4 Network Assignment Options

The page in Figure 15.4 lets you decide what network neighbour element to use when you have chosen the network assignment method.

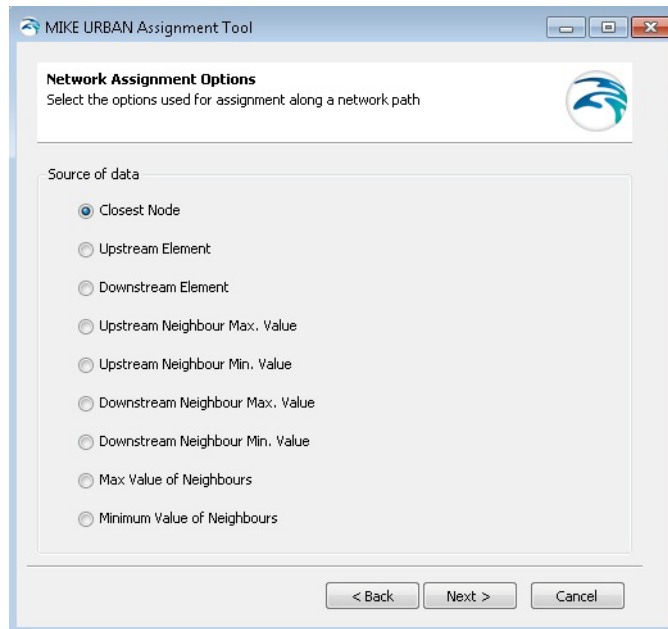


Figure 15.4 The Network Assignment Options dialog

- **Closest Node** - This will use the node that is closest to the one being assigned to. This option is only relevant if both target and source are nodes.
- **Upstream Element** - This option will assign from the upstream element (upstream/downstream is as defined by the GIS geometric network and may differ from the actual flow direction (which may not be constant))
- **Downstream Element** - This option will assign from the downstream element (upstream/downstream is as defined by the GIS geometric network and may differ from the actual flow direction (which may not be constant))
- **Upstream/Downstream Neighbour Max. Value** - These two options will scan the connected network neighbours upstream/downstream and use the maximum source value found as data source. Example: for assigning ground level and diameters
- **Upstream/Downstream Neighbour Min. Value** - These two options will scan the connected network upstream/downstream neighbours and use the minimum source value found as data source. Example: for assigning invert levels.
- **Max. Value of Neighbours**- This option will scan the connected network neighbours and use the maximum source value found as data source. Example: for assigning groundlevel and diameters
- **Min. Value of Neighbours** - This option will scan the connected network neighbours and use the minimum source value found as data source. Example: for assigning invert levels.

15.5 Network Interpolation Parameters

The page displayed in Figure 15.5 allows you to control how many network 'hops' the interpolation will search for a value. The search continues until the max number is reached or a non-null value is found. When the values is set to 5 or higher a warning will be displayed as setting the number to high may cause instability (particularly in looped networks). A value of 0 means that only immediate neighbours are taken into consideration. Large values may be time consuming if a large number of features are selected for update.

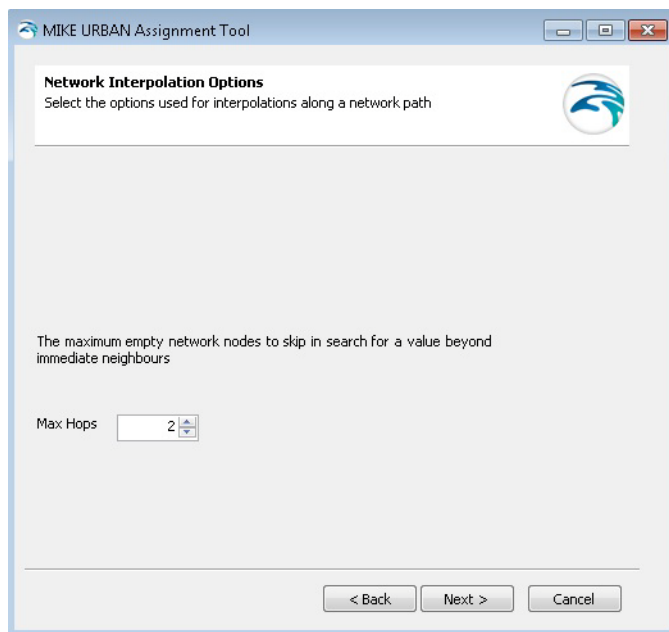


Figure 15.5 The Network Interpolation dialog

15.6 Overall Selection Options

On the dialog shown in Figure 15.6 you can control what features are taken into account for the assignment operation.

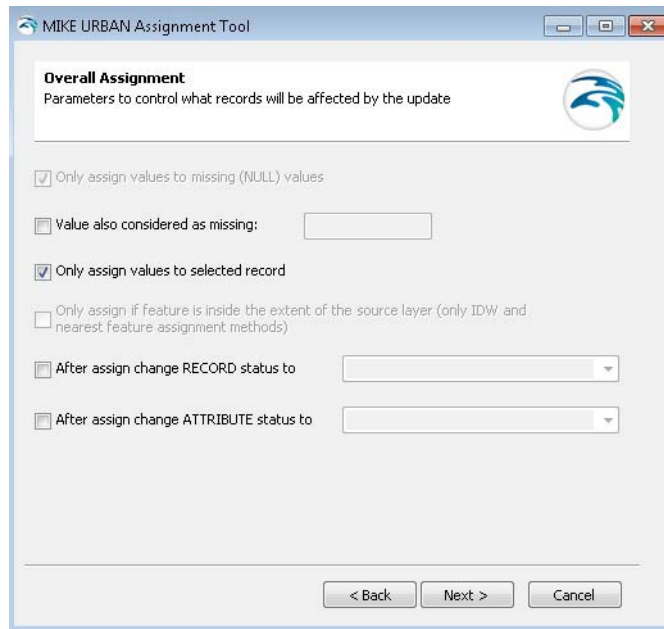


Figure 15.6 The Overall Assignments dialog

- Only assign to missing (NULL) values - means that features already having a value in the target field will not be updated. Removing this tick mark will overwrite any feature.
- Only assign to selected records - this means that only records that are selected before you started the wizard are taken into consideration for updates. You can select either by attribute or by location before you start.
- Only assign to features inside the extent of the source layer - this option prevent the tool from extrapolating outside the boundaries of the source layer when looking for the closest feature or when doing IDW interpolation.

It is optional also to assign a change in the record and attribute status. This can e.g. be used to flag the record status to 'modified' when changing records with the interpolation and assignment tool.

15.7 Finishing the Wizard

The last page gives a summary of your inputs. When pressing 'Finish' the calculation will start.

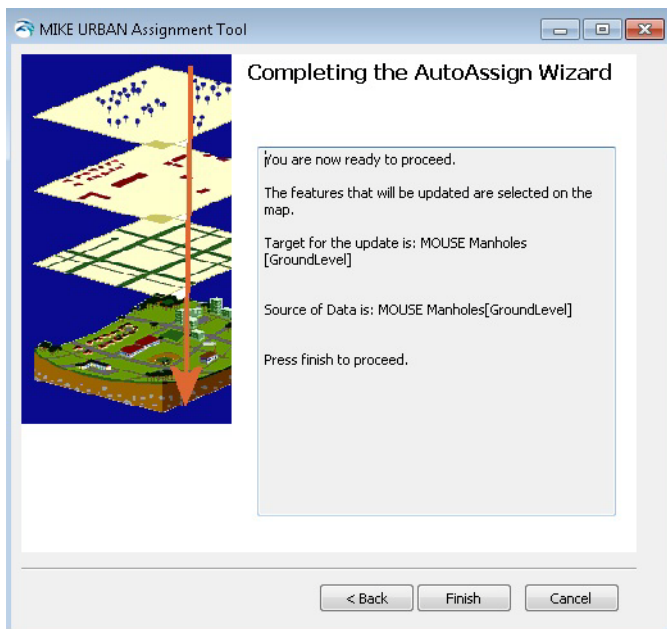


Figure 15.7 The summary dialog

Note that the features to be updated are selected on the map to make it easy to check that only the expected features are included. This selection does not check for other constraints i.e. null values may still prevent some of the selected features from being updated.



16 Simplification Tool

16.1 Introduction

'Simplification' is the term associated with the process of removing disconnected and unnecessary network elements, removal of network parts outside an area of interest and eliminating internal nodes which appear as redundant and insignificant for the hydraulic computations. Simplification reduces the complexity of a network, which improves the efficiency of the hydraulic computations without compromising the integrity and accuracy of the model.

Simplified networks are used in different contexts such as online model applications, long term simulations, strategic versus detailed models, etc.

16.2 Launching the Tool

The tool is found under Tools|Simplification. When activated the dialog shown in Figure 16.1 appears.



Figure 16.1 Welcome simplification wizard page

It is possible to load simplification settings from an earlier configuration. When continuing the settings in all dialogs will reflect the previous configuration.



If one or more configurations should be executed without editing the 'Load multiple configurations' can be used. See 16.3.9 for further details.

16.3 Simplification Procedure - Wizard

The simplification procedure is built into a simplification wizard and can be described by the following steps:

1. Selection of model network
2. Selection of simplification method (Scrubbing, Trimming, Merging)
3. Selection of model area of interest for which simplification is valid
4. Specification of criteria for elements to be selected
5. Specification of merge methods - valid only for Merging
6. Specification of reconnection of connected element
7. User interaction - changing the selection of elements to be simplified

Each of the above steps are explained in detail below.

16.3.1 Selection of MIKE URBAN model network

The simplification tool can be used for the model network layers in MIKE URBAN, such as Collection System (MOUSE, SWMM) and Water Distribution (EPANET). The Asset model layers in MIKE URBAN can not be simplified. The selection of the model network layers (MOUSE, SWMM and EPANET) is set automatically to the active layers in the current project, but can be changed by the user.



The simplification can be limited to work only on certain model network types for Collection Systems, such as the Waste, Storm or Combined network.

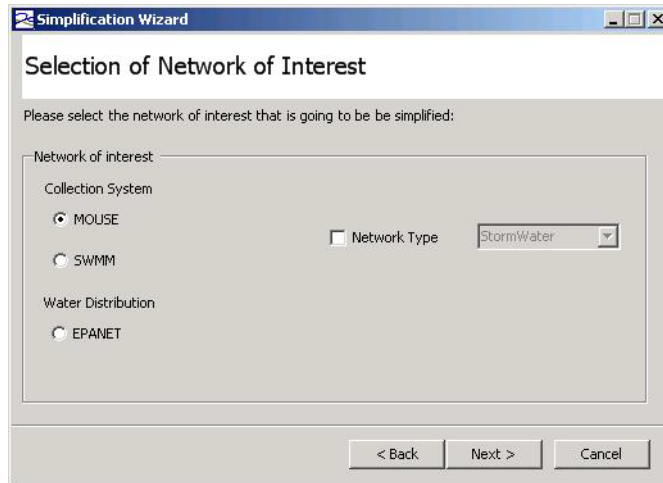


Figure 16.2 Selection of network of interest

16.3.2 Selection of simplification method

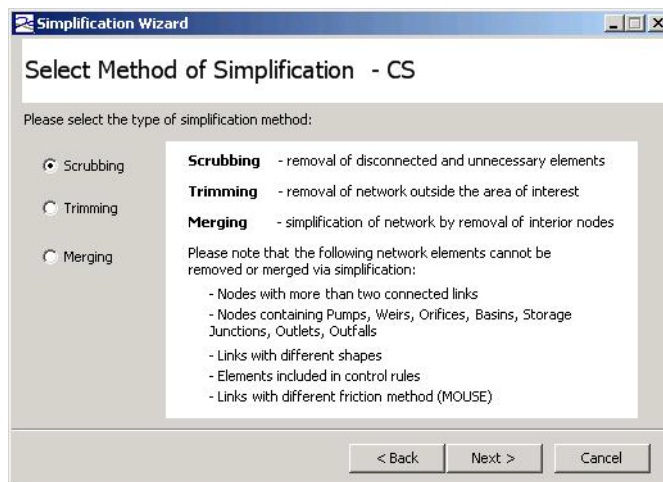


Figure 16.3 Selection of simplification method

The simplification of a network is organized into three main simplification methods



1. Scrubbing
2. Trimming
3. Merging

Scrubbing

Scrubbing is associated with the process of removing disconnected network elements, such as orphan nodes and links. Additionally, the purpose of Scrubbing is the elimination of relatively insignificant network branches at upstream end of network and service pipes - so-called dead-end pipes. Removal of dead-end pipes can be done based on user specified criteria, such as pipe lengths shorter than a certain length and removal of pipes with a diameter less than a certain diameter.

Unlinked Nodes, Unlinked Links and Dead-end pipes, which have some of the special features as listed in 16.3.4, are not selected automatically for removal. Selection of such nodes and pipes can only be done manually. Reconnection of connected elements, such as demands and catchments is not possible.

Trimming

Trimming is associated with the process of removing network parts outside an area of interest, e.g. larger network branches. Connected elements, such as catchments or demands to those network parts that are trimmed off, will be reconnected to remaining network elements based on user specification.

Merging

Frequently, flow-through nodes in network branches appear as redundant and can be removed without any important loss of accuracy. Removing interior nodes is associated with merging the connected links into a single link with similar hydraulic properties equivalent to the original links.

Connected elements, such as catchments or water demands to those nodes that are removed, will be reconnected to remaining network elements based on user specifications.

Nodes and links, which have some of the special features as listed in 16.3.4, are not selected automatically for removal. Selection of such nodes and pipes can only be done manually.

16.3.3 Selection of model area of interest

The simplification of a model can be limited to a certain area of interest.

The options for specifying this are given below

- Complete Network (default)



- Network inside polygon - select existing polygon in map
- Network inside a polygon - manually digitize polygon
- Stretch of pipes - select path with flags
- Network based on selection - specify selection
- Network based on attribute - specify selection filter

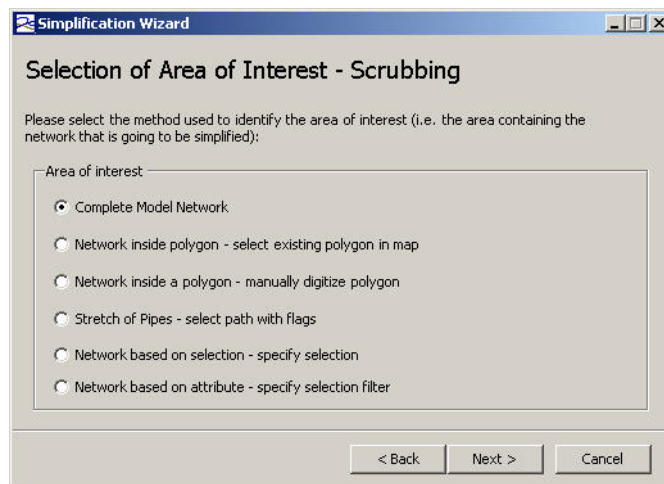


Figure 16.4 Selection of area of interest

For the **'Complete Model Network'** option, the selection of network elements that can be simplified is not limited.

For the **'Network inside polygon - select existing polygon in map'** option, the selection of network elements that can be simplified is limited to those elements inside the selected polygon. For the **'Network inside a polygon - manually digitize polygon'** option, the selection of network elements that can be simplified is limited to those elements inside the digitized polygon.

For the **'Stretch of pipes - select path with flags'** option, the idea is mainly to supply the user with the possibility of simplifying a branch of pipes through merging removing interior nodes only on this branch of pipes.

For the **'Network based on selection - specify selection'** the selection of network elements that can be simplified is limited to those elements inside the defined selection.

For the **'Network based on attribute - specify selection filter'** the selection of network elements that can be simplified is limited to those elements that fulfil the selection filter.

Additionally to these options, the simplification will as other tools in MIKE URBAN only work on the current selection - so the network area of interest



can be limited further based on some user specified selection updated manually on a later page.

After selecting the area of interest it is possible to specify elements that should be excluded from the simplification. The user required network elements are specified through a selection file. The user required network elements are only applicable during scrubbing and merging.

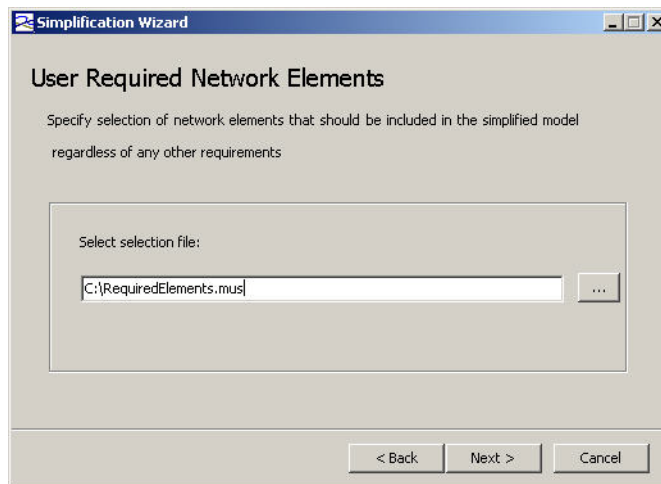


Figure 16.5 User required network elements

16.3.4 Specification of criteria for elements to be selected

Different criteria are available for the different simplification methods.

Scrubbing

For Scrubbing four different criteria can be set.

1. Remove unlinked nodes
2. Remove unlinked pipes
3. Remove dead-end pipes shorter than
4. Remove pipes with diameter less than

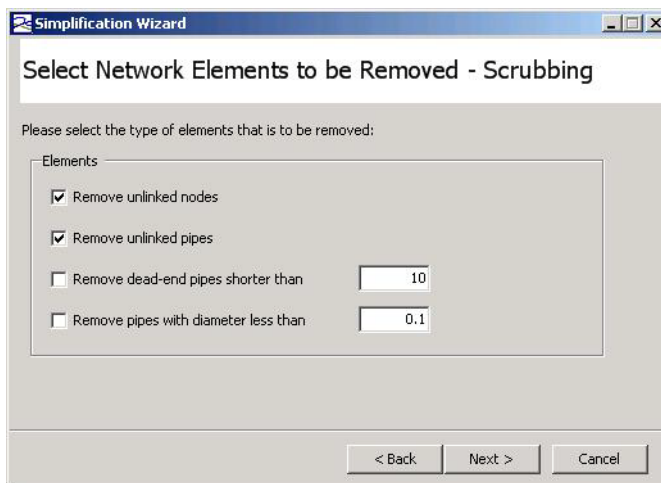


Figure 16.6 Selection of network elements to be removed

The above criteria are identical for both Collection System and Water Distribution network models.

Unlinked Nodes, Unlinked Pipes and Dead-end pipes, which have the features as listed in Table 16.1, are **not** selected automatically for removal. Selection of such nodes and pipes can only be done manually.

Table 16.1 Features for unlinked nodes, unlinked pipes and dead-end pipes in CS MOUSE, CS SWMM and Water Distribution

	Dead-end node features	Dead-end pipe features
CS MOUSE	Basin Outlet Storage Node Assigned a Network load Nodes containing pumps, weirs and orifices/gates	Additional Regulation Non-return valve Assigned a Network Load



Table 16.1 Features for unlinked nodes, unlinked pipes and dead-end pipes in CS MOUSE, CS SWMM and Water Distribution

CS SWMM	Storage Unit Outfall Flow Divider Junctions containing pumps, weirs and orifices Assigned a Network load	Flap Gate
Water Distribution	Tank Air-Chamber Emitter Air-Valve	Pump Valve Closed pipe Check-valve

The network elements that fulfil the criteria are selected for removal.

Trimming

Trimming will remove all network parts inside the area of interest.

Merging

For Merging there are different selection criteria for Collection System and Water Distribution networks.

Collection System

- Similar Size
- Similar Pipe Invert Level
- Same Material
- Similar Slope (slope diff.) applied at slopes lower than specified limit
- Similar Slope (diff. as %) applied at slopes higher than specified limit
- Similar Friction Loss
- Similar Pipe Direction



Simplification Wizard

Selection of Merge Parameters Simplification - CS

Please select the parameters for merging of pipes and removing interior nodes. All selected criteria must be fulfilled in order to merge the pipes:

Merge Criteria

Criteria	Max Difference:	Criteria	Max Difference:
<input checked="" type="checkbox"/> Similar Size	10 %	<input checked="" type="checkbox"/> Similar Friction Loss	10 %
<input checked="" type="checkbox"/> Similar Pipe Invert Level	0.1	<input checked="" type="checkbox"/> Similar Pipe Direction	10 °
<input checked="" type="checkbox"/> Same Material			
<input checked="" type="checkbox"/> Similar Slope (slope diff.)	10	applied at slopes lower than:	1000
<input checked="" type="checkbox"/> Similar Slope (diff. as %)	10	applied at slopes higher than:	0

Merge Method

Size: Friction Loss:

< Back Next > Cancel

Figure 16.7 Merge criteria and methods for CS

The pipe direction is defined as shown in Figure 16.8.

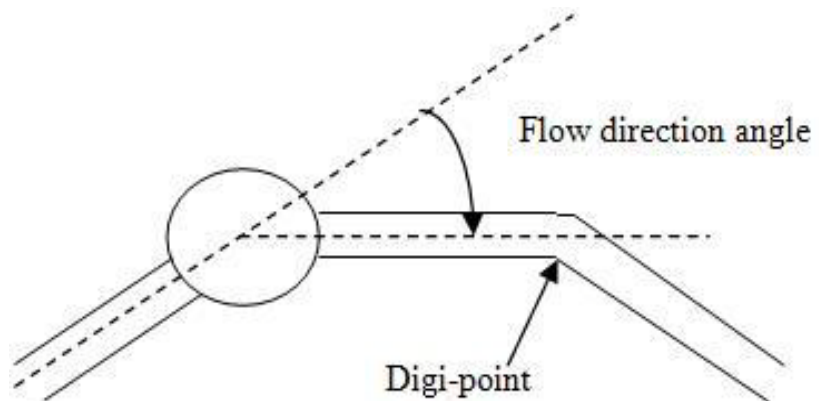


Figure 16.8 Definition of pipe direction angle

Water Distribution

- Similar Diameter
- Similar Friction
- Similar Age
- Same Material

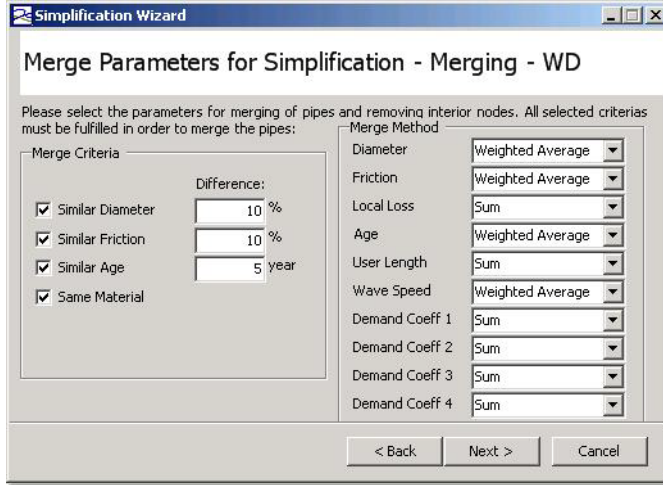


Figure 16.9 Merge criteria and methods for WD

Nodes and links, which have the features as listed in Table 16.2 are not selected automatically for removal. Selection of such nodes and pipes can only be done manually.

The percentage criteria will be evaluated relative to the lowest value in order to have the most conservative evaluation. This means that if e.g. two pipes are evaluated for similar diameter and the first pipe has a diameter of 0.20 and the other 0.25 they will only be merged with criteria higher than 25%.

Table 16.2 Features for nodes and links in CS MOUSE, CS SWMM and Water Distribution

	Node features	Link features
CS MOUSE	Basin Outlet Storage Node Assigned a Network load Nodes containing pumps, weirs and orifices/gates Nodes connecting links specified with different directions Assigned a Network load	Additional Passive Regulation and RTC Non-return valve Assigned a Network Load



Table 16.2 Features for nodes and links in CS MOUSE, CS SWMM and Water Distribution

CS SWMM	Storage Unit Outfall Flow Divider Junctions containing pumps, weirs and orifices Nodes connecting links specified with different directions Assigned a Network load	Flap Gate
Water Distribution	Tank Air-Chamber Emitter Air-Valve Nodes connecting links specified with different directions	Pump Valve Closed pipe Check-valve

16.3.5 Specification of merge methods - valid only for Merging

When the elements fulfilling the merge criteria have been selected then considerations have to be made about how the selected elements should be merged. The merge methods are different for Collection System and Water Distribution. Only pipes of identical shape can be merged.

Table 16.3 Collection System merge methods (MOUSE and SWMM)

Merge methods	CS MOUSE and SWMM
Size (size of merged pipe)	Weighted Average - average pipe size weighted by length used as merged size Minimum - minimum size used as merged size Maximum - maximum size used as merged size
Friction Loss	Weighted Average - average friction loss weighted by length used as merged friction loss Minimum - minimum friction loss used as merged friction loss Maximum - maximum friction loss used as merged friction loss



Table 16.4 Water Distribution merge methods

Merge methods	Water Distribution
Diameter	<p>Weighted Average - average pipe diameter weighted by length used as merged diameter</p> <p>Minimum - minimum diameter used as merged diameter</p> <p>Maximum - maximum diameter used as merged diameter</p>
Friction	<p>Weighted Average - average friction loss weighted by length used as merged friction loss</p> <p>Minimum - minimum friction loss used as merged friction loss</p> <p>Maximum - maximum friction loss used as merged friction loss</p>
Local Loss	<p>Sum - sum of local losses</p> <p>Weighted Average - average local loss weighted by length used as merged local loss</p> <p>Minimum - minimum local loss used as merged local loss</p> <p>Maximum - maximum local loss used as merged local loss</p>
Age	<p>Weighted Average - average pipe age weighted by length used as merged age</p> <p>Minimum - minimum age used as merged age</p> <p>Maximum - maximum age used as merged age</p>
User Length	Sum - sum of user lengths
Wave Speed	<p>Weighted Average - average wave speed weighted by length used as merged wave speed</p> <p>Minimum - minimum wave speed used as merged wave speed</p> <p>Maximum - maximum wave speed used as merged wave speed</p>
Demand Coeff1	<p>Sum - sum of Demand Coeff1</p> <p>Weighted Average - average Demand Coeff1 weighted by length used as merged Demand Coeff1</p> <p>Minimum - minimum Demand Coeff1 used as merged Demand Coeff1</p> <p>Maximum - maximum Demand Coeff1 used as merged Demand Coeff1</p>



Table 16.4 Water Distribution merge methods

Demand Coeff2	<p>Sum - sum of Demand Coeff2</p> <p>Weighted Average - average Demand Coeff2 weighted by length used as merged Demand Coeff2</p> <p>Minimum - minimum Demand Coeff2 used as merged Demand Coeff2</p> <p>Maximum - maximum Demand Coeff2 used as merged Demand Coeff2</p>
Demand Coeff3	<p>Sum - sum of Demand Coeff3</p> <p>Weighted Average - average Demand Coeff3 weighted by length used as merged Demand Coeff3</p> <p>Minimum - minimum Demand Coeff3 used as merged Demand Coeff3</p> <p>Maximum - maximum Demand Coeff3 used as merged Demand Coeff3</p>
Demand Coeff4	<p>Sum - sum of Demand Coeff4</p> <p>Weighted Average - average Demand Coeff4 weighted by length used as merged Demand Coeff4</p> <p>Minimum - minimum Demand Coeff4 used as merged Demand Coeff4</p> <p>Maximum - maximum Demand Coeff4 used as merged Demand Coeff4</p>

16.3.6 Reconnection of connected Catchments and Demands

For both Trimming and Merging connected Catchments and Demands are reconnected to the remaining network. How the catchments and demands should be moved is specified by the user. The following choices exist:

Collection Systems, MOUSE and SWMM

- Move to upstream node along the path
- Move to downstream node along the path
- Move to closest node along the path

Water Distribution

- Move to closest node along the path

Please note that moving to the closest node along the path will move the catchment or demand to the closest along the upstream or downstream path of the currently connected node. If no such path is found the catchment/demand will be disconnected.

After choosing the area of interest connected elements to parts of the network that are now removed will be reconnected to remaining elements based on user specification, see e.g. Figure 16.10.

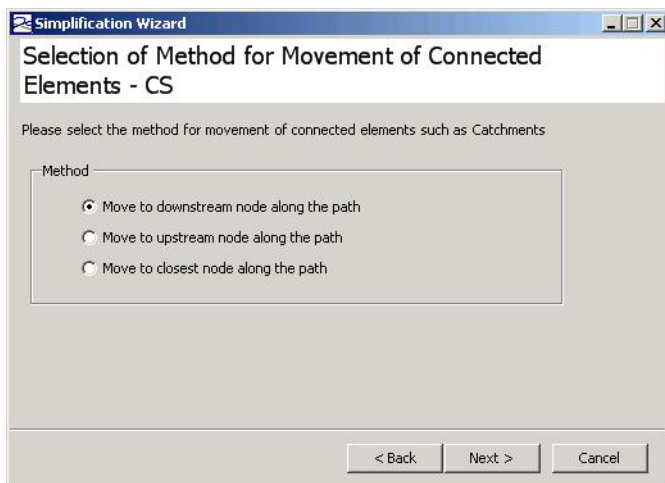


Figure 16.10 Reconnecting connected elements

16.3.7 Save Options

When working on a MOUSE network it is possible to create a result selection based on the remaining network. This will include e.g. all the nodes which are included in both the original model and the simplified model. The functionality can be used to create identical result files from both the original and the simplified model used for verification.

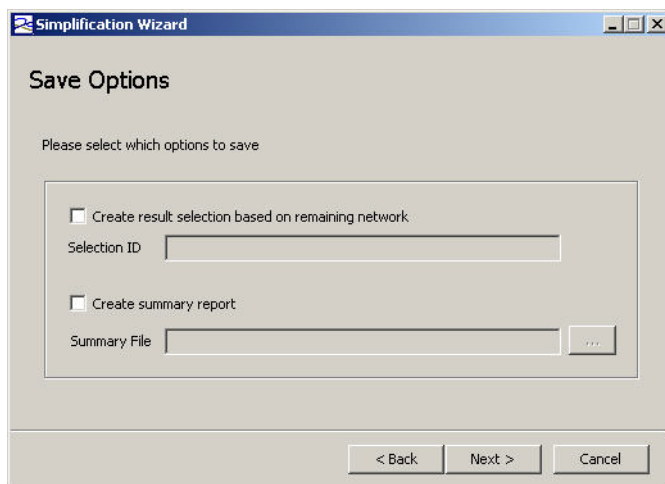


Figure 16.11 Save options



Additionally, it is possible to create a summary report, which will include information on which modifications were done to the network during the simplification.

Summary reports from merging simplifications will further include information on the calculation of each of the criteria. All nodes fall under 3 categories:

- **EXCLUDE - Network Constraint:** These nodes cannot be removed due to a network constraint in the network. This can be a node connected to three pipes, connections to structures etc. These are marked with blue in the report.
- **EXCLUDE - Criteria:** These nodes cannot be removed since one or more of the specified criteria have been exceeded. These are marked with red in the report.
- **DELETE:** These nodes are removed in the simplification since none of the specified criteria are exceeded. These are marked with green in the report.

Status	Node ID	Similar size	Similar Slope(slope diff.)	Similar Slope(diff.%)	Simili
		10%	10	10%	
EXCLUDE - Criteria	1	0		128,7955	
EXCLUDE - Criteria	10	0	0,02429546		
EXCLUDE - Criteria	100	0	0,898275	121,794	
DELETE	101	0	0,09694254		
EXCLUDE - Network Constraint	102				
EXCLUDE - Network Constraint	103				
EXCLUDE - Network Constraint	104				
EXCLUDE - Network Constraint	105				
EXCLUDE - Network Constraint	106				
EXCLUDE - Network Constraint	107				
EXCLUDE - Network Constraint	108				
EXCLUDE - Network Constraint	109				
DELETE	11	0	0,1156906		
EXCLUDE - Network Constraint	110				
EXCLUDE - Criteria	111	0	0,01456326		
EXCLUDE - Network Constraint	112				
EXCLUDE - Network Constraint	113				

Figure 16.12 Part of summary report for merging simplification

16.3.8 Manually editing of selection

Scrubbing and Trimming

The selected elements by the Scrubbing and Trimming process are the elements that will be removed. This selection can be changed in the horizontal plan manually by adding and removing from the selection in the horizontal plan using the general selection tools or by utilizing a .mus file. This can be done at any time in the wizard. When reaching the final page in wizard and you wish to make changes to the selection, then go to the horizontal plan, do the changes and then press 'Update' on the wizard page to include these

changes, see Figure 16.13. It is only when clicking Finish on the last page action will be taken.

Merging

The selection for Merging, represents the data that are valid for Merging actions such as removed, merged etc. The selection can be changed in the horizontal plan manually by removing (not adding) from the selection using the general selection tools or by utilizing a .mus file. This selection can again be changed by the user in the horizontal plan - i.e. removing nodes from the selection that are wished to be kept in the model - e.g. nodes where a flow meter has been installed. When clicking Finish, the merging process on the data is performed.

Please note that the entire simplification process is done in an Edit Session thus the undo functionality is available after completing the wizard.

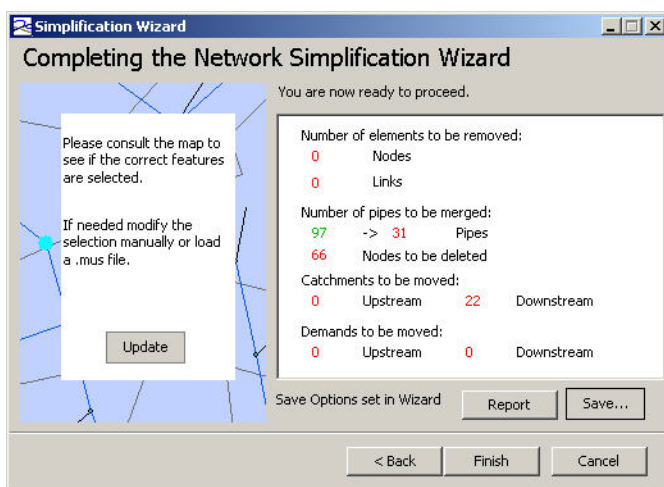


Figure 16.13 The final page in the simplification wizard

16.3.9 Multiple Configurations

If previous simplification settings have been saved and should be executed on the current model without further editing it is possible to use the tick mark option on the first page of the wizard.

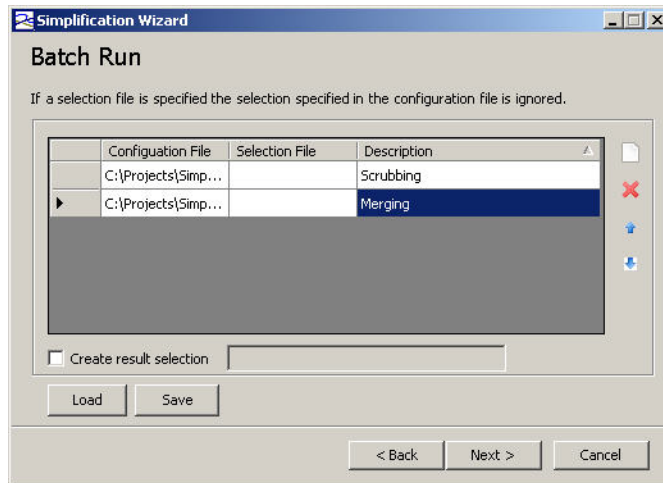


Figure 16.14 Batch Run

On the batch run page a list of configurations can be specified. They will be executed in the order listed such that the simplification in the second row will be applied to the network resulted from the simplification specified in the first row.

The area of interest is specified within the configuration file. However, if another area is needed the area of interest can be updated by specifying a selection file.

The option of saving a result selection (see 16.3.7) in a configuration is disabled running in batch. However on the batch page it is possible to specify a result selection including the elements that exists both in the original network and in the simplified network after all simplification configurations have been executed.





17 Scenario management

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

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

Analysis of the scenarios as separate projects creates major inconveniences, such as:

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

In other words, working with the scenarios as separate projects is inefficient and cumbersome. Instead, the MIKE URBAN Scenario Manager provides an easy way of managing the multiple scenarios, within a single MIKE URBAN project (i.e. single geo-database).

17.1 What is a Scenario Manager

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



- Augmentation of existing trunk sewer mains;
- Increased wastewater loading from increased population;
- Increased water demands from increased population;
- Alternative design loads, e.g. rainfall-runoff of different return period;
- Alternative alignment of sewer and storm mains;
- Building of a new sewer trunk and water supply mains in order to cater for a new development area;
- Etc.;

within the same MIKE URBAN project.

With MIKE URBAN Scenario Manager, user can work with an unlimited number of scenarios in a single MIKE URBAN project.

17.2 Design of the MIKE URBAN scenario manager

17.2.1 Data Groups, Alternatives and Scenarios

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

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

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

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

The MIKE URBAN Scenario Manger is accessed by 'Tools|Scenario Manager'.



17.2.2 Alternatives

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

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

A right-click on a certain alternative will display the list of editors belonging to that alternative. The list serves as a shortcut to these editors, i.e. clicking on a certain editor name opens the respective editor.

Collection System Alternatives

For Collection Systems, the scenarios are composed of the following data groups (see also Figure 17.1):

- Asset
- Catchment Connection
- Catchment
- Hydrological
- Loads and Boundary
- LTS
- Network
- Operational
- Profiles and Curves
- WQ

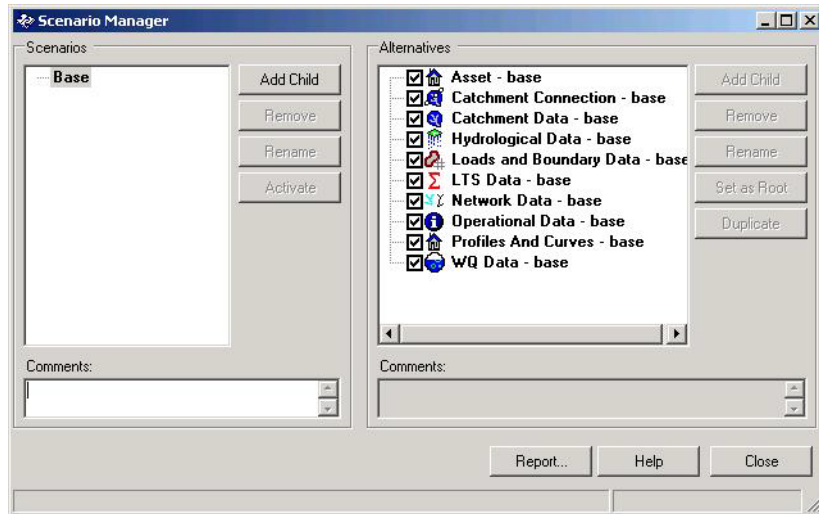


Figure 17.1 Alternative groups for a Collection System

Water Distribution Alternatives

For Water Distribution Systems input data can be grouped the following way, corresponding to the different types of available alternatives (see also Figure 17.2):

- Asset
- Computational
- Control Rules
- Cost and Energy
- Demand Allocation
- Demands
- Extended
- Fireflow
- Miscellaneous
- Network
- Profiles and Curves
- Water Quality

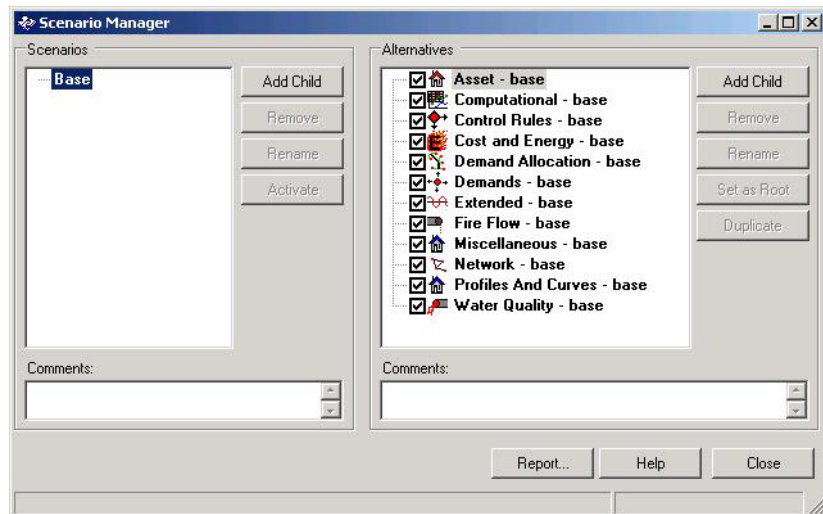


Figure 17.2 Alternative groups for a Water Distribution system

An easy overview over the changes made to scenarios and alternatives are provided through different reports of the changes.

17.2.3 Base data vs. child data

When the scenario manager is activated for the first time, the system provides 'Base' alternatives for each data group automatically. The 'Base' data is the root of all the alternative trees.

A 'Base' alternative for any data group can be empty, if no data are specified in any of the tables belonging to this data group. E.g. no operational data may be specified to begin with, thus leaving the Operational Data 'Base' alternative empty. So, although the Operational Data is a part of the 'Base' scenario, it does not necessarily mean that any operational data are specified. It is possible to add a 'child' to the Operational Data 'Base' alternative containing operational data and include this alternative in a new scenario. This way, the scenario containing operational data can be tested and the reports of the changes will reflect that the operational data have been changed in the 'child'.

There may be many reasons for adding child alternatives. E.g. it can be for testing performance of the system if the diameters for certain pipes are upsized, testing the effects of the population growth or testing the effects of applying different real time control strategies.

When making a scenario active and starting to edit the data, all the alternatives that are a part of the scenario are automatically made active and can thus be edited. The title bar of each dialog will state the name of the alternative edited in that dialog as well as the title bar of the MIKE URBAN application will display the scenario name.



When the best suited alternative has been found for a given system it is possible to merge the changes from the chosen alternative to the 'Base' alternative.

17.2.4 Inheritance principles

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

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

17.2.5 Data not specific to any alternative/scenario

There are some data tables which are not included in the Scenario Manager. These are typically tables containing data of general usability, i.e. data without a reference to the current network - e.g. in MIKE URBAN CS these include cross sections, parameter sets, etc. These data should be understood as belonging to a general project database.

The exception are single-record tables containing various parameters (e.g. water quality parameters). They do take part in Scenario Manager, in order to allow for applying various parameters within the same project.

The data not included in the Scenario Manager can be accessed from any scenario, regardless of the alternatives that make up that specific scenario.



Please note that the computed values are not part of Scenario Manager (all fields ending with _C) and are not automatically re-computed after switching scenarios.

17.3 Managing Scenarios and Alternatives

The Scenario Manager has two parts:

- The Scenario part
- The Alternatives part

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

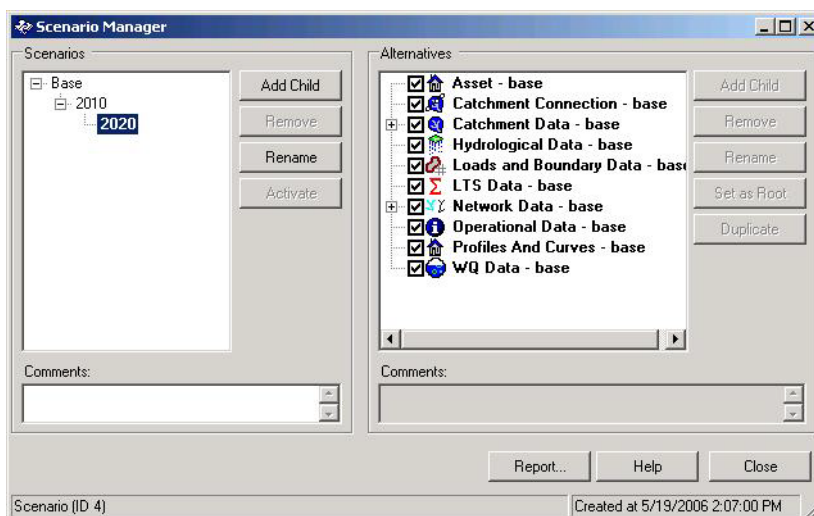


Figure 17.3 Example of the scenario page for the scenario manager

17.3.1 Scenarios

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

Add Child,

The add child button adds a scenario that is a child of the highlighted (not to be confused with the active/current scenario), i.e. to begin with the alternatives of a new scenario will be that of the highlighted scenario. A name for the new scenario is suggested by default. The name can be changed either by using the rename button or by left-clicking on the name once and then editing the name.



Remove,

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

Rename,

The rename button will make the scenario name active so it can be easily renamed. The same functionality is obtained by left-clicking on the scenario name.

Activate,

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

17.3.2 Alternatives

Alternatives can be edited only if the appropriate scenario is made active. Alternatives can, however, be added regardless of the active scenario. When a scenario is loaded, the project data is manipulated so that all editors contain the appropriate data. The title bar of each dialog will display the name of the alternative that is currently being displayed and edited.

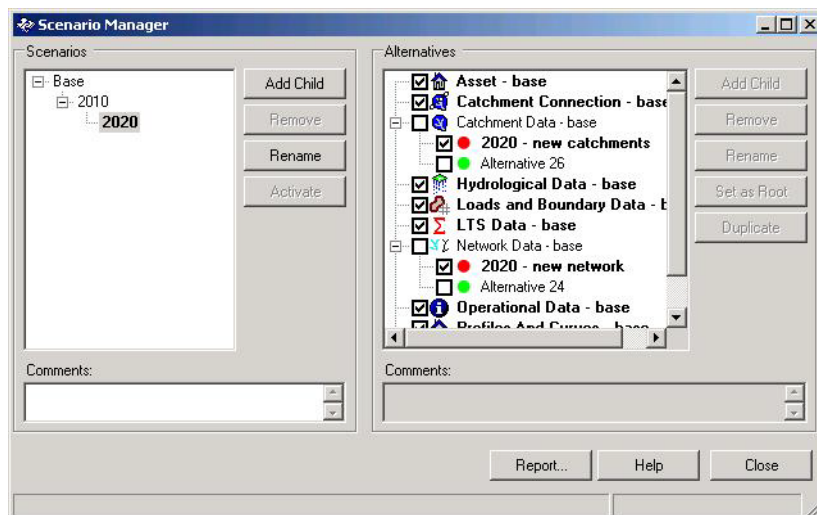


Figure 17.4 The alternatives applied in the active scenario are displayed in bold

The alternative part of the dialog consists of a number of buttons along the right side. The alternatives that are referenced from the active scenario are displayed in bold. By right-clicking on the active alternative a local menu opens that provides a short-cut to all the editors related to that alternative, see Figure 17.5

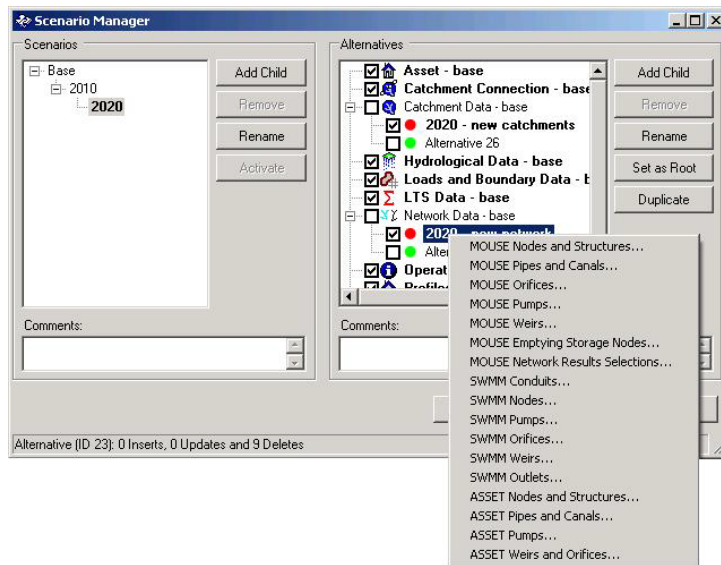


Figure 17.5 The local menu of alternatives in the active scenario provides direct access to the editors containing data of that alternative

Add Child,

The add child button adds an alternative that is a child of the highlighted (not to be confused with the active/current alternative). A name for the new alternative is suggested per default. The name can be changed using either the rename button or by left-clicking on the name once and then editing the name.

Remove,

The remove button will delete the highlighted alternative. The Base alternative cannot be deleted. Remember: Deleting an alternative will delete the changes made to that alternative.

Rename,

The rename button will make the alternative name active so it can be easily renamed. The same functionality is obtained by left-clicking on the scenario name.

Set as Root,

Can only be executed from the active alternative. The data belonging to the active alternative will be used as the base alternative and only children to that alternative will remain. All other alternatives in the specific data group will be removed.



Duplicate,

The duplicate button will make a duplicate of the highlighted alternative. This means that all the changes made to the highlighted alternative will be transferred to the new alternative. Once the new alternative has been made, the original and the duplicate alternative are edited independently of one another.

17.3.3 Example

If e.g. you would like to investigate how an upsizing of certain pipes and adding some real time control to the system can affect the performance of the system, start by making two child alternatives: one for the network data (as the pipes are a part of this alternative) and one for the operational data (as the real time control is a part of that alternative). Then, create a scenario that contains the new network alternative and the new operational data alternative and activate this scenario. Then, start editing the data (e.g. upsizing the pipes and adding real time control). Observe that the editors indicate that the active alternative has changed from 'Base' to the alternative you just created. Once the data is edited, you can perform a simulation to test the effect of the changes.

You can also choose to make a new scenario that contains e.g. the network alternative (but not the operational data alternative), to see what change in performance the upgrading of the pipes alone will have. And so on - the combinations are endless.

17.3.4 Reporting of the changes

When making all these alternatives and scenarios one of the most important aspects is keeping track of the changes that have been done. In MIKE URBAN a number of very informative reports are available for tracking and documenting the changes made in the different scenarios and alternatives. All reports can be produced in colors or in black/white. The reports are all in xml format.

Basically two different types of reports exist: Reports that display the data content of a scenario or alternative and difference reports that display the changes between two scenarios or alternatives.

By local data below, we refer to data that is modified in the current alternative and thus is defined locally for that alternative.

Data included in scenario/Data included in alternative

- For a scenario: This will report the entire content of all the alternatives belonging to that scenario, displaying local as well as inherited data.



- For an alternative: This will report the content of the alternative, local as well as inherited data.

Difference between two scenarios

This will create a difference report between the two scenarios chosen. It will also report the deleted records.

Color coding is used to signify the origin of the record:

- White - original record, no changes
- Green – record added
- Blue – record has been changed (updated)
- Read - record has been deleted

Setting the tickmark 'Include sequence of modifications' will trace all the changes done to a specific record. Setting the 'Include all data' will also include records that have not been changed.

Difference between two alternatives

This will create a difference report between the specified alternatives for the data group chosen. It will also report the deleted records.

The color coding used is the same as in the difference reports for scenarios.

Setting the tickmark 'Include sequence of modifications' will trace all the changes done to a specific record. Setting the 'Include all data' will also include records that have not been changed.

Scenario hierarchy

Will create a report with the scenario hierarchy.

Alternative hierarchy

Will create a report with the alternative hierarchy.

Customizing the reports

The design of the reports generated can be customized under 'Settings', see Figure 17.6. The report style can be either 'DHI Default' (with the color coding mentioned in the above), 'Printer friendly' (suited when printing the report afterwards) and 'None' (no style-sheet is applied, only an xml-file is shown).

It is also possible to customize how empty fields are displayed in the report. Two types of empty records exist:

- If the attribute for a given record has no value.
- If the attribute has not changed (in difference reports)

The default implementation uses 'NULL' to indicate empty fields in the database and 'NOTHING' to indicate no changes. Both strings can be configured, e.g. left blank.

If entire tables are empty these can be either shown or omitted (by setting the radio button under 'Show empty tables').

Comments can be added in the heading of the report by writing in the 'Comments' field.

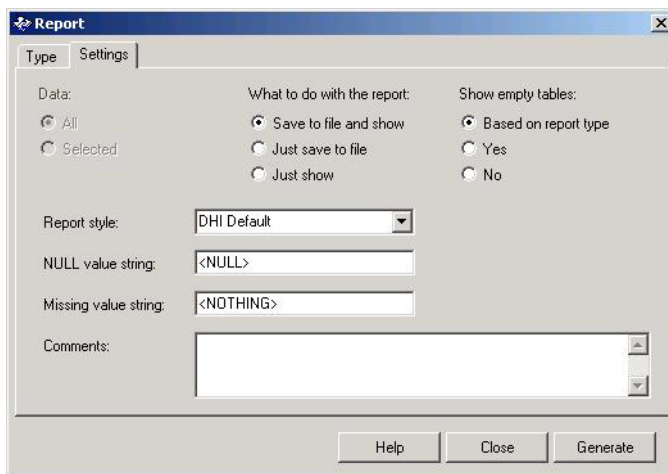


Figure 17.6 Customizing the reports

17.4 How to Start Working with Scenarios?

17.4.1 Creating alternatives and scenarios

- Go to Tools|Scenario Manager
- Create a child scenario by pressing 'Add Child'
- Select the alternative group that you wish to add an alternative to and press the 'Add Child' button
- You can now rename it and/or continue to make alternatives
- Once you have created the alternatives that you need, highlight the scenario you created and put tickmarks next to the alternatives that you wish to include in the scenario
- Finally, activate the scenario that you wish to work with (press 'Activate' button).

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

Upon activation of a certain scenario (which may take a while), the caption of each editor would display the name of the current alternative. So, the user has a full control over the data he edits at any time.



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





18 Climate Change Scenarios

18.1 Introduction

The Climate Change Tool is designed to generate future climate change scenarios based on existing setups. The climate change tools utilizes results published by the Intergovernmental Panel for Climate Change (IPCC). The main results used from the IPCC data are changes in air temperature and precipitation as a function of CO₂ emission rates and prediction year.

The tool is found under Tools|Climate Change Tool.

The functionality of the climate change tool is based on the user selecting a limited number of input parameters for a future climate change scenario. The editor is then used to construct a future scenario based on the base scenario supplied by the user with modified input for air temperature, precipitation and potential evapo-transpiration. The parameters that the user needs to supply are namely

- One or more global circulation models (a climate model for the atmosphere) also referred to as GCMs.
- A CO₂ emission scenario
- A future prediction year

Based on these values a climate change scenario setup will be generated and the results may be compared with the results of the unaltered setup.

The overall work flow is

1. Select the climate scenario year (values up to the year 2100 will typically be covered by the data).
2. Select the emission scenario to be modelled
3. Choose one or more GCMs for which the modification factors are to be used.
4. Click the 'Generate default delta change values' button
5. Inspect the modification factors for precipitation, evaporation and temperature in the subsequent pages
6. Modify the modification factors if needed
7. If satisfied with the values click "Generate Climate Change Scenario" and a dialog will confirm that the setup has been generated
8. Run the model and compare with the unaltered setup results

18.2 Model Location

For the climate change tool to select the correct modification factors the geographic location must be specified. See the figure below. By default the mid point of the model area in the setup is chosen. These values will be displayed in the latitude and longitude fields. If you are not satisfied with these values or if your setup does not hold geographic information (using a NON-UTM as projection) specific values must be supplied.

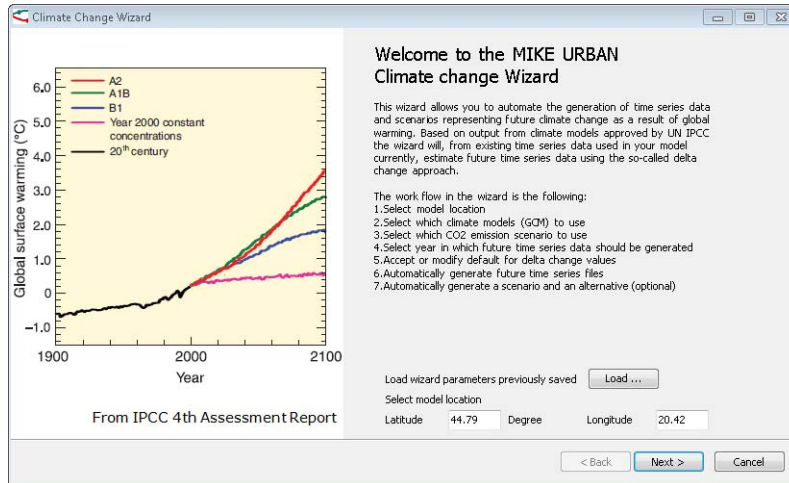


Figure 18.1 Specification of model location

Additionally, if settings for this tool have been saved during previous session such setting can be loaded by clicking the "Load" button.

18.3 Choosing the Climate Model and CO2 Emission Scenario

The next step is to choose the climate model and the CO2 emission scenario (Figure 18.2).

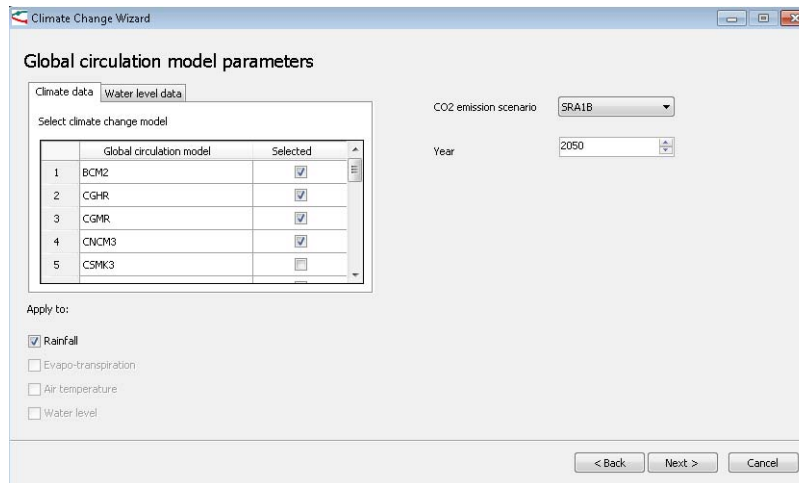


Figure 18.2 Selection of climate change parameters

Here the choice of the following parameters is made:

- Climate data: GCM models. The GCM models for which output is available is listed and the user can choose which models to include.
- Water level data: Sea level rise models. The sea level rise models for which output is available is listed and the user can choose which models to include. Please note that this part will only be active if time varying (i.e. time series) are used for the water level. A constant water level will not be modified.
- CO2 emission scenario. One of the three CO2 emission scenarios SRA1B, SRA2 and SRB1 has to be selected.
- Year. The user will have to choose the year within the range 2020-2100 for which the climate change prediction is to be made.
- Apply to. The user specifies which of the data types rainfall, air temperature, evapo-transpiration and water level should be included in the climate change scenario to be generated. The possible choice of data type is limited to the data types actually represented in the model currently in use.

The scientific documentation for the climate change tool provides further details and references regarding GCM and emission scenarios.

18.4 Delta Change Values

Delta change values for rainfall

Default value

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec
1	1.01...	0.98...	1.09...	1.01...	0.93...	0.93...	0.86...	0.87...	0.89...	0.97...	0.95...	1.03...

Apply global value

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec
1	1.01...	0.98...	1.09...	1.01...	0.93...	0.93...	0.86...	0.87...	0.89...	0.97...	0.95...	1.03...

Apply individual values

Boundary ID	Item Name	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec
1	CDS5	1.01...	0.98...	1.09...	1.01...	0.93...	0.93...	0.86...	0.87...	0.89...	0.97...	0.95...	1.03...
2	CDS2	1.01...	0.98...	1.09...	1.01...	0.93...	0.93...	0.86...	0.87...	0.89...	0.97...	0.95...	1.03...
3	zero_rain	1.01...	0.98...	1.09...	1.01...	0.93...	0.93...	0.86...	0.87...	0.89...	0.97...	0.95...	1.03...
4	Petlovo Brdo	1.01...	0.98...	1.09...	1.01...	0.93...	0.93...	0.86...	0.87...	0.89...	0.97...	0.95...	1.03...

< Back Next > Cancel

Figure 18.3 Delta change values

The page shown above will be repeated for each of the data types (rainfall, air temperature, evapo-transpiration) selected on the previous page.

In the upper part of the page the default delta change values will be displayed. These values are calculated from the GCM output selected on the previous page and the values cannot be edited.

In the middle part of the page the user can choose to apply the same delta change values for all time series data used in the model. Default values can be applied by clicking the button "Use default" and subsequently the values can be edited.

In the lower part of the page the user can choose to apply individual delta change values for each time series data used in the model. Default values can be applied by clicking the button "Use default" and subsequently the values can be edited.



18.5 Creating Climate Change Scenarios

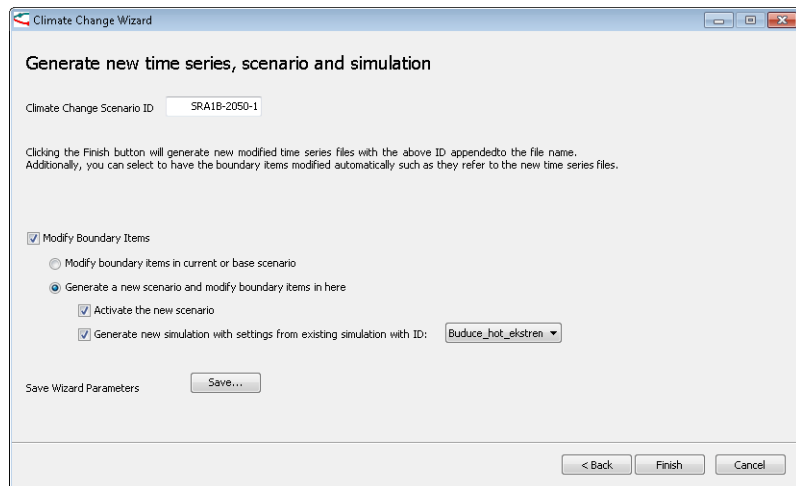


Figure 18.4 Creating climate change scenarios

On the page shown above the user makes the final choices before clicking the "Finish" button upon which the tool will generate the climate change scenario. This always includes that the tool will make a copy of the time series data files involved and it will modify the copied time series data according to the delta change values selected on the previous page.

In addition to modifying the time series data the user can choose to have the boundary items modified such that these refer to the modified time series data files. This modification of the boundary items can be done either in the current scenario or the user to choose to have a new scenario generated in which the boundary items are modified automatically by this tool.





19 Project Check Tool

19.1 Introduction

The project check tool will check the consistency of the MIKE URBAN project database against a large number of rules. The rules are combined so that both the data structure as well as the values of attributes are checked in order to ensure an error free model.

19.2 Launching the Tool

The tool is found under Tools|Project Check Tool. When activated the dialog shown in Figure 19.1 comes up:

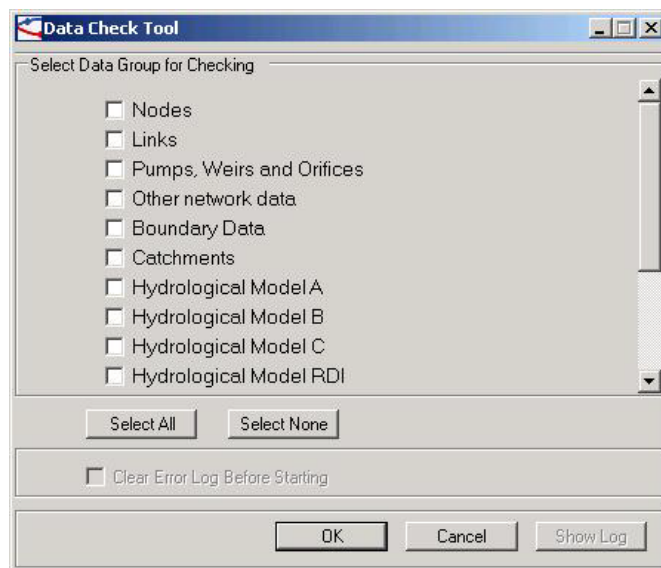


Figure 19.1 The Project Check dialog

The various data checks are divided into groups that can be selected individually from the list. The contents of the list varies depending upon the current modelling mode of MIKE URBAN.

If an existing error log is present in the database an option allows for clearing the log before starting the check tool.

In case you just want to return to the existing list without performing any new checks just press the button 'Show Log'

The check is started is started by pressing the 'OK' button.



19.3 Running the Check

The project check will run for a while depending upon the database size, number of selected check groups and number of errors actually found. Expect an execution time between approximately 1 minute to several minutes.

If the option 'Show Log' is chosen the initialisation will still take up to a couple of minutes.

When the check is finished the resulting error log will show.

19.3.1 Computed values and topology

Please note that errors/warnings related to computed values (such as e.g. slope) will be based on the computed values found in the database. This means that if you have moved e.g. a node up or down and haven't recomputed the slope afterwards then the project check tool will be based on the previously computed slope.

Errors corrected by changing the topology of the network (i.e. by moving pipe or nodes or changing the connections) will not be detected by the 'Rerun' function. This means that even after pressing re-run the error will still be flagged in the list even though it has been corrected. You will need to restart the tool in order to have this type of errors/corrections checked.

19.4 Using the Error Log

The error log window is a list of the encountered errors and warning as seen in Figure 19.2.



Data Check List

Verified: All Marked: All E/W Type: All

MUID: E/W No: Feature Class: All

Verified	Marked	E/W	E/W No	MUID	Description	Comment	FeatureClass
✗	<input type="checkbox"/>	E	1201031	Node_1	Nodes: Invert level not specified!		msm_Node
✗	<input checked="" type="checkbox"/>	E	1201031	Node_2	Nodes: Invert level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201031	Node_3	Nodes: Invert level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201031	Node_4	Nodes: Invert level not specified!		msm_Node
✗	<input checked="" type="checkbox"/>	E	1201031	Node_5	Nodes: Invert level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201031	Node_6	Nodes: Invert level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201031	Node_7	Nodes: Invert level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201031	Node_8	Nodes: Invert level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201031	Node_9	Nodes: Invert level not specified!		msm_Node
✗	<input checked="" type="checkbox"/>	E	1201031	Node_10	Nodes: Invert level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201057	Node_1	Nodes: Manhole diameter not specific		msm_Node
✗	<input type="checkbox"/>	E	1201057	Node_2	Nodes: Manhole diameter not specific		msm_Node
✗	<input checked="" type="checkbox"/>	E	1201057	Node_3	Nodes: Manhole diameter not specific		msm_Node
✗	<input type="checkbox"/>	E	1201057	Node_4	Nodes: Manhole diameter not specific		msm_Node
✗	<input type="checkbox"/>	E	1201057	Node_5	Nodes: Manhole diameter not specific		msm_Node
✗	<input checked="" type="checkbox"/>	E	1201057	Node_6	Nodes: Manhole diameter not specific		msm_Node
✗	<input type="checkbox"/>	E	1201057	Node_7	Nodes: Manhole diameter not specific		msm_Node
✗	<input type="checkbox"/>	E	1201057	Node_8	Nodes: Manhole diameter not specific		msm_Node
✗	<input type="checkbox"/>	E	1201057	Node_9	Nodes: Manhole diameter not specific		msm_Node
✗	<input checked="" type="checkbox"/>	E	1201057	Node_10	Nodes: Manhole diameter not specific		msm_Node
✗	<input type="checkbox"/>	E	1201041	Node_1	Nodes: Ground level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201041	Node_2	Nodes: Ground level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201041	Node_3	Nodes: Ground level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201041	Node_4	Nodes: Ground level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201041	Node_5	Nodes: Ground level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201041	Node_6	Nodes: Ground level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201041	Node_7	Nodes: Ground level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201041	Node_8	Nodes: Ground level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201041	Node_9	Nodes: Ground level not specified!		msm_Node
✗	<input type="checkbox"/>	E	1201041	Node_10	Nodes: Ground level not specified!		msm_Node

* Print Remove Fixed Rerun Close

Figure 19.2 The error log

Apart from the list itself the error log window contains fields for filtering the list (top section) and a set of action buttons.

The columns in the list are:

- Verified

The icon shows the current status of the actual row:



Means that an error/warning exist.



Means that the error has been fixed and the fix verified by the tool.



Signifies that a warning was ticked off by the user but that the verify run failed.

- Marked

This column is used by the user to 'tick off' errors that have been fixed.

- E/W



'E' this is an error that will prevent the model from running, 'W' a warning condition exist that may produce incorrect results.

- E/W No

An error number used to categorize the error.

- MUID

This is the ID of the failing record, for the main elements like (pipe, nodes etc.) this is the MUID field of the table, for certain errors in subtables like pump-curves and other tabular data the ID refers to the ID of the attached element.

- Description

A description of the error, the text will usually start with a reference to the editor in which the error should be corrected.

- Comment

A field for comments entered by the user. Can e.g. be used to make notes about clarifications to be made etc. The comments are saved along with the error list and will come up when choosing the 'Show Log' option from the main window.

- Feature Class

This is the database name of the Feature Class / Table in which the error was found.

Below a number of buttons is found:

- Copy to ClipBoard: Allows to copy the list of errors and warnings and paste them into e.g. Excel.
- Print: Prints the list on the default printer
- Remove Fixed: Deletes rows that have been fixed AND verified.
- Rerun: This will rerun all the rules that have been given a tickmark in the marked column. Certain network related errors cannot be verified by 'Rerun' but requires that you close the list and make a complete rerun. This pertains to errors that have been fixed by adding new elements and/or adjusting the network connectivity.
- Close: Will close the list and save it to the database.

19.4.1 Additional functionality

- The list can be sorted by any column by clicking the column header.
- Right clicking in the 'Marked' Column header can mark/unmark all rows.



- Clicking or dragging in the row header will select the corresponding element in the MIKE URBAN map view, making it possible to identify the element to be fixed. The selection only work on errors directly on map element such as pipes, pumps and manholes.

19.5 Typical Use Cases

In a typical session the user will start the tool and perform a new check. Once the error list is shown errors can usually be categorized as individual or general errors.

Individual errors are typically from wrong data entries, whereas the general errors most often stems from an import session with data missing.

Individual errors are typically corrected by using the normal MIKE URBAN editors. For instance using the following procedure:

1. Unselect all elements
2. Click the row header of the error you want to fix, the corresponding element is now selected on the map.
3. Open the relevant MIKE URBAN editor, or use the 'open editor' tool. In both cases will open positioned on the correct record ready for editing.
4. Once edited, save the edits and tick off the error in the marked column.
5. You can now press the Rerun button to verify the fix.

Dealing with general errors can often be done by using the assignment tool in connection with the error list.

1. Use the error list to select all erroneous records.
2. Open the assignment tool and find a the most appropriate way to find the correct data (external layer, interpolation etc.)
3. Use the option of 'Only Assign to Selected Records' to limit the assignment process to the erroneous records.





20 Submodel Manager

20.1 Introduction

The 'Submodel Manager' tool is used to create models where a specified area of interest is detailed and the remainder of the model is simplified.

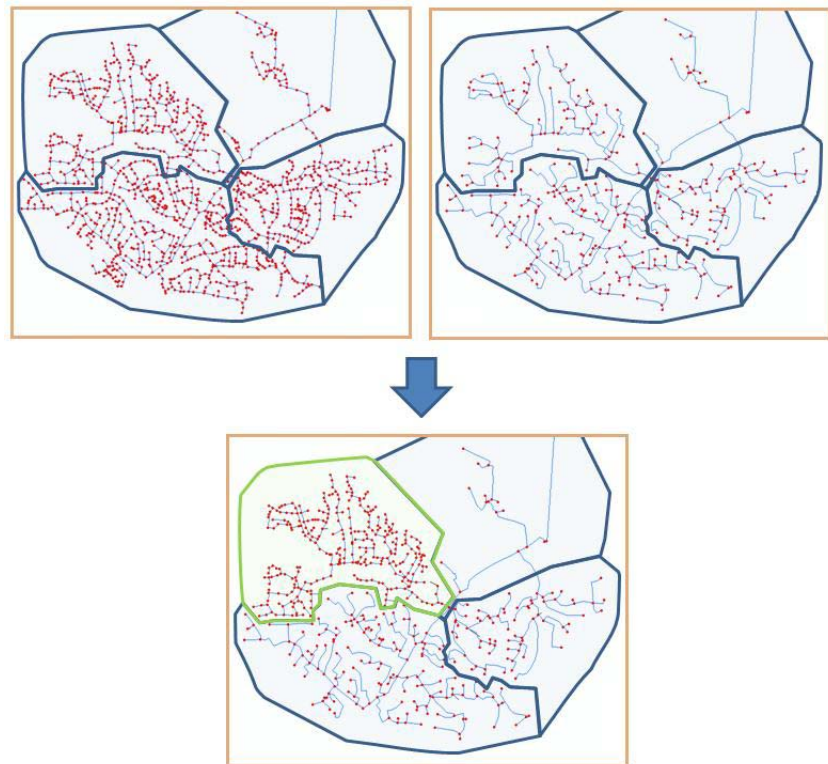


Figure 20.1 Submodel Manager concept

The requirements for the Submodel Manager are a complete detailed and simplified model of the same network. Based on user specified polygons the models are split into submodels of the network. These submodels can be merged to a new model containing one version of all submodels either from the detailed or simplified model in any combination.

The tool is found under 'Tools | User Tools| Submodel Manager' - XXX (depends on implementation). The tool has three main functionalities:

- Create Selection File
- Split Submodels
- Merge Submodels



The three options are detailed in the following sections. They can be executed separately or in sequence. Settings can be saved and loaded again for later execution.

20.2 Create Selection File

Submodels are defined by using polygons to divide the complete model into submodels. Some model elements will cross the borders defined by the polygons. These elements must be identical in both detailed and simplified versions of the submodel parts.

The Simplification Wizard can be configured to leave these elements unchanged in the simplification process. This is done by using a selection file with a list of these elements.

The Submodel Manager can be used for creating this selection file.

'Create Selection File' will create a MIKE URBAN selection file (*.mus) containing the elements that exists in more than one submodel.

The selection file can be included as a selection of user required elements when creating the simplified model using the Simplification Procedure - Wizard (p. 276).

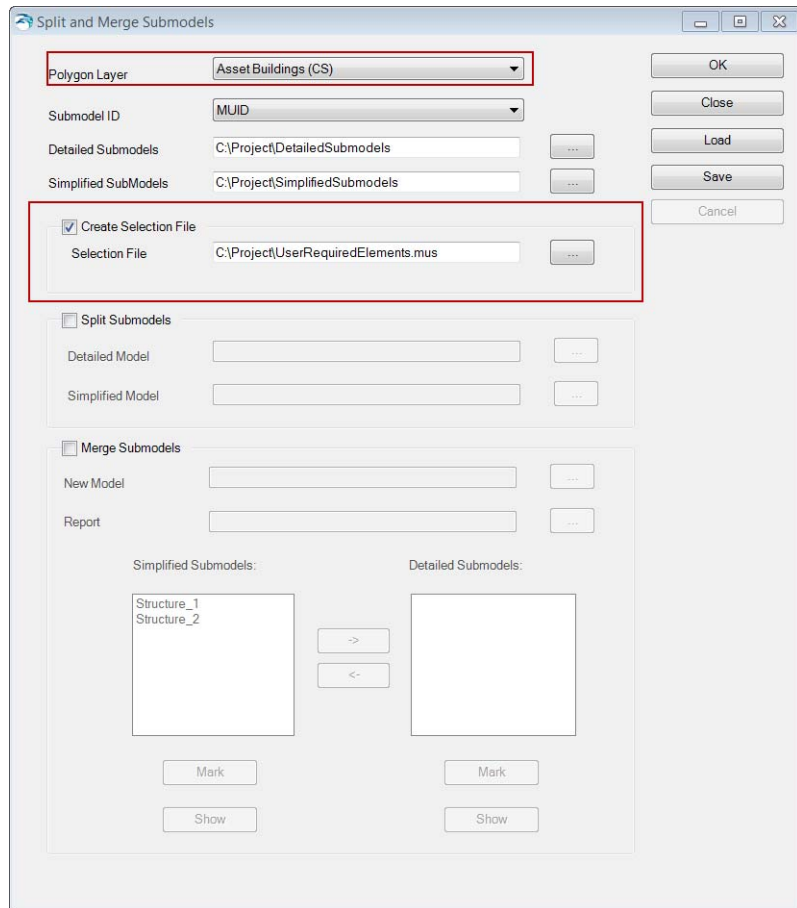


Figure 20.2 Create Selection file

In order to create a selection file the polygon layer with the submodel areas must be specified and the checkbox checked.

20.3 Split Submodels

A model can be split into submodels based on a polygon background layer in MIKE URBAN. Every polygon must contain a unique ID which will be used as reference to the individual submodel.

The background layer must cover the entire model area and the polygons should only intersect with pipes, structures and connections (catchments, demands etc.).

A submodel will contain all nodes (manholes, junctions, etc.) inside the polygon and all elements connected to these nodes. E.g. a catchment located outside the submodel polygon will still be included in the submodel if it is connected to a manhole located inside the submodel area.

In order to split the model into submodels, a polygon layer and submodel ID must be provided to specify the extent and name of the submodel. The directories for the detailed and simplified submodels in the dialog specify where the created submodels will be located. If the directory is specified without full path the directory is a relative path to the database currently loaded in MIKE URBAN.

The check mark for 'Split Submodels' must be checked and the detailed and simplified models specified. If only submodels should be created from the detailed model the field for the simplified model is left empty and vice versa.

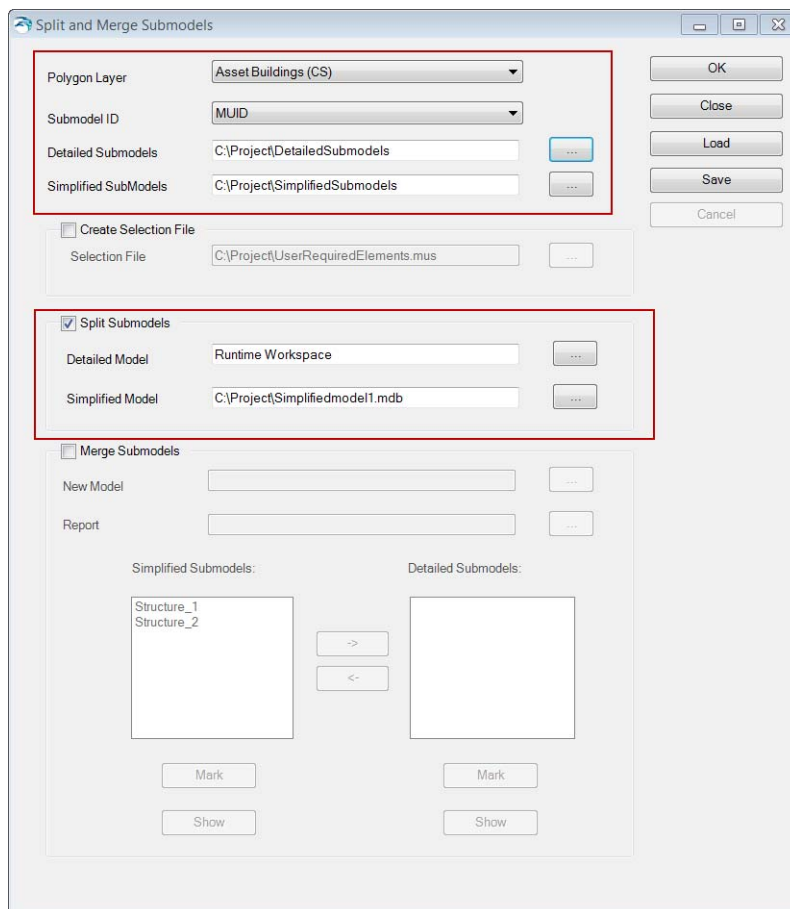


Figure 20.3 Split Submodels

It should be noted that the created submodels are not functioning individual models but parts for a complete model. Hence a submodel may not be valid for a simulation without further manual editing.



20.4 Merge Submodels

When merging submodels the same polygon layer as used to split models should be loaded in MIKE URBAN. For each submodel it should be decided if they should be included as a detailed or simplified submodel.

Per default all submodels will be included in a simplified version. By updating the lists at the bottom of the dialog it is possible to choose which submodels are to be simplified or detailed. The submodels are moved between the lists by marking the relevant IDs and using the arrows to move the marked submodels.

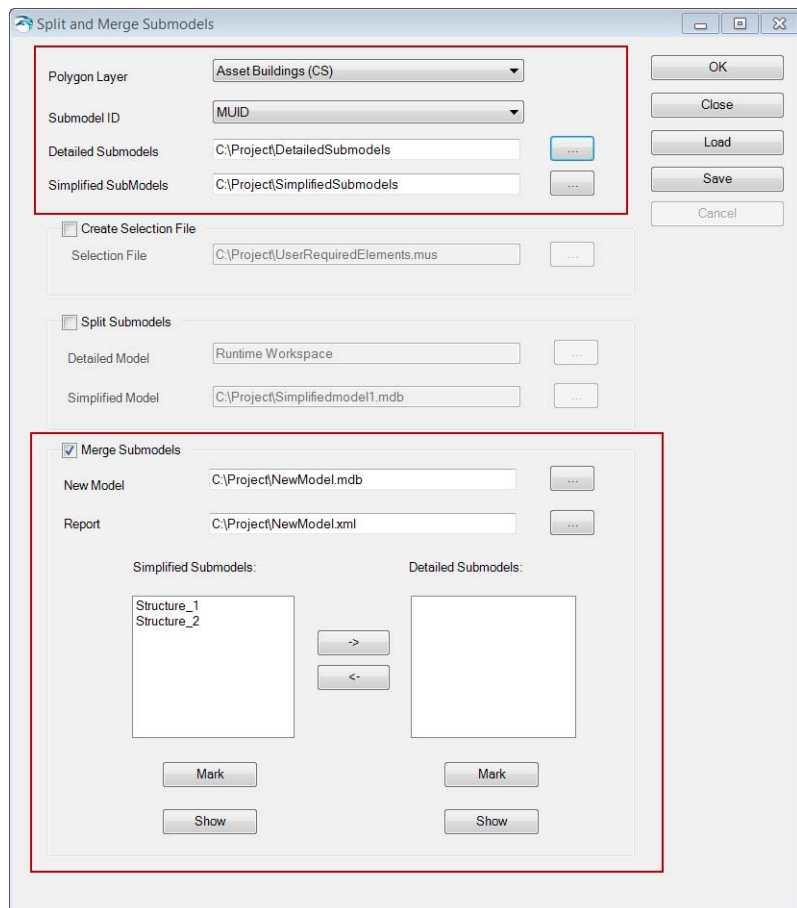


Figure 20.4 Merge Submodels

It is possible to mark the submodels either directly in the list by ID or by selecting the equivalent polygons on the map and using the button 'Mark' to highlight them in the list.



Using the 'Show' button beneath each list will select the equivalent polygons on the map to give an overview of the areas included in either detailed or simplified versions.

In order to merge submodels, the polygon layer, submodel ID and directories for the detailed and simplified submodels are specified at the top of the dialog. If the location of the submodels is specified without a full path the directory is a relative path to the database currently loaded in MIKE URBAN.

The check mark for 'Merge Submodels' must be checked and the name of the new model is given together with the name of a report. The report will include information of when the model was created and which version of each submodel was used.

20.5 Model Requirements

When a new model is created it is based on a combination of submodels originating from the detailed model and from the simplified model. In order to ensure that the individual submodels fit together in both versions there are some limitations to which changes are allowed between the detailed and simplified model.

When a model is split into submodels the individual submodel will contain:

- parts that only exists in the individual submodel; and
- parts that also exists in other submodels.

The parts that only exists in one submodel are the physical parts of the network e.g. manholes, pipes and catchments (what can be seen on the map in MIKE URBAN) including data that is directly related to the physical elements such as catchment data.

The parts that exist in more than one submodel are general data such as materials, basin geometries, curves and other tabulated data, catchment parameter sets etc.

When creating a simplified model from the detailed model the physical elements can be changed or deleted since they only appear in one sub-model. The exception is the pipes or structures that links two submodels together and hence exists in two submodels. These are found by creating a selection file see section Create Selection File (p. 320).

For the general data none of the existing data must be altered, however new data can be inserted and used. As an example a basin geometry specified in the detailed model cannot be changed as it exists in multiple submodels.

However if a different basin geometry should be used in the simplified model a new basin geometry must be created and can be used at any nodes, since the nodes are only included in one submodel.



21 Model Configuration Tool

21.1 Introduction

‘Model Configuration’ is a term associated with a set of changes made to a specific model. The model configuration gives the possibility of re-applying the same changes to a new version of a model as previously made to an earlier version.

The model configuration is used in a situation where a model is maintained outside the MIKE URBAN model e.g. in an asset system. When the data is imported in MIKE URBAN it will often require manual modifications in order to make a final model. Modifications can entail addition of data not available in the asset system, reconfiguration of complex structures and removal of small pipes or an area which are outside the area of interest.

These modifications can be saved in a model configuration so that when an update is available from the asset system the same modifications can be re-applied without manual editing.

The Model Configuration Tool is used in two steps. First when creating the model configuration and second when applying it. When creating a new model configuration the tool finds all changes that the user has made between two versions of the same model. When a new version of the model exists the same changes are applied.

For an overview the changes can be shown visually on the map.

21.2 Launching the Tool

The tool is found under “Tools | Model Configuration”. When activated the dialog shown in Figure 21.1 appears.

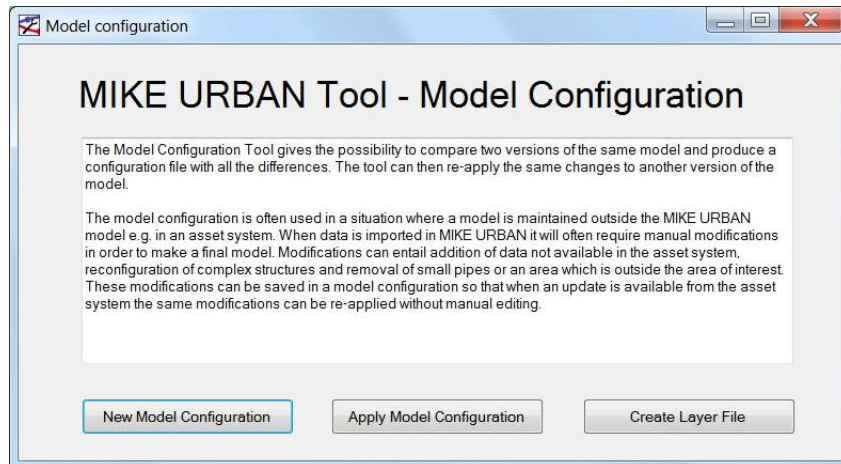


Figure 21.1 Main dialog in Model Configuration Tool

The dialog has three main functionalities “New Model Configuration”, “Apply Model Configuration” and “Create Layer File”, which will be described in the following sections.

21.3 New Model Configuration

When creating a new model configuration the tool will compare two MIKE URBAN databases with each other and capture all differences between the two versions. Hence it is important to keep a copy of the database from the point where further changes should be included in a model configuration.

The tool only compares the active scenario so any differences in the alternatives will not be taken into account.

In order to create a model configuration it is required that the settings for the two databases are the same. E.g. the databases must be in the same version, using the same unit system etc. The tool will automatically upgrade to current version and issue error messages if these conditions are not fulfilled.

When this functionality is activated the dialog shown in Figure 21.2 appears.

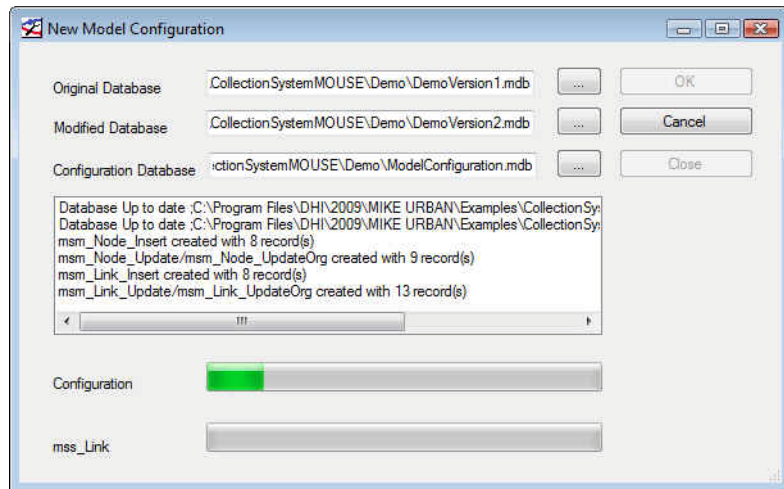


Figure 21.2 New Model Configuration dialog

The dialog requires three input databases. The “Original Database” and “Modified Database” are the two databases to be compared. These databases can either be a personal geodatabase (*.mdb) or a file geodatabase (*.gdb) but must be of the same type. The “Configuration Database” is the model configuration and is saved as a personal geodatabase (*.mdb).

When the tool is executed it will compare all tables based on the unique ID and mark all differences as either inserted, deleted or updated.

The differences are stored in a personal database, so that they can be viewed and/or edited either through Microsoft Access, MIKE URBAN or ArcMap. For each table in MIKE URBAN there can be up to four tables in the model configuration:

- TableName_Insert
- TableName_Delete
- TableName_Update
- TableName_UpdateOrg

The *_Insert tables contain the records of all inserted elements including all attributes.

The *_Delete tables contain the records of all deleted elements including all attributes.

The *_Update tables contain the records of all updated elements. Only attributes which have changed for one or more of the records will be shown. If an attribute is filled this is the new value. If an attribute is empty it can either mean that the value was deleted or that there is no change.



The *_UpdateOrg tables contain the same records as the *_Update however the values shown are the original values. If a specific attribute is shown in the *_Update table and not in the *_UpdateOrg it means that an empty value has been filled. And vice versa, if a specific attribute is shown in the *_UpdateOrg table and not in the *_Update it means that the value was deleted.

An additional attribute “ShapeChange” is added to all *_Update and *_UpdateOrg feature class tables to indicate if the GIS shape has been updated. If this is the case the *_Update table will contain the new GIS shape and *_UpdateOrg will contain the original.

The model configuration database also includes a general table “ConfigurationSettings” that stores general information about which databases were used to create the model configuration, the version of MIKE URBAN and the unit system.

A layer file will be created in the same directory as the model configuration. The layer file contains a reference to all features in the model configuration. For information about the layer file please see 21.5.

The original and modified database will not be changed, however they will both be upgraded to the current MIKE URBAN version when the tool is executed.

21.4 Apply Model Configuration

When applying a model configuration, the saved differences will be re-applied to another database.

When this functionality is activated the dialog shown in Figure 21.3 appears.

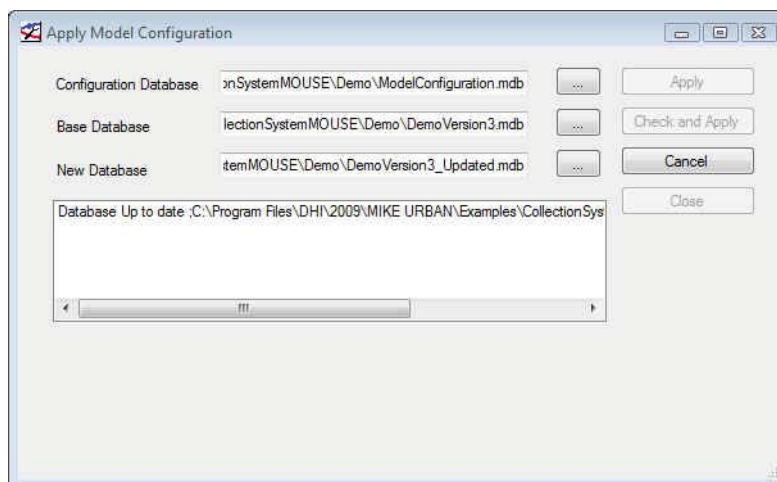


Figure 21.3 Apply Model Configuration dialog



The dialog requires three databases. The “Configuration Database” is the model configuration. The “Base Database” is the database you wish to apply the model configuration to and the “New Database” is the resulting database. These databases can either be a personal geodatabase (*.mdb) or a file geodatabase (*.gdb) but must be of the same type.

The base database will be upgraded to the current version of MIKE URBAN, but will otherwise not be changed. All changes are applied to the new database.

When the functionality is executed it is possible either to directly apply the model configuration or to first check for possible conflict before committing the changes.

Different types of possible conflicts can be detected and for some cases the user can choose different options for resolution.

Possible conflicts are:

- The model configuration specifies that an element should be inserted which already exists in the base database. The default is to keep the element existing in the base database. When using “Check and Apply” it is possible to specify that the element in the model configuration overwrites the element in the base database.
- The model configuration specifies that an element that does not exist in the base database should be deleted. The element will not exist in the new database.
- The model configuration specifies that an element that does not exist in the base database should be updated. The element will not exist in the new database.
- The model configuration specifies that an element should be updated. The element exists in the base database, but the model configuration was made on a different assumption. An example could be that a model configuration specifies that a pipe diameter is changed from 600 mm to 700 mm. The same pipe diameter is now 650 mm in the base database. The default is that the model configuration is applied i.e. the pipe diameter is changed to 700 mm in the new database. When using “Check and Apply” it is possible to specify that the element should keep the value from the base database i.e. the pipe diameter is 650 mm in the new database.

When using the “Apply” button the model configuration is applied directly without further input from the user.

When using the “Check and Apply” button the model configuration is checked against the base database for possible conflicts. If any possible conflicts are found the dialog in Figure 21.4 is shown before continuing.

Table	Primary Key	Attribute	Current Base DataBase	Configuration Base DataBase	Configuration Modified To	Apply Current	Apply Conf
msm_Node	E74150R	Deleted	Deleted	Deleted	Deleted	<input checked="" type="checkbox"/>	<input type="checkbox"/>
msm_Node	E74780R	GroundLevel	38.301	38.441	38.641	<input type="checkbox"/>	<input checked="" type="checkbox"/>
msm_Node	E74790R	GroundLevel	38.646	38.786	38.986	<input type="checkbox"/>	<input checked="" type="checkbox"/>
msm_Node	E74795R	GroundLevel	38.886	39.026	39.226	<input type="checkbox"/>	<input checked="" type="checkbox"/>
msm_Link	Merged_4	Inserted	Inserted	Inserted	Inserted	<input checked="" type="checkbox"/>	<input type="checkbox"/>
msm_Link	Merged_87	Inserted	Inserted	Inserted	Inserted	<input checked="" type="checkbox"/>	<input type="checkbox"/>
msm_Link	R_96778	Deleted	Deleted	Deleted	Deleted	<input checked="" type="checkbox"/>	<input type="checkbox"/>
msm_Link	R_96778_0	Deleted	Deleted	Deleted	Deleted	<input checked="" type="checkbox"/>	<input type="checkbox"/>
msm_Link	Merged_1	Deleted	Deleted	Deleted	Deleted	<input checked="" type="checkbox"/>	<input type="checkbox"/>

Figure 21.4 Conflicts dialog

The dialog contains first the table and the primary key (usually MUID) to identify the record. The following columns display different information based on the origin of the conflict.

If the conflict comes from an element that is updated the “Attribute” column will show the attribute that is updated. The “Current Base Database” shows the value of the attribute in the base database. The “Configuration Base Database” and “Configuration Modified To” show the value from the base database when the model configuration was made and the value it was changed to. If the columns show “Deleted” it means that the element no longer exists in the new base database and cannot be updated.

If the conflict comes from a element that should have been deleted in the model configuration the “Attribute”, “Current Base Database”, “Configuration Base Database” and “Configuration Modified To” shows “Deleted”. Likewise if the conflict comes from a element that should have been inserted all columns will show “Inserted”.

The last two columns contain checkmarks. Only one of the checkmarks can be checked at a time. When the first column “Apply Current” is checked the model configuration will be ignored for this record and attribute and the current values in the base database will remain in the new database. When the second column “Apply Conf” is checked the model configuration will be applied for this record and attribute.

When conflict occurs because the element is deleted in the base database the last two columns will be greyed out and set to “Apply Current” e.g. that the element is deleted.

The button “Set Current Flag” will set a checkmark in “Apply Current” for all records in the table where there is neither a checkmark in “Apply Current” or “Apply Conf”. Likewise the button “Set Conf Flag” will set a checkmark in “Apply Conf”.



21.5 Create Layer File

A layer file (*.lyr) is a configuration file for background layers in MIKE URBAN. When creating a model configuration file a layer file is automatically created with the same name as the model configuration. The layer file contains references to all features in the model configuration, such that a visual representation of the model configuration can be shown in MIKE URBAN.

Inserted elements will be shown in green, deleted elements in red and updated elements in blue.

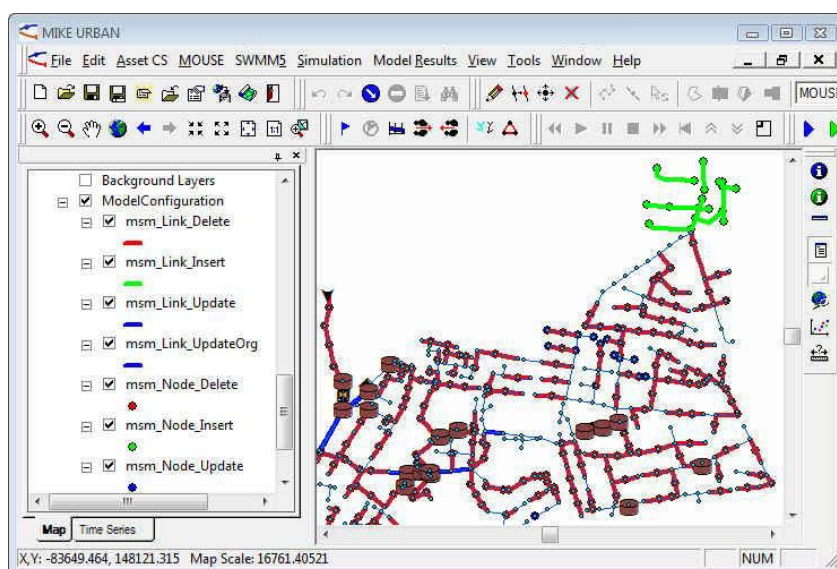


Figure 21.5 Loaded layer file shows the model configuration visually

In order to add a layer file in MIKE URBAN go to “Edit | Insert layer”.

Using the identify button in MIKE URBAN clicking on an update element can show the contents of e.g. the updates before applying them.

A layer file contains absolute paths to the model configuration, so if the database is moved the layer file must be manually updated which is done through the “Create Layer File” button.

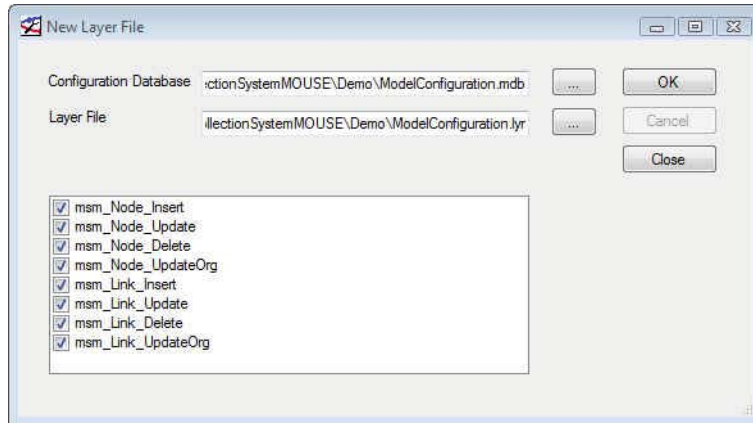


Figure 21.6 New Layer File dialog

It is also possible to create a layer file on a MIKE URBAN database and choose which features should be included in the layer file.



22 CS Toolbox

22.1 Introduction

Urban stormwater flood modeling can be carried out using a 1D/1D or a 1D/2D approach. The 1D/1D approach has a simplified representation of the overland surface hydraulics compared to the 1D/2D approach.

In recent years coupled 1D/2D modeling applications have been widely used in urban stormwater analysis. The advantage of the 1D/2D approach is that the model is faster to configure and it provides a more realistic hydraulic description of the surface flow. Model simulation times are however generally long, which makes the design and testing of stormwater improvements and installations difficult and time-consuming.

The advantage of the 1D/1D approach is that the simulation time is considerably faster than the 1D/2D approach and is therefore more suitable for detailed design option runs. However configuring the 1D/1D model is more time-consuming than the 1D/2D model.

A 1D/1D stormwater model typically comprises three main components: the sewer/stormwater system, the overland flow system, and rainfall-runoff hydrology.

The toolbox is a new set of tools developed to specifically address some normally time-consuming aspects of building a 1D/1D stormwater model.

For a catchment study a staged approach is usually needed, for example:

- The first stage involves catchment-wide modeling in a combined 1D/2D model to assess the performance of the system and identify potential improvement locations;
- For the second stage, a full 1D/1D model is built only for the improvement location including the 1D overland flow path network. The 1D/1D model runtimes are fast and allow for quicker assessment of stormwater improvements.

The stormwater tools focus on building the stormwater model within a 1D framework, which include:

- cross section extraction from the DEM
- lateral snapping of nodes according to the DEM
- auto connection of overland network to stormwater network
- sequential labeling of nodes

The tools can be activated from the menu 'Tools | Toolbox' or the 'Toolbox' toolbar.

22.2 Cross Section Generation Tool

When overland flow paths are defined as MIKE URBAN links, a cross section (CRS) needs to be assigned to each of these overland flow links. Often, a standard road profile will fulfil the modeling requirements. However in other urban flood modeling situations, individual cross sections are required for open spaces, rural areas, park areas, etc.

The Cross Section Generation tool uses cross section alignments drawn in a line feature layer to extract cross sections from a DEM for links intersected by the alignments, see Figure 22.1. It generates cross sections for each link and sets the reference between the link and the generated cross section ID.

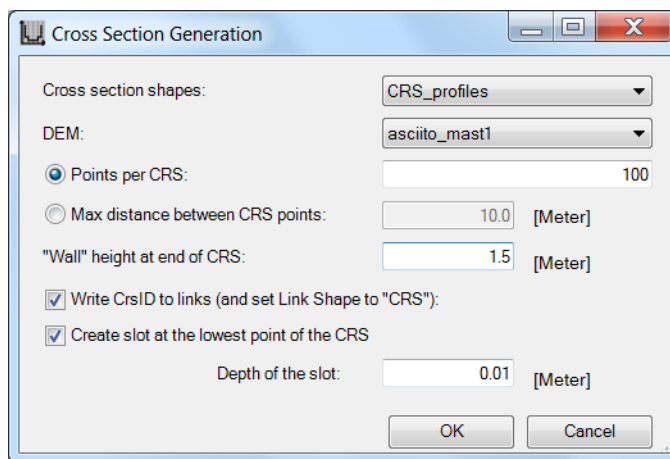


Figure 22.1 Cross Section Generation tool

The manual digitalization of the CRS alignment lines can be guided by MIKE URBAN FLOOD simulations results (1D/2D results) or an uncoupled 2D model where the precipitation is applied to the surface assuming that the sub-surface network is completely full. Overland flood results can be used as a background layer in MIKE URBAN. The only requirement for the digitized CRS lines is that they intersect the pertinent MIKE URBAN links. An example is shown in Figure 22.2.

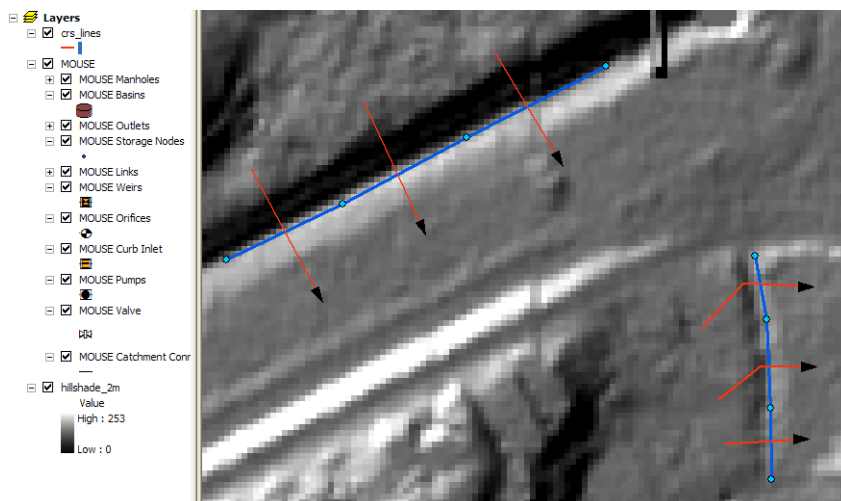


Figure 22.2 Example of defined CRS lines

Either the number of points or the maximum distance between points for a CRS can be specified. To fully capture the information from the DEM the resolution of the CRS should correlate to the resolution of the DEM.

The ID of the new CRS will be identical to the ID of the corresponding link. The CRS will be created with the type X-Z open and with a width equal to the length of the corresponding CRS alignment line. The pertinent link will also be updated to use the new CRS by default, but this option can be disabled in the tool.

If the water level in a CRS rises above the defined CRS height, the simulation will stop. This is very common in stormwater models, so the tool allows for side-walls to be automatically added. Use a side-wall of 1 to 2 meters for an overland flow path such as a road, and 3 to 5 meters for waterways. Adding side-walls adds 2 points to the number of CRS points specified in the tool.

Another consideration is that if a CRS is very flat, too much numerical water will be generated when the link is running dry. Thus, a slot can automatically be inserted at the lowest point of the cross section

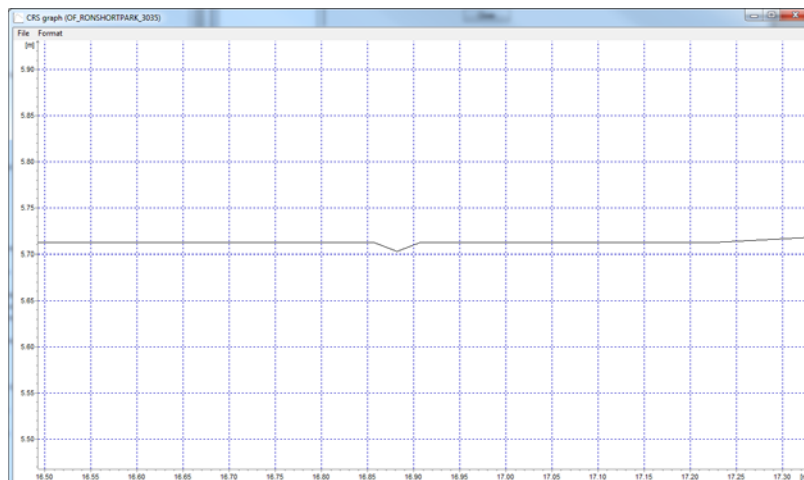


Figure 22.3 Example of CRS with inserted slot

The tool will generate CRSs for selected links or for all links intersecting alignments if no particular link is selected.

22.3 Lateral Snapping Tool

It can be difficult to exactly place nodes along the invert when digitizing an overland flow path whether it is along the gutter in a road or the invert of a waterway.

The Lateral Snapping tool, shown in Figure 22.4, is used for automatically moving nodes and snapping them laterally to the lowest DEM value along a lateral snap alignment, which is shown in Figure 22.5. The length of the lateral snap alignment is specified in the tool.

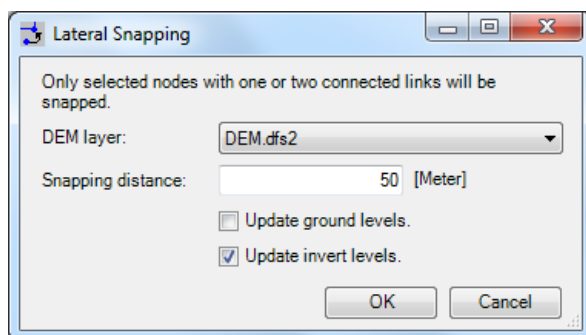


Figure 22.4 Lateral Snapping tool

When the selected node is connected to two links, the direction of the lateral snapping alignment will be created such that the internal angles between the upstream and downstream links are equal ($\theta_1 = \theta_2$), see Figure 22.5. If the



selected node is only connected to one link, the direction of the lateral snapping alignment will be perpendicular to the link.

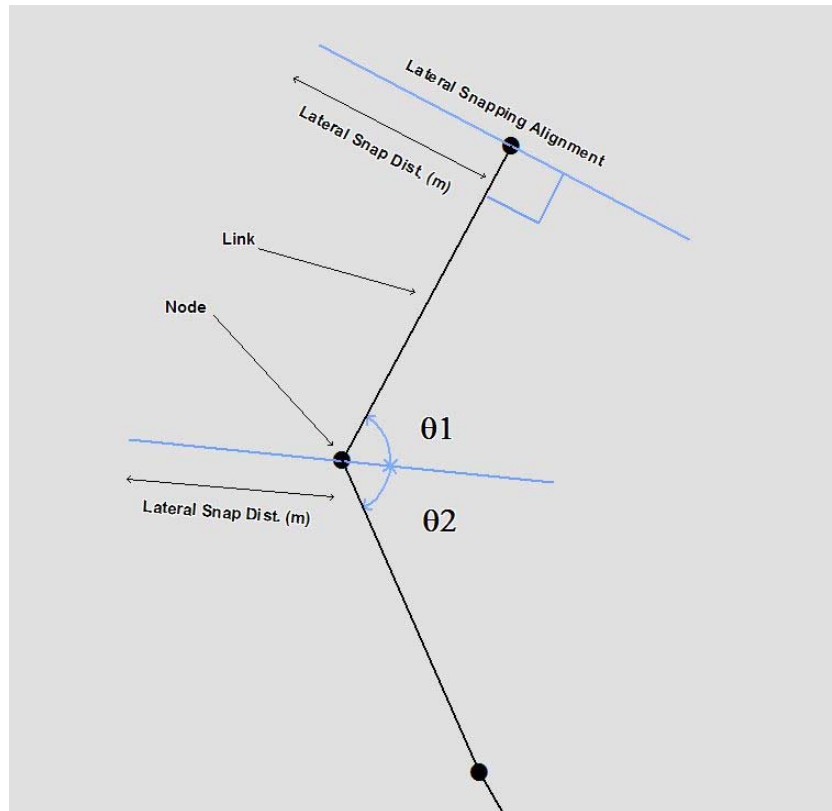


Figure 22.5 Lateral snapping concept

When nodes are moved to the lowest point, the tool can update either the ground level or the invert level according to the DEM. Note that the ground level should be updated for nodes belonging to the subsurface network and the invert level for nodes belonging to the overland flow network.

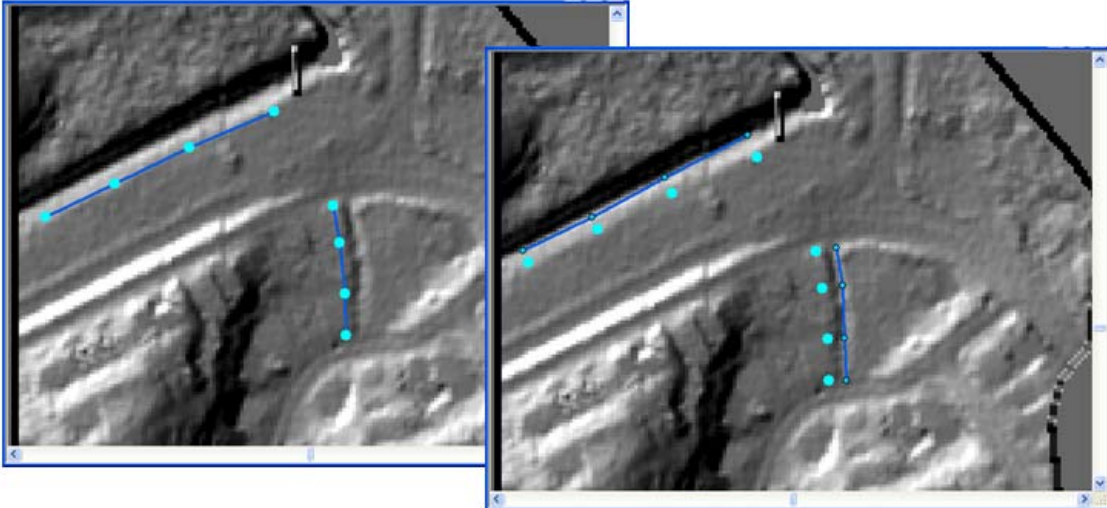


Figure 22.6 Example of nodes moved with the lateral snapping tool

The tool will only laterally snap selected nodes.

22.4 Auto Connection Tool

When overland flow paths have been digitized and snapped to a correct alignment, the overland flow links have to be connected to each other (overland to overland) or connections have to be made between the overland flow network and the subsurface network.

The tool can generate both connections between the same type of network and connections between two different types of network. A network can be defined as i.e. Stormwater, Combined Sewer, Sanitary Sewer or a user-defined network type. A user-defined network could be Overland Flow network (OF) or Rising Main (RM). User-defined network types are set under “Tools | Customize | Status Codes” (select Net Type).

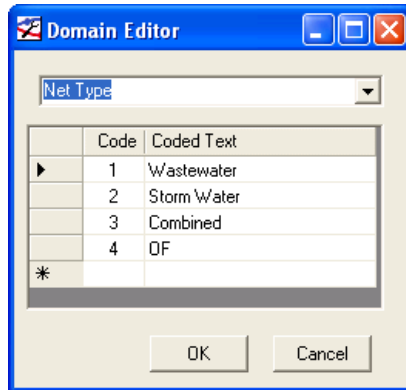


Figure 22.7 Domain Editor

The nodes in the model must be provided with the appropriate network type before proceeding to set up the connections.

A connection can be a weir, orifice or curb-inlet for a MOUSE network, and a weir or orifice for a SWMM network. Connections between an overland flow network and a subsurface network are usually either orifices or curb-inlets while connections between two overland flow networks are usually weirs. The connective structure is created according to the direction specified in the tool. When connecting an overland flow network with a subsurface network, the direction should be specified from the overland network to the subsurface network.

A user-specified search radius is used around the nodes from the network listed under “From nodes” to control where connections can be made. If the two different networks are connected, the connection will be made to the closest node with the type of the “To nodes”. If identical networks are connected then connections will be made between all nodes within the search radius that are not already connected.

The structures are created with an invert level according to the network, but other parameters for the individual structures must be set manually. If two different networks are connected, the default invert for the structure will come from the invert of the upstream node. If two identical networks are connected, the default invert for the structure will be the maximum of the upstream and downstream node inverts. An offset for the invert can be specified in the tool. It is recommended to insert a small offset of i.e. 0.05 m between overland flow networks and subsurface networks to ensure that flow is not always occurring through the connection.

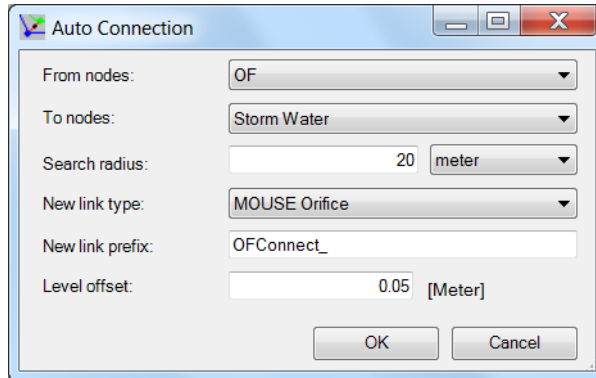


Figure 22.8 Auto Connection Tool

An example from a 1D/1D model is provided in Figure 22.9 where the white network represents the stormwater network and the red network represents the overland flow network. The overland flow network is connected to the stormwater network by orifices. The overland flow network elements are connected together at junctions by weirs.

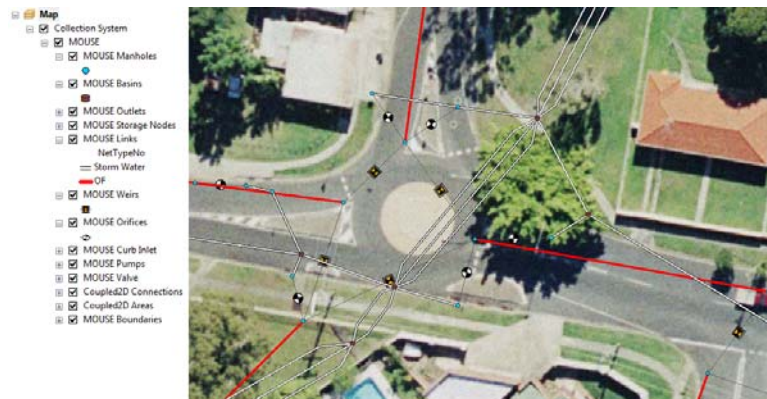


Figure 22.9 Example of connections

The tool will try to make connections for selected nodes, and if no node is selected, connections will be attempted for all nodes according to the specifications.

22.5 Sequential Labeling Tool

When an overland flow path or other network is digitized, it is often useful to provide ID's that are descriptive and logical. The ID's can be made up by street name, type of network, sub-catchment identifier, etc. and the sequential labeling tool can automatically achieve this.



A format for the automatic ID's can be specified for selected nodes or links in the map. A label prefix and suffix may be given together with sequential numbering parameters.

Only selected items of the chosen Map layer (nodes or links) will be given new ID's.

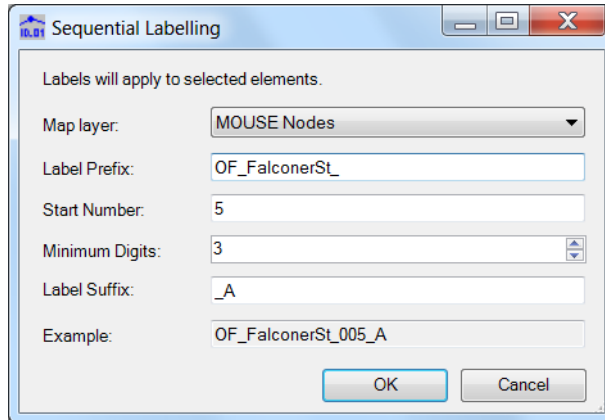


Figure 22.10 Sequential Labeling Tool





23 Presenting results

With MIKE URBAN you can present your results in a number of ways:

- Time series
- Profile plots
- Animations
- Result comparisons
- Statistical analysis

The MIKE by DHI freeware result viewer MIKE VIEW can also be used for viewing results. See the MIKE VIEW documentation for further on this. Please note that results from the MIKE 1D numerical engine can only be viewed in MIKE VIEW and not in MIKE URBAN.

23.1 Loading results

To load results:

- Go to Model Results|Load Results| ... and choose you result file - see Figure 23.1

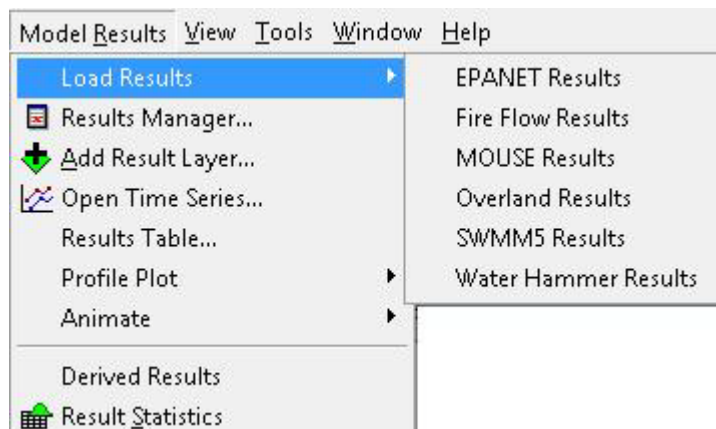


Figure 23.1 Loading model results

- A dialog with choices of which data to load from the result then appears. The items displayed are dependent on the type of result file. An example is shown in Figure 23.2.
- Pressing 'OK' with everything selected chooses all the items in the result file. The number of time series belonging to each item is also displayed.

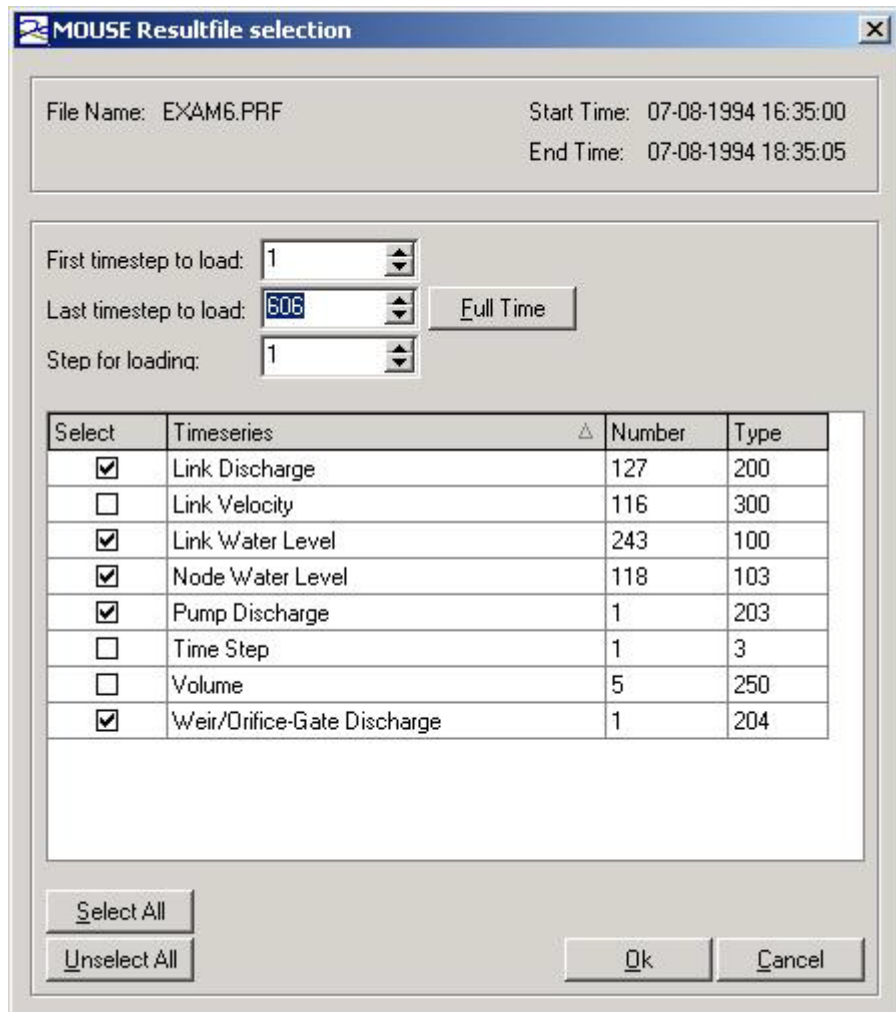


Figure 23.2 Example of the result file selection dialog

The results are loaded into the memory and hence will not be available after closing MIKE URBAN.

23.2 Displaying results on the horizontal plan

In order to display results on the horizontal plan you need to add a result layer to the Table Of Contents (TOC).

- Go to Model Results|Add Result Layer, see Figure 23.3

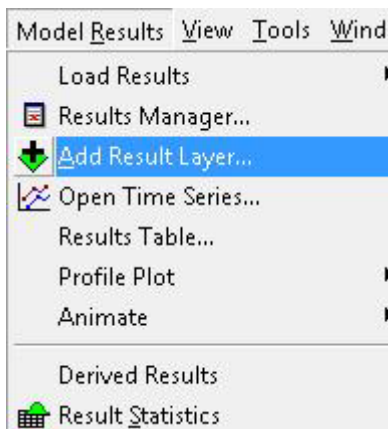


Figure 23.3 Adding a result layer to the Table Of Contents

- On the dialog that comes up, see Figure 23.4, choose between the loaded result files and the data type to display on the TOC (e.g. water level in nodes, discharge in branches etc.). The data types are dependent on the result file

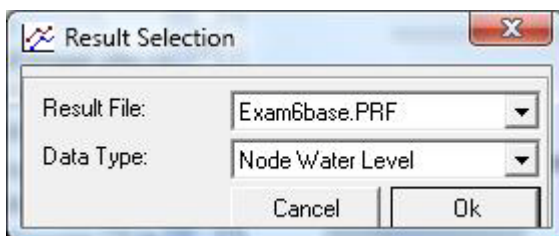


Figure 23.4 Choosing the type of data to display on the TOC

- After pressing the 'OK' button the result layer is added to the TOC. The symbols and color chosen is pre-defined. The symbology and labelling can then be altered.

23.3 Results browser



The results browser is activated on Model Results|Results Browser. The results browser opens a window where information on the element that you point to, as well as results for that element, are displayed, see Figure 23.5



Property	Value	Minimum	Maximum
Link ID	5511		
Type	CRS		
Up Level			
Down Level			
Length			
Diameter			
Width			
Height			
Cross Section	EXAM*RK2		
Topography ID			
Material ID	MOUSE_Smooth_Con		
Description			
Time	07-08-1994 18:29:00	07-08-1994 16:35:00	07-08-1994 18:35:10
Link Water Level	195.011	195.005	195.209
Link Discharge	0.001	0.000	0.294
Link Velocity	0.010	0.009	0.571

Figure 23.5 The results browser

With the result browser you can easily view results for various elements, but also information on the element as such are displayed (e.g. ID and materials etc.). The results displayed are the current value (in the 'Value' column), the minimum value and the maximum value. 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.

23.4 Results manager

The results manager is activated by Model Results|Results Manager. The results manager, see Figure 23.6, displays the result files that are currently loaded. By selecting a result file and then clicking on the 'Remove' button the result file is removed from the memory.

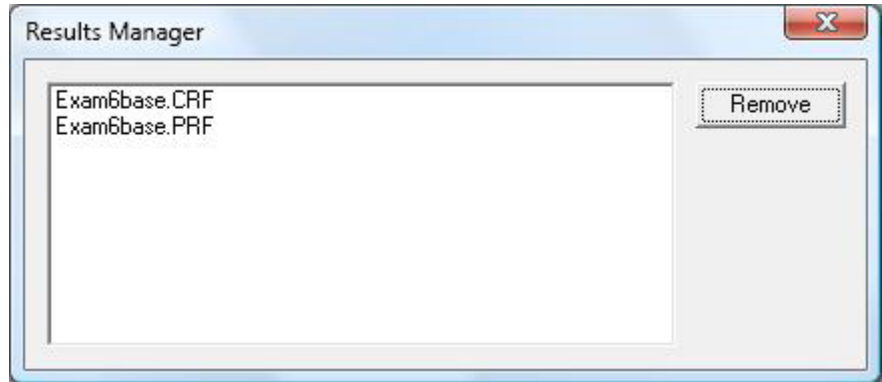


Figure 23.6 The results manager

23.5 Derived results

MIKE URBAN can, in addition to the standard results, compute derived results (e.g. flood in nodes, tap pressure). In order to compute these choose Model Results|Derived Results.

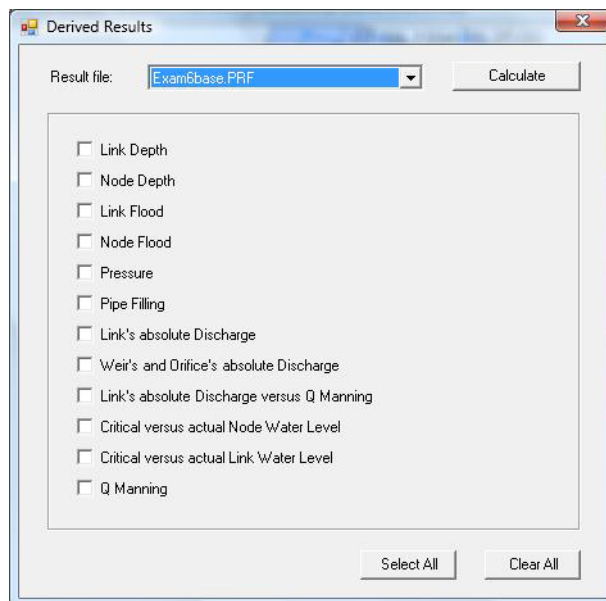


Figure 23.7 Example of the additional variables that can be computed on request.

MOUSE results

The following additional results can be computed for relevant MOUSE result files:



- Link depth: The link depth is calculated as link water level minus link invert/bed level, for each H-point.
- Node depth: The node depth is calculated as node water level minus node invert/bed level for all nodes except storage nodes
- Link flood: The link flood is calculated as link water level minus link ground level for all links. The link ground level is calculated as: $H_{ground}(X) = \text{GroundLevel}(\text{upstream node}) - ([\text{GroundLevel}(\text{upstream node}) - \text{GroundLevel}(\text{downstream node})] * [\text{Chainage}(X) / \text{Length}])$
- Node flood: The node flood is calculated as node water level minus node ground level for all nodes except outlets and storage nodes
- Pressure: Pressure is calculated as water level minus pipe top level (i.e. it is calculated as height of water column. It is not calculated for open cross sections or natural channels.
- Pipe filling: The pipe filling is calculated as the depth divided by the pipe height, e.g. if the pipe is running under pressure the ratio will be above 1.0.
- Link's absolute discharge: Calculated as absolute value of computed discharges
- Weir's and orifice's absolute discharge: Calculated as absolute value of computed discharges
- Critical versus actual node water level: Calculated for all manholes and basins. For nodes where critical level is specified it is calculated as: Actual node level - Critical Node level. For nodes where critical level is not specified it is calculated as: Actual node level - Ground level (i.e. in these places it is equivalent to Node Flood)
- Critical versus actual link water level: Calculated as link water level minus link critical level. The link critical level is calculated as: $H_{ground}(X) = \text{GroundLevel}(\text{upstream node}) - ([\text{CriticalLevel}(\text{upstream node}) - \text{CriticalLevel}(\text{downstream node})] * [\text{Chainage}(X) / \text{Length}])$. For nodes where no critical level is specified, Critical level is replaced by GroundLevel (i.e. it is equivalent to link flood)
- Q Manning: The Q/Q-manning (where Q-manning is the discharge computed with the Manning formula for the full flowing pipe) is computed as: $Q_{actual}/Q_{manning}$.

SWMM Results

The following additional results can be computed for relevant SWMM result files:

- Water level (in nodes)
- Node flood
- 'Discharge' absolute value (in links)



- Q/Q-manning: The Q/Q-manning (where Q-manning is the discharge computed with the Manning formula for the full flowing pipe) is computed as: $Q_{\text{actual}}/Q_{\text{manning}}$

EPANET Results

The following additional results can be computed for relevant EPANET result files:

- Link: Absolute value of flow: Calculated as absolute value of the existing result flow item
- Link: Absolute value of velocity: Calculated as absolute value of the existing result velocity item
- Link: Headloss (difference in hydraulic grade line at the beginning and ending pipe nodes)
- Node: Tap pressure (difference between hydraulic grade line and the node elevation plus node estate height)

Once the additional results are loaded into MIKE URBAN, you can display them in the same way as any other results i.e. you can use Results Browser, Profile Plot, Time Series, and Results Table.

23.6 Result statistics

With Model Results|Result statistics you can compute minimum, maximum and the accumulated value (for relevant time series) of your result time series. You can also do this for the derived results. Simply choose the statistics you wish to calculate and press 'Calculate'. This way you can e.g. compute the maximum flood in nodes. The result statistics are only stored in the memory, i.e. when you close the project and re-open it, they are gone. To retain the calculated information for future reference, the data can be exported to a shape file (refer to 23.17) and loaded into the MU project (create the appropriate symbology based on the 'Results' field in the generated shape file).

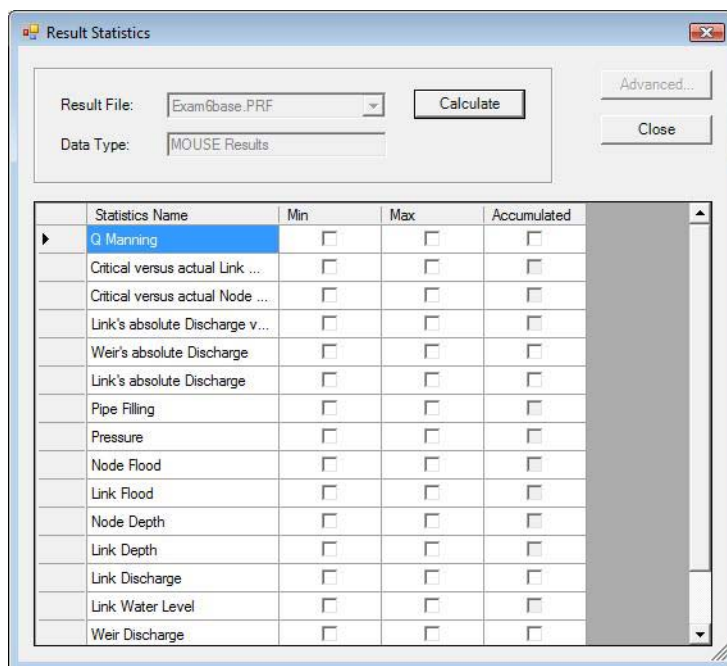


Figure 23.8 Computing result statistics

23.7 Result conversion

In MIKE URBAN it is possible to convert result files from the EPA-SWMM5 engine to file format compatible with the MIKE URBAN - MOUSE result files. This is done by applying the tool, 'SWMM to MOUSE', found under Tools\Send Data.

The tool uses the SWMM5 result file (.out) and the associated SWMM5 input file (.inp) as input. The tool writes two new result files: <Name>.PRF and <Name>.CRF where <Name> is the name of the specified SWMM5 result file.

The PRF file will contain the results related to the pipe network, while the CRF file will contain the runoff hydrographs from the surface catchment runoff. If no subcatchments exist in the SWMM5 input file then only the PRF file will be produced by the tool. This tool hence makes it possible to use the runoff hydrographs from a SWMM5 computation as input for a MOUSE pipe flow computation. It is thus also possible to use the freeware result viewer, MIKE VIEW, on the SWMM5 results.

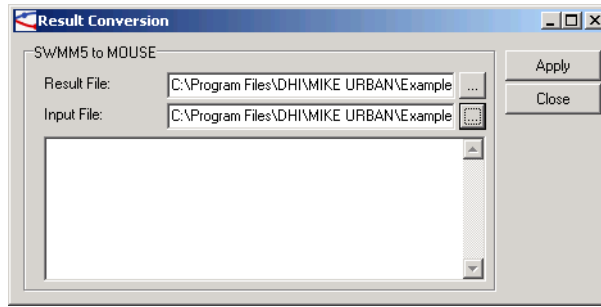


Figure 23.9 Converting SWMM5 results to format compatible with MOUSE results

23.8 Labelling and symbology

To change the labelling and symbology on e.g. a result layer do the following:

- Right-click on the layer that you wish to display differently and choose “Properties“. The dialog seen in Figure 23.10 is then displayed.

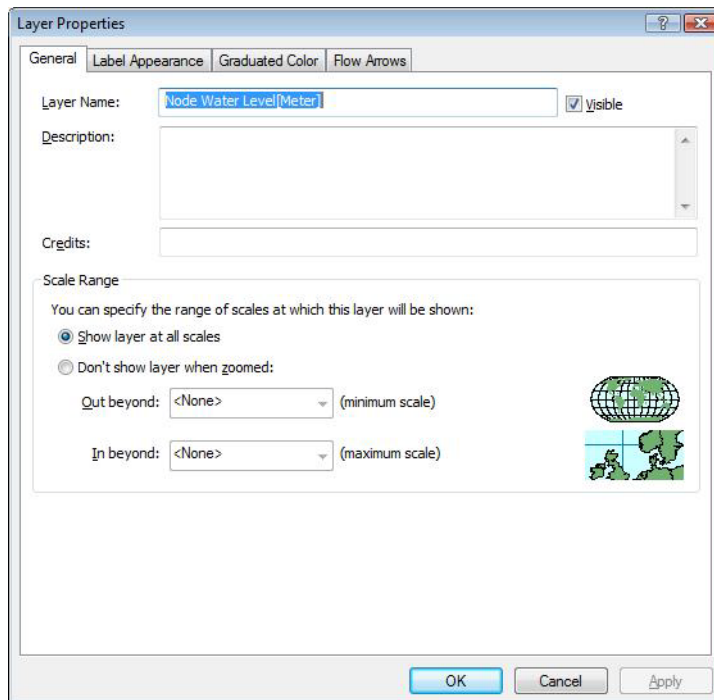


Figure 23.10 Layer properties dialog



- The four tab-pages on the dialog are: General, Label Appearance, Graduated Color, Flow Arrows. In order to change labelling and symbology you go to the relevant tabpage and click 'Apply' - or 'OK' if you would like to apply the new settings and close the dialog as well.

General

The 'General' tabpage, see Figure 23.10, allows you to change the scale range for which the layer will be displayed.

Label Appearance

The 'Label Appearance', see Figure 23.11, allows to add labels on the horizontal plan. The position of the label, font label, number of decimals displayed and when to display labels can be set.

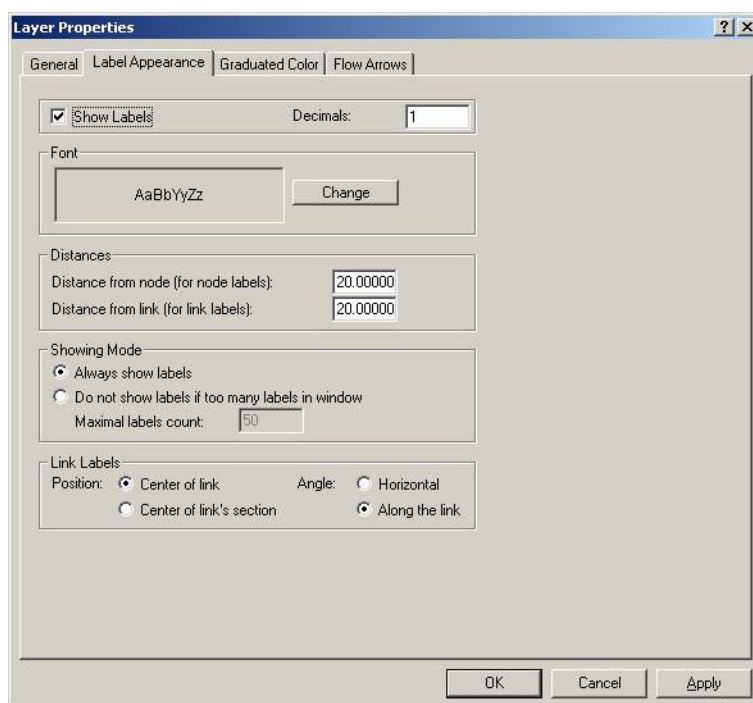


Figure 23.11 Label Appearance

Graduated Color

The 'Graduated Color', see Figure 23.12, allows to change the symbols in size and color and the number of classes to allow. E.g. if you wish to have your links displayed by color and you wish to apply four different colors to do this you choose the number of classes to four.



It is possible to save the colour legend used for the thematic mapping of results layers into a file and load it from the save file next time you want to use it for the same or another model.

In order to save or load the colour legend, click the result layer properties and select "Graduated Colour", "Load" and "Save" buttons allow you o save the settings into a configuration file or load it from the previously saved configuration file.

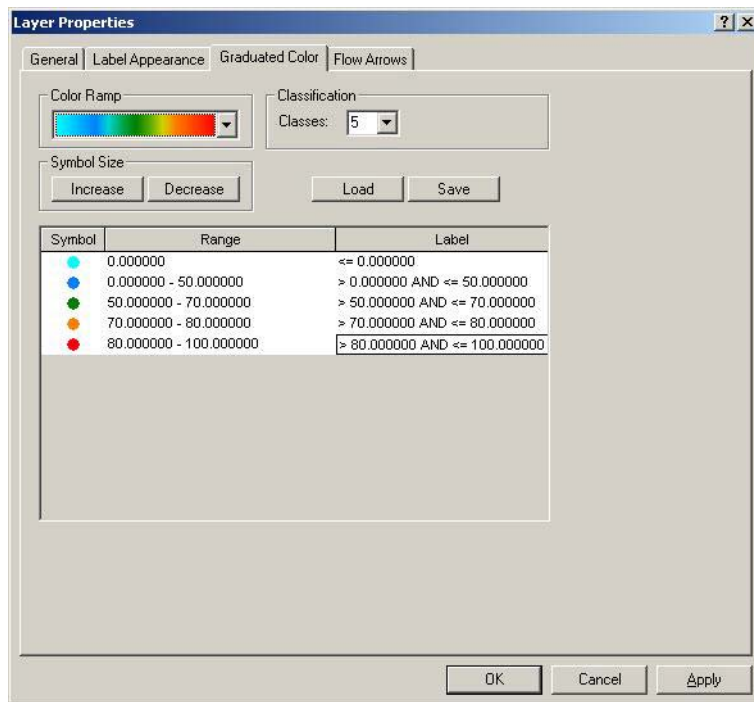


Figure 23.12 Graduated Color

Flow Arrows

On the 'Flow Arrows' tab page, see Figure 23.13, you can toggle the display of flow arrows on/off, as well as set color, placement and showing mode of the flow arrows.

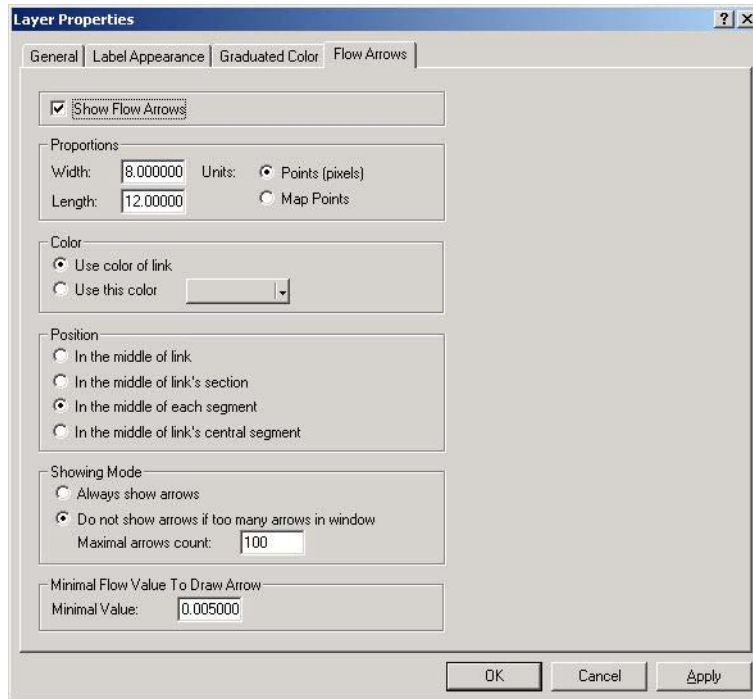


Figure 23.13 Flow Arrows

23.9 Thematic maps and statistics

In MIKE URBAN it is easy to create thematic maps.

Example: Creating a map where all manholes closer than 95% to the flood level are displayed in a significant color

In below we assume that a result file is loaded.

Step 1: Start by creating a statistics layer with Hmax

This is done by:

- Enter Edit mode ('Edit|Start Editing')
- Go to 'Model Results|Results Statistics'. The dialog displayed in Figure 23.14 appears. Choose to calculate the maximum water level and press the 'Calculate' button.

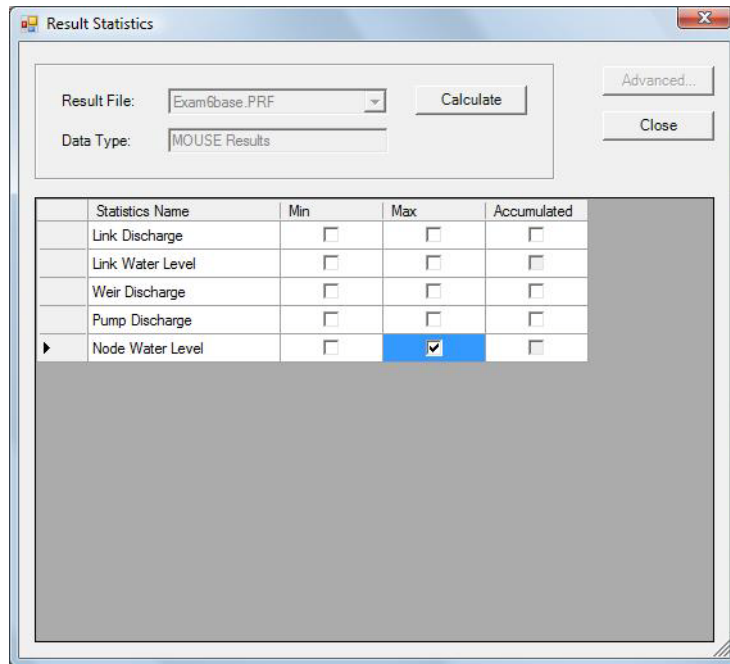


Figure 23.14 Computing results statistics

- Create a statistics layer with Hmax by going to 'Model Results|Add Statistics Layer'. The dialog in Figure 23.15 comes up, choose 'Max. H in Node' and press the 'Ok' button.
- A layer is now added to your TOC - this may be moved up by selecting the layer and accessing the local menu by a right-click. On this local menu choose 'Move Layer Up' or 'Move Layer to Top' if you would like to the layer.

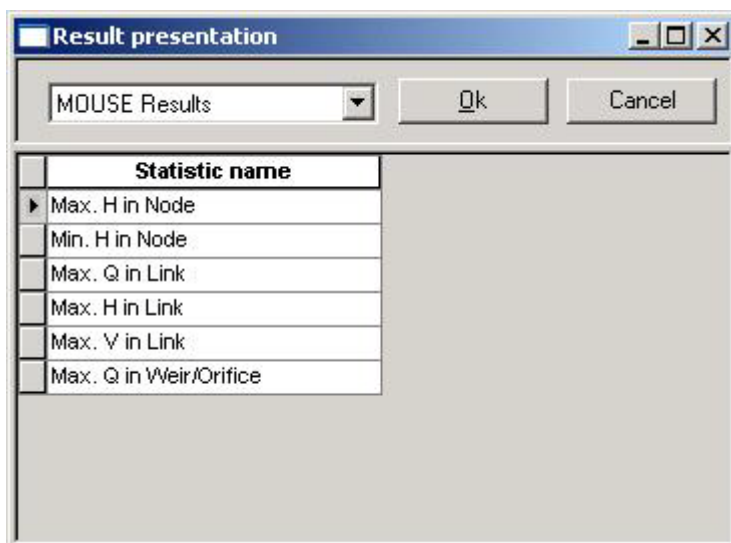


Figure 23.15 Adding a statistics layer

Step 2: Change the symbology so that the scale is 'normalized' with respect to ground level

This is done by:

- Right-click on the layer and choose 'Properties' on the local menu
- Go to the tab-page that says 'Symbology' and choose the ground level in the field to normalize the Hmax with. This is shown in Figure 23.16.

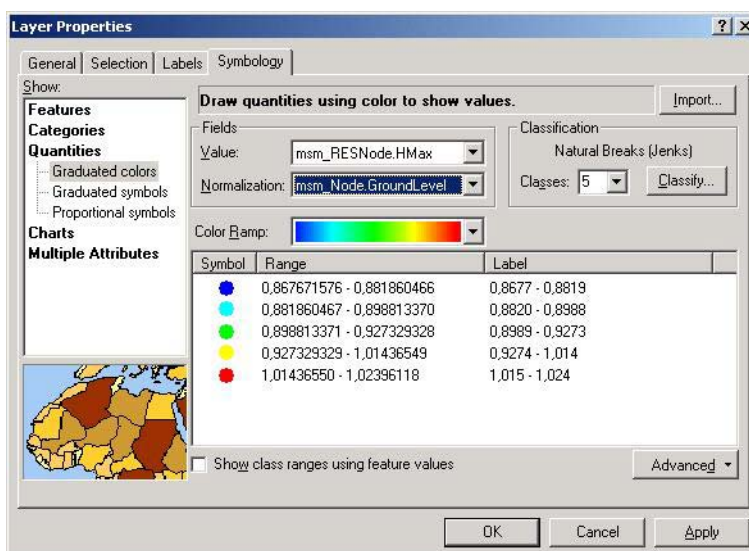


Figure 23.16 Normalizing the Hmax with the ground level



Step 3: Change the classification so that the top group is e.g. covering everything above 0.95

This is done by:

- Press the 'Classify' button in Figure 23.16. In the dialog that comes up, Figure 23.17, choose the break value for the top group to be 0.95. This can be done both by graphically moving the break value and by typing the value. Press 'OK' after doing so.

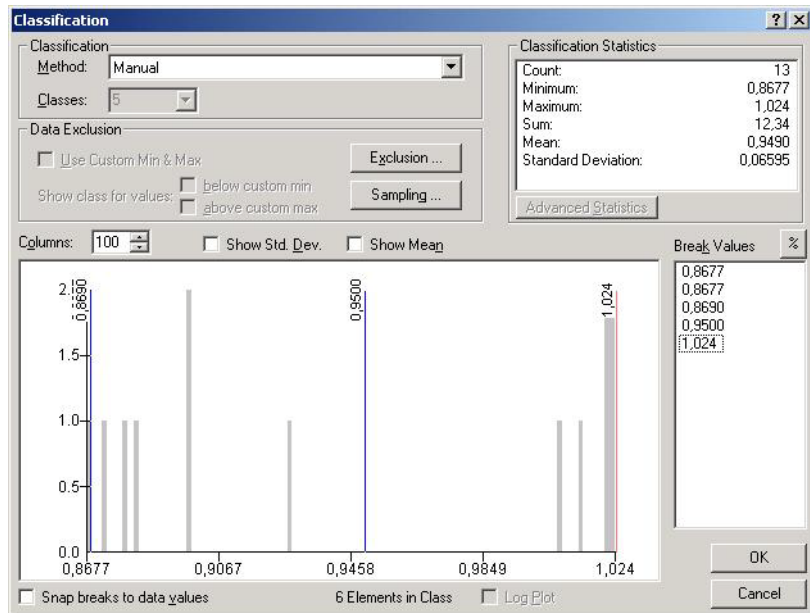


Figure 23.17 Changing the classification

Step 4: Change the color for the top group to red (or another distinct color)

This is done by:

- Right-click on the symbol for the top group and choose 'Properties for the Selected Symbol(s)', see Figure 23.18

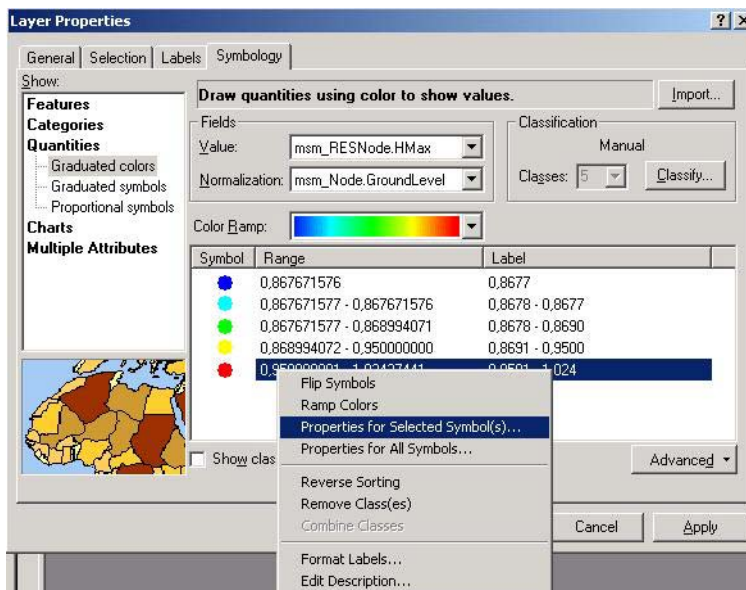


Figure 23.18 Changing the symbol for the selected symbols

- Then choose the symbol, color, size etc. for the top group on the dialog that comes up, see Figure 23.19 and press 'OK'

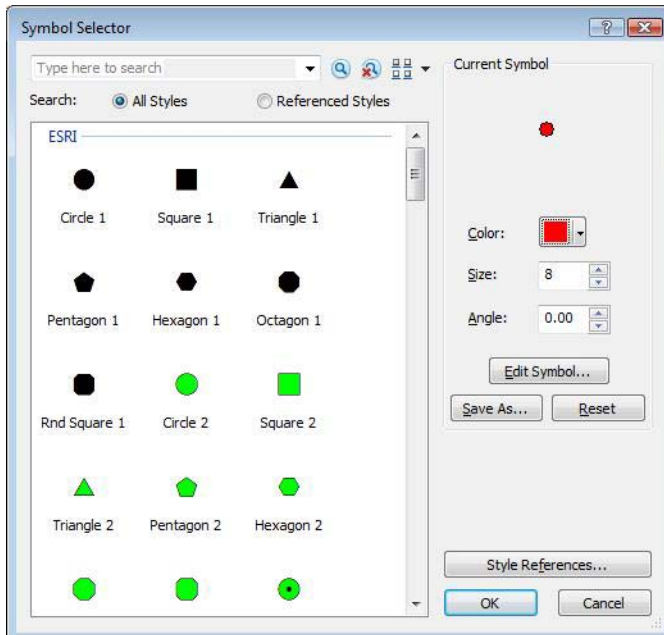


Figure 23.19 Selecting symbols



Now you will have all manholes that are closer than 95% to the flood level displayed in red (or the color chosen in the above). You can of course also use 'critical level' instead of flood level. Or you can choose to change symbols for the other groups as well.

23.10 Time series

Time series can both be input time series or time series taken from the various types of result files.

In this section the focus is displaying time series from the result files. Start by loading a result file, see 23.1.

- Go to Model Results| Time Series
- This will open the result selection dialog, see Figure 23.4
- Choose the result file and data type and press the 'OK' button
- The cursor will now change to include a time series icon - and you can point on the node, link etc. where you wish to display a time series from

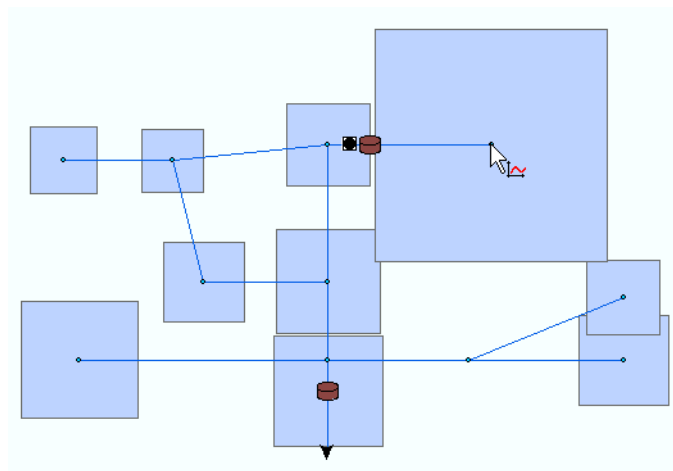


Figure 23.20 Select and display time series from a node

23.11 Results table

The results table (Model Results|Results table) provides an overview of all or selected results in tabular form. Depending on the type of result file different information is available. Beneath the main grid summary fields are available. The summary fields can be displayed for all elements or for the filtered elements.



Name	Node: Demand	Node: HGL	Node: Pressure	Node: Quality
2607	0.00160000007	264.5300903	54.91009902954	0
2610	0.06399999958	264.5301208	54.38010406494	0
2611	0.02449999749	264.5319213	54.55190658569	0
2614	0.00805000029	264.5318908	54.50190353393	0
2615	0.05999999985	264.5331726	53.96316146850	0
2671	0.11320000141	264.5396728	54.00967025756	0
2674	0.03079999797	264.5394897	52.90949630737	0
5623	0.04440000280	264.5319824	54.41199111938	0
5630	0.03709999844	264.5320129	54.58202743530	0
8999	0.04080000147	264.5298461	54.90985870361	0
11473	0.03079999797	264.5328674	54.14285278320	0
12446	0.07244999706	264.5318908	54.58189773559	0
14903	0.05960000306	264.5304565	54.32044219970	0
15230	0.06054999306	264.53125	54.15124511718	0
15432	0.01924999989	264.5303955	54.22039413452	0
15433	0.04440000280	264.5300750	54.67005710000	0

Statistics	Node: Demand	Node: HGL	Node: Pressure	Node: Quality
Minimum	0.00160000007	264.5298461	52.90949630737	0
Maximum	0.11320000141	264.5396728	54.91009902954	0
Average	0.04177368358	264.5323052	54.32809408087	0

Sum of inflow: 0; Sum of demand: 0.79

Result table time: 0 d. 00:00:00 Animation time: 0 d. 00:00:00

Figure 23.21 The summary fields are below the main grid

The summary fields are generated for every numerical result item:

- Minimum, maximum, average: minimum, maximum, and average value
- Sum of inflow, sum of demand: represents the total junction node demand or inflow within the selected area (or the whole network). This information is available only in the WD mode.

23.12 Reports

MIKE URBAN provides a reporting tool for setting up reports (in HTML or CSV format) with a combination of input data and selected results. It is also possible to join information from different tables in the report. The tool is found under Model Results|Reporting Tool). The report tool uses a wizard approach for configuring the reports.

23.12.1 Setting up a report

The first page of the report tool allows to load an existing configuration or to create a new (empty) one.

Pressing 'Next' will lead you to a number of dialogs that specify the content and the layout of the report. At each point in time you can move back to previ-



ous pages by using the 'Back' button. If you press 'Back' until you reach the first page, a radio button 'Keep configuration' is enabled and checked. 'Keep configuration' allows you to go to the next page again and continue with the current configuration settings. 'Keep configuration' is checked per default every time, you go back to this page.

In order to set up a report for the first time hence choose 'New configuration' on the first tab page (see Figure 23.22).

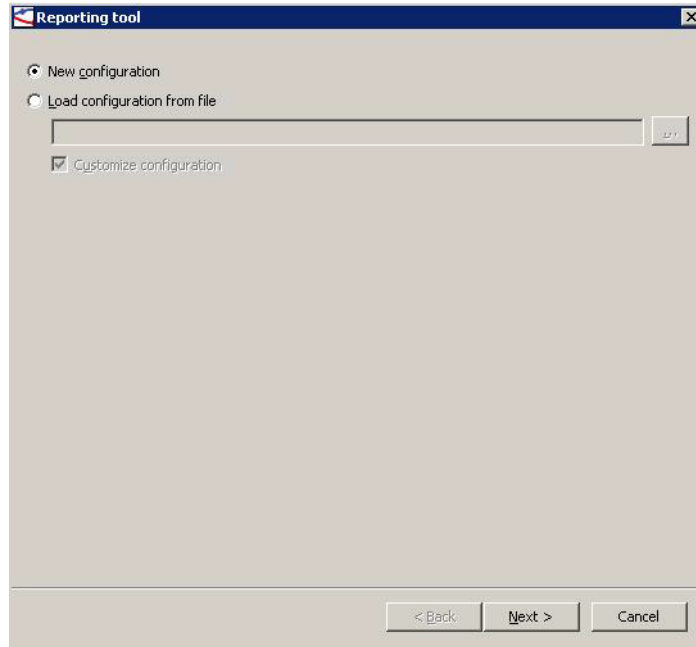


Figure 23.22 Creating a new report or loading in an existing report configuration

On the next page that comes up (see Figure 23.23) it is possible to define the actual content of the report:

- The 'Add' button allows to add either information from tables in the MIKE URBAN database (Add|Database table) or information extracted from result files that are loaded into MIKE URBAN (Add|Results)
- The 'Edit' button allows to edit content of already added tables and/or results
- The 'Remove' button will delete the selected attributes from the report
- The 'Move up' and 'Move down' button allows to move the added columns in the report. The input specified as rows (see Figure 23.25) will be equal to the columns of the report.

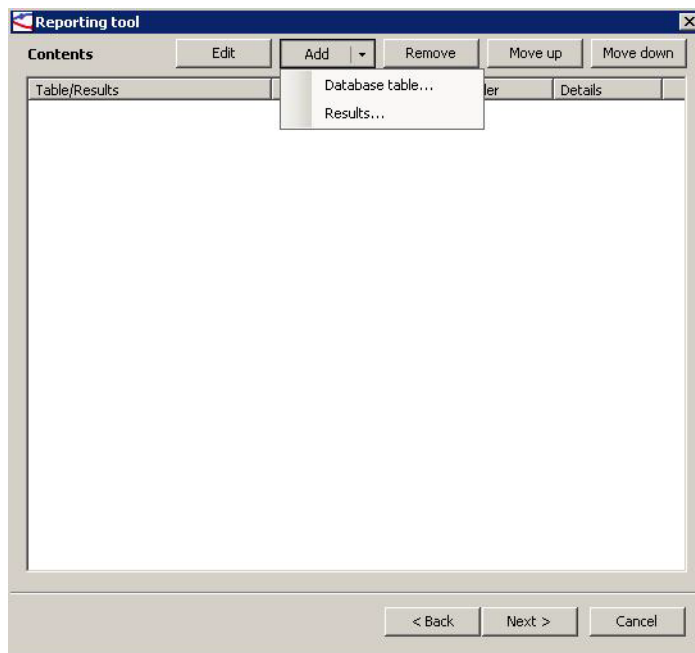


Figure 23.23 The report can contain information from both the database tables and from result files

After choosing the tables to add to the report, you afterwards choose which of the individual attributes of the table you wish to add to the report (see Figure 23.24). The 'Column title' field allows to provide a heading for the column in the report that is different from the attribute name in the database. Pressing 'Apply' after having chosen all the attributes from the table in question allows for adding more tables to the report. Pressing 'OK' closes the dialog and returns to the wizard.

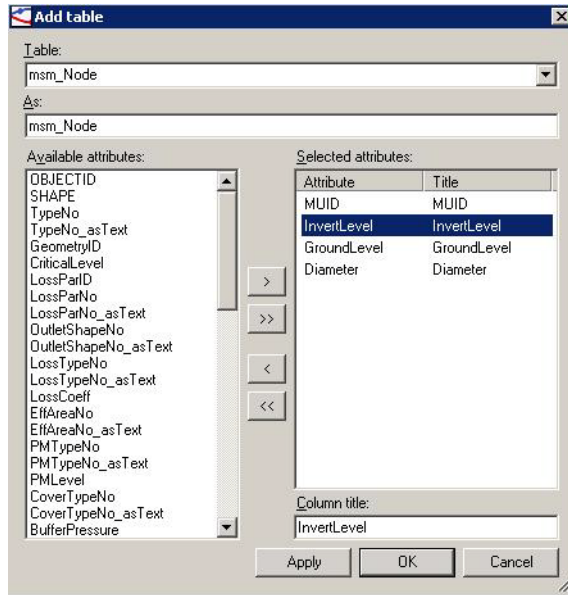


Figure 23.24 Specification of content from the table added to the report as well as the column title

An example of how the page looks after having specified some input from one of the MIKE URBAN tables is seen in Figure 23.25.

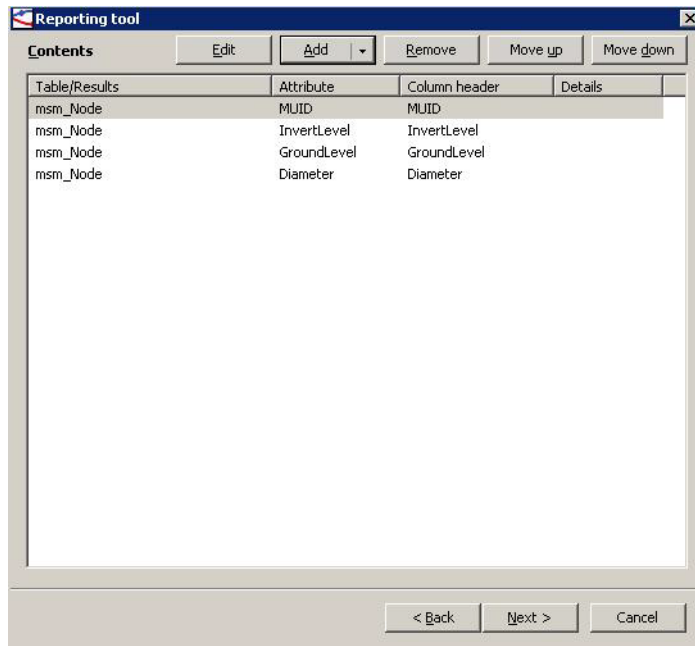


Figure 23.25 Information from tables as well as from result files can be added to the report.



Adding results is done in a similar manner. It is not necessary to have the result file added already, as it can be done from the dialog directly (see Figure 23.26). When adding results you can choose to add the accumulated value, the average value or maximum or minimum value. You can also specify a critical value. When you have chosen a critical value for a specific result type you can later on choose to use a different font color in the report for values that are above or below the critical value (depending on your choice).

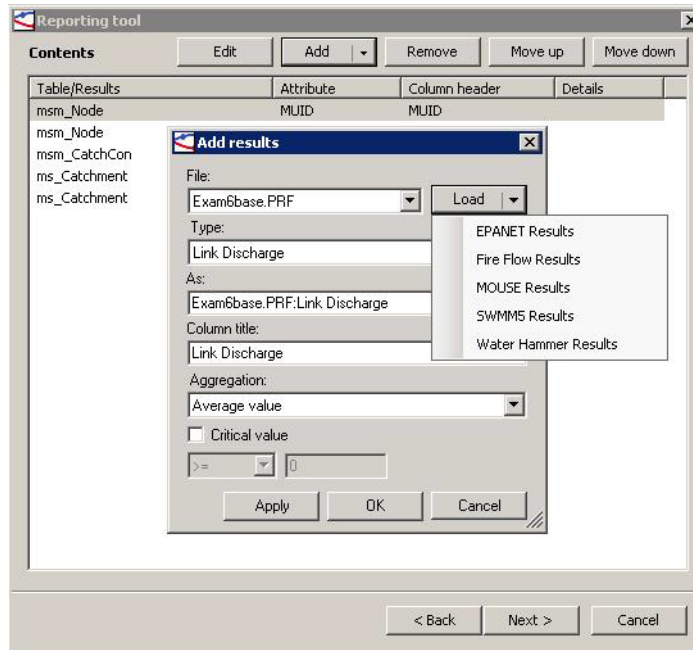


Figure 23.26 When adding results you can choose to display e.g. the average value or the maximum value

Join of tables and results

It is also possible to display joined tables in the report. Default MIKE URBAN comes up with a suggestion on which fields to use for the join. Typically this will be the unique ID for the table in question.

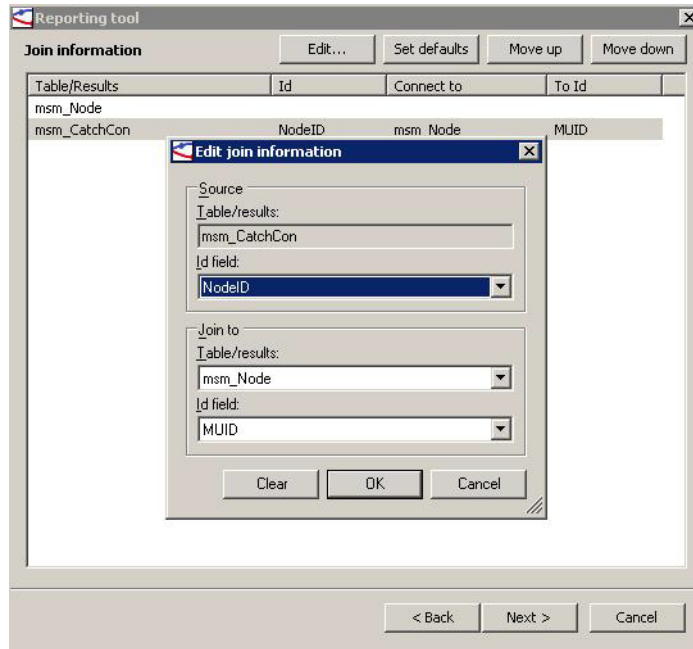


Figure 23.27 When joining tables you need to specify the common field in the two tables used for the join

So you start out by choosing the field that you will base the join on. In the example shown in Figure 23.27 the join is based on the 'MUID' in the msm_node table. The table that we wish to join information from is the msm_CatchCon table (containing the catchment connections for the MOUSE model) - the field in this table to base the join on is 'NodeID'. This means that the report will list all the catchments connected to each node.

Unmatched records will contain no values for fields being appended from the join table, e.g. if no catchments are attached to a specific node then the joined columns in the report will be left empty.

Additional tables can afterwards be joined - e.g. the catchment table can be joined in order to retrieve attributes from this table to the report (e.g. catchment area), see Figure 23.28.

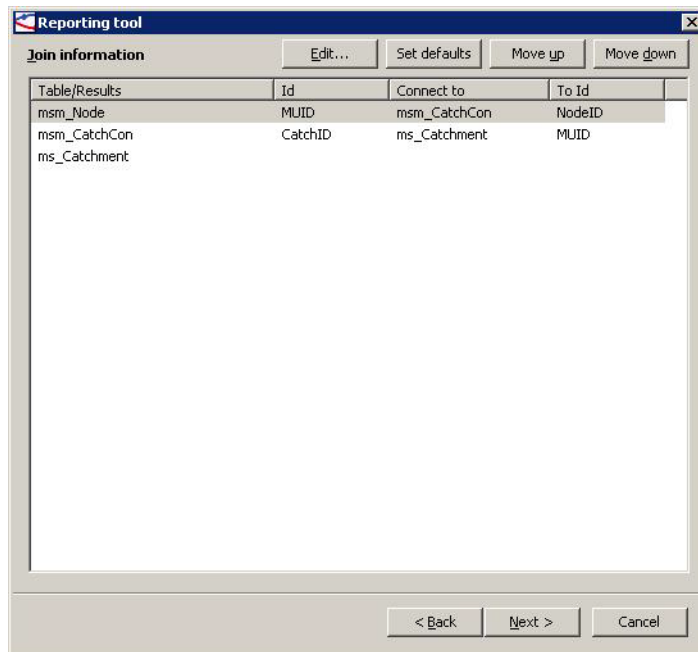


Figure 23.28 The joins as they appear in the wizard

Using filters

If you wish to set up the report only to show information from selected elements you can do so by specifying a filter. You can use 3 different types of filters (profile paths, LSE and MUS selection files). If a profile path filter is used, nodes and links are ordered according to the appearance on the profile path in the report. If no filter is used, all elements are reported in the order as they are extracted from the project's geodatabase.

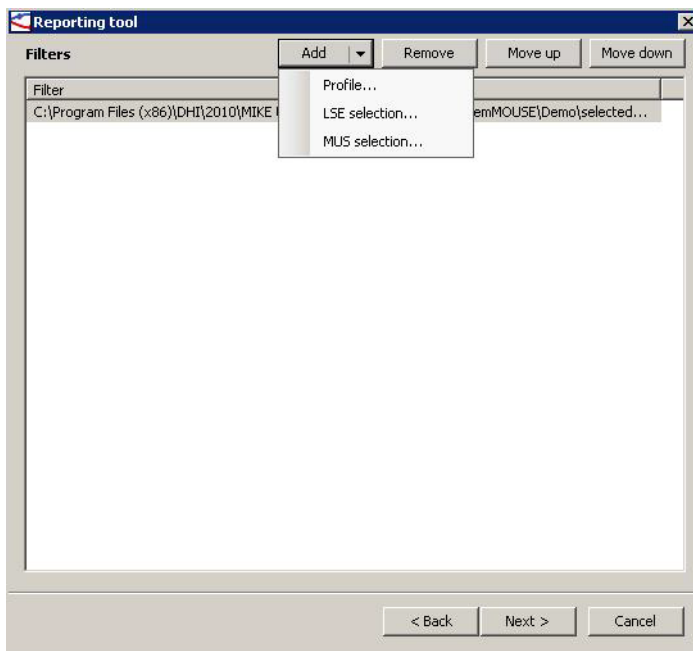


Figure 23.29 A report can be limited to selected elements only

Overall layout

Reports can be generated in two formats: HTML and CSV (see Figure 23.30). Style-sheets are used for the generation of these. Default MIKE URBAN comes with two different style-sheets, one for the HTML format (ResultsTableHTML.xsl) and one for the CSV format (ResultsTableCSV.xsl). The default style-sheets are installed in the configuration folder of your MIKE URBAN installation (the 'Cfg' folder in the MIKE URBAN installation directory).

It is also possible to use a custom style-sheet using the browse option to browse for the location of this.

The 'Split output' check box controls how the output is produced if several filters have been specified. If checked, separate output files are created for each filter; if unchecked all filters are merged into one output file. If there are no or just one filter specified to be used, 'Split output' is unchecked and disabled.

If 'Split output' is checked, the list shows output file names for each filter.

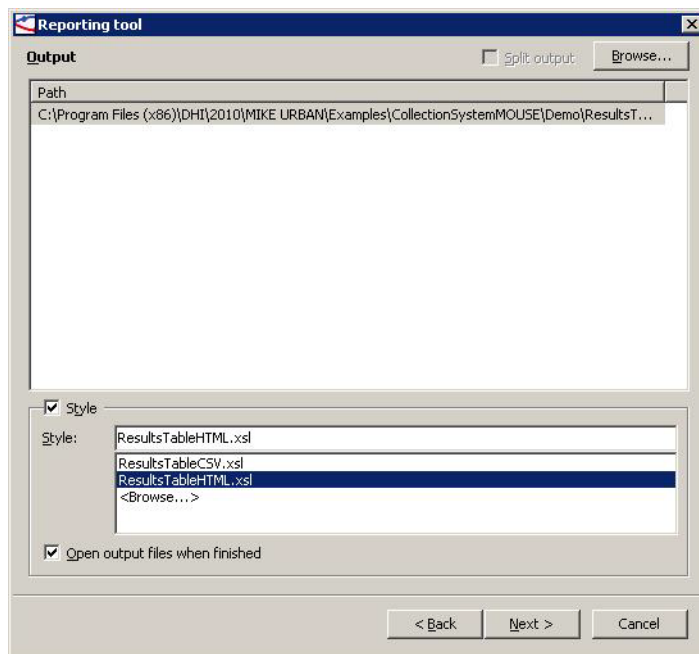


Figure 23.30 Specifying the format of the generated report

After having specified the output file and the format to be used you will be directed to a page where you can specify some additional layout options when using the HTML format (see Figure 23.31). If you have chosen the CSV format you will be directed directly to a page where you can choose to save the configuration of the report.

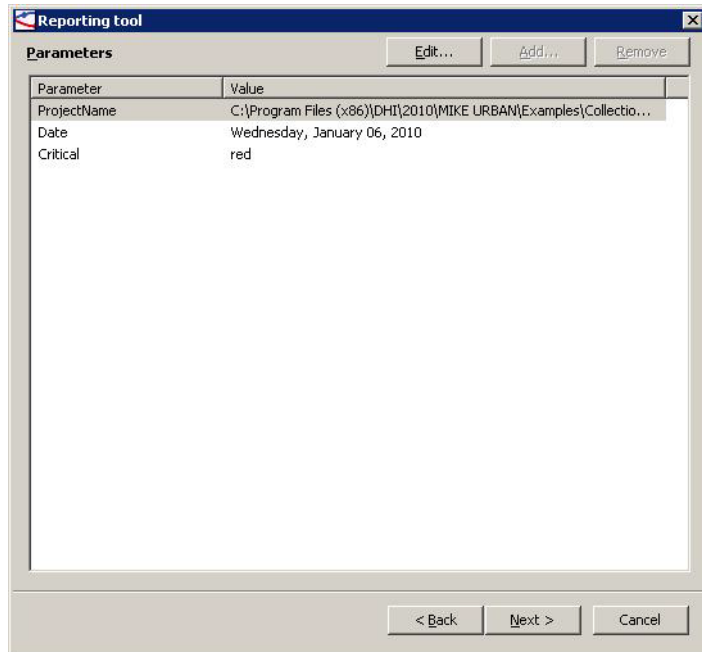


Figure 23.31 Additional layout options are available when the HTML format has been chosen for the report

The additional formatting allows to add a header for the report consisting of the 'Project name' and the 'Date'. Both fields are text fields. In addition to this the color of the critical levels can be specified. For available colors, please refer to Figure 23.32. The color can hence be specified as both the name (e.g. purple) or the color value ("#800080 in the case of purple). Most browsers accept a wider set of color names but use of these is not recommended.



Figure 23.32 Colors that can be applied in the html format



Saving the configuration file

Once the report layout is also in place it is recommended to save the report configuration (so that it may be reused later on), see Figure 23.33.

The next step produces the report(s).

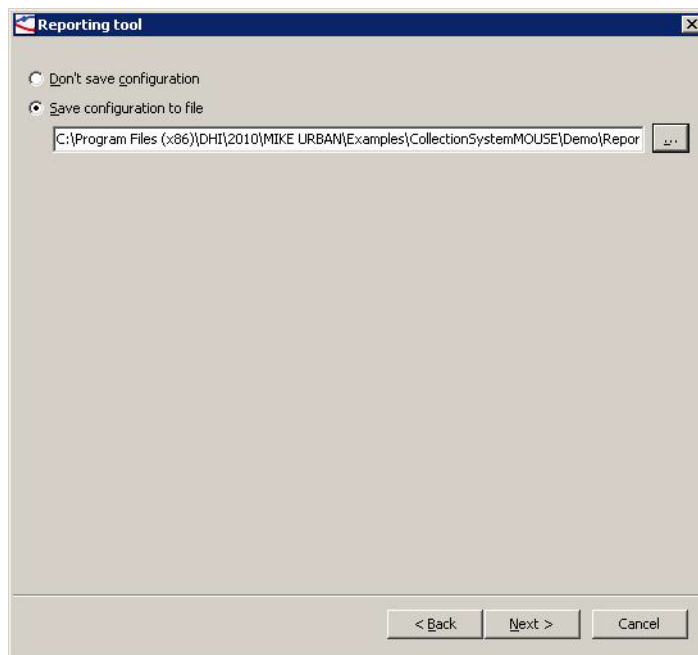


Figure 23.33 Saving the report configuration is recommended

23.13 Interpolate Raster

This feature interpolates a raster layer from a point layer attribute. This can either be an attribute in a data layer such as MOUSE nodes or WD junctions or it can be a statistics layer such as the maximum water level in nodes. With the latter combined with a DEM it is possible to generate approximate flood maps.

The feature is invoked by choosing Model Results|Interpolate Raster. This will open the dialog shown in Figure 23.34.

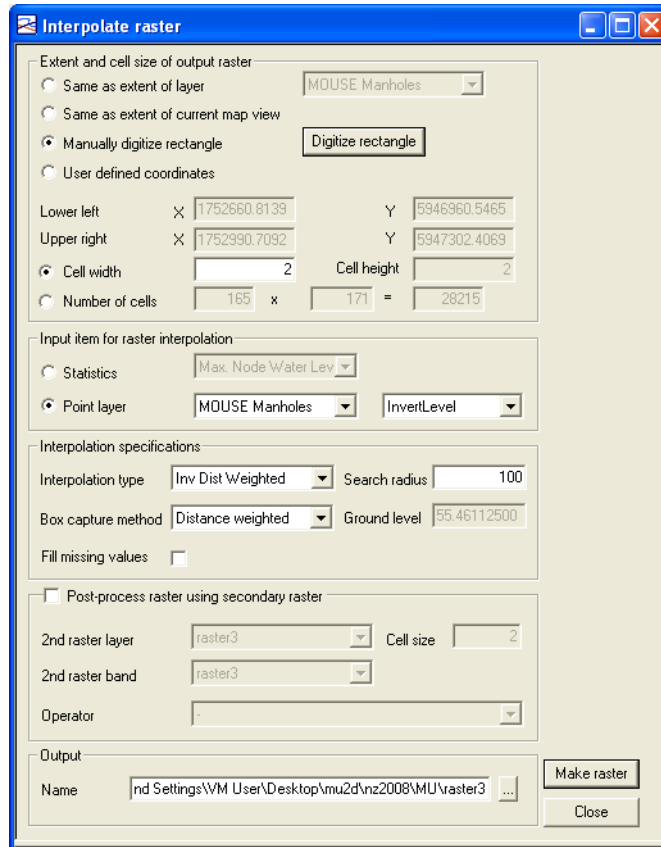


Figure 23.34 Raster Layer dialog

The dialog offers the following choices:

Extent and cell size of output raster

The extent of the raster layer to be generated is defined by the X and Y coordinate of the lower left and upper right corner of the extent. These coordinates can be defined in one of the following four ways:

1. Same as extent of layer. A MOUSE or WD data layer is chosen and the raster extent will be chosen such that it encapsulates the data sets in the chosen data layer.
2. Same as the extent of the current map view. The extent of the raster layer will equal the extent of the area currently visible in the map view.
3. Manually digitize rectangle. Clicking this button will make the dialog disappear and the user should subsequently digitize a polygon in the map view. Each point in the polygon is defined with a single click and double-clicking will add the last point as well as terminate the polygon. The dialog will re-appear and the raster extent will be chosen such that it encapsulates the digitized polygon.



4. User defined coordinates. With this option chosen the X and Y coordinate of the lower left and upper right corner will be enabled and the user should enter the values.

Input item for interpolation

The data set used as input for the interpolation can either be chosen as a layer generated with the Result Statistics feature or as an attribute in one of the MOUSE or WD data sets currently holding data and being a point layer.

Interpolation specifications

The method used to interpolate the raster from the point data set is selected here. Three methods are available: Inverse Distance Weighted, Inverse Square Distance Weighted and Kriging. The parameters associated to each interpolation method are different and they should be chosen with care as the interpolation results may strongly depend on the selection of parameters. Unless very familiar with this feature, different parameters should always be tested to evaluate sensitivity.

Post-processing raster using secondary raster

The raster generated can be post-processed by combining it with another raster layer. This is called the 2nd raster layer. Possible post-processing methods includes +/- as well as two methods that cut off the generated raster if it is lower or higher than the 2nd raster layer. The latter may be used to cut off approximate flood maps when the interpolated flood level is less than the DEM.

Output

When clicking the Make Raster button the generated raster will be saved to the selected file name as well as added to the TOC and map view.

23.14 Profile plots

Profile plots can be produced with or without results. The profile plot is drawn between the flags that are being specified. If two or more ways can be found between two flags any one of these is being used. Hence, in order to control the path chosen, more flags must be set until a unique path between flags can be determined.

- Go to Model Results|Profile Plot. Choose Define Path Flags. The cursor then changes to a small flag. Then point to the first node in the profile. This will place a small flag next to the node along with the number of the flag



- Now continue until the path is well-defined. The horizontal plan will then look as seen in Figure 23.35. If you choose 'Save Profile' from the local menu on the horizontal plan the profile path is saved to a .profile file with a user defined name. A profile path can be loaded again by using 'Load Profile' in the local menu.

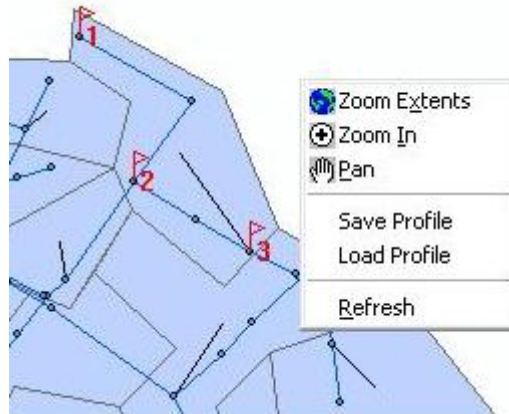


Figure 23.35 The flags defining the path of the profile plot have been set

- Finally choose Model Results|Profile|Create Profile Plot. This will create a new window with the profile plot. When the plot is drawn all elements in the profile path will be marked as selected in the horizontal plan making it easy to determine if the profile followed the desired path. In case any results are loaded the profile will automatically add the water level as an item in Graphical items.

23.14.1 The context menu options

When you right-click on the longitudinal profile the context menu opens (see Figure 23.36).

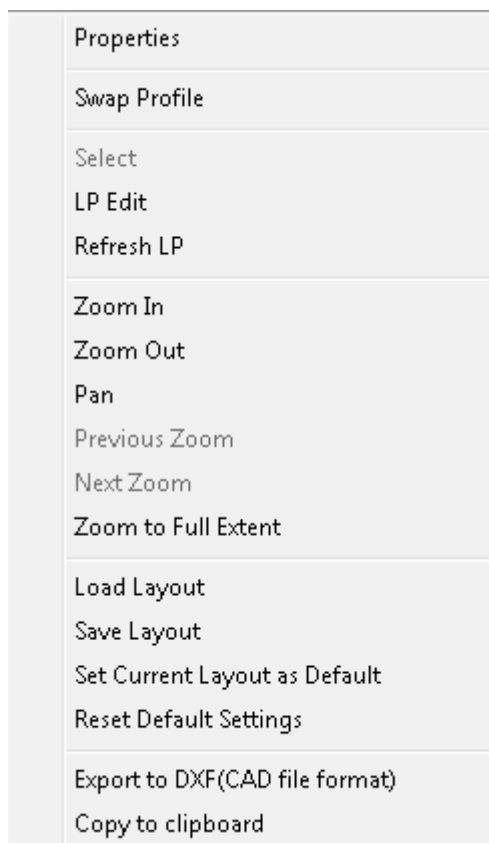


Figure 23.36 The context menu on the longitudinal profile

Properties

See section 23.14.2.

Swap Profile

Will swap the profile, e.g. swap profile from being drawn from node A to node B to being drawn from node B to node A.

Select

After choosing a select tool on the normal selection tool bar in MIKE URABN the 'Select' menu point will get activate and allow to select elements in the longitudinal profile.

LP Edit

When in an edit session the 'LP Edit' can be activated. This turns the cursor into a small hand with which you can move the pipe in the longitudinal profile. If you position the hand in the middle you can move the entire pipe up and



down. If you position the hand at either end you can move the connection of the pipe at that end. It is not possible to move the pipe above ground or below the inverts of the nodes. Once you leave the edit session you will be asked if you wish to save the changes or not.

Refresh LP

Refreshes the plot and zooms out to full extent. If you have made changes e.g. to a node invert level in the node editor while the longitudinal profile with the node in question was open, the refresh will ensure that the new invert level is drawn.

Zoom In, Zoom Out, Pan, Previous Zoom, Next Zoom, Zoom to Full Extent

Allows to zoom in and out as well as return to previous zoom positions and back again. It is also possible to pan. Zoom to full extent brings you back to the full view of the longitudinal profile.

Load Layout, Save Layout

The 'Save Layout' saves the current profile layout (i.e. display properties) to a .XML file. This is the same file that can be saved using the [Save] button in the Profile Properties dialog (see also section 23.14.2). The 'Load Layout' loads the profile layout from a .XML file. Note, the profile layout file includes references to specific result file names in the Graphical Items table and in the Raster Items table, so by loading the profile layout file you may be including references to result files that you do not want displayed in the profile.

Set Current Layout as Default

Saves the current profile layout as the default layout for all profiles for all projects.

Reset Default Settings

Resets the default profile layout for all MOUSE model projects to the original manufacturer settings.

Export to DXF (CAD file format)

Allows to save the profile in a DXF-file.

Copy to clipboard

Copies the longitudinal profile displayed to the clipboard and allows it to be pasted into other applications.

23.14.2 Longitudinal profile properties

The properties of the longitudinal profile can be changed by choosing 'Properties' on the local menu of the longitudinal profile. This opens the dialog as seen in Figure 23.37.

Any changes made in the properties pages will be temporarily saved, and re-applied the next time the longitudinal profile is opened within the same session. This means that it is possible to retain the configuration of the profile plot when viewing pipe stretches in different parts of the model.

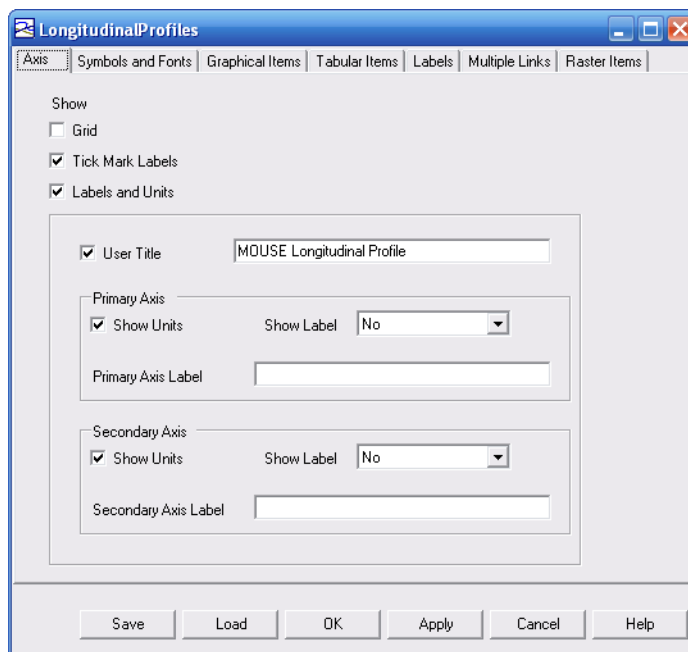


Figure 23.37 Setting the properties of the profile plot

The various tab pages will allow you to change the properties for the longitudinal profile. In the bottom of the 'Properties' page basic set of buttons are available:

Save

Will save the settings for the longitudinal profile in an xml-file with a user specified filename. The saved file includes the settings of both graphical and tabular result items including references to specific result file names.

Load

Will load setting for the longitudinal profile from the specified file.



OK

Will apply the settings specified and close the properties dialog.

Apply

Will apply the settings specified, but leave the properties dialog open.

Cancel

Will cancel any changes made and close the properties dialog.

Help

Will activate the online help.

Axis

On the Axis page various properties related to the axes can be set, see Figure 23.37.

Show

- Grid: Toggles grid on/off
- Tick mark labels: Toggles values for the axes on/off.
- Labels and Units: Toggles labels and units on/off for the axes. Please note that this switch will overrule the settings under primary and secondary axis.

User Title

If a tick mark is set in 'User Title' the text written in the edit field following the tick mark is used as the main title for the longitudinal profile.

Primary Axis

- Show units: Toggles on/off units on the primary axis
- Show Label: No, Horizontal, Vertical are the choices. These specify where the label will be drawn
- Primary Axis Label: Edit field for writing user defined label for the primary axis

Note: The primary axis settings are overruled by the setting 'Show | Labels and Units'

Secondary Axis

- Show units: Toggles on/off units on the secondary axis
- Show Label: No, Horizontal, Vertical are the choices. These specify where the label will be drawn
- Secondary Axis Label: Edit field for writing user defined label for the secondary axis



Note: The secondary axis settings are overruled by the setting 'Show | Labels and Units'

Symbols and Fonts

On the Symbols and Fonts page various properties related to the these can be set, see Figure 23.38.

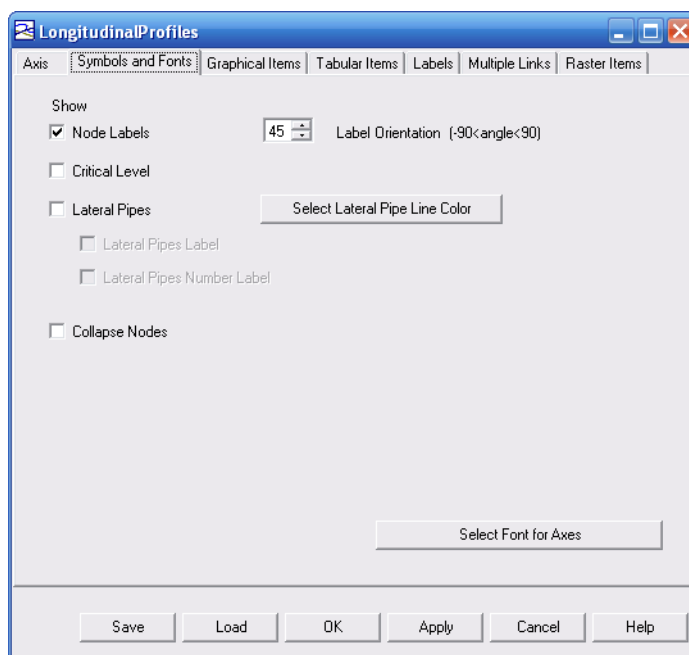


Figure 23.38 Symbols and Fonts Settings

- Node Labels: Toggles on/off the label of the node above the node in the profile. The orientation of the label is given in the 'Label Orientation' field
- Critical level: Toggles on/off if a red line marking critical levels (specified by the user) are drawn on the longitudinal profile
- Lateral Pipes: Toggles on/off if a line is drawn in nodes that have connected links that are not in the path of the lateral profile. The line is drawn at the invert level of the incoming pipe. The line color can be user specified by pressing 'Select Lateral Pipe Line Color'
 - Lateral pipes Labels: Toggles on/off display of ID-labels for the lateral pipes
 - Lateral Pipes Number Labels: Toggles on/off display of a number label showing the total number of lateral pipes connected to a node
- Collapse Nodes: Toggles on/off if nodes are drawn on the longitudinal profile. Useful for long profiles with many nodes.



- **Select Font for Axes:** Allows to change font for the axes. A dialog will come up where the font may be changed.

Graphical Items

On the Graphical Items page it is possible to specify which data from results are to be drawn in the longitudinal profile, see Figure 23.39. The various display options available for the graphical items can be reviewed and changed on this page.

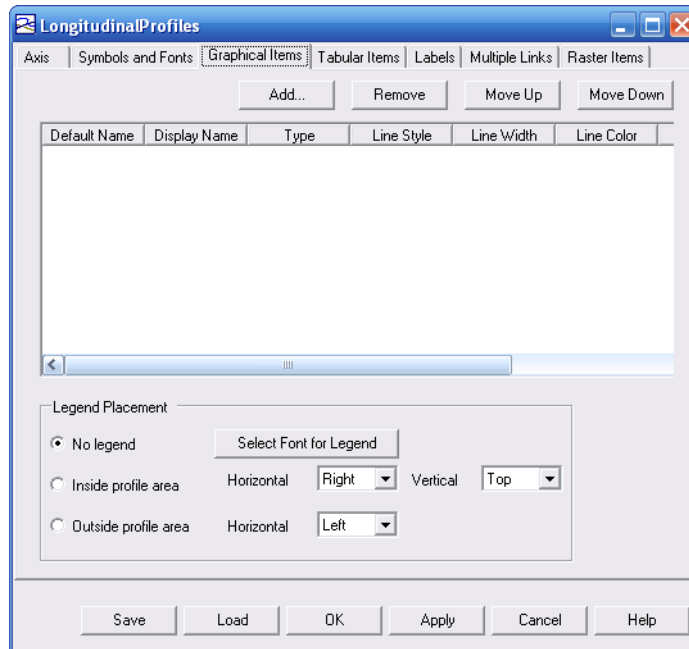


Figure 23.39 The Graphical Items properties

The buttons in the top of the Graphical Items page have the following functionality:

- **Add ...:** will open up an "add item" dialog, shown in Choosing results to display on the profile plot. From this dialog it is possible to select between the various result items found in the loaded result files.
- **Remove:** will remove the active graphical item in grid.
- **'Move Up' and 'Move Down':** will move the active graphical item up or down respectively to determine the drawing order.

The results in the profile plot may be animated. The actual time of the animation is seen in the lower right corner of the longitudinal profile.

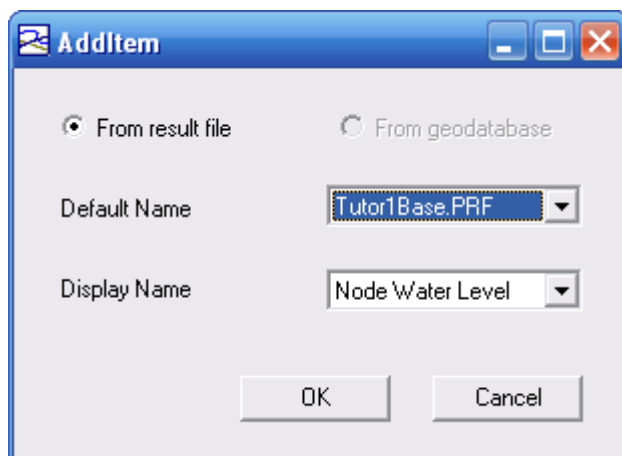


Figure 23.40 Choosing results to display on the profile plot

Graphical items in grid

The fields in the grid are activated by a click with the pointer. Upon activation many fields will display the current setting and next to this a 'down'-button. Pressing the 'down'-button will reveal the available options.

The columns in the grid in of the Graphical Items page are:

- Default Name: Name taken from the result item (cannot be edited)
- Display Name: User specified name - used for display in the legend
- Type: Field where you choose which type of graph to show. This option is mainly used for dynamic items. Choose between:
 - Not-filled animated: draws data as an animated graph
 - Filled animated: draws data as an animated graph where the area under the graph is filled. Well suited to display i.e. the water level in sewers.
 - Min: draws a static graph with the minimum values found in the time series for each point in the profile.
 - Max: draws a static graph with the maximum values found in the time series for each point in the profile.
 - Min and Max: draws static graphs with the minimum and maximum values found in
- Line Style: Field where you choose line style. Pressing the button-down field will reveal the various styles.
- Line Width: Field where you choose line width. Pressing the button-down field will reveal the various widths.
- Line Color: Field where you choose line color. Pressing the button-down field will reveal the various colors.
- Display Line: Ability to toggle on/off display of line.



- **Marker Style:** Field where you choose the symbol to use as marker. Pressing the button-down field will reveal the various symbols.
- **Marker Size:** Field where you choose size of the marker.
- **Marker Color:** Field where you choose marker color. Pressing the button-down field will reveal the various colors.
- **Display Marker:** Toggles on/off display of marker.
- **Fill Color:** Color of the filled items.

Legend Placement

The legend placement is fixed for the entire profile - hence all legends are displayed as a list within the same frame. The following options exist:

- **No legend:** No legend is drawn
- **Inside profile area:** The legend is drawn within the profile area. The placement of the legend is determined by the horizontal and vertical option. Horizontal box contains the values: Left, Middle, Right. Vertical box contains the values: Top, Middle, Bottom.
- **Outside profile area:** The legend is drawn outside of the profile area. The placement of the legend is determined by the horizontal option. Horizontal box contains the values: Left, Right.

Tabular Items

On this page, see Figure 23.41, it is possible to specify the content of the table below the longitudinal profile. The buttons in the top of the Tabular Items page have the following functionality:

- **Add ... :** will open up an "add item" dialog. From this dialog it is possible to select between the various result items found in the loaded result files as well as data from the tables in the geodatabase.
 - **Remove:** will remove the active tabular item in grid.
 - **'Move Up' and 'Move Down':** will move the active tabular item up or down respectively to determine the display order.
- 'Select Font for Table' controls the font settings for the table.

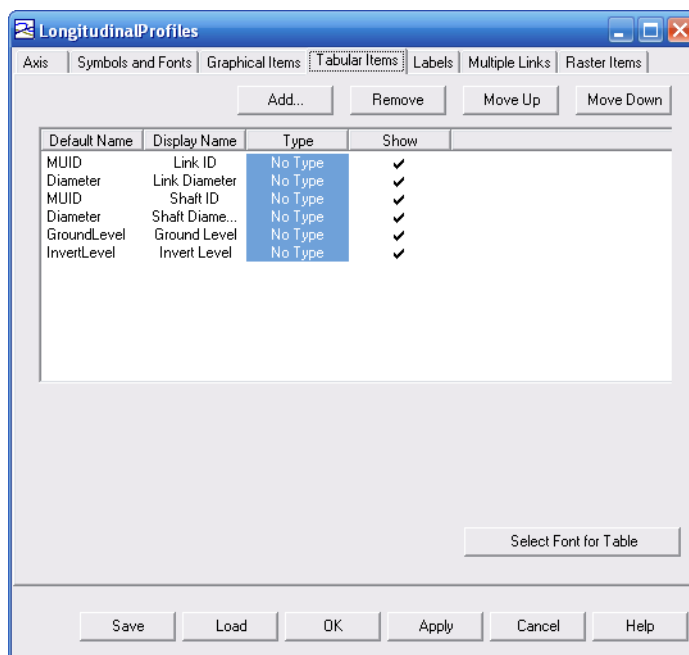


Figure 23.41 The Tabular Items properties

The columns in the grid in of the Tabular Items page are:

- Default Name: Name of the result item in the result file (cannot be edited)
- Display Name: User specified name - displayed as row label in the table
- Type: Field where the data type is selected. Fields with blue background indicate that the data are static and the type cannot be changed by the user. If the source data for a tabular item are dynamic the options are:
 - Dynamic: displays the dynamic values found in the time series for each valid point in the profile. The tabular values will be updated in synchronisation with the graphical items and other result items.
 - Min: displays the minimum values found in the time series for each valid point in the profile.
 - Max: displays the maximum values found in the time series for each valid point in the profile.
- Show: option that toggle on/off display of the selected tabular item.

When two adjacent cells in a row have the same value the cells will merge and only one value is displayed. This is illustrated in Figure 23.42 where the merged cells are marked with a red punctured line.



Link ID	8111	4511	7811	7911	
Link Diameter	0.4000	1.0500	0.6500		
Shaft ID	81	45	44	78	79
Shaft Diameter		1.0000			
Ground Level	198.84	198.61	198.29	197.83	198.33
Invert Level	195.22	193.66	193.64	194.05	195.55
Node Water Level	195.222		193.675	194.073	195.572
					07-08-1994 17:11:00

Figure 23.42 Adjacent cells in the same row will result in a merge of cells in the table

The two cells marked with a red punctured line are merged and represented by one value because they contain identical values.

Labels

On this page, see Figure 23.43, page it is possible to define labels that will appear on the specified location on the longitudinal profile.

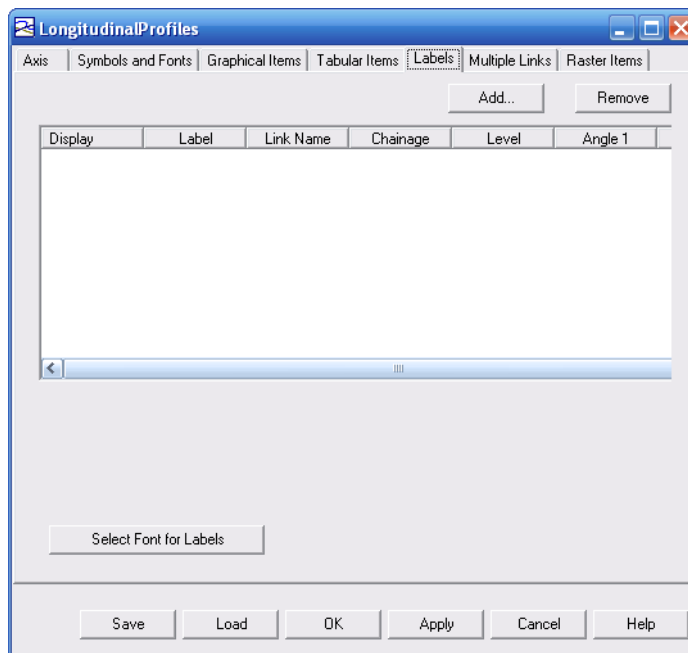


Figure 23.43 The Labels properties

The buttons in the top of the Tabular Items page have the following functionality:

- Add... : Adds a new item that then must be configured by the user.
- Remove: Removes the currently selected label.

The columns in the grid in of the Labels page are:



- Display: Tick mark to toggle on/off display of label
- Label: Label string that will be displayed. The text is reset when the Link name is changed.
- Link Name: Combo box that will display available link ID's in the profile
- Chainage: The distance to place the label from the start of link. Is used to shift the label in the horizontal plan.
- Level: Determines the vertical position of the place the label line points to.
- Angle 1: The angle between the label line and a horizontal line. The angle is given counter clockwise.
- Angle 2: The angle of label itself. The angle is given counter clockwise.
- Label length: Length of the line connecting the place the label points to and the label itself.

Example of how it may look is seen in Example on how the label appears on the longitudinal profile (Label = Link:79|1, Chainage = 20 Level = 196 m, Angle 1 = 70 , Angle 2 = 20 , Label length = 10).

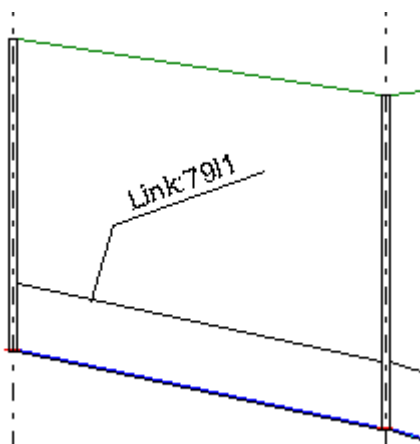


Figure 23.44 Example on how the label appears on the longitudinal profile

Multiple Links

On the Multiple Links page it is possible to control the display in places where two or more parallel links exist between two nodes. The grid on this page is automatically populated with the parallel links whenever these exist within the path of longitudinal profile.

Per default all links between two nodes are drawn when making a longitudinal profile, even if this means that some links will appear to cross one another. Results will be shown in all visible pipes.

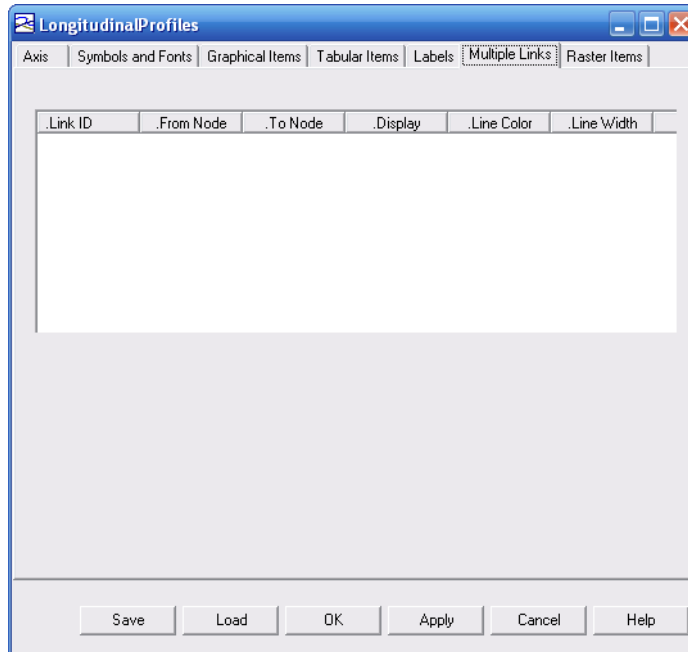


Figure 23.45 The multiple links page

An example of a network with two parallel pipes is seen below

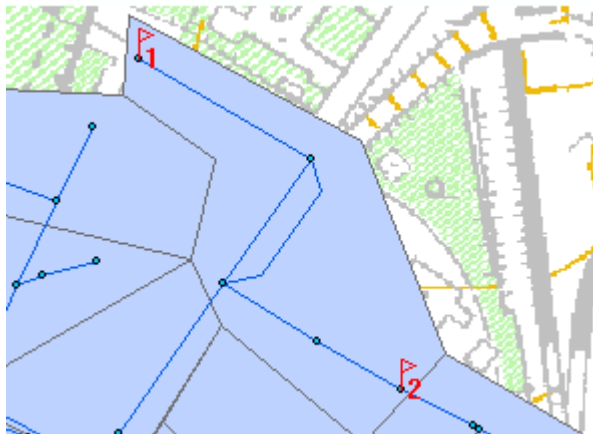


Figure 23.46 Example of two pipes between the same nodes

In the given example the computed length of the one link will be longer than the length of the other. In cases where the length of parallel pipes differs the following logic is applied:



- Computed length: In case the computed length of the two parallel pipes are different, the distance of the longest pipe is used as distance between manholes, and a gap between the downstream node and the short pipe will appear.
- User defined length: In case one of the parallel links is specified with a user defined length that is longer or shorter than the computed length, that link will be drawn longer/shorter at the downstream node.

The columns in the grid in of the Multiple Links page are:

- Link ID: Name that identifies the link (field is not user editable)
- From Node: Name that identifies the upstream node for the current link. The sets of links that are parallel can be identified by the fact that they share the same From node and To node. (field is not user editable)
- To Node: Name that identifies the downstream node for the current link. The sets of links that are parallel can be identified by the fact that they share the same From node and To node. (field is not user editable)
- Display: field that toggles display on/off for the individual link. It is not allowed to turn display off on all links between two nodes; at least one link must be visible.
- Line Color: Field where line color is specified. Can be used to give each parallel link an individual appearance.
- Line Width: Field where line width is specified. Can be used to give each parallel link an individual appearance.

Raster Items

On the Raster Items page it is possible to specify which data from raster files are to be drawn in the longitudinal profile. The various display options available for the raster items can be reviewed and changed on this page.

Raster items like digital terrain models (dtm/dem) and flooding results (dfs2) are drawn as graphs in the longitudinal profile. The graphed values are found by projecting the raster data onto the profile path.

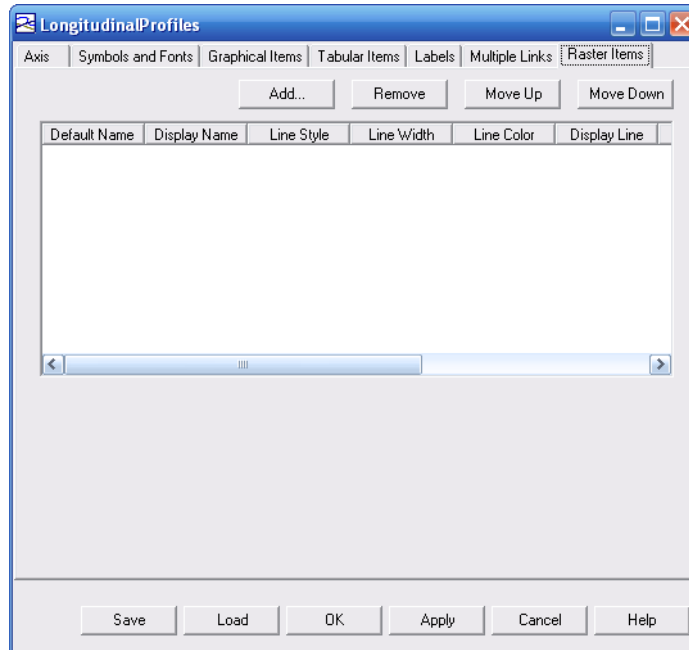


Figure 23.47 The Raster Items page

The buttons in the top of the Raster Items page have the following functionality:

- Add...: will open up an "add item" dialog. From this dialog it is possible to select between the various items found in the loaded raster files.
- Remove: will remove the active raster item in grid.
- 'Move Up' and 'Move Down': will move the active raster item up or down respectively to determine the drawing order.

If the raster item contains dynamic values the plot may be animated. The actual time of the animation is seen in the lower right corner of the longitudinal profile.


The columns in the grid in of the Raster Items page are:

- Default Name: Name that identifies the raster dataset
- Display Name: User specified name - used for display in the legend
- Line Style: Field where you choose line style.
- Line Width: Field where you choose line width.
- Line Color: Field where you choose line color.
- Display Line: option that toggle on/off display of line.
- Marker Color: Field where you choose marker color.
- Display Marker: option that toggle on/off display of marker.



23.14.3 Selecting

Selecting features directly in the longitudinal profile is possible by activation of the selection tool in the main menu and subsequent activation of 'Select' in the local menu of the longitudinal profile.

When in selection mode the pointer changes to  thus indicating that the selection tool is active.


The selection mode is controlled through the main menu. The ordinary rectangular selection mode found in MIKE URBAN is available in the longitudinal profile; while select by polygon is not available.

The displayed selection in the profile is synchronized with both the horizontal plot and the editors. When elements are selected in the longitudinal profile they are highlighted by showing a turquoise line surrounding each of them.

To exit selection mode choose another pointer mode in the local menu i.e. zoom in - which is the default pointer mode.

23.14.4 LP Edit

Editing links in the longitudinal profile is possible by starting an edit session in the main menu and subsequent activation of 'LP Edit' in the local menu of the longitudinal profile. The LP edit tool is synchronized with the ordinary editing in MIKE URBAN undo is available.

When in edit mode the pointer changes to  thus indicating that the edit tool is active.

The LP Edit tool allows the user to graphically change the invert levels of links by simple click-and-drag. When in LP edit mode the pointer will work in the following way:

- To move a link: Grab a link by holding down the left mouse button, and then move the link by sliding the pointer up or down.
- Grab in middle of a link: Will shift the whole link up and down while maintaining the same slope. The limitation to the movement is that ground and invert levels of the nodes in either end of the link cannot be exceeded.
- Grab in the ends of a link: Will move the invert level of that end of the link - with the limitation that ground or invert level of the connecting node cannot be exceeded.

If levels are already exceeded to begin with, starting to edit will snap to ground/invert level. This means that a click in the middle of a pipe will shift the whole pipe when snap occurs, while a click on end of a pipe will only move that end of the pipe when snap occurs.



For all structures it is possible to open the related editing dialog by double-clicking on the element while in "LP Edit" mode.

To exit LP Edit mode choose another pointer mode in the local menu i.e. zoom in - which is the default pointer mode.

23.14.5 Refresh LP

Use Refresh LP to update the longitudinal profile graphics without closing the profile.

While editing Refresh LP can be used to see how the latest changes made look in the longitudinal profile.

23.14.6 Export to CAD

Export of the longitudinal profile into a CAD file-format can be done from the local menu under the menu point 'Export to DXF (CAD file format)'.

The elements found in the profile are exported into separate layers in a dxf-file. Loaded results or items added to LP will also be exported to the dxf-file. The following separate layers will, if relevant, be created when exporting: Legend, Labels, Tabular items, Graphical items.

Pressing 'Export to DXF (CAD file format)' will allow to save the profile to a dxf-file.

The created CAD file is primarily for presentation purposes. It is not recommended to use the dxf-file as basis for real life work or maintenance of sewer pipes.

23.15 Animations

After having loaded in a result file, see 23.1, and having added a result layer to the TOC, see 23.2, it is possible to make an animation of the results. To do so:

- Go to Model Results|Animate, see Figure 23.48
- Press Play in order to play the animation. The various options for playing the animation can be seen in Figure 23.48.

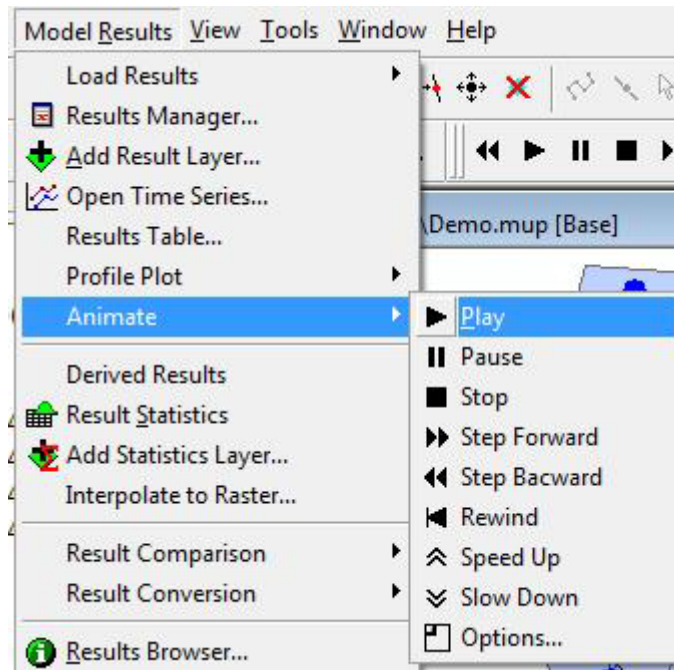


Figure 23.48 Animation of results

Some more advanced settings for the animation can be seen in Figure 23.49. It is e.g. possible to animate only a part of the result file.

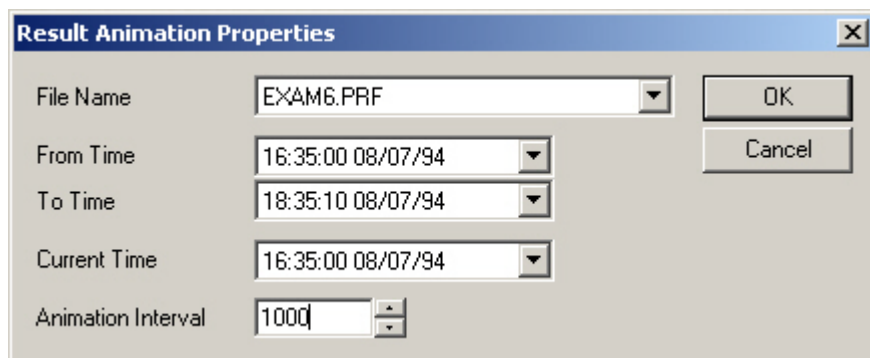


Figure 23.49 Setting the properties for the result animation

If a time series plot is open or a longitudinal profile is open all animations can be synchronized (default).



23.16 Result comparison

It is possible to compare results that are computed for the same network. The comparison basically subtracts the results from one another allowing a difference plot to be plotted on e.g. the horizontal plan or in the time series.

In order to compare results do the following:

- Run a simulation twice with various input parameters - but not with differences in the physical model.
- Go to Model Results| Compare Results| ... and choose the relevant result files to compare, see Figure 23.50
- After choosing a file type and the first result file in question, e.g. MOUSE Results the ordinary result file selection dialog comes up, see I - after pressing OK it will load the result file and then immediately after open the dialog where you can load in the second result file.

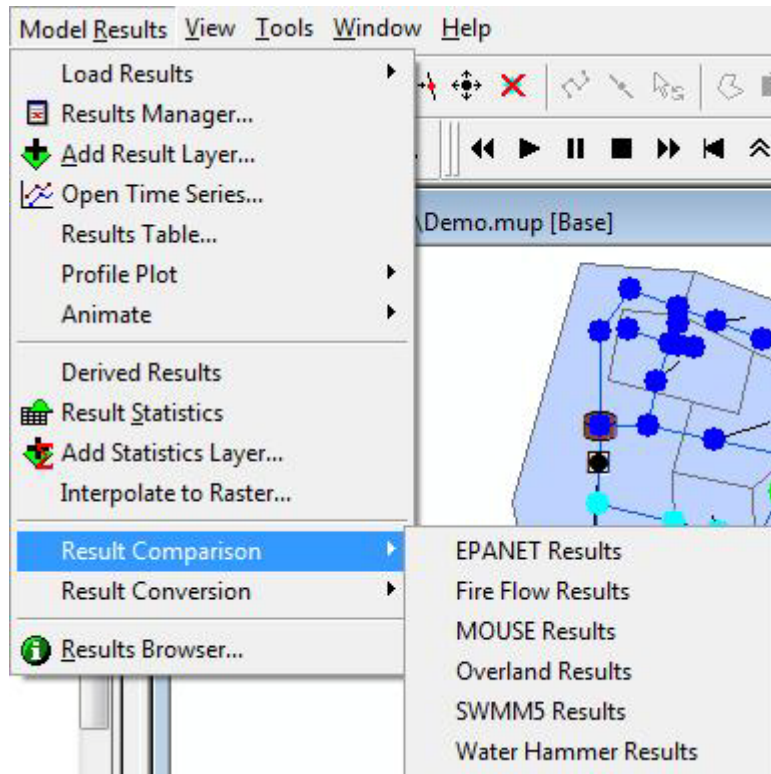


Figure 23.50 Loading result files for comparison of results

- After loading in the second result file, go to Model Results|Add Result|Layer to add the comparison to the horizontal plan or go to Model Results|Time Series to plot the compare time series. On the result selection dialog that comes up choose the result file type that starts with 'Comp' (for comparison) and has the first result file name afterwards, see an example in Figure 23.51

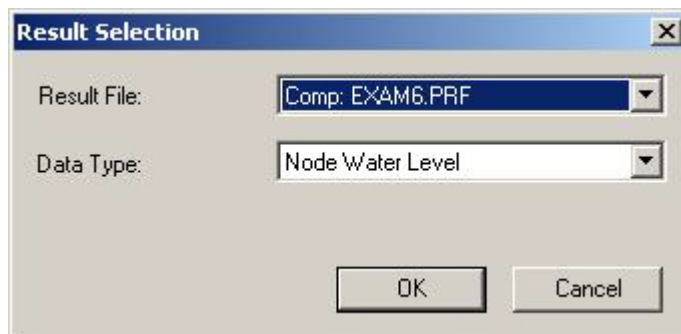


Figure 23.51 The result selection dialog where the comparison between resultfiles is chosen

- Then choose the data type to compare with. The comparison will take a value from the first result file and subtract the corresponding value of the second result file. And this difference will then be plotted.

23.17 Export of results to ESRI shapefiles

In order to export MIKE URBAN simulation results to ESRI shapefiles click on the results layer in the Table of Contents (ToC) and select 'Export layer to shapefile' from the pop-up menu on the right mouse click.

Specify the name of the shapefile and click OK. This will export the selected results layer to ESRI shapefile. The exported shapefile contains the layer geometry, the simulated result item value, and the unique MUID. Note, that using the unique MUID you can link (join) the result layer with the input data such as pipes, manholes, junctions, and similar.

23.18 Layout and advanced GIS

If you wish to do more advanced layouts of your MIKE URBAN results, e.g. for reports, choose Tools|Layout and Advanced GIS. This will then take your current map in MIKE URBAN and open it up in ArcMap.



23.18.1 Adding and animating results in ArcMap

After having transferred the horizontal plan from MIKE URBAN you can also choose to add result layers on top of this and perform an animation of the results (as in MIKE URBAN). In order to do so you need to utilize the MIKE URBAN extension for ArcMap. This is activated in ArcMap by Custom-ize|Toolbars|MIKEURBAN Extension, see Figure 23.52.



Figure 23.52 The MIKE URBAN extension toolbar in ArcMap



Allows to open a result file. The dialog show in Figure 23.53 will open. After choosing a file type you then choose the relevant result file.



Allows you to reconnect result layers.



Allows to add result layers to the map.

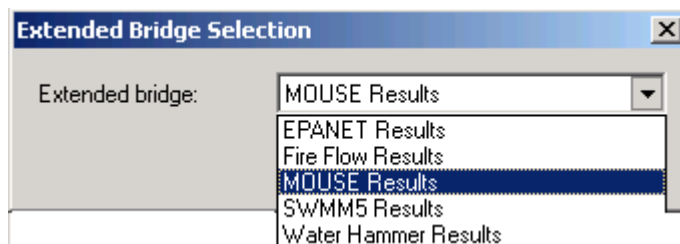


Figure 23.53 Loading a result file

If you already had result layers added in MIKE URBAN when activating Lay-out and Advanced GIS you only need to open the result file and reconnect the result layers in ArcMap. This is done by first opening the result file from the toolbar and then reconnecting the result layers (all layers will be recon-nected). It is always possible to add additional result layers.

If you do not have results layers added before activating Layout and Advanced GIS, you need to open the result file and then add the result layers.



23.19 Presentation of Water Distribution results

Some items under the Model Results menu are only visible when working in Water Distribution mode, see Figure 19.65. These will be described in the following.

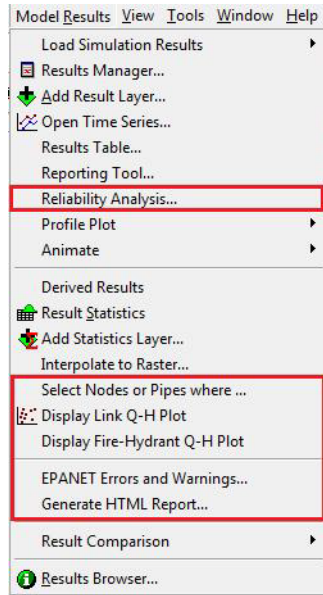


Figure 23.54 Menu items only visible in WD mode

23.19.1 Select Nodes or Pipes where ...

Makes it 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 (e.g. pressure) have changed above a specified threshold.

Example 1

In order to select (display) pipes, where the flow direction has changed during the simulation period, load the simulation results and select Tools | Select nodes and pipes where... .

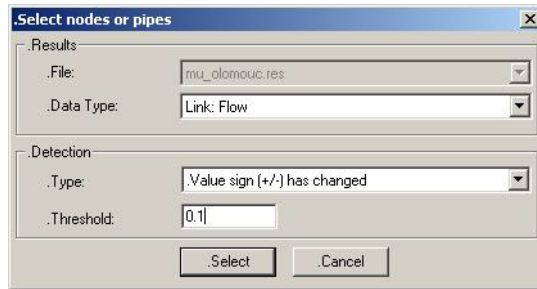


Figure 23.55 Select nodes or pipes where...

Select "Link: Flow" result item from the results data type and select "Value sign (+/-) has changed" from the detection type. Define the threshold value, which is used to detect the change. Click "Select" when ready.



Figure 23.56 Pipes where the flow direction has changed are selected on the Map

Example 2

In order to select (display) nodes, where the node pressure has changed during the simulation period more than 15metres, load the simulation results and select Tools | Select nodes and pipes where... .

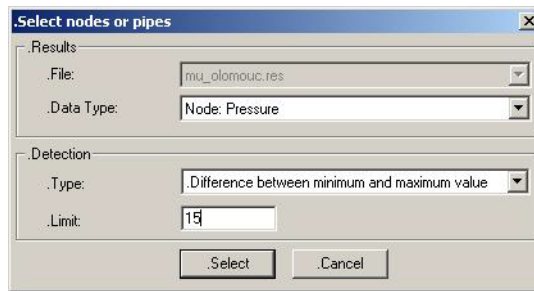


Figure 23.57 Select nodes or pipes where ...

Select "Node: Pressure" result item from the results data type and select "Difference between minimum and maximum value" from the detection type. Define the limit value, which is used to detect the change. Click "Select" when ready.

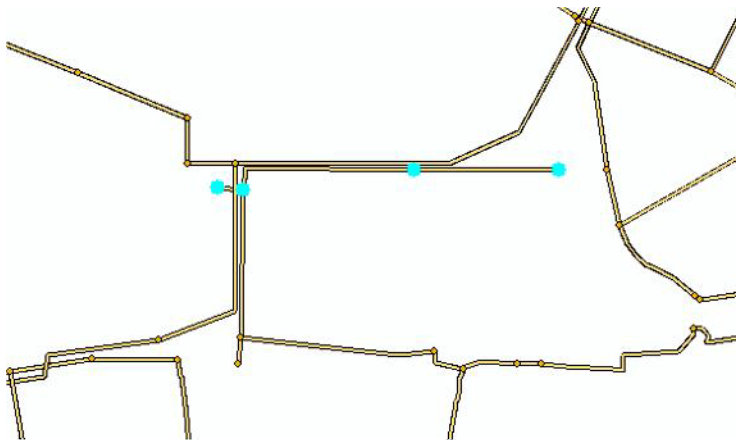


Figure 23.58 Nodes where the node pressure has changed more than 15 metres are selected on the Map

Nodes where the node pressure has changed more than 15 metres are selected on the Map

23.19.2 Display link flow Q-H plots

In order to display the Q-H plot for any pipe within the model, load the simulation into MIKE URBAN, select Model Results | Display Link Q-H Plot..., and click the pipe in the Map. The Q-H plot will be displayed in the new window and you can customize its settings. It is also possible to copy the graph window into Windows clipboard as graphics or as text values.



23.19.3 Display fire hydrant Q-H plots

MIKE URBAN allows you to display the fire hydrant Q-H plot for any node, which is used in a fire hydrant simulation.

In order to display the fire hydrant Q-H plot for the requested node, run the fire flow simulation for the selected node(s) with the "Simulate fire hydrant" options and load the simulation of the fire flow simulation results into MIKE URBAN. Select Model Results | Display Fire Hydrant Q-H Plot... and select the FQH file. The list of available hydrant nodes will be displayed in the list box and you can select it from this list.



Figure 23.59 Select the fire-hydrant from the list

Once selected, click 'Plot' and the Q-H plot will be displayed in the new window and you can customize its settings. It is also possible to copy the graph window into Windows clipboard as graphics or as text values.

23.19.4 Generate HTML Report

'Model Results | Generate HTML Report' you to create a HTML based report for your WD project including input data and results of the simulation. The report detail can be chosen to include general project information, nodes, pipes, tanks, pumps, valves, pump energy and other sections.

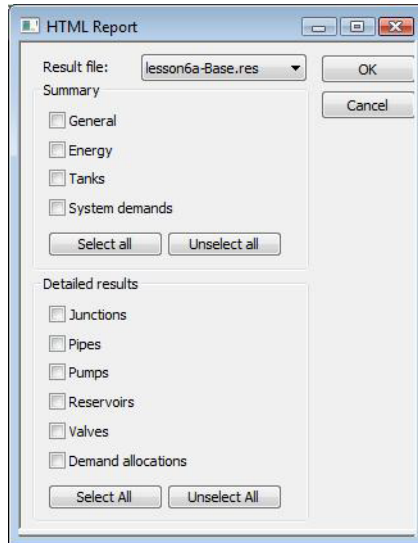


Figure 23.60 Setting up the HTML report

Once you have chosen what to include in your HTML report press the 'OK' button to both generate and open the HTML report (see Figure 23.61). The HTML report is saved to the project folder and given the name `ExportedResults.html`.

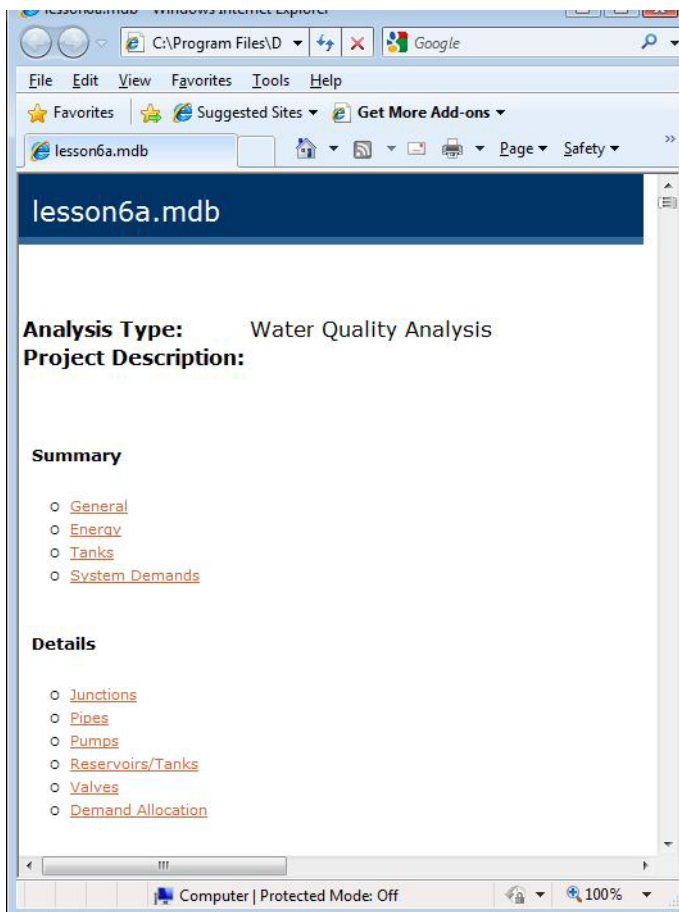


Figure 23.61 Example of the generated HTML report

23.19.5 Store results in the database

It is possible to store the complete model results (all time steps) for a selected part of the network into the database. This is useful when using Microsoft Access or Excel tools for analyzing 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 select all nodes and links.

Next, select Model Results | Results Table to open the results table. You can also re-define the selection set from within the Results Table. Click "Database" when ready and the active tab results (such as Junction, for example) will be stored to the database. See the table below to see what database tables are used for the results.



Table 23.1 Database tables used to store the simulation results

GEO type	Database table
Junctions	mw_RES_junction
Tanks	mw_RES_tank
Pipes	mw_RES_pipe
Pumps	mw_RES_pump
Valves	mw_RES_valve

Click "Database" from within the Results Table to store the results to the database. Once the results are stored in the database, you can open and work with them within Microsoft Access or other database management tools.

23.19.6 Display current operating pump points on the Q-H curve

MIKE URBAN allows you to display the current operating pump point for the selected time step and/or animate the position of the operating point during the simulation period.

In order to display the current operating Q-H point for a pump, load the model results into MIKE URBAN, open the Pump Editor and click Graph to display the Q-H plot. The pump Q-H plot will be displayed in the new window and the position of the current operating point will be red circled.

It is also possible to animate the position of the operating point during the simulation period by clicking Run from the results animation.

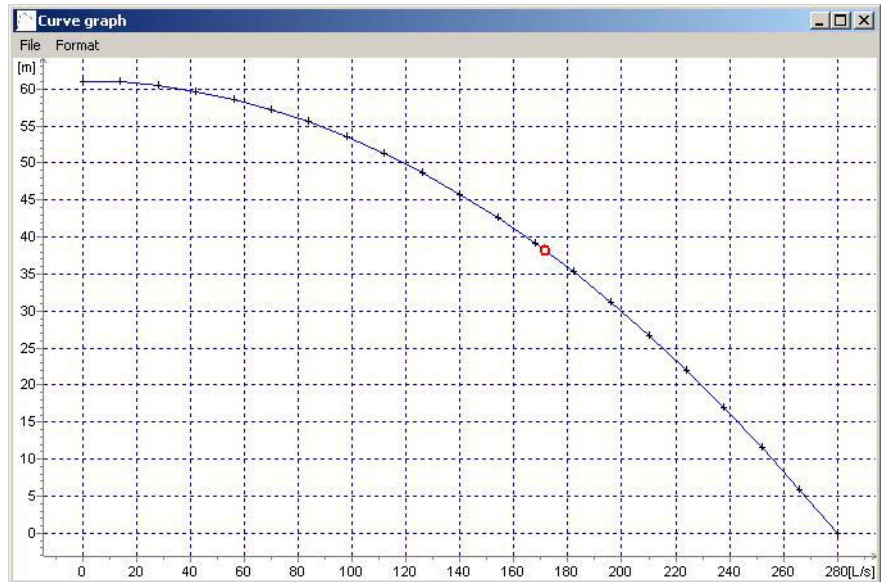


Figure 23.62 Current operating Q-H point for a pump

23.19.7 Reliability Analysis

The Reliability Analysis Tool (Model Results|Reliability Analysis) allows you to analyze hydraulic and water quality results of any steady state or extended period analysis and to investigate the reliability in terms of exceeded maximum and/or minimum values of pressures, flows, water quality, or any other result items. Figure 23.63 illustrates how the hydraulic reliability is calculated. Minimum and maximum limits can be defined for any result item e.g. node pressure and these limits will be used for the reliability analysis. The analysis will determine the duration of situations when the minimum and maximum limits were exceeded and the result will be presented as a percentage between 0-100 percent. Risk factor can be calculated as a product of the reliability and the "damage factor" e.g. node demand, node population, etc. The results can be presented in both graphical and tabular forms.

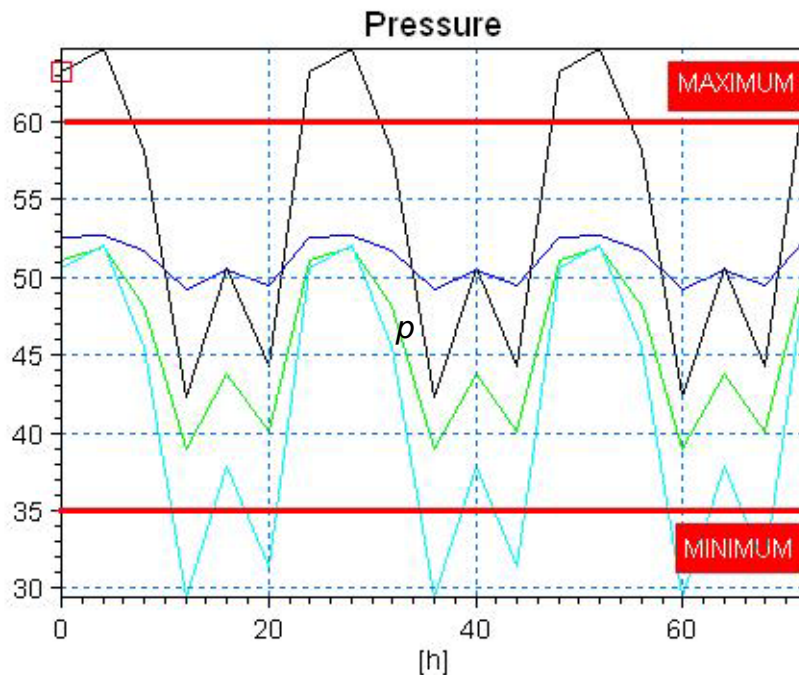


Figure 23.63 Minimum and maximum limits can be defined for any result item such as a node pressure and these limits will be used for the reliability analysis

When the tool is launched below dialog comes up:

Result	Reliability (r)	Risk (R=F*D)
Minimum limit overrun		
Maximum limit overrun		
Combined limit overrun		

Figure 23.64 Specifying input for the reliability analysis

In the dialog the following is specified:



- Result file: select a result file
- Reliability parameter: select a result item e.g. node pressure, node water quality, link velocity
- Minimum value: define a minimum allowable value (that you do not want to be exceeded)
- Maximum value: define a maximum allowable value (that you do not want to be exceeded)
- Starting time level: select starting "from" time level
- Ending time level: select ending "to" time level
- Damage parameter (D): select a attribute that can be used as a weighted parameter to calculate risk. The parameter could be any pipe attribute that makes sense to be used for this type of a calculation e.g. pipe demand coefficient corresponding to the number of connected consumers, water demand, or another numerical field.
-

Calculate

When activating this button the hydraulic or water quality reliability will be calculated

Report

Once the reliability as been calculated a report can be generated when pressing 'Report'. This opens the report as seen in Figure

	Time Level	Reliability Min	Reliability Max	Reliability Combined	Risk Min	Risk Max	Risk Combined
▶	0 hrs	88.93 %	83.82 %	72.74 %	0.03	0.05	0.07
	4 hrs	95.23 %	80.92 %	76.15 %	0.01	0.03	0.04
	8 hrs	72.40 %	98.81 %	71.21 %	0.13	0.01	0.14
	12 hrs	9.03 %	100.00 %	9.03 %	0.73	0.00	0.73
	16 hrs	33.22 %	100.00 %	33.22 %	0.42	0.00	0.42
	20 hrs	10.73 %	100.00 %	10.73 %	0.68	0.00	0.68
	24 hrs	88.93 %	83.82 %	72.74 %	0.03	0.05	0.07
	28 hrs	95.23 %	80.92 %	76.15 %	0.01	0.03	0.04
	32 hrs	72.40 %	98.81 %	71.21 %	0.13	0.01	0.14
	36 hrs	9.03 %	100.00 %	9.03 %	0.73	0.00	0.73
	40 hrs	33.22 %	100.00 %	33.22 %	0.42	0.00	0.42
	44 hrs	10.73 %	100.00 %	10.73 %	0.68	0.00	0.68
	48 hrs	88.93 %	83.82 %	72.74 %	0.03	0.05	0.07
	52 hrs	95.23 %	80.92 %	76.15 %	0.01	0.03	0.04
	56 hrs	72.40 %	98.81 %	71.21 %	0.13	0.01	0.14
	60 hrs	9.03 %	100.00 %	9.03 %	0.73	0.00	0.73
	64 hrs	33.22 %	100.00 %	33.22 %	0.42	0.00	0.42
	68 hrs	10.73 %	100.00 %	10.73 %	0.68	0.00	0.68
	72 hrs	88.93 %	83.82 %	72.74 %	0.03	0.05	0.07

Figure 23.65 Results of a reliability analysis of pressures (to be within 40-55 m interval)

Reliability (r)

Reliability (0-1, resp. 0-100%) represents a value that tells you what is the time percentage (occurrence) that the selected result item value is within the minimum and maximum limits

Failure (f)

Failure (f) is defined as $f = 1 - R$, where R =reliability R is within the range of $<0..1>$

Damage parameter (D)

The damage parameter (D) represents any pipe or node data that can be used for the calculation e.g. pipe demand coefficient corresponding to the number of connected consumers, water demand or another numerical field.

Risk ($R=f*D$)

Risk ($R=f*D$): risk is the product of a failure (f) multiplied by a damage parameter (D). The calculation of $f*D$ will quantify the risk that is associated with the probability of a pipe to fail the hydraulic reliability criteria.

Example: suppose that the reliability is 90%, risk is $1-0.9=0.1$, and if the $D = 50\text{l/s}$ (demand consumption) the risk = $0.1 * 50\text{l/s} = 5\text{l/s}$ of demand that might not be delivered.



Or, if reliability is 75%, risk is $1-0.75=0.25$, and if $D = 200$ connected consumers, risk = $0.25 * 200 = 50$ consumers that might not get enough of water.

In the report the following columns are found:

- Time level
- Reliability Min: reliability related to a non-exceeded minimum limit
- Reliability Max: reliability related to a non-exceeded maximum limit
- Reliability Combined: reliability related to a non-exceeded minimum and maximum limit
- Risk Min: risk related to a non-exceeded minimum limit
- Risk Max: risk related to a non-exceeded maximum limit
- Risk Combined: risk related to a non-exceeded minimum and maximum limit

Results of the reliability analysis can be displayed for the entire length of the simulation or per each time level (in case of extended period analysis). The example (Figure 23.65) displays the hydraulic reliability for pressures; 88.93% is the percentage of all nodes where the pressure is above the minimum pressure at the time level 0, for example.

It is also possible to display the reliability results in the Map; once the reliability analysis is executed; additional result items are available for the display. The Map below shows nodes where pressure limits are exceeded i.e. where the reliability is < 1 .

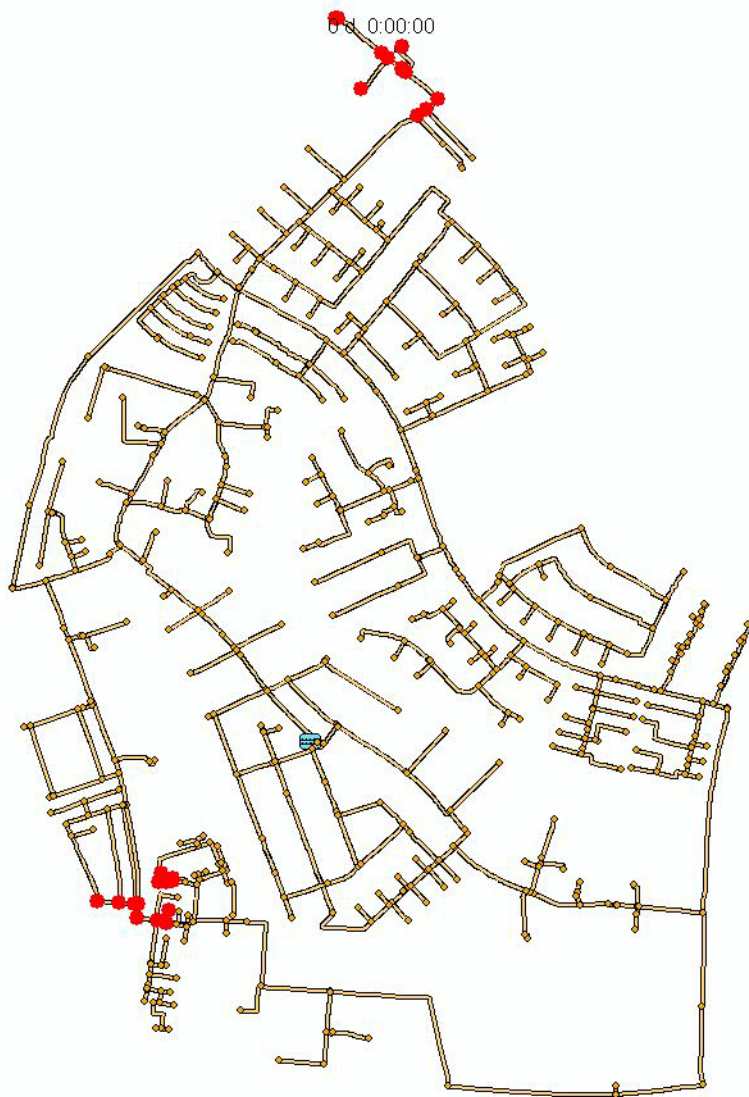


Figure 23.66 Nodes where pressure limits are exceeded are marked on the Map



24 Calibration Plots

Model calibration is important to ensure that model predictions represent the actual hydraulic and water quality conditions in the system. An attempt to calibrate a model should always be made when model results are used in decisions concerning possible remedial actions, augmentation works, forecasting etc.

The calibration is primarily focused on reproducing the observed hydraulics and water quality behaviour of the system in terms of flow depth/pressure, flow discharges and velocities. The model calibration should include comparisons between model simulation results and field measurements for, but not limited to, the following data:

- Flow
- Water level / Pressure / Hydraulic head
- Velocity
- Water mass balance
- Contaminant concentrations
- Contaminant migration rates
- Degradation rates

Usually these comparisons are presented in maps, tables or graphs. The calibration results needs to be evaluated by the modeller using engineering professional judgement. There are no universally accepted "goodness-of-fit" criteria that can be applied in all cases. However it is important that the user make every attempt to minimize the difference between model simulations and measured field data.

The model calibration procedure is supported in MIKE URBAN in which calibration plots of simulated and measured value can be set up and compared visually. Also, statistical analysis can be performed to determine the goodness of fit in MIKE URBAN. The calibration plots and statistics are written to an HTML report as documentation and further report processing.

24.1 Measurement Stations

Measurement stations in terms of location of flow gauges, pressure meters etc. can be defined in MIKE URBAN. The stations can be viewed in the horizontal plan providing the user an overview of monitoring locations. Additional information such as image and description of the monitoring site can be applied to the individual station.

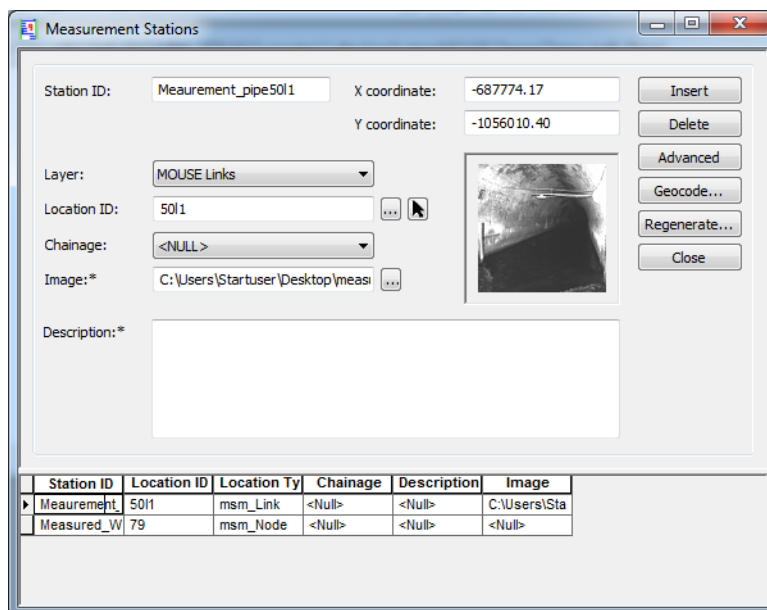


Figure 24.1 Measurement stations editor

It is possible to insert a measurement station on the map graphically as well as directly in the editor. The station can be inserted graphically using the insert point features 'Stations' accessible in the 'Active Layer' combo box for MOUSE, SWMM and EPANET network. As with other point layers the 'Create feature', 'Change Feature Type', 'Move Selected' and 'Delete Selected' tool bar buttons are enabled when Stations are selected in the 'Active Layer' combo box.

The '...' button after 'Location ID' edit box opens a Node/Junction/Link/Pipe selection list depending on the selected 'Layer type'. The arrow button allows one to pick a Node/Junction/Link/Pipe on the map depending on the selected Layer type.

In order to link the measurement station with a modelling result item, the stations need to be Geocoded to the model network (MOUSE, SWMM or EPANET). The button 'Geocode...' will launch the geocoding wizard which is also used for geocoding i.e. catchments, load allocations and demands. The wizard will generate station connections and populates either `m_Station.NodeID` or the `m_Station.LinkID` depending on the layer type. The wizard works either on selected measurement stations or, if none is selected, on all stations. For stations connected to links there is an option to specify the chainage or computational grid point of the connection - either Upstream, Middle or Downstream of the link. In case of measurement stations connecting to a link in a MOUSE model the `m_Station.Chainage` will be set to 'Downstream'. The reason for this is that flow gauges are most likely located in the 'Downstream' end of pipe.



The 'Regenerate' button will regenerate the connection lines between the Stations and the modelling network on the map.

24.2 Stations and Station Connections Layer

In the TOC the Measurement Stations group contains both the Stations and the Station Connections layers. The symbology can be altered

Edit fields on the Measurement Stations dialog:

Table 24.1 Edit fields on the Measurement Stations dialog

Edit field	Description	Used or required	Field name in data structure
Station ID	Station Identifier	Yes	m_Station.MUID
Layer	Network element	Yes	m_Station.LocationType
Location ID	Model Element ID	Yes	m_Station.LocationID
Chainage	Selection of grid point	If LocationType is msm_Link Domain: Upstream/Downstream/Middle	m_Station.Chainage
Image	An image may be added for each Measurement Station	Optional	m_Station.ImagePath
Description	A descriptive text can be added	Optional	m_Station.Description


24.3 Calibration Plots and Reports


The Calibration Plots and Reports dialog is accessible from the "Model Results |Calibration Plots and Reports" and allows the user to define relations between externally measured data and simulation results, produce calibration plots and specify the level of reporting for the calibration. The editor consists of a measurement group where an external time series file (dfs0) and item can be selected. A report section can be defined for the calibration plots.



Calibration	Order	Include	Station ID	Item ID	Component	Report Head	Report Description
Calibration 1	<Null>	True	Measured_WL	WL49	<Null>	<Null>	<Null>

Figure 24.2 Defining relations between measured data and simulation results

A calibration plot is defined by selecting a Station from a list by clicking on the button '...' or graphically by clicking on the  next to the Station ID field. The Result File field takes as default the loaded result file but it is also possible to select the result file manually.

The '...' button after the 'Station ID' edit box opens Station selection list like node selection list in 'Pipe and Channels' editor. The  opens a 'pick station on the map' also found in the 'Pipe and Channels' editor.

The '...' button after the 'File' edit box opens a file browser and the '...' button after the 'Item ID' edit box opens an item picker, in which the item ID and item type appears. The '...' button after the 'Result File' edit box opens a list showing all loaded result files. If no results are loaded the list will be empty.

Please note: some of the edit fields on the Calibration Plot and Reports dialog are only found in the grid.



Table 24.2 Edit fields on the Calibration Plots and Reports dialog

Edit field	Description	Used or required	Field name in data structure
Calibration ID	Each time a record is added an automatic ID is given of the form: Calibration_1, Calibration_i.... This may be changed to something more descriptive	Yes	m_Measurement.MUID
Include	When the tick mark is set the calibration ID gets included when pressing 'Report All'	Yes	m_Measurement.Include
Station ID	The ID of the Measurement Station	Yes	m_Measurement.StationID
File	External file name and path - this is where the measured time series is linked to the measurement station	Yes	m_Measurement.FilePath
Item ID	Choice of item in the measured time series (if more than one exists)	Yes	m_Measurement.FileItemID
Component	The AD components to be used (if relevant) is specified	Yes	m_Measurement.Component
Start	Auto filled with the dfs0 file start time. Used to control the Start time for comparison	Yes	m_Measurement.PlotStart
End	Auto filled with the dfs0 file end time. Used to control the End time for comparison	Yes	m_Measurement.PlotEnd



Table 24.2 Edit fields on the Calibration Plots and Reports dialog

Edit field	Description	Used or required	Field name in data structure
File	Result file name and path. If only one result file is loaded in this field may be left blank as the result file loaded will be used. If more than one file is loaded it is necessary to specify the result file to compare with.	Yes	m_Measurement.ResultFilePath
Simulation Start	EPANET result-files are only specified with relative time. This field allows to specify when the simulation started for easy comparison	Only when in WD mode	
File	The calibration report is named automatically. This may be changed afterwards.	Yes	m_Measurement.ReportFilePath
Header	Header used in report and in the calibration plot	Yes	m_Measurement.ReportHeader
Description	Description in report	Yes	m_Measurement.ReportDescription
Order (found only in the grid)	The Calibration ID is placed in the report dependent of the order specified. The Calibration ID with the lowest number is placed first in the report.		m_Measurement.PlotOrder
TS Config file (found only in the grid)	The settings of the calibration plots are saved in config files. Deleting the file resets the settings to default.		m_Measurement.TSCfgFilePath



Report Group

In the report group it is possible to select the output report file and type in simple text information such as 'Header' and 'Description' information. The 'Header' will be shown on the calibration plot report as the title.

24.3.1 Calibration Plots

Plot Button

Pressing the plot button launches a time series plot with the two time series for the current active row (Calibration ID).

- If only one result file is loaded, this file will be used automatically even if it is not specified on the dialog.
- If no result file is specified and more than one result file is loaded, it will only display the measurement in the calibration plot.
- If a result file name is specified but the result file is not loaded, it will only show the measurement in the TS plot

It is possible to open multiple plots simultaneously.

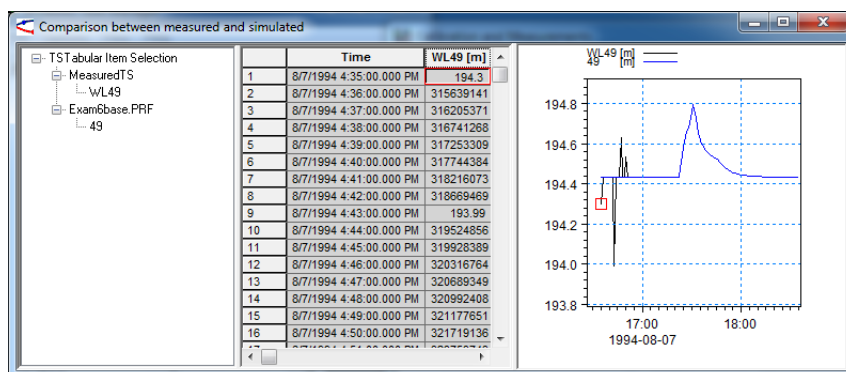


Figure 24.3 The calibration plot displays the 'Header' specified in the report section on the dialog

Changes made to the plot appearance (zoom, colors, line width etc.) are automatically saved when closing the plot window. When a time series plot is opened these settings will be loaded automatically. The settings are stored in the cfg-file seen in the grid below (specified in the column with the header 'TS Config file'). Deleting the file will remove the settings set. The cfg-files are placed in the project directory.

Report / Report All Button

The 'Report All' button produces and displays an html report for all Calibration ID's specified in the dialog where the 'Include' flag is toggled on.

The 'Report' button will generate an html report only for the currently active calibration ID. The rules for including results in the calibration plots are also valid for the html report.



Figure 24.4 Example of an html report

If an order is defined in the grid (this is done in the column with the heading 'Order') the order of the graphs in the report will correspond to this. This means that the first plot will be of the Calibration ID with the lowest value in the Order column.

The plots will be regenerated if the result or measurement files are updated. The plots will contain statistical information as shown and the Measurement Station ID in the summary section links to the actual plot in the report.

You can find more information on the plots and the statistical information in the documentation for the Result Verification tool (separate documentation, link to it is found in the documentation overview of MIKE URBAN).



WD Simulations

Results generated from WD simulation currently only have relative time tags (as the EPANET engine is used). The user should enter the simulation start time for the WD results in the dialog (under Simulation Start).





INDEX

**Numerics**

3D view of network 59

A

Appending catchments 235

Assigning values 267

B

Background layers 57

C

Catchment delineation 242

Catchment overlays and gaps 238

Catchment parameter processing 249

Climate Change 307

Connecting catchments 239

Coordinate system 43

D

Data models 93

Derived results 347

Desktop Workspace 27

E

Examples 29

Export to DXF 389

F

Feature Edit toolbars 176

Floating Toolbars 28

G

Geo-coding of load points 262

H

Help 28

I

Identify 28

Imperviousness 250

Import and Export of data 99

Import Assignments 103

Interpolation tool 267

K

Keyboard shortcuts 178

L

languages 42

Load allocation 255

Loading results 343

M

Map Window 27

Merging 278

MIKE URBAN modules 18

Model configuration 325

Moving catchments 234

P

Pipe Editor 202

Project check 313

R

rebuild network 53

Report to XML 42

Reports 360

Result statistics 349

Results table 359

S

Saving animations 56

Scrubbing 278

Select Dead-End Pipes 48

Selection tools 47

Simplification of networks 275

Splitting catchments 234

Submodel Manager 319

T

Toolbars 70

Tooltips 29

Tracing tools 49

Trimming 278

U

Units in MIKE URBAN 65

User defined flags 167

User written extensions 74