MIKE URBAN

Collection System
Modelling of storm water drainage networks and sewer collection systems

User guide

The expert in **WATER ENVIRONMENTS**

MIKE 2020

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 A/S (hereinafter referred to as “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 CS - MOUSE User Guide . . . 13

1 Modelling Collection Systems . . . 15

2 Modelling Collection Systems with MOUSE . . . 17

3 Hydraulic Network Modeling with MOUSE . . . 19
3.1 Introduction . . . 19
3.2 Definition of a MOUSE Network . . . 19
3.2.1 Modelling real network elements . . . 20
3.3 Nodes and Structures . . . 22
3.3.1 Identification group . . . 24
3.3.2 MOUSE model data group . . . 26
3.3.3 Q-H relations for nodes . . . 28
3.3.4 Outlet head loss . . . 28
3.3.5 Model Concept of Soakaway . . . 29
Soakaway tab . . . 32
3.4 Pipes and Canals . . . 35
3.4.1 Identification group . . . 37
3.4.2 Geometrical properties . . . 38
3.4.3 Hydraulic friction losses . . . 39
3.4.4 Miscellaneous . . . 40
3.5 Weirs . . . 41
3.5.1 Identification and connectivity . . . 43
3.5.2 Model data . . . 44
3.6 Orifices . . . 44
3.6.1 Identification and connectivity . . . 47
3.6.2 Model data . . . 47
3.6.3 Defining a gate or a weir in an orifice . . . 48
3.7 Stormwater Inlets . . . 49
3.7.1 Curb Inlet (Lintel) . . . 50
3.7.2 On-grade Capture . . . 52
3.7.3 Capacity curves . . . 53
3.8 Pumps . . . 54
3.8.1 Pump types . . . 54
Constant flow pumps . . . 55
Constant speed pumps . . . 55
Variable speed pumps . . . 56
3.8.2 Identification and connectivity . . . 61
3.8.3 Model data . . . 62
3.9 Valves . . . 62
3.10 CRS & Topography . . . 66
3.11 Emptying Storage Nodes . . . 69

4 Rainfall-Runoff Modelling with MOUSE . . . 71
4.1 Terms and Concepts . . . 71
4.1.1 MIKE URBAN Catchments . . . 72
4.1.2 Connecting Catchments to the Network . . . 73
4.1.3 Hydrological Models . . . 73

4.1.4 Creating Hydrological Models for a Catchment 74
4.2 Time-Area Method (A) 75
4.2.1 Model Data 75
4.2.2 Parameter Sets 77
4.2.3 Time-Area Curve Editor 79
4.3 Kinematic Wave (B) 80
4.3.1 Model Data 80
4.3.2 Parameter Sets 82
4.4 Linear Reservoir (C1 and C2) 84
4.4.1 Model Data 84
4.4.2 Parameter Sets 87
4.5 Unit Hydrograph Method (UHM) 89
4.6 Additional Flow and RDI 92
4.6.1 Model Data 95
4.6.2 MOUSE RDI Parameter Sets 96
4.7 Rainfall Data and Other Meteorological Variables - Boundary Conditions for Hydrological models 99
4.8 Running the Runoff Computations 102
4.9 MOUSE RDI - Guidelines for application 104
4.9.1 Choice of calculation time step 104
4.9.2 The RDI hotstart 105
4.9.3 The RDI result files 105
4.9.4 MOUSE RDI Validation 106
Surface runoff model 107
General hydrological model - RDI 107
4.9.5 Overflow within the model area 111
4.9.6 Non-precipitation dependent flow components 112
4.10 Using the Computed Runoff as Network Hydraulic Load 112
4.11 Low Impact Development (LID) 114
4.11.1 LID Controls 115
Bio Retention Cells 116
Infiltration Trenches 117
Porous Pavement 118
Rain Barrels 118
Vegetative Swales 119
Rain Garden 119
Green Roof 120
4.11.2 The LID Controls Editor 121
Identification 125
LID Control data Specification 126
4.11.3 LID Deployment 130
Identification and Connectivity 131
LID Deployment Properties 132
The LID deployment result file 132
The LID Simulation Summary 134

5 Time Series 137
5.1 Inserting New Time Series 137

5.1.1 Properties of time series object 138
5.1.2 Properties of time series item 141
5.1.3 Time series plot properties 144
5.2 Example: How to enter a rain time series 148
5.3 Example: How to Import a Time Series from Excel 151
6 Curves and Relations 157
6.1 Introduction 157
6.1.1 Capacity curves 158
6.1.2 Pump acceleration curves 158
6.1.3 Regulation curves Qmax(H) and Qmax(dH) 159
6.1.4 QH relation 159
Manholes, Basins 159
Outlets 159
Storage node 159
6.1.5 Time-Area curve 159
6.1.6 Removal efficiency 160
6.1.7 Basin geometry 160
6.1.8 Valve rating curve 161
6.1.9 DQ and QQ relations 161
6.1.10 Capacity curve QdH & Power 161
6.1.11 Undefined type 162
7 Boundary Conditions 165
7.1 Catchment Boundary Condition 166
7.1.1 Visualization of catchment boundaries 167
7.2 Network Boundary Conditions 169
7.2.1 Visualization of network boundaries 170
7.3 Boundary Condition Editors 171
7.3.1 Catchments Loads and Meteorological Items Editor 172
7.3.2 Network Loads Editor 173
7.3.3 External Water Levels Editor 174
7.3.4 Boundary Items Editor 175
7.4 Examples 177
7.4.1 How to add a varying water level at an outlet? 177
7.4.2 How to add infiltration in a pipe? 178
7.4.3 How to add a rainfall as a boundary condition to the catchments? . . 179
7.4.4 How to add a discharge to a node? 180
7.4.5 How to add runoff results as input for the network computation? . . 181
7.4.6 How do I add DWF in my network dependent on number of inhabitants? . 182
7.4.7 How to attach a pollutant concentration to a network load? 183
7.5 Repetitive Profile Editors 184
7.6 Diurnal Patterns 185
7.7 Profiles Calendar 185
7.8 Cyclic Profiles 186
7.9 Special Days 187

8 MOUSE Simulations . . . 189
8.1 The General Simulation Settings . . . 189
8.1.1 Choice of numerical engine . . . 192
8.1.2 Hotstart files . . . 192
8.1.3 Export to remote simulation . . . 194
8.1.4 Batch simulations . . . 195
8.2 The Runoff Simulation Settings . . . 196
8.3 The Network Simulation Settings . . . 198
8.4 The Summary Simulation Settings . . . 201
8.5 The 2D Overland Parameters . . . 204
8.6 MOUSE Result Selections . . . 204

9 2D Overland Flow . . . 207
9.1 Introduction . . . 207
9.2 Input Required - Overview . . . 207
9.3 Input Required - Details . . . 208
9.3.1 Choosing 2D model . . . 208
Flood Screening Tool in MIKE URBAN . . . 209
9.3.2 Adding the DEM to the map view . . . 210
9.3.3 Defining the 2D model domain and the resolution . . . 210
9.3.4 2D model settings . . . 211
9.3.5 2D initial conditions . . . 213
9.3.6 2D initial condition polygon layer . . . 214
9.3.7 2D bed resistance polygon layer . . . 216
9.3.8 2D boundaries . . . 218
9.3.9 Automatic model adjustments along water level and discharge boundaries 221
Adjustment of DEM . . . 221
Adjustment of initial conditions . . . 222
9.3.10 Defining couplings . . . 222
9.3.11 Flow parameters at manholes and basins . . . 226
Calculation method . . . 228
9.3.12 Outlets . . . 230
9.3.13 Pumps and weirs . . . 230
9.4 Running the Combined 1D and 2D Simulations . . . 230
9.4.1 Setting the simulation type and requesting 2D results . . . 230
9.4.2 Starting simulation . . . 234
9.5 Visualising Simulation Results . . . 234
9.6 Understanding the Files Generated . . . 236

10 MOUSE Control module . . . 239
10.1 RTC in Urban Drainage and Sewer Systems . . . 239
10.2 Architecture of RTC Systems . . . 240
10.3 MOUSE Control vs. Real Life . . . 242
10.4 Sensors . . . 242
10.5 Logical Conditions . . . 243
10.6 Control Actions . . . 245

10.7 PID parameter sets . . . 246
10.7.1 Calibration of the PID constants . . . 247
10.8 Controllable Devices . . . 249
Control Type and PID-ID . . . 251
10.8.1 Pumps . . . 252
10.8.2 Weirs . . . 253
10.8.3 Orifices with weirs and gates . . . 253
10.8.4 Difference between weir and orifice with weir . . . 254
10.8.5 Valves . . . 256
10.8.6 Control rules . . . 256
10.9 MOUSE Control Computations . . . 257
10.10 User Written Control . . . 258

11 Long term statistics . . . 265
11.1 Data Input . . . 266
11.1.1 Job list . . . 266
11.1.2 Job list criteria . . . 267
11.1.3 Initial conditions for simulated events . . . 269
11.1.4 Generating job list . . . 271
11.1.5 Edit job list . . . 272
11.1.6 Runtime stop criteria . . . 272
Run-Time Stop Criteria Evaluation Matrix . . . 273
11.2 LTS Computations . . . 275
11.3 Result Files . . . 277
11.3.1 User-Specified result files . . . 277
11.3.2 Statistics result file . . . 278
11.4 Specification of Statistic and Result Files . . . 278
11.4.1 LTS Global Statistics Parameters - Event definitions . . . 279
11.5 LTS Statistics Presentation . . . 287

12 Automatic pipe design with MOUSE . . . 289
12.1 Design Principles . . . 289
12.2 Design Input . . . 290
12.2.1 Example of an ADP-file . . . 292
12.2.2 Design Type . . . 292
12.2.3 Design Criteria . . . 292
12.2.4 Design Group Type . . . 293
12.2.5 Lower Limit . . . 293
12.2.6 Commercial Diameters . . . 294
12.2.7 Creating the ADP file for the design simulation . . . 294
12.3 Design Simulation and Output . . . 295

13 Modelling Water Quality with MOUSE . . . 301
13.1 Key Features and Application Domain . . . 301
13.1.1 Surface Runoff Quality (SRQ) . . . 301
13.1.2 Pipe Sediment Transport (ST) . . . 301
13.1.3 Pipe Advection-Dispersion (AD) . . . 302
13.1.4 Biological Processes (BP) . . . 302

13.1.5 Interaction between water quality modules 303
13.2 Surface Runoff Quality (SRQ) 304
13.2.1 Surface Sediment Data Dialogs 305
MOUSE| Water Quality| SRQ Main Parameters 305
MOUSE| Water Quality| SRQ Attached Pollutants 306
MOUSE| Water Quality| SRQ Gully Pot Data 307
13.3 Advection-Dispersion (AD) 308
13.3.1 Advection-Dispersion Data Dialogs 309
MOUSE| Water Quality| AD Components 309
MOUSE | Water Quality | AD Dispersion 311
Advection-Dispersion and Open Boundary Conditions 312
13.4 Biological Processes (BP) 313
13.4.1 Biological Processes Dialog (MOUSE|Water Quality|WQ Process Model) 313
13.5 Water quality (MIKE ECO Lab) 317
13.5.1 MIKE ECO Lab Templates and Assignments 318
13.5.2 MIKE ECO Lab State Variables 320
13.5.3 MIKE ECO Lab Forcings 321
13.5.4 MIKE ECO Lab Constants 323
13.5.5 Running MIKE ECO Lab simulation 324
13.6 Sediment Transport (ST) 325
13.6.1 The Sediment Transport Models in MOUSE ST 325
The Explicit Sediment Transport Models 325
The Morphological Models 326
13.6.2 The Transport Formulae - Short Description 326
The Ackers-White formulae 327
The Engelund-Hansen formula 327
The Engelund-Fredsøe-Deigaard formulae 327
The van Rijn formulae 327
13.6.3 The Flow Resistance in Sewer Systems with Sediment Deposits . . 328
13.6.4 Sediment Transport Data Dialogs 328
MOUSE | Water Quality | ST Main Parameters 328
MOUSE| Water Quality | ST Sediment Fractions 330
MOUSE | Water Quality | ST Initial Sediment Depth Local 331
MOUSE| Water Quality| ST Sediment Removal Basins 332
MOUSE | Water Quality| ST Sediment Removal Weirs 333
13.6.5 Boundary Conditions for the Sediment Transport Model 334
13.7 Storm Water Quality 335
13.7.1 Assigning concentrations of pollutants to runoff and infiltrations . . . 335
Cst. concentration (method 1) 338
Table concentration (method 2) 338
EMC formula (method 3) 338
13.7.2 Local treatment nodes 339

MIKE URBAN CS - SWMM User Guide 347

14 Hydraulic Network Modelling with SWMM 349
14.1 Terms and Concept 349
14.2 Nodes 350
14.3 Conduits 358
14.4 Orifices 361
14.5 Pumps 364
14.6 Weirs 366
14.7 Outlets 369
14.8 Transects 372
14.9 Tabular Data (Curves) 375
14.10 Controls 379

15 Hydrological Modelling with SWMM 381
15.1 Terms and Concepts 381
15.2 Catchments 381
15.3 Surface Routing 384
15.4 Infiltration 387
15.5 RDII 389
15.6 Aquifers 394
15.7 Groundwater 396
15.8 Snowpack 398
15.9 Climatology 402
15.10 Coverage 412
15.11 LID Controls 413
15.12 LID Deployment 422

16 Time Series and Boundary Conditions in SWMM 427
16.1 Time Series 428
16.2 Time Patterns 430
16.3 Raingauges 434
16.4 Dry Weather Flow 436
16.5 Inflow 438

17 Project Options and Simulations in SWMM 441
17.1 The General Simulation Settings 441
17.2 The Runoff Simulation Settings 443
17.3 The Network Simulation Settings 446
17.4 The Summary Simulation Settings 450

18 Water Quality Modelling with SWMM 453
18.1 Terms and Concepts 453
18.2 Land Uses 453
18.3 Buildup 455
18.4 Washoff 457
18.5 Loading 460
18.6 Pollutant 461
18.7 Local Treatments 463

MIKE URBAN CS - MOUSE

User Guide

1 Modelling Collection Systems

When modelling a collection system with MIKE URBAN you can choose to model the collection system with either the SWMM5 engine or the MOUSE engine.

In order to run SWMM5 simulation a Model Manager module is required, while running MOUSE simulations require some further modules depending on type of simulation being carried out (e.g. pipeflow, rainfall runoff simulations).

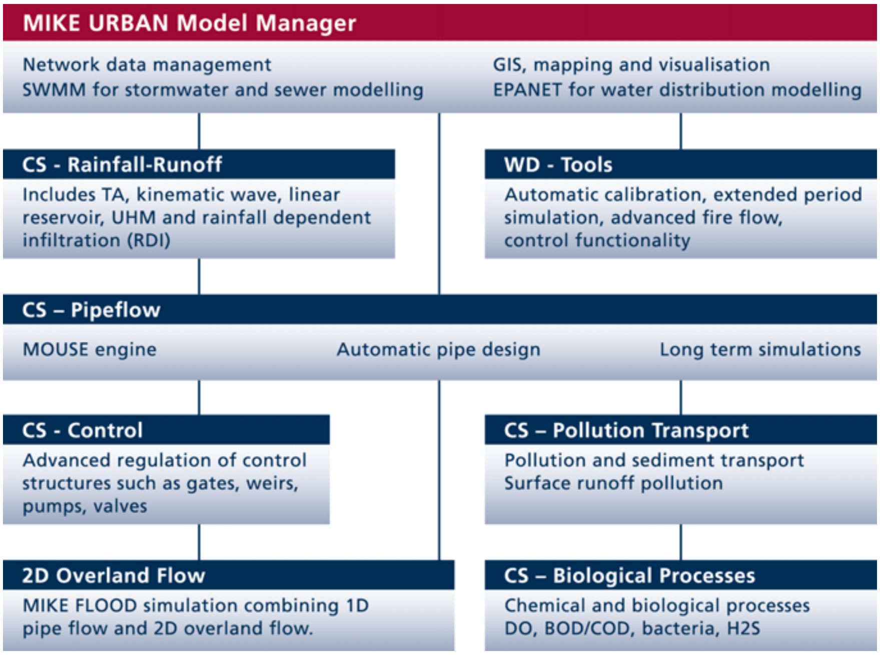

MIKE URBAN Model Manager
Network data management
SWMM for stormwater and sewer modelling
GIS, mapping and visualisation
EPANET for water distribution modelling
CS - Rainfall-Runoff
Includes TA, kinematic wave, linear reservoir, UHM and rainfall dependent infiltration (RDI)
WD - Tools
Automatic calibration, extended period simulation, advanced fire flow, control functionality
CS – Pipeflow
MOUSE engine
Automatic pipe design
Long term simulations
CS - Control
Advanced regulation of control structures such as gates, weirs, pumps, valves
CS – Pollution Transport
Pollution and sediment transport
Surface runoff pollution
2D Overland Flow
MIKE FLOOD simulation combining 1D pipe flow and 2D overland flow.
CS – Biological Processes
Chemical and biological processes
DO, BOD/COD, bacteria, H2S

Figure 1.1 The modular structure of MIKE URBAN

2 Modelling Collection Systems with MOUSE

MOUSE is a powerful and comprehensive engine for modelling complex hydrology, advanced hydraulics in both open and closed conduits, water quality and sediment transport for urban drainage systems, storm water sewers and sanitary sewers.

MOUSE owes its exceptional power to the advanced software implementation techniques, the efficient algorithmic formulations and the application versatility. And finally, it is the reliability of MOUSE, tested and proven in great many applications since the late 70s by more than one thousand users all around the world, which makes MOUSE the perfect choice.

Typical applications of MOUSE include studies of combined sewer overflows (CSO), sanitary sewer overflows (SSO), complex Real Time Control (RTC) schemes development and analysis, design of new site developments, regulatory consenting procedures and analysis & diagnosis of existing storm water and sanitary sewer systems.

By applying MOUSE, it is possible to answer questions, such as:

- What are the return periods for overloading of various parts of the existing sewer system?
- What are the main causes of that overloading - backwater or insufficient local pipe capacity?
- What are the implications of replacing critical sewers, installing new basins, weirs, etc.?
- How is the long-term environmental impact affected by changing the operational strategy?
- Where and why are sediments deposited in the sewer network?
- What are the peak concentrations of pollutants at the overflow weir or at the treatment plant after a rainstorm?

3 Hydraulic Network Modeling with MOUSE

3.1 Introduction

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

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

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

The modelling process consists of the following distinct steps:

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

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

3.2 Definition of a MOUSE Network

A MOUSE network within MIKE URBAN can be defined in one of the following ways:

- Import of existing MOUSE Project
- Import of external data (e.g. GIS) into MIKE URBAN CS MOUSE network
- Copying network data from MIKE URBAN CS Asset network into MIKE URBAN CS MOUSE network

- Copying network data from MIKE URBAN CS SWMM network into MIKE URBAN CS MOUSE network
- Graphically digitizing and manual data typing within MIKE URBAN

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

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

A model consists of the following hydraulic elements:

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

3.2.1 Modelling real network elements

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

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

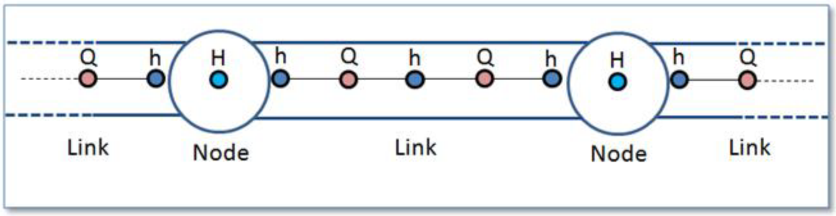


Figure 3.1 The computational grid

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

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

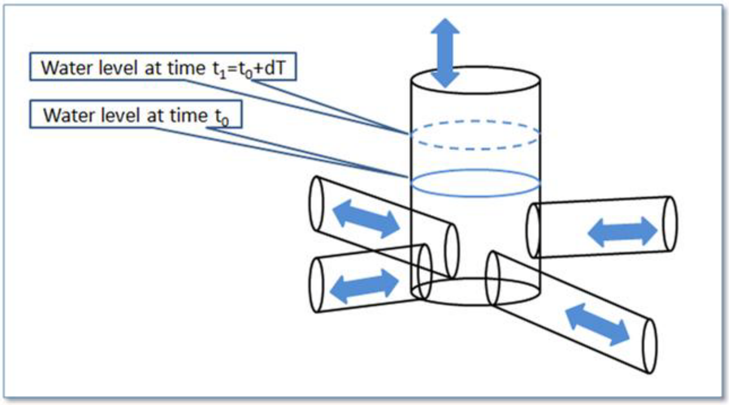


Figure 3.2 Water flowing through a node

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

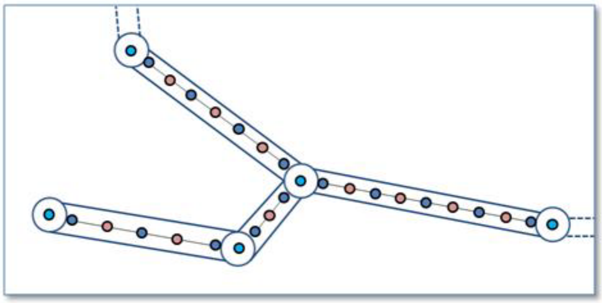


Figure 3.3 The computational grid for a given network

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

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

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

In MIKE URBAN we have five functional elements which from the model building point of view are related to nodes like manholes or basins. These are "Pumps", Weirs", "Valves", "Orifices" and “Stormwater inlets”. The concept of elements related to nodes is reflected in the design of the dialog for editing the parameters for these elements. Here you find a field named "Location:" for all of the elements. The field takes the ID of a node as input. All elements also have a field for "To:" which also takes a node ID as input.

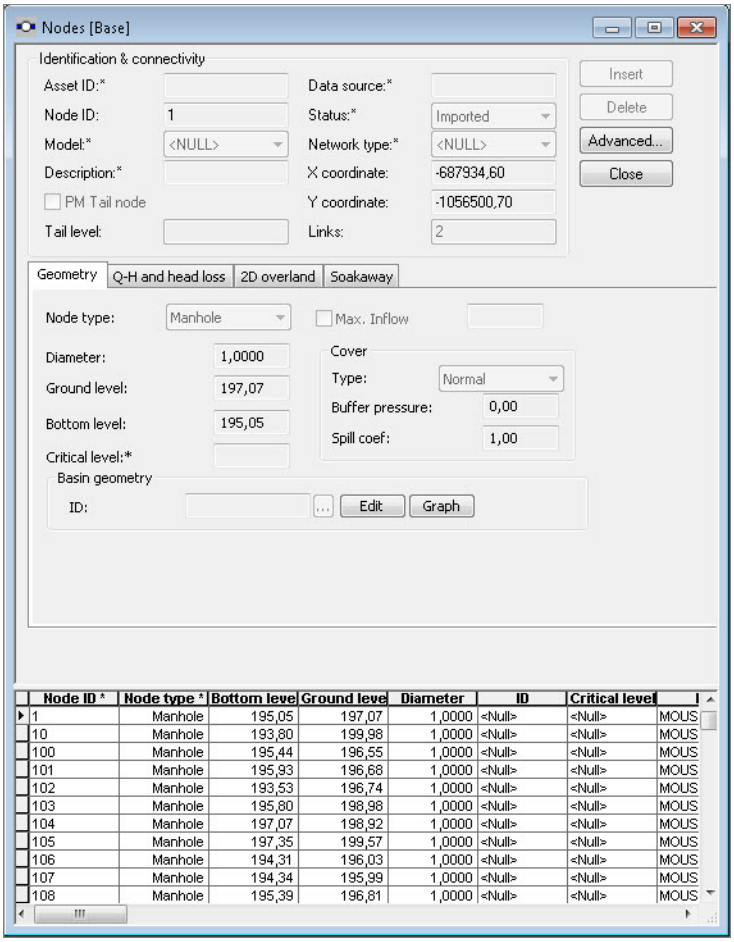
Seen from the computational solution point of view the five elements are actually connections from one node to another node. This is similar to how pipes are defining the link for flow between nodes as reflected in the dialog where you find fields for entering "From node:" and "To node:".

3.3 Nodes and Structures

The “Nodes and Structures” editor makes it possible to define the elements used to model manholes, outlets and basins in a CS MOUSE storm and sewer collection system.

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

- Identification - General identification and location information
- MOUSE model data - Model related data
- Basin Geometry - Geometry information
- Q-H relation and Outlet head loss - Q-H relations and information on the head loss approach and coefficients
- 2D overland flow - used for coupling to 2D flood (requires MIKE FLOOD license, see separate description on the 2D overland flow chapter)
- Soakaway tab - defines the type of infiltration, porosity and initial water level is selected



Asset ID:*
Node ID: 1
Model:* <NULL>
Description:*
☐ PM Tail node
Tail level:
Data source:*
Status:* Imported
Network type:* <NULL>
X coordinate: -687934,60
Y coordinate: -1056500,70
Links: 2

Node type: Manhole
☐ Max. Inflow
Diameter: 1,0000
Ground level: 197,07
Bottom level: 195,05
Critical level:*
Type: Normal
Buffer pressure: 0,00
Spill coef: 1,00
ID:

Node ID *	Node type *	Bottom leve	Ground leve	Diameter	ID	Critical level	I
1	Manhole	195,05	197,07	1,0000	<Null>	<Null>	MOUS
10	Manhole	193,80	199,98	1,0000	<Null>	<Null>	MOUS
100	Manhole	195,44	196,55	1,0000	<Null>	<Null>	MOUS
101	Manhole	195,93	196,68	1,0000	<Null>	<Null>	MOUS
102	Manhole	193,53	196,74	1,0000	<Null>	<Null>	MOUS
103	Manhole	195,80	198,98	1,0000	<Null>	<Null>	MOUS
104	Manhole	197,07	198,92	1,0000	<Null>	<Null>	MOUS
105	Manhole	197,35	199,57	1,0000	<Null>	<Null>	MOUS
106	Manhole	194,31	196,03	1,0000	<Null>	<Null>	MOUS
107	Manhole	194,34	195,99	1,0000	<Null>	<Null>	MOUS
108	Manhole	195,39	196,81	1,0000	<Null>	<Null>	MOUS

Figure 3.4 Nodes and Structures editor

MOUSE distinguishes between four types of nodes: circular manholes, basins, outlets and storage nodes. The same dialog is used for all four node categories, but the dialog adapts according to the selected node type.

Each node is geographically determined by 'x' and 'y' co-ordinates. The co-ordinates may be specified in any local co-ordinate system.

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

Alternatively, it is possible to specify a sealed/locked node (Cover type equal to 'Sealed'), i.e. a node with a fixed lid on the top - at the ground level - so water cannot escape although the pressure still builds up inside.

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

If inflow to a node from catchments is limited this can be modelled by specifying the ‘Max. Inflow’ parameter for the specific node. This limit will sustain as long as the node is not coupled to the 2D model. When the node is coupled, “Max.inflow” is ignored, and the coupling parameters take control over the inflow capacity.

It is possible to get the information about the volume held back at the node due to the inflow limitation by adding an entry in the ‘dhiapp.ini’ file. Please refer to the documentation on this file for further information.

In the tables given below each data variable is described shortly and if it is required as input.

3.3.1 Identification group

Table 3.1 The edit fields in the Identification group

Edit field	Description	Used or required by simulations	Field name in datastructure
Asset ID	Reference to an ID used in external data sources	No	AssetName
Data source	Reference to an external data source (table ID) where the record has been imported from	No	DataSource
Node ID	A unique name for the node. Up to 40 characters (letters, numbers, blank spaces and underscore characters)	Yes	MUID
Model	Associates the current node to a specified submodel	No	SubModelNo
Description	User's descriptive information related to the node	No	Description

Table 3.1 The edit fields in the Identification group

Edit field	Description	Used or required by simulations	Field name in datastructure
PM Type	Definition of the node´s role in the pressure main as the downstream point of the pressure main’s connection to the network. Manholes and basins can be declared as a “Tail Node”. Please refer to the MOUSE Reference Manual (Pipe Flow) for further on pressure mains.	Yes	PMTypeNo
Tail level	Water level (absolute elevation) in the "receiving node", used as lower boundary for permanently pressurized parts of the system. Please refer to the MOUSE Reference Manual (Pipe Flow) for further on pressure mains.	Yes	PMLevel
Status	Data status for the entire record, serves for keeping track on the source of information	No	Element_S
Network type	Attributes the link to a certain type of network. Used in cases when two or more different networks are included in the same project	No	NetTypeNo
X coordinate	X coordinate of the node position	Yes	Shape_X
Y coordinate	Y coordinate of the node position	Yes	Shape_Y
Links	Number of links going to or from the node. Non-editable value.	-	-

3.3.2 MOUSE model data group

Table 3.2 The edit fields in the MOUSE model data group

Edit field	Description	Used or required by simulations	Field name in datastructure
Node type	MOUSE Node Types: 1. Manhole - node with shaft and chamber storage 2. Basin 3. Outlet - node where water leaves the system (no storage) 4. Storage node 5. Soakaway	Yes	TypeNo
Diameter	Diameter of the manhole - not enabled for any other node types	Yes	Diameter
Ground level	Ground level of the node	Yes	GroundLevel
Bottom level	The bottom level of the manhole	Yes	InvertLevel
Critical level	User defined critical level. Used in result presentations and in the Pipe Design module	Yes	CriticalLevel
[Tickmark]	Activates the inlet delimiter function	Yes	InletControlNo
Max. Inflow	Value of maximum possible inflow into the node from runoff	Yes	MaxInlet
Cover sub-group			
Type	Choose among available types: 1. Normal 2. Sealed 3. Spilling	Yes	CoverTypeNo
Buffer pressure	Buffer pressure is only active for type = spilling. Equal to the pressure above the ground level needed to cause spills from the manhole	Yes	BufferPressure
Spill coeff.	Spill coefficient is only active for type = spilling. Controls the spill capacity	Yes	SpillCoeff

Table 3.2 The edit fields in the MOUSE model data group

Edit field	Description	Used or required by simulations	Field name in datastructure
Basin geometry sub group			
ID	Reference to a tabulated area-elevation function for the basin geometry. The H-column for the basin geometry can start at any value, e.g. 0.0 for interpretation of H as depth in the basin. The MOUSE Engine will associate the first H-value to the bottom level of the node.	Yes	GeometryID
Q-H relation sub group			
[Tickmark]	Switch for choice of the comp. method for hydraulic conditions at the outlet, in a manhole or basin. When toggled on uses Q-H tabulated function, toggled off calculates on the basis of water level (see note below this table)	Yes	QHTypeNo
[Text string]	Reference to a tabulated Q-H relation	Yes	OutletQHID
Outlet head loss sub group			
ID	Reference to a head loss parameter set	Yes	LossParID
Method	Method selected for the actual head loss parameter set	-	-
Use local data	Switch for use of local head loss data	Yes	LossParNo
Coeff.	Locally defined interpretation of head-loss coefficient. Km="shape coefficient", Cont. HLC= outlet contraction head losshead loss coeff. (relative to velocity head) , Total HLC = outlet total head losshead loss (relative to velocity head)	Yes	LossTypeNo
[Field next to coeff field]	Local value of loss coefficient	Yes	LossCoeff
Eff. flow area	Locally defined choice of method for the calculation of wetted area	Yes	EffAreaNo
Soakaway sub group			

Table 3.2 The edit fields in the MOUSE model data group

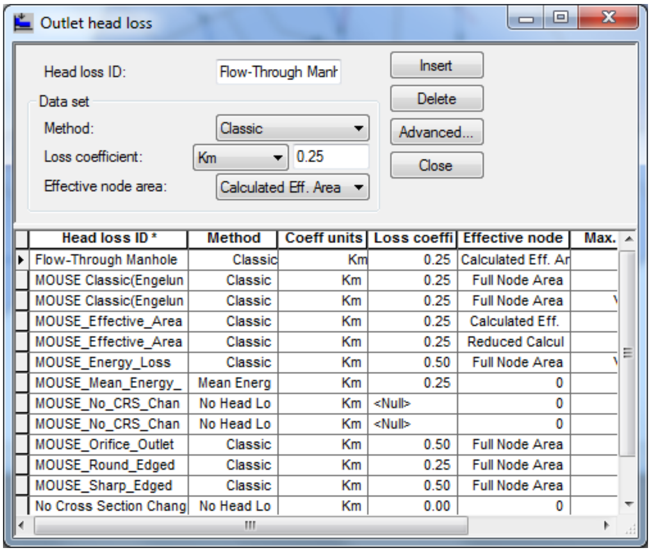
Edit field	Description	Used or required by simulations	Field name in datastructure
Porosity of fill material	Porosity of filling material	Yes	Porosity of fill material
Kfs, bottom	Conductivity of soil	Yes	Kfs, bottom
[Tickmark]	Activates the bottom conductivity function	Yes	
Infiltration method	Method of infiltration	Yes	Infiltration method
Infiltration rate	Defines the infiltration rate	Yes	Infiltration rate
Initial water level	Initial water level in soakaway	Yes	Waterlevel

3.3.3 Q-H relations for nodes

Specifying a Q-H relation for an outlet controls the flow at the outlet. When specifying a Q-H relation for a manhole or basin the Q-H relation controls infiltration to the node. The Q-H relation specifies the relation between the water level in the manhole (or basin) and the infiltration flow. The flow (Q) value in the Q-H relation should be given as a positive value when water enters the node and a negative value for specifying a loss of water from the network model.

3.3.4 Outlet head loss

If you wish to make changes to the head loss parameter set that you have made a reference to, you can change this by accessing MOUSE|Local Head Losses. The editor is shown in Figure 3.5.



Outlet head loss
Insert
Delete
Advanced...
Close

Head loss ID: Flow-Through Manh

Data set

Method: Classic

Loss coefficient: Km 0.25

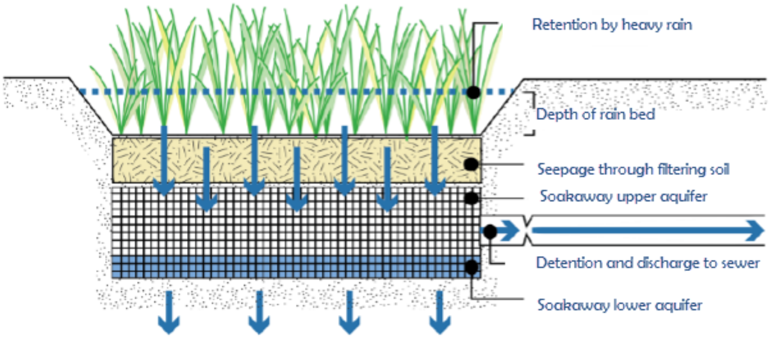
Effective node area: Calculated Eff. Area

	Head loss ID *	Method	Coeff units	Loss coeffi	Effective node	Max.
▸	Flow-Through Manhole	Classic	Km	0.25	Calculated Eff. Ar	
	MOUSE Classic(Engelun	Classic	Km	0.25	Full Node Area	
	MOUSE Classic(Engelun	Classic	Km	0.25	Full Node Area	
	MOUSE_Effective_Area	Classic	Km	0.25	Calculated Eff.	
	MOUSE_Effective_Area	Classic	Km	0.25	Reduced Calcul	
	MOUSE_Energy_Loss	Classic	Km	0.50	Full Node Area	
	MOUSE_Mean_Energy_	Mean Energ	Km	0.25	0	
	MOUSE_No_CRS_Chan	No Head Lo	Km	<Null>	0	
	MOUSE_No_CRS_Chan	No Head Lo	Km	<Null>	0	
	MOUSE_Orifice_Outlet	Classic	Km	0.50	Full Node Area	
	MOUSE_Round_Edged	Classic	Km	0.25	Full Node Area	
	MOUSE_Sharp_Edged	Classic	Km	0.50	Full Node Area	
	No Cross Section Chang	No Head Lo	Km	0.00	0	

Figure 3.5 Outlet head loss editor

3.3.5 Model Concept of Soakaway

Detailed hydraulic modelling of the green solutions can be done by means of the network point node type in MIKE URBAN - named soakaway. The soakaway can be connected to the pipe network as any other node elements for detailed hydraulic studies. With is implementation in MIKE URBAN the soakaway represents a generic type of LID control as it can represent a number of different WSUD controls. The soakaway can be digitized graphically and it has its own feature layer which can be viewed in the horizontal plan view.



Retention by heavy rain
Depth of rain bed
Seepage through filtering soil
Soakaway upper aquifer
Detention and discharge to sewer
Soakaway lower aquifer

Figure 3.6 Conceptual drawing of a soakaway

A schematic drawing of a soakaway (Bio-retention cell) is illustrated in Figure 3.6. The stormwater drains of the surface and enters the soakaway at the upper vegetated layer. Then the stormwater infiltrates vertically through the soakaway and infiltrates out of the sides and bottom of the soakaway.

In some cases the soakaway is not connected to any drainage network and captured runoff to the soakaway is infiltrated and in case of extreme rainfall and exceedance of its infiltration and storage capacity storm water is surcharged to the surface.

In other cases the soakaway is connected to the drainage network by a flow controlled outlet pipe as illustrated in Figure 3.6. During extreme rainfall causing exceedance of its infiltration, storage and outlet flow capacity, the soakaway also surcharges to the surface.

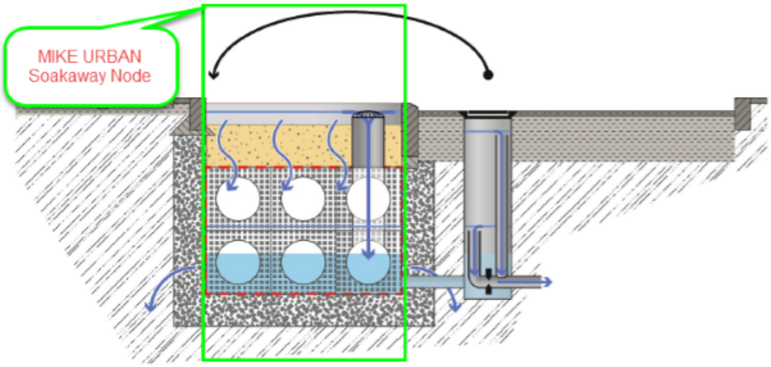


Figure 3.7 Soakaway concept in MIKE URBAN

A Soakaway will be represented in MIKE URBAN as a node (point) as shown in Figure 3.7.

Since a Soakaway is defined as a node type then the remaining configuration is unlimited in terms of

- Inlet pipe(s) + Flow Regulation
- Outlet pipe (s) + Flow Regulation
- Weirs (s) or Orifice (s)

Connection to the existing drainage system can be configured to match the various types of soakaway installations. Soakaway nodes can i.e. be coupled in series to support the modelling of constructed infiltration trenches.

The soakaway can be added graphically and/or imported from any Asset GIS system hence existing soakaways can be illustrated in the plan view of MIKE URBAN.

The Soakaway node has the following attributes:

- NodeTypeNo = 5 (Soakaway node)
- Invert Level
- Ground Level
- Geometry - defined as basin geometry
- CoverType = 2 (Spilling)
- Headloss = No Cross Section Changes

The inflow to the Soakaway can be provided as:

- Direct Inflow (constant, time series)
- Inflow from Rainfall Runoff (Runoff model A, B, C, Unit Hydrograph).
- Various infiltration rate options
  - No infiltration
  - Qinf - constant or as time series through boundary condition system
  - Qinf, side - function of wetted side area. This is calculated by MIKE 1D based on basin geometry
  - Qinf, bottom - function of bottom area
- Saturated hydraulic conductivity [m/s]
- Porosity of the soakaway material [ ]

The soakaway is modelled as a regular basin in MIKE URBAN, and it is also MIKE URBAN that calculates the water level based on the inflow (runoff), outflow to the drainage network (via overflow pipe, weir, orifices etc.) and the infiltration. The soakaway has, unlike a basin, a porous filling material that affects the water level calculations, this is accounted for in the MIKE 1D and the basin geometry is defined as a usual basin geometry description.

The overflow pipe, weir or orifice is modelled in MIKE URBAN. Make sure that the up level of the pipe, weir or orifice is set correctly, and not at the bottom of the basin which will be the case if no invert level is specified. If the pipe is placed at the bottom then make sure that flow regulation is applied to the out-going pipe. The inflow pipe is generally not modelled; instead the catchment that generates the runoff to the soakaway (e.g. a road section, a roof) is connected directly to the basin via the catchment connection tools in MIKE URBAN.

A new point feature layer has been created to illustrate the soakaway in the plan view and to be used for graphical digitization and graphical connection of soakaway to existing drainage network.

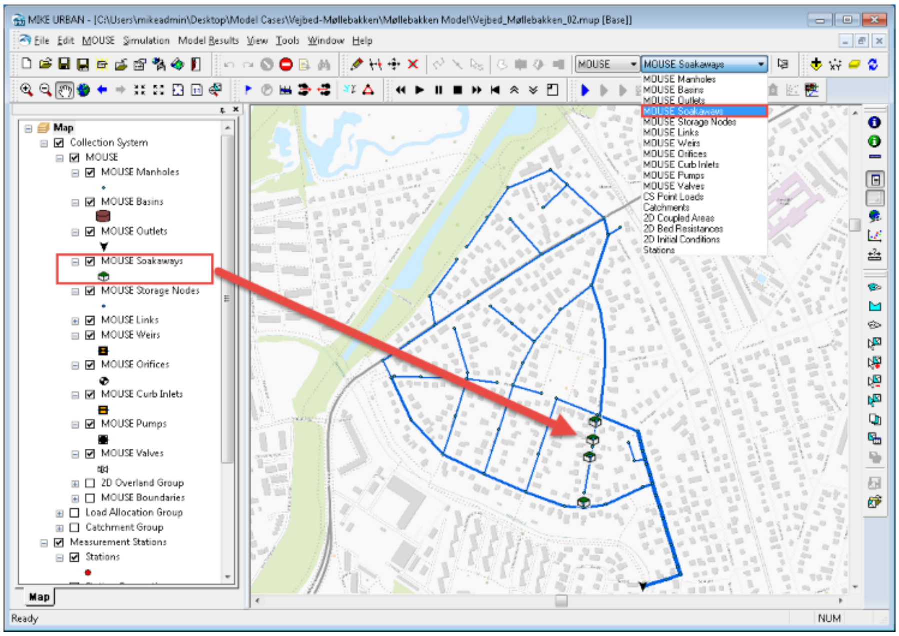

MIKE URBAN - [C:\Users\mikeadmin\Desktop\Model Cases\Vejbed-Møllebakken\Møllebakken Model\Vejbed_Møllebakken_02.mup [Base]]
File Edit MOUSE Simulation Model Results View Tools Window Help
MOUSE
MOUSE Soakaways
MOUSE Manholes
MOUSE Basins
MOUSE Outlets
MOUSE Soakaways
MOUSE Storage Nodes
MOUSE Links
MOUSE Weirs
MOUSE Orifices
MOUSE Curb Inlets
MOUSE Pumps
MOUSE Valves
CS Point Loads
Catchments
2D Coupled Areas
2D Bed Resistances
2D Initial Conditions
Stations
Map
☑ Collection System
☑ MOUSE
☑ MOUSE Manholes
☑ MOUSE Basins
☑ MOUSE Outlets
☑ MOUSE Soakaways
☑ MOUSE Storage Nodes
☑ MOUSE Links
☑ MOUSE Weirs
☑ MOUSE Orifices
☑ MOUSE Curb Inlets
☑ MOUSE Pumps
☑ MOUSE Valves
☐ 2D Overland Group
☐ MOUSE Boundaries
☐ Load Allocation Group
☐ Catchment Group
☑ Measurement Stations
☑ Stations
Map
Ready
NUM

Figure 3.8 Soakaway shown in Feature layer

At the 'Geometry' tab in the dialog the type of node is selected in the Node type combobox and the following attributes are set:

- Ground Level
- Invert Level
- Basin Geometry
- Cover Type (Normal, Spilling and Sealed).

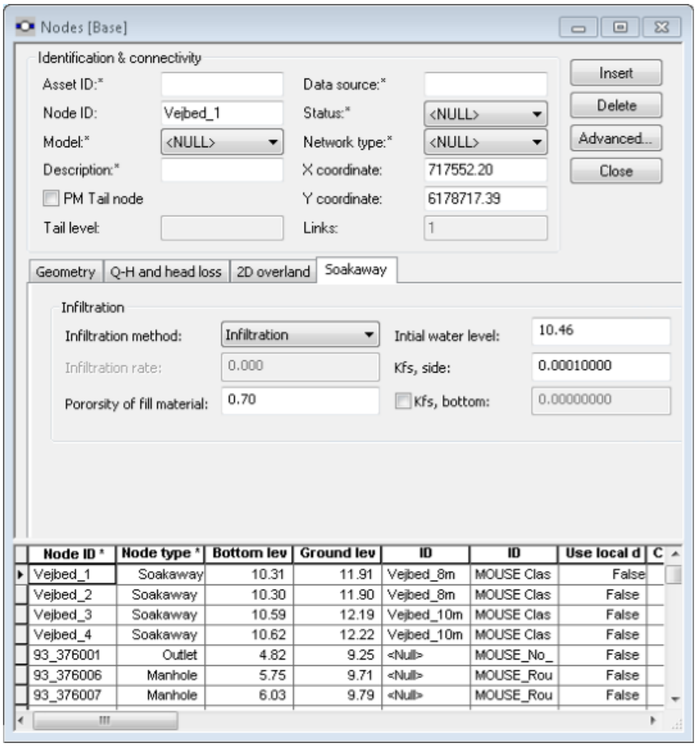
The geometry of the soakaway is defined as a standard basin.

Soakaway tab

At the 'Soakaway' tab in the dialog the type of infiltration, porosity and initial water level is selected

The following types of infiltration are available:

- No Infiltration
- Constant Infiltration
- Infiltration



Asset ID:*
Node ID: Vejbed_1
Model:* <NULL>
Description:*
☐ PM Tail node
Tail level:
Data source:*
Status:* <NULL>
Network type:* <NULL>
X coordinate: 717552.20
Y coordinate: 6178717.39
Links: 1

Infiltration method: Infiltration
Infiltration rate: 0.000
Porosity of fill material: 0.70
Intial water level: 10.46
Kfs, side: 0.00010000
☐ Kfs, bottom: 0.00000000

Node ID *	Node type *	Bottom lev	Ground lev	ID	ID	Use local d	C
Vejbed_1	Soakaway	10.31	11.91	Vejbed_8m	MOUSE Clas	False	
Vejbed_2	Soakaway	10.30	11.90	Vejbed_8m	MOUSE Clas	False	
Vejbed_3	Soakaway	10.59	12.19	Vejbed_10m	MOUSE Clas	False	
Vejbed_4	Soakaway	10.62	12.22	Vejbed_10m	MOUSE Clas	False	
93_376001	Outlet	4.82	9.25	<Null>	MOUSE_No_	False	
93_376006	Manhole	5.75	9.71	<Null>	MOUSE_Rou	False	
93_376007	Manhole	6.03	9.79	<Null>	MOUSE_Rou	False	

Figure 3.9 Node Editor

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

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

The 'Infiltration' option provides the functionality of having a variation in the infiltration based in the water level in the soakaway. A schematic drawing of the soakaway is provided in Figure 1.5 and Equation 1 describes the water balance of the model. Equation 2 and Equation 3 describe how the infiltration rate is calculated. Parameters and variables are listed and explained and how they are used in MIKE 1D.

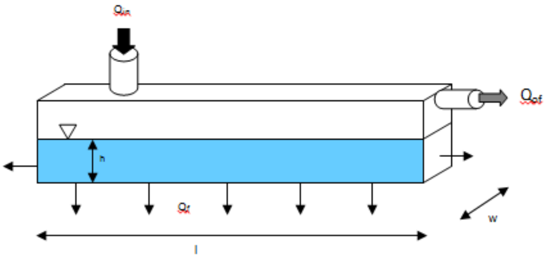


Figure 3.10 Schematic of the soakaway model

The 'Infiltration' option is based on the infiltration rate calculated by Equation (3.1) and Equation (3.2):

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

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

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

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

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

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

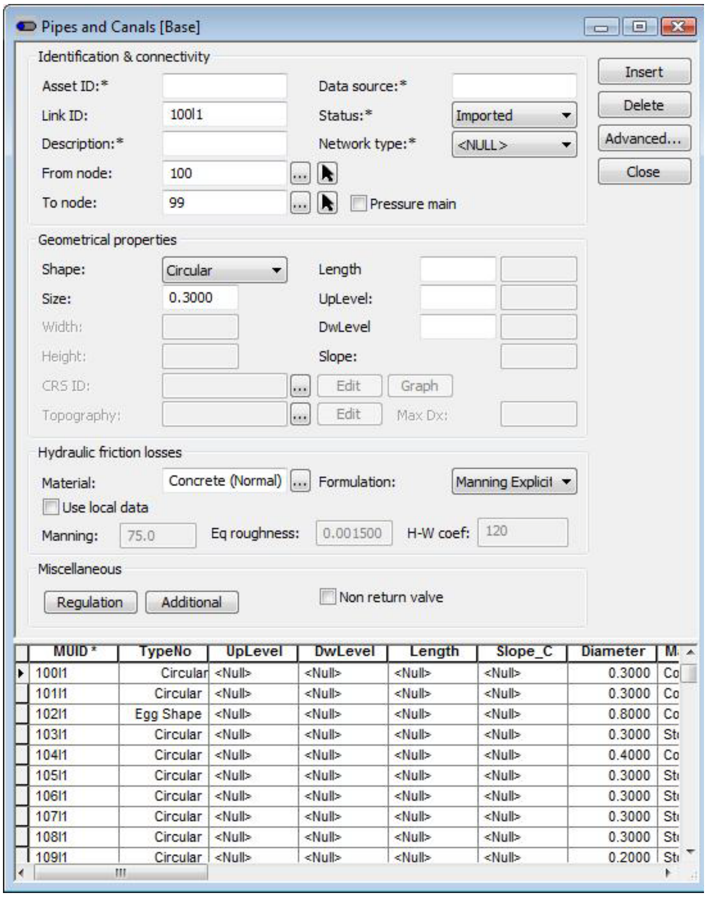
The infiltration from the bottom can be turned off by a flag. However the infiltration from both side and bottom can be shut off by setting the field-saturated hydraulic conductivity to zero.

The porosity of the fill material is used to calculate the water level in the soakaway and the initial water level is used to set the initial water level in the soakaway. Table values of hydraulic conductivity, K_{fs}, for different soil classes are provided in Table 3.3. Within each soil type the hydraulic conductivity varies significant why it is important to the measure the hydraulic conductivity at the site.

Table 3.3 Hydraulic conductivity for different soil classes

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

3.4 Pipes and Canals


Pipes and Canals [Base]
Insert
Delete
Advanced...
Close
Identification & connectivity
Geometrical properties
Hydraulic friction losses
Miscellaneous
Edit
Graph
Edit
Max Dx:
Regulation
Additional

Asset ID:*
Link ID: 100l1
Description:*
From node: 100
To node: 99
Data source:*
Status:* Imported
Network type:* <NULL>
☐ Pressure main

Shape: Circular
Size: 0.3000
Width:
Height:
CRS ID:
Topography:
Length
UpLevel:
DwLevel
Slope:

Material: Concrete (Normal)
Formulation: Manning Explicit
☐ Use local data
Manning: 75.0
Eq roughness: 0.001500
H-W coef: 120

☐ Non return valve

MUID *	TypeNo	UpLevel	DwLevel	Length	Slope_C	Diameter	M
100l1	Circular	<Null>	<Null>	<Null>	<Null>	0.3000	Co
101l1	Circular	<Null>	<Null>	<Null>	<Null>	0.3000	Co
102l1	Egg Shape	<Null>	<Null>	<Null>	<Null>	0.8000	Co
103l1	Circular	<Null>	<Null>	<Null>	<Null>	0.3000	St
104l1	Circular	<Null>	<Null>	<Null>	<Null>	0.4000	Co
105l1	Circular	<Null>	<Null>	<Null>	<Null>	0.3000	St
106l1	Circular	<Null>	<Null>	<Null>	<Null>	0.3000	St
107l1	Circular	<Null>	<Null>	<Null>	<Null>	0.3000	St
108l1	Circular	<Null>	<Null>	<Null>	<Null>	0.3000	St
109l1	Circular	<Null>	<Null>	<Null>	<Null>	0.2000	St

Figure 3.11 Pipes and Canals editor

A link is specified as a conduit between two nodes. A link is considered as either a straight line or a drawn polyline between two nodes and per default is assumed to connect the adjacent nodes at bottom levels. Pipes permanently running under pressure are specified by setting the tickmark in “Pressure main”. Please refer to the MOUSE Reference Manual (Pipe Flow) for further on pressure mains.

The respective node bottom levels are displayed in the grey areas of the “UpLevel” and “DwLevel” fields by selecting “Recompute” when clicking the “Advanced” button.

In case of a step-wise connection (but not allowed below node bottom level), the elevations of both the upstream and downstream connection must be specified in the editable “UpLevel” and “DwLevel” fields.

Specification of nodes as 'upstream' and 'downstream' does not have any impact on the computations, apart that positive flow is considered from upstream to downstream. Therefore, it is recommended to specify the upstream and downstream in the direction of predominant flows. The specification of ‘upstream’ and ‘downstream’ can be swapped by selecting “Swap nodes” when clicking on the “Advanced” button.

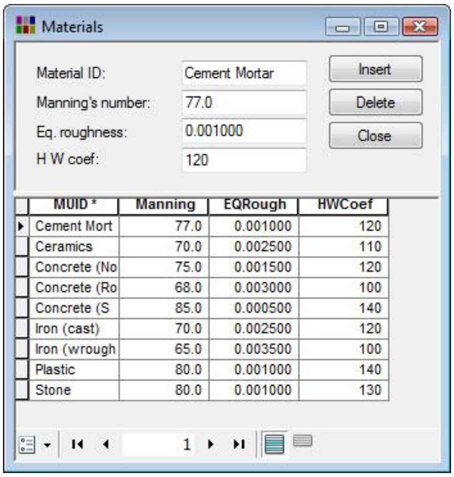
Depending on the selected type, a link may take the form of one of the 'standard' pipes (Circular, Rectangular, O Shaped, Egg-Shaped), or any closed or open cross section shape (CRS) and Natural Channels. The CRS and Natural Channels are defined in the CRS and Topography Editors.

Standard pipes are defined by diameter (or cross section width and height for non-circular pipes), the geometry of special cross sections is as mentioned specified under the cross section editor. In this dialog, only the reference to the CRS ID.

For natural channels a topography, defined in the Topography editor, is specified. The topography is specified with a series of CRS, where the first is placed in chainage 0. The chainage of the last CRS defines the length of the topography. If the length of the topography is shorter than the computed length for the link (or user specified length if specified) the last cross section will be used for the remaining part of the natural channel. And vice versa, if the length of the topography is longer than the link length only the topography specified until the computed length (or user specified length) will be used, the CRS at the end of the link may be an interpolation from two CRS. When the topography length differs from the computed length (or the user defined length if specified) a warning will be issued. It is possible to define an optional maximum length, dx between to h-points between two CRS. I.e. the distance between two chainages is 235 m and the max dx = 150 m, then MOUSE will add an h-point at the middle between the two CRS at 117.5 m.

A link is characterised by material, which determines the Manning friction coefficient (Manning), the Colebrook White coefficient (Equivalent roughness) or Hazen-Williams coefficient. It is optional to use either the default roughness values for specific materials or local values.

Specification of the different kind of materials and roughness coefficients is done through the ‘MOUSE | Materials’ Editor



Material ID: Cement Mortar
Manning's number: 77.0
Eq. roughness: 0.001000
H W coef: 120

MUID *	Manning	EQRough	HWCoef
Cement Mort	77.0	0.001000	120
Ceramics	70.0	0.002500	110
Concrete (No	75.0	0.001500	120
Concrete (Ro	68.0	0.003000	100
Concrete (S	85.0	0.000500	140
Iron (cast)	70.0	0.002500	120
Iron (wrough	65.0	0.003500	100
Plastic	80.0	0.001000	140
Stone	80.0	0.001000	130

Figure 3.12 Materials Editor

The length of a link is calculated from the shape of the line in MIKE URBAN. The length is displayed in the 'Length_C' field, but is not updated until the 'Recompute' command is executing (from the ‘Advanced’ button). If a user defined length is specified this will overwrite the calculated one during simulation.

3.4.1 Identification group

Table 3.4 The edit fields in the Identification group

Edit field	Description	Used or required by simulations	Field name in datastructure
Asset ID	Reference to an ID used in external data sources	No	AssetName
Data source	Reference to an external data source (table ID) where the record has been imported from	No	DataSource

Table 3.4 The edit fields in the Identification group

Edit field	**Description**	**Used or required by simulations**	**Field name in datastructure**
Link ID	A unique name for the node. Up to 40 characters (letters, numbers, blank spaces and underscore characters)	Yes	MUID
Status	Data status for the entire record, serves for keeping track on the source of information	No	Element_S
Description	User's descriptive information related to the link	No	Description
Network type	Attributes the link to a certain type of network. Used in cases when two or more different networks are included in the same project	No	NetTypeNo
From node	Upstream Node	Yes	MUID
To node	Downstream Node	Yes	MUID
Pressure main	Defines a link as pressure main. A link connected to a manhole or basin, can only constitute a pressure main if the manhole/basin is declared to be “tail node”. Please refer to the MOUSE Reference Manual (Pipe Flow) for further on pressure mains.	No	PMApprNo

3.4.2 Geometrical properties

Table 3.5

Edt field	Description	Used or required by simulations	Field name in datastructure
Shape	Shape of pipe	Yes	TypeNo
Size	Nominel size of pipe (diameter of circular pipe, height of Egg-shape pipe and width for O-shaped)	Yes, if Shape = Circular, Egg-Shape and O-Shaped	Diameter

Table 3.5

Edt field	Description	Used or required by simulations	Field name in datastructure
Width	Width of rectangular shape	Yes, if Shape = Rectangular	Width
Height	Height of rectangular shape	Yes, if Shape = Rectangular	Height
CRS ID	ID of cross section	Yes, if shape = CRS	CrsID
Topography	ID of topography	Yes, if Shape = natural channel	TopographyID
Max Dx	Max distance between gridpoints	Yes, if Shape = natural channel	Maxdx
Length	Length of link	Yes	Length
UpLevel	Upstream invert level of link	Yes	UpLevel
DwLevel	Downstream invert level of link	Yes	DwLevel

3.4.3 Hydraulic friction losses

Table 3.6

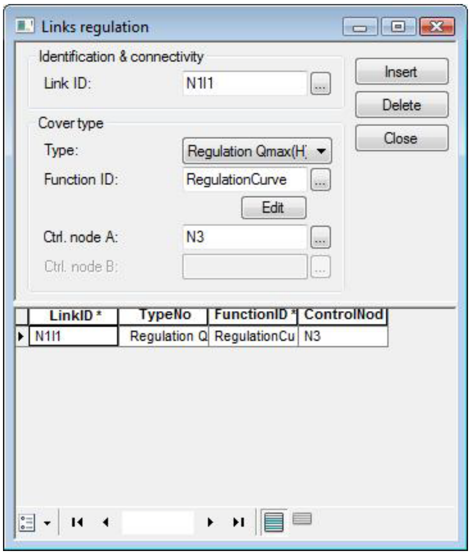
Edit field	Description	Used or required by simulations	Field name in datastructure
Material	Material of link	Yes	MaterialID
Formulation	Formula for calculation of the friction loss (Manning Explicit, Manning Implicit, Colebrook White, Hazen-Williams)	Yes	FricTypeNo
Use local data	Determines if roughness values from the material are overwritten by local values	Yes	FricNo
Manning	Manning roughness value	Yes, if ‘Manning Explicit’ or ‘Manning Implicit’ is chosen	Manning

Table 3.6

Edit field	Description	Used or required by simulations	Field name in datastructure
Eq. roughness	Equivalent roughness	Yes, if 'Colebrook White' formulation is chosen	Rough
H-W coef	Hazen-Williams roughness coefficient	Yes, if 'Hazen-Williams' is chosen	HWCoef

3.4.4 Miscellaneous

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



Link ID: N1l1

Type: Regulation Qmax(H

Function ID: RegulationCurve

Ctrl. node A: N3

Ctrl. node B:

LinkID *	TypeNo	FunctionID *	ControlNod
N1l1	Regulation Q	RegulationCu	N3

Figure 3.13 Links regulation dialog

The 'Additional' button provides access to more advanced options for a link such as specifying a depth-variable Manning number.

3.5 Weirs

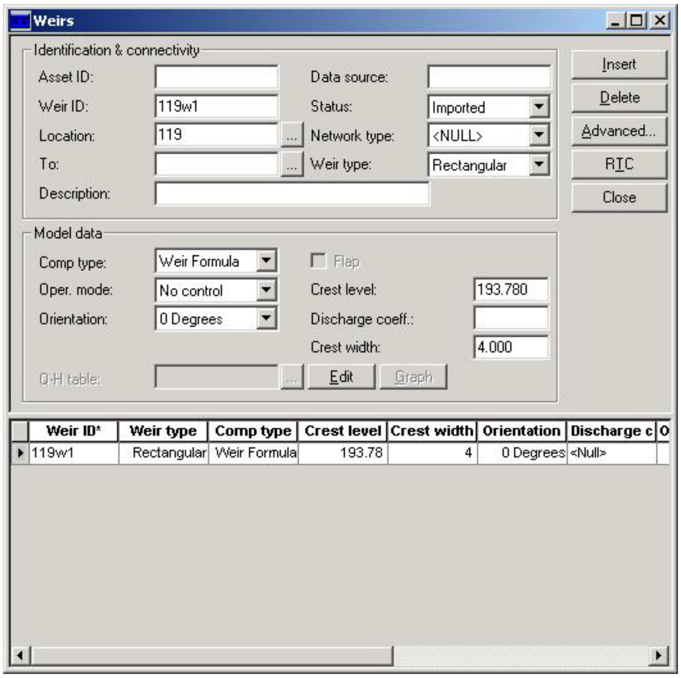
A weir is actually a functional relation, which connects two nodes of a MOUSE network (two-directional flow and submerged flow possible), or is associated with only one node (free flow 'out of the system'). The latter case is achieved if the 'To' field is left empty.

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

It is possible to define several weirs between the same two nodes if this is required. This is similar to the possibility of having more than one pipe as the link between nodes. The generation of the computational grid shown in Figure 3.15 for the orifice is also applied for pumps, weirs and valves. The numerical solution of the flow equations will depend on the selected device. Please refer to the reference manual on more on this.

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

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



Asset ID:
Weir ID: 119w1
Location: 119
To:
Description:
Data source:
Status: Imported
Network type: <NULL>
Weir type: Rectangular

Comp type: Weir Formula
Oper. mode: No control
Orientation: 0 Degrees
☐ Flap
Crest level: 193.780
Discharge coeff.:
Crest width: 4.000
Q-H table:

Weir ID*	Weir type	Comp type	Crest level	Crest width	Orientation	Discharge c	O
119w1	Rectangular	Weir Formula	193.78	4	0 Degrees	<Null>	

Figure 3.14 Weirs editor

A weir is characterised by the computational method, weir type, crest level, crest width, and orientation. If the Q-H relation is specified, only the crestlevel and a DataSetID are specified. With the built-in weir formula, the results are affected by the specified parameters. The weir type can be selected among 'Rectangular', ‘V-notch’, ‘Trapezoidal’, ‘Irregular’and ‘Long weirs’. For ‘Rectangular’ and ‘Long Weirs’ the ‘Weir formula’ option is used. For the other weir types the Q-H relation must be used.

Orientation ('degrees') plays an important role (as long as the head loss coefficient is undefined), since depending on the specified orientation, kinetic energy of the flow is included (90^{o}) or is not included (0^{o}) in calculations of the weir flows.

The dimensionless head loss coefficient is optional. If the coefficient is specified, it will overwrite the default and change the computation mode (so that the effect of ‘orientation’ will not be included) during the simulation.

Weirs are per default static (No Control) but can be controlled through Real Time Control (RTC). Clicking on the “RTC” button to the right gives quick access to the RTC specification dialog.

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

3.5.1 Identification and connectivity

Table 3.7

Edit field	Description	Used or required by simulations	Field name in datastructure
Asset ID	Reference to an ID used in external data sources	No	AssetName
Data source	Reference to an external data source (table ID) where the record has been imported from	No	DataSource
Weir ID	A unique name for the weir. Up to 40 characters (letters, numbers, blank spaces and underscore characters)	Yes	MUID
Status	Data status for the entire record, serves for keeping track on the source of information	No	Element_S
Location	ID of Node where Weir is located	Yes	MUID
To	ID of Node where Weir is discharging to. If field left empty, then water is discharging out of the system	Yes	MUID
Network type	Attributes the weir to a certain type of network. Used in cases when two or more different networks are included in the same project	No	NetTypeNo
Weir type	Specification of type of weir	Yes	TypeNo
Description	User's descriptive information related to the weir	No	Description

3.5.2 Model data

Table 3.8

Edit field	Description	Used or required by simulations	Field name in datastructure
Comp type	Selection of computation Method	Yes	MethodNo
Flap	Flap indicating a flap-gate built-in weir (i.e. no return flow possible)	Yes	FlapNo
Oper. mode	No control or RTC controllable weir	Yes	ControlTypeNo
Orientation	Weir orientation relative to the main flow direction. “0” is Side weir, “90” is a transversal weir	Yes, if discharge coeff. is not specified	AngleNo
Crest level	Crest level of weir	Yes	CrestLevel
Discharge coeff.	Discharge coefficient	Yes, if weir formula is chosen	Coeff
Crest width	Width of rectangular weir	Yes, if weir formula is chosen	CrestWidth
Q-H table	Reference to tabulated Q-H funtion	Yes, if Q-H is chosen	QHID
Source Channel	The ID of the source or the upstream channel of the weir	Yes, if fragmented is chosen	SourceLinkID
Destination Channel	The ID of the destination or the downstream channel of the weir	Yes, if fragmented is chosen	DestinationLinkID
Weir Crest Geometry	Reference to tabulated variation of the weir crest along the weir	Yes	WeirCrestID

3.6 Orifices

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

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

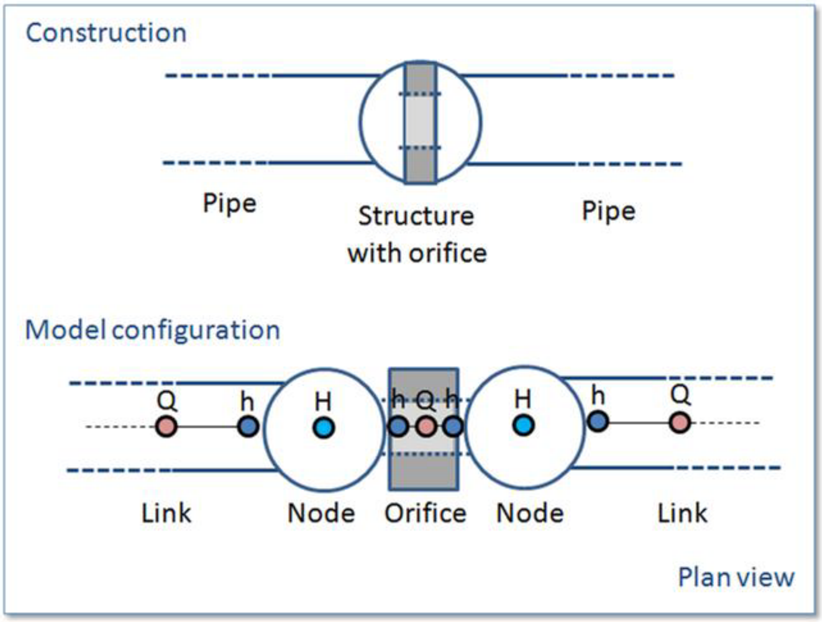


Figure 3.15 The difference between real world orifice and model configuration of orifice

It is possible to define several orifices between the same two nodes if this is required. This is similar to the possibility of having more than one pipe as the link between nodes. The generation of the computational grid shown in Figure 3.15 for the orifice is also applied for pumps, weirs and valves. The numerical solution of the flow equations will depend on the selected device. Please refer to the reference manual on more on this.

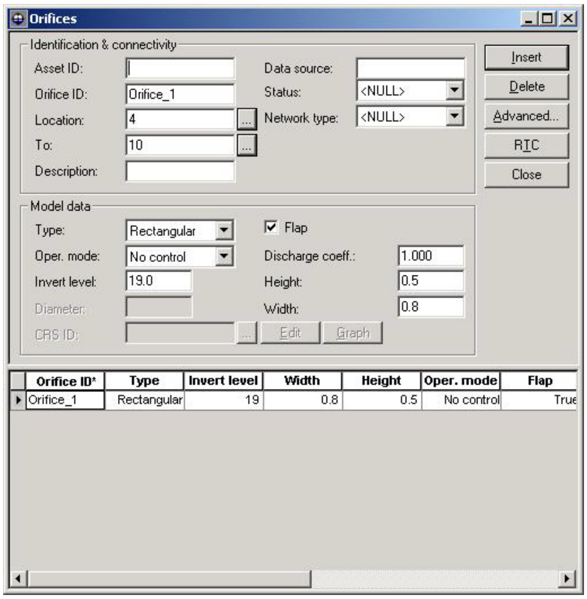
It is recommended not to place the two nodes in the same spot, instead place the nodes a short distance apart. The reason is that the node head loss computation will have a component from change of flow direction. If the two nodes surrounding the device are placed exactly at the same location then the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

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

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

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

Orifices are per default static (No Control) but an orifice can be controlled through Real Time Control (RTC). Clicking on the “RTC” button to the right gives quick access to the RTC specification dialog.



Identification & connectivity

Asset ID:
Orifice ID: Orifice_1
Location: 4
To: 10
Description:
Data source:
Status: <NULL>
Network type: <NULL>

Model data

Type: Rectangular
Oper. mode: No control
Invert level: 19.0
Diameter:
CRS ID:
☑ Flap
Discharge coeff.: 1.000
Height: 0.5
Width: 0.8

Orifice ID*	Type	Invert level	Width	Height	Oper. mode	Flap
Orifice_1	Rectangular	19	0.8	0.5	No control	True

Figure 3.16 Orifice editor

3.6.1 Identification and connectivity

Table 3.9

Edit field	Description	Used or required by simulations	Field name in datastructure
Asset ID	Reference to an ID used in external data sources	No	AssetName
Data source	Reference to an external data source (table ID) where the record has been imported from	No	DataSource
Orifice ID	A unique name for the orifice. Up to 40 characters (letters, numbers, blank spaces and underscore characters)	Yes	MUID
Status	Data status for the entire record, serves for keeping track on the source of information	No	Element_S
Location	ID of Node where orifice is located	Yes	MUID
To	ID of Node where orifice is discharging to. If field left empty, then water is discharging out of the system	Yes	MUID
Network type	Attributes the link to a certain type of network. Used in cases when two or more different networks are included in the same project	No	NetTypeNo
Description	User's descriptive information related to the orifice	No	Description

3.6.2 Model data

Table 3.10

Edit field	Description	Used or required by simulations	Field name in datastructure
Type	Type of orifice according to shape	Yes	TypeNo
Flap	Flap indicating a flap-gate built-in (i.e. no return flow possible)	Yes	FlapNo

Table 3.10

Edit field	Description	Used or required by simulations	Field name in datastructure
Oper. mode	No control or RTC controllable orifice	Yes	ControlTypeNo
Discharge coeff	Calibration coefficient. Value = 1 results in the flow as determined by orifice algorithm	Yes	DischargeCoeff
Invert level	Absolute elevation of the orifice invert	Yes	InvertLevel
Height	Height of rectangular orifice	Yes	Height
Width	Width of rectangular orifice	Yes	Width
Diameter	Diameter of circular orifice	Yes	Diameter
CRS ID	Reference of a cross-section ID for irregularly-shaped orifice	Yes, if CRS type is chosen	CrsID

3.6.3 Defining a gate or a weir in an orifice

The orifice itself is just an opening with a static shape. In real constructions orifices are often equipped with a controlled gate or weir which can be used in real time control for regulating the flow through the orifice. The gate device will move from the top of the orifice opening and downwards until the orifice is fully closed. The weir moves from the bottom of the orifice upwards and closes fully when the weir crest reaches the top of the orifice opening (see Figure 3.17 for an illustration). It is possible to apply both types of movable devices in the computations. In both cases the device is "added" to a defined orifice. This is done from the Controllable Devices dialog (MOUSE|Control|Controllable Devices). See more on this in section 9.8.

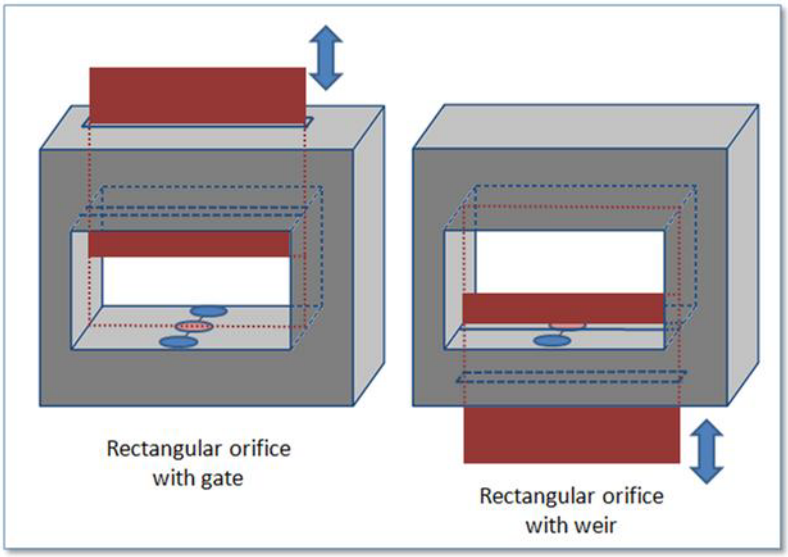

Rectangular orifice
with gate
Rectangular orifice
with weir

Figure 3.17 Examples on a rectangular orifice with a gate and a weir

3.7 Stormwater Inlets

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

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

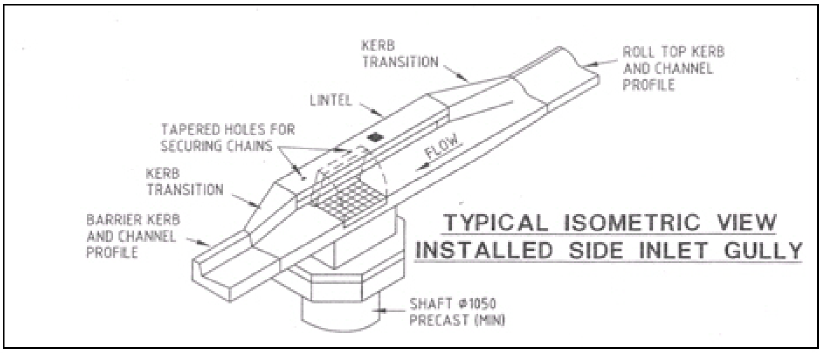

KERB
TRANSITION
ROLL TOP KERB
AND CHANNEL
PROFILE
LINTEL
TAPERED HOLES FOR
SECURING CHAINS
FLOW
KERB
TRANSITION
BARRIER KERB
AND CHANNEL
PROFILE
TYPICAL ISOMETRIC VIEW
INSTALLED SIDE INLET GULLY
SHAFT ⌀1050
PRECAST (MIN)

Figure 3.18 A typical curb inlet configuration

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

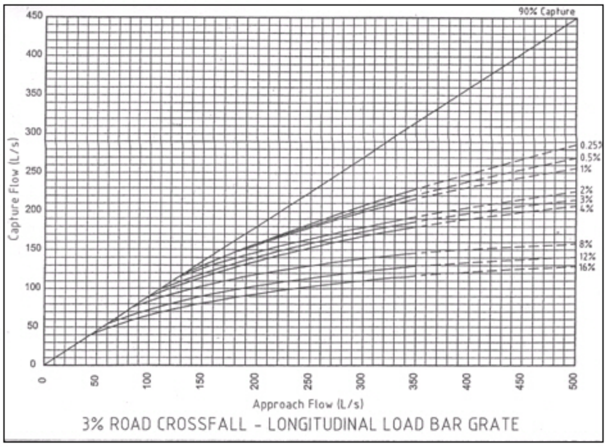


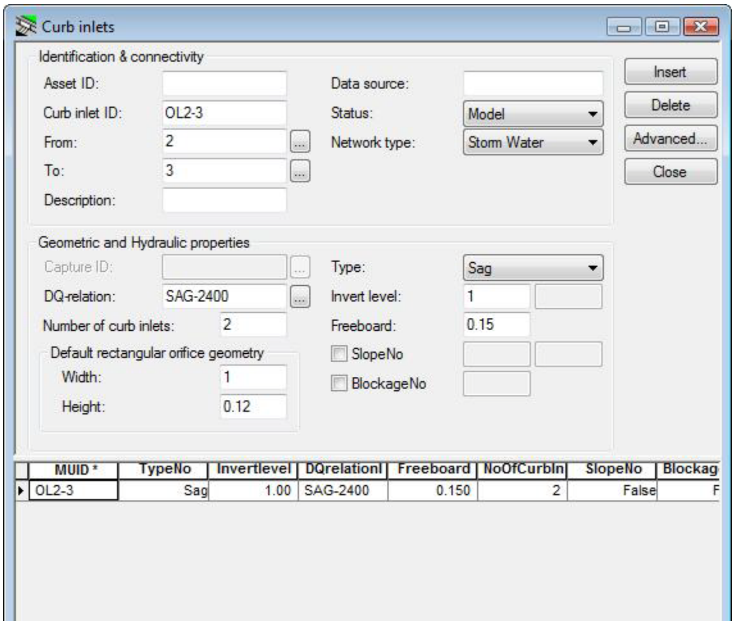
Figure 3.19 Example of empirical curves for On-Grade type

3.7.1 Curb Inlet (Lintel)

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

There are two types of Curb Inlet:

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



Identification & connectivity

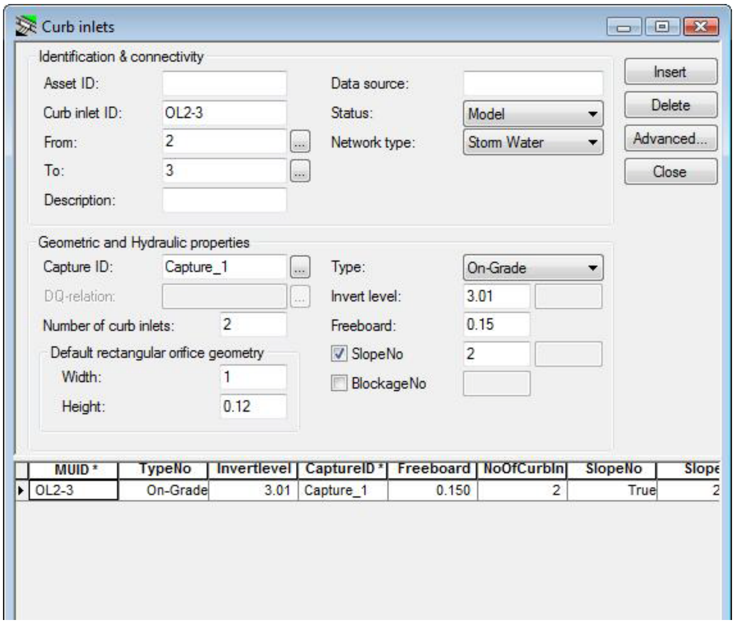
Asset ID:
Curb inlet ID: OL2-3
From: 2
To: 3
Description:
Data source:
Status: Model
Network type: Storm Water

Geometric and Hydraulic properties

Capture ID:
DQ-relation: SAG-2400
Number of curb inlets: 2
Default rectangular orifice geometry
Width: 1
Height: 0.12
Type: Sag
Invert level: 1
Freeboard: 0.15
☐ SlopeNo
☐ BlockageNo

MUID *	TypeNo	Invertlevel	DQrelationI	Freeboard	NoOfCurbIn	SlopeNo	Blockag
OL2-3	Sag	1.00	SAG-2400	0.150	2	False	F

Figure 3.20 The Curb Inlet data dialog (SAG Type)



Identification & connectivity

Asset ID:
Curb inlet ID: OL2-3
From: 2
To: 3
Description:
Data source:
Status: Model
Network type: Storm Water

Geometric and Hydraulic properties

Capture ID: Capture_1
DQ-relation:
Number of curb inlets: 2
Default rectangular orifice geometry
Width: 1
Height: 0.12
Type: On-Grade
Invert level: 3.01
Freeboard: 0.15
☑ SlopeNo 2
☐ BlockageNo

MUID *	TypeNo	Invertlevel	CaptureID *	Freeboard	NoOfCurbIn	SlopeNo	Slope
OL2-3	On-Grade	3.01	Capture_1	0.150	2	True	2

Figure 3.21 The Curb Inlet data dialog (On-Grade Type)

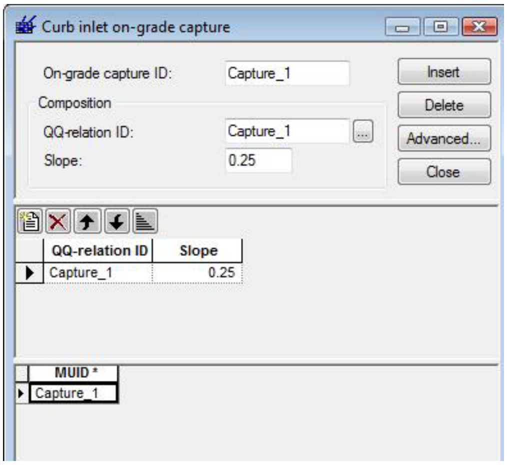
User defined parameters in the Curb Inlet dialog include:

- Invert level (m) defining the point at which spilling starts (similar to weir crest level). The user is shown a system calculated invert level which is the same as the invert of the connection node in the overland flow network. As with weir flow, a crest level at least 0.01 m higher than the connection node invert level is recommended for initial condition stability.
- Freeboard (m), defining a critical water level (Invert - Freeboard) at the connection node in the pipe network below which the defined DQ and QQ-relations apply. For submerged and reverse flow (surcharge), the transfer capacity of the connection reverts to a standard orifice relationship.
- Slope (%), representing the slope of the steepest link in the overland flow network entering the connection node (only applies to ON-GRADE Type). The system calculated slope is used in the calculation unless a user defined slope is specified.
- Blockage factor (%) which can be used to account for debris blockage at the grate/inlet. This linear factor is applied to the tabular data sets defining the transfer capacity of the connection.
- Number of Curb Inlets, allowing multiple curb inlets of the same specified geometry (transfer capacity) applied at the same location within a single connection.
- Default rectangular orifice geometry, applies to those flow cases (submerged and reverse flow) were the defined DQ and QQ-relations do not apply. This generally applies when water levels at the connection node in the pipe network exceed the critical level defined by the Freeboard, including reverse flow (surcharge).

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

3.7.2 On-grade Capture

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



On-grade capture ID: Capture_1

Composition

QQ-relation ID: Capture_1

Slope: 0.25

QQ-relation ID	Slope
Capture_1	0.25

MUID *
Capture_1

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

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

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

3.7.3 Capacity curves

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

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

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

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

3.8 Pumps

A pump is actually a functional relation, which connects two nodes of a MOUSE network or is associated with only one node (free flow 'out of the system'). The latter case is achieved if the 'To' field is left empty. In this case it is also possible to let the water pumped out of the system enter a 2D model (in the case of coupling MIKE URBAN to MIKE 21, i.e. MIKE FLOOD). This is done by setting the tickmark in ‘Coupling to 2D overland flow’.

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

It is possible to define several pumps between the same two nodes if this is required. This is similar to the possibility of having more than one pipe as the link between nodes. The generation of the computational grid shown in Figure 3.15 for the orifice is also applied for pumps, weirs and valves. The numerical solution of the flow equations will depend on the selected device. Please refer to the reference manual on more on this.

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

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

3.8.1 Pump types

Several pump types can be specified in MIKE URBAN.

Constant flow pumps

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

When specifying a constant flow pump, set the Speed to ‘Constant’ and choose ‘Constant’ in the Cap. curve type field and type in the constant flow in the Constant flow field.

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

Constant speed pumps

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

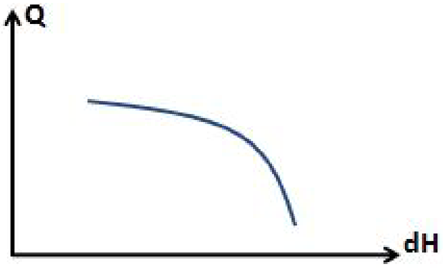


Figure 3.23 Typical example of pump curve

Pump curves are defined in MIKE URBAN by tabulating the curve (see more in section 5). This can be done by selecting representative points on the pump curve as shown in the next figure. Information about pump curves are typically provide by the manufacturer of the pumps.

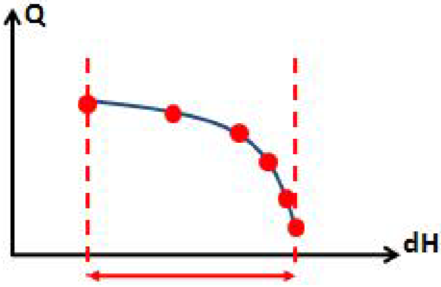


Figure 3.24 Tabulating the curve

When a pump curve is tabulated like this and used in the simulations executed with the MOUSE Engine, then the pump is allowed to operate within the range of the dH values in the table. At any time during the simulation the MOUSE Engine will compute the actual dH and determine the corresponding Q value from the table, resulting in the actual duty point position for the pump.

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

Variable speed pumps

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

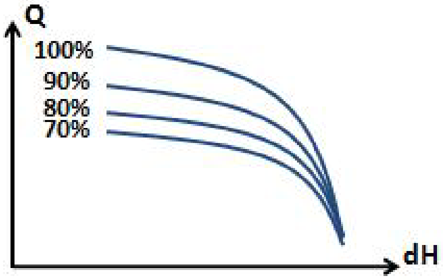


Figure 3.25 Pump curves at various percentages of maximum rotation speed or maximum power input

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

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

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

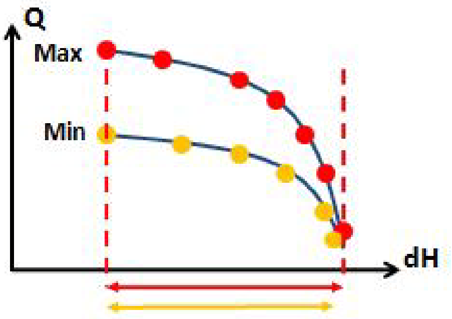


Figure 3.26 The RPMmax and RPMmin pump curves

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

Operation of variable speed pumps

If the 'Speed' selection field is set to 'Variable' and the 'Oper. Mode' field is set to 'Wet Well Set Point' a value for the wet well set point is entered.

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

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

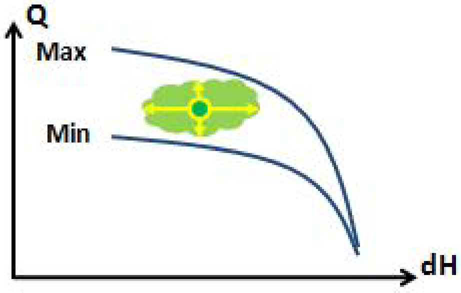


Figure 3.27

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

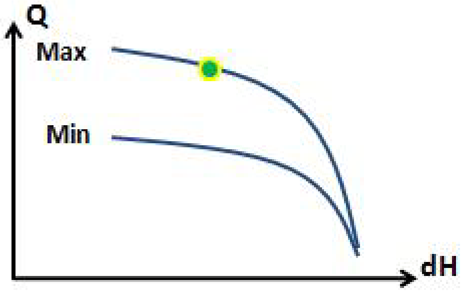


Figure 3.28

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

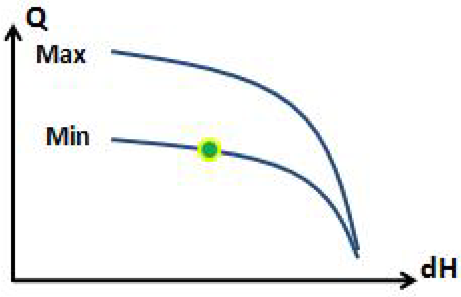


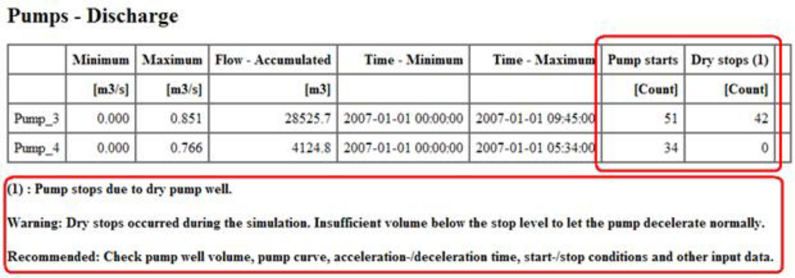
Figure 3.29

More detailed output information may be added to the standard result file for variable speed pumps. Please refer to the documentation on the DHIAPP.INI file found in the “DHIAPP INI and ADP Files Reference Manual”.

Registration of 'dry stops' of pumps

The hydrodynamic network simulation provides a simulation summary report at the end of the computation. The summary for the pumps will show the number of pump starts during the simulation and if the individual pumps have been forced to stop due to lack of water in the pump wet well.

There can be various reasons for situations where pumps are running dry. When this happens in the model simulation it indicates a mismatch in the configuration. E.g. too small wet well volume, errors in the applied pump capacity curves or deceleration times.



Pumps - Discharge

	Minimum	Maximum	Flow - Accumulated	Time - Minimum	Time - Maximum	Pump starts	Dry stops (1)
	[m3/s]	[m3/s]	[m3]			[Count]	[Count]
Pump_3	0.000	0.851	28525.7	2007-01-01 00:00:00	2007-01-01 09:45:00	51	42
Pump_4	0.000	0.766	4124.8	2007-01-01 00:00:00	2007-01-01 05:34:00	34	0

(1) : Pump stops due to dry pump well.

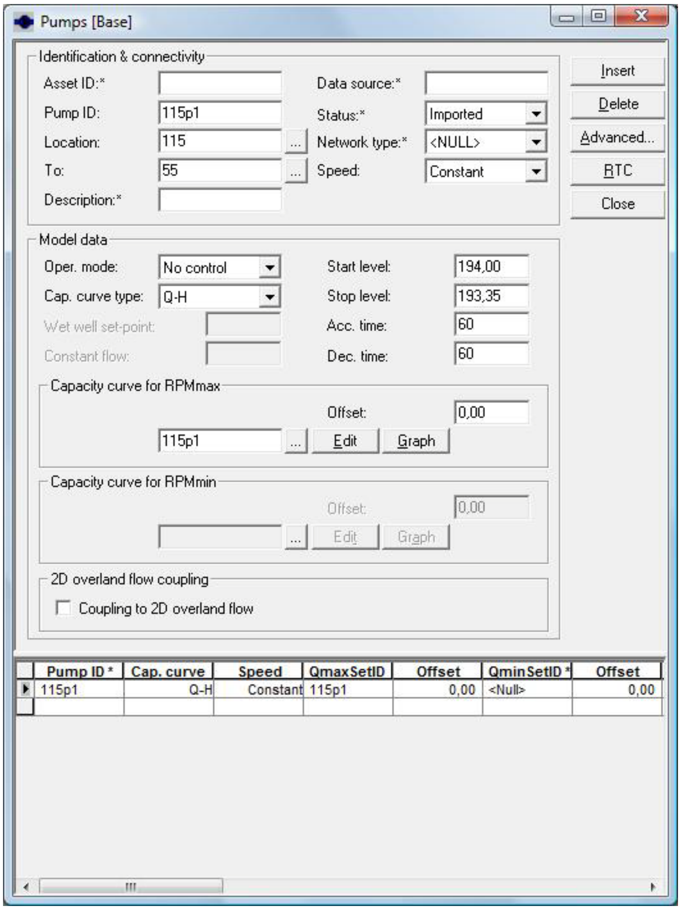
Warning: Dry stops occurred during the simulation. Insufficient volume below the stop level to let the pump decelerate normally.

Recommended: Check pump well volume, pump curve, acceleration-/deceleration time, start-/stop conditions and other input data.

Figure 3.30 Summary reveals dry stops during simulation

Pumps will be switched off when the water level drops to the defined stop level. But the pump flow continues during the specified deceleration period by a linear decrease to zero flow. During this time interval the water level in wet well may eventually drop to the bottom of the wet well during a single time step. In this situation the MOUSE Engine will force an immediate 'dry stop' of the pump discharge.

The 'Pumps - Discharge' section is only output to the summary report if 'pumps' has been selected in the summary selection dialog (found on the Simulation dialog).



Identification & connectivity
Asset ID:*
Pump ID: 115p1
Location: 115
To: 55
Description:*
Data source:*
Status:* Imported
Network type:* <NULL>
Speed: Constant

Model data
Oper. mode: No control
Cap. curve type: Q-H
Wet well set-point:
Constant flow:
Start level: 194,00
Stop level: 193,35
Acc. time: 60
Dec. time: 60

Capacity curve for RPMmax
Offset: 0,00
115p1

Capacity curve for RPMmin
Offset: 0,00

2D overland flow coupling
☐ Coupling to 2D overland flow

Pump ID *	Cap. curve	Speed	QmaxSetID	Offset	QminSetID *	Offset
115p1	Q-H	Constant	115p1	0,00	<Null>	0,00

Figure 3.31 Pumps editor

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

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

Pumps are per default static (No Control) but can be controlled through Real Time Control (RTC). Clicking on the “RTC” button to the right gives quick access to the RTC specification dialog.

3.8.2 Identification and connectivity

Table 3.11

Edit field	Description	Used or required by simulations	Field name in datastructure
Asset ID	Reference to an ID used in external data sources	No	AssetName
Data source	Reference to an external data source (table ID) where the record has been imported from	No	DataSource
Pump ID	A unique name for the pump. Up to 40 characters (letters, numbers, blank spaces and underscore characters)	Yes	MUID
Location	ID of Node where pump is located	Yes	MUID
To	ID of Node where Pump is pumping to.	Yes	MUID
Status	Data status for the entire record, serves for keeping track on the source of information	No	Element_S
Network type	Attributes the pump to a certain type of network. Used in cases when two or more different networks are included in the same project	No	NetTypeNo
Speed	Selection between constant and variable speed pumps. Only variable speed pumps can be used for PID control	Yes	SpeedNo
Description	User's descriptive information related to the pump	No	Description

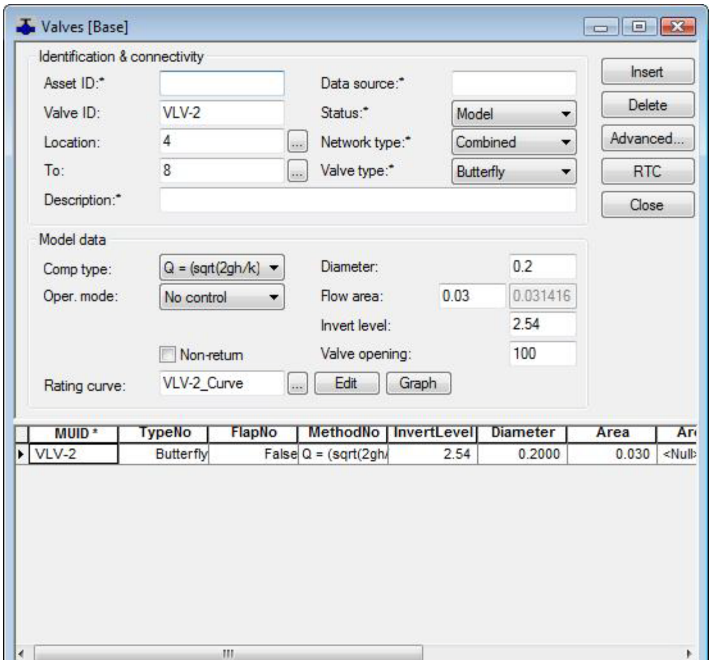
3.8.3 Model data

Table 3.12

Edit field	Description	Used or required by simulations	Field name in datastructure
Oper. mode	Switch between "normal" (i.e. ON/OFF at fixed level) and RTC operation. "Wet-well set point" is a special RTC control, which is accessible also without RTC add-on module	Yes	ControlTypeNo
Cap. curve type	Type of capacity curve. Type 1 is applicable for archimed Screw, type 2 for all rotodynamic pumps	Yes	CapTypeNo
Start level	Water level in pump sump which triggers the pump to start	Yes	StartLevel
Stop level	Water level in pump sump which triggers the pump to stop	Yes	StopLevel
Acc. time	Pump acceleration time. Used to dampen sudden increase of flow at the pumps START events	Yes	AccTime
Dec. time	Pump decceleration time. Used to dampen sudden decrease of flow at the pumps STOP events	Yes	DecTime
Offset	Offset of the capacity curve. Applies to Q-H types, which may be specified relative to different datums	Yes. if Q-H type	Offset1
Offset	Offset of minimum capacity curve. Applies only for variable speed pumps	Yes, if variable speed pump	Offset2
Wet Well set-point	Absolute water level in the pump sump (i.e. wet well) which the pump is supposed to maintain	Yes	WetWellSetPoint
QMaxSetID	Capacity curve for the nominal (max) rotation speed	Yes	QMaxSetID
QMinSetID	Capacity curve for the minimum speed. Applies only for variable speed pumps	Yes	QMinSetID

3.9 Valves

A valve is a functional relation which connects two nodes of a MOUSE network.



Asset ID:*

Valve ID: VLV-2

Location: 4

To: 8

Description:*

Data source:*

Status:* Model

Network type:* Combined

Valve type:* Butterfly

Comp type: Q = (sqrt(2gh/k)

Oper. mode: No control

☐ Non-return

Rating curve: VLV-2_Curve

Diameter: 0.2

Flow area: 0.03 0.031416

Invert level: 2.54

Valve opening: 100

MUID *	TypeNo	FlapNo	MethodNo	InvertLevel	Diameter	Area	Ar
VLV-2	Butterfly	False	Q = (sqrt(2gh/	2.54	0.2000	0.030	<Null

Figure 3.32 Valve dialog

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

It is possible to define several valves between the same two nodes if this is required. This is similar to the possibility of having more than one pipe as the link between nodes. The generation of the computational grid shown in Figure 3.15 for the orifice is also applied for pumps, weirs and valves. The numerical solution of the flow equations will depend on the selected device. Please refer to the reference manual on more on this.

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

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

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

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

The valve is by default static, in which case the valve opening must be specified. It is also possible to define a valve to be the RTC controlled and then the valve opening will be controlled by any specified control rules.

Table 3.13 Identification and connectivity edit fields of the MIKE URBAN valve editor.

Edit field	Description	Used or required by simulations	Field name in data structure
Asset ID	Reference to an ID used in external data sources	No	AssetName
Valve ID	A unique name for the valve. Up to 40 characters (letters, numbers, blank spaces and underscore characters)	Yes	MUID
Location	ID of Node where valve is located	Yes	FROMNODE
To	ID of Node where valve is discharging to	Yes	TONODE
Description	User’s descriptive information related to the valve	No	Description
Data source	Reference to an external data source (table ID) where the record has been imported from	No	DataSource
Status	Data status for the entire record, serves for keeping track on the source of information	No	Elements

Edit field	Description	Used or required by simulations	Field name in data structure
Network Type	Attributes the valve to a certain type of network. Used in case where two or more different networks are included in the same project	No	NetTypeNo
Valve Type	Specification of type of valve, this information is not used in the simulation engine	No	TypeNo

Table 3.14 Geometrical and hydraulic properties, edit fields of the MIKE URBAN valve editor

Table 3.15

Edit field	Description	Used or required by simulations	Field name in data structure
Comp type	Selection of computation Method.	Yes	MethodNo
Oper.mode	No control or RTC controllable valve	Yes	ControlTypeNo
Non Return	Indicating a flap-gate built-in valve (i.e. no return flow possible)	Yes	FlapNo
Rating Curve	Reference to the tabulated k-opening function	Yes	RatingCurveID
Diameter	The default value of the area (the field “Flow Area”) is calculated on the assumption of a circular valve, with the diameter specified in this field. Furthermore, the transition to a pressurized valve is defined by the invert level plus the diameter.	Yes	Diameter
Flow Area	A user specified flow area overwrites the default valve area computed on the basis of a circular cross section.	Yes	Area

Table 3.15

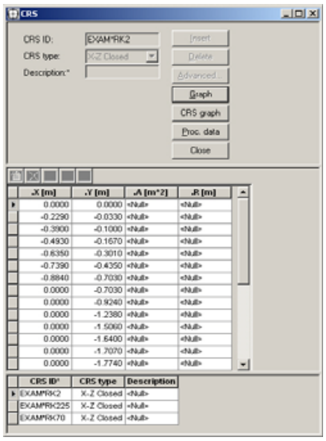
Edit field	Description	Used or required by simulations	Field name in data structure
Invert Level	The invert level defines the minimum water level, which generates flow through the valve	yes	InvertLevel
Valve Opening	Defines the opening of the valve in percentages (value between 0 and 100). For an RTC controlled valve, this value is not applied, the opening is controlled by the MIKE URBAN build in RTC algorithm	Yes	Opening

3.10 CRS & Topography

The CRS and Topograpy editors allow the definition of the conduit cross-sections and natural streams topographies, according to 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 **MOUSE | CRS & Topography**.



CRS ID: EXAM*RK2
CRS type: X-Z Closed
Description:*

X [m]	Y [m]	A [m^2]	R [m]
0.0000	0.0000	<Null>	<Null>
-0.2290	-0.0330	<Null>	<Null>
-0.3900	-0.1000	<Null>	<Null>
-0.4930	-0.1670	<Null>	<Null>
-0.6350	-0.3010	<Null>	<Null>
-0.7390	-0.4350	<Null>	<Null>
-0.8840	-0.7030	<Null>	<Null>
0.0000	-0.7030	<Null>	<Null>
0.0000	-0.9240	<Null>	<Null>
0.0000	-1.2380	<Null>	<Null>
0.0000	-1.5060	<Null>	<Null>
0.0000	-1.6400	<Null>	<Null>
0.0000	-1.7070	<Null>	<Null>
0.0000	-1.7740	<Null>	<Null>

CRS ID*	CRS type	Description
EXAM*RK2	X-Z Closed	<Null>
EXAM*RK225	X-Z Closed	<Null>
EXAM*RK70	X-Z Closed	<Null>

Figure 3.33 The CRS editor

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

- X, Z open: The CRS geometry is described by points defined by co-ordinate pairs (x, z), where 'x' is a horizontal axis, and 'z' a vertical axis. The points are specified in a counter-clockwise direction.
- X, Z closed: The CRS geometry is described by points defined by co-ordinate pairs (x, z), where 'x' is a horizontal axis, and 'z' a vertical axis. The points are specified in a counter-clockwise direction. The first and last points are connected to close the cross section.
- H, W open: The CRS geometry is described by pairs (h, w), where 'h' is relative height, and 'w' is the corresponding cross section width. The pairs are specified in an upward direction.
- H, W closed: The CRS geometry is described by pairs (h, w), where 'h' is relative height, and 'w' is the corresponding cross section width. The pairs are specified in an upward direction. The last specified (h, w) pair defines the top of the closed cross section.
- Processed open: The CRS geometry is described directly through their hydraulic parameters; Length (L), Width (W), cross section area (A) and hydraulic radius (R). For this type of CRS a graph is not available.
- Processed closed: The CRS geometry is described directly through their hydraulic parameters; Length (L), Width (W), cross section area (A) and hydraulic radius (R). For this type of CRS a graph is not available.
- X-Z-R-M open: The CRS geometry is described by points defined by co-ordinate pairs (x, z), where 'x' is a horizontal axis, 'z' a vertical axis, the relative resistance (R) and the marker (M). The points are specified in a counter-clockwise direction.

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

A series of lateral resistance factors can hence be specified for the cross section. The markers (see Figure 3.34) are used for splitting the cross section into parallel channels defined by the markers. The marker (1) and the marker (3) define where the cross section is ‘cut off’, i.e. outside of these markers the cross section will not be taken into account. If marker (1) and (3) are not specified the entire cross section will be used. Marker (2) is not used for MOUSE simulations. Marker (8) and (9) indicate the left and right channel banks respectively. The markers are used for splitting the cross section into three parallel channels. Thus the reach of the left flood plain is given by markers (1) and (8), the reach of the main channel is defined by markers (8) and (9) and finally the right flood plain is defined through markers (9) and (3). The calculation of the hydraulic parameters is carried out for each of the three channels.

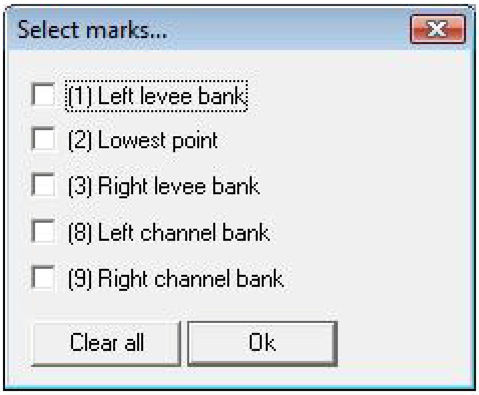
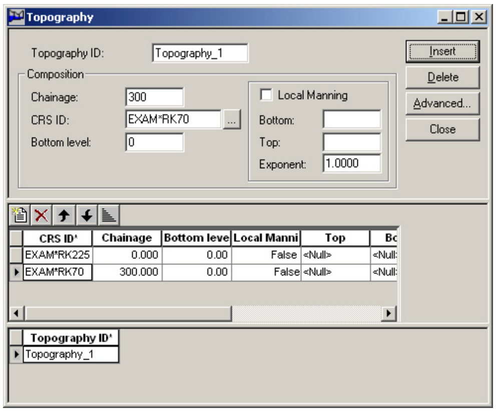

Select marks...
☐ (1) Left levee bank
☐ (2) Lowest point
☐ (3) Right levee bank
☐ (8) Left channel bank
☐ (9) Right channel bank
Clear all
Ok

Figure 3.34 The marker options for the cross section type X-Z-R-M.

The X, Z types are appropriate for irregular cross sections, while H, W are best for symmetric cross sections.

For X,Z and H,W closed cross section types, MOUSE automatically provides the Preissmann slot, in order to facilitate the flow computations in pressurised conditions.


Topography
Composition
Insert
Delete
Advanced...
Close

Topography ID: Topography_1

Chainage: 300

CRS ID: EXAM*RK70

Bottom level: 0

☐ Local Manning

Bottom:

Top:

Exponent: 1.0000

CRS ID'	Chainage	Bottom leve	Local Manni	Top	Bc
EXAM*RK225	0.000	0.00	False	<Null>	<Null:
EXAM*RK70	300.000	0.00	False	<Null>	<Null:

Topography ID'
Topography_1

Figure 3.35 The Topography Editor

A topography is made up from a number of cross sections (specified in the cross section editor), each combined with the corresponding chainage, bottom level and three optional parameters: The Manning number at the top of the cross section, the Manning number at the bottom of the cross section and the Manning's number variation exponent. These three parameters allow for a non-linear variation of the Manning number as a function of the water depth in the cross section. If the fields for specification of the parameters are left empty MOUSE will use the Manning's numbers specified for friction loss under 'MOUSE Materials'.

The 'Chainage' is the location for which the CRS is valid and 'Bottom Level' is the elevation of the CRS at the specific chainage. MOUSE will interpolate between each of the inserted CRS according to chainage. A CRS must always be specified at chainage = 0.0. If the last CRS is specified at a chainage lower than the length of the pipe the last CRS will be extrapolated to the remainder of the link.

3.11 Emptying Storage Nodes

Storage nodes are dimensionless elements used for a controlled routing of the flows outside the MOUSE network. They are typically used for simulating the partial return of surcharged water into the network in case of urban flooding.

Storage nodes are defined only by their name (ID-string), a receiving node, a control node and Emptying function (QH-relation).

Storage nodes are not connected to the MOUSE network by links, which is the case with all other types of nodes.

Water arrives into a storage node over a weir or a pump. This process is controlled by the actual hydraulic situation in the system and the weirs or pumps capacities. Weirs and pumps behave in this case as if they discharge "out of the system". The volume of water that can be stored in a storage node is unlimited.

Return of water from storage basins back into the MOUSE network is controlled by the emptying functions specified in the dialog "Emptying Storage Nodes". An emptying function is actually a Q-H function, where 'Q' is the flow from the storage node into a 'receiving node', and 'H' is the water level in a 'control node'. The flow according to the emptying function is possible only until the storage node is completely empty.

Table 3.16

Edit field	Description	Used or required by simulations	Field name in data structure
Node ID	Identifier of a storage node	Yes	NodeID
Receiving node	Receiving node for the flow from a storage node	Yes	RecNodeID
Control node	Identifier of a node for the flows from the storage node	Yes	ContrNodeID
Emptying function	Reference to tabulated emptying function	Yes	EmptFunctID

4 Rainfall-Runoff Modelling with MOUSE

4.1 Terms and Concepts

MOUSE provides a versatile set of tools and computational models aimed at modelling surface storm runoff and infiltration on urban and semi-rural catchments. As a result, user can quickly prepare a precipitation-runoff model set-up of desired level of detail (in terms of spatial discretization and in terms of input data) and utilize the computed runoff as a load of the collection network.

This Chapter provides a comprehensive guideline for the preparation of the MOUSE hydrological models and execution of the hydrological computations.

Modelling of urban storm runoff and infiltration in MOUSE requires understanding of the information requirements and the involved processes. This understanding is supported by the illustration in Figure 1-1.

The user must be aware that runoff computation and its subsequent use as a network load are, in principle two distinct steps in the modeling process.

Accordingly, the runoff modeling (Step 1) engages the following:

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

Upon execution of the runoff computation, the runoff is typically used as a hydraulic load of the collection network (Step 2). This requires further steps:

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

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

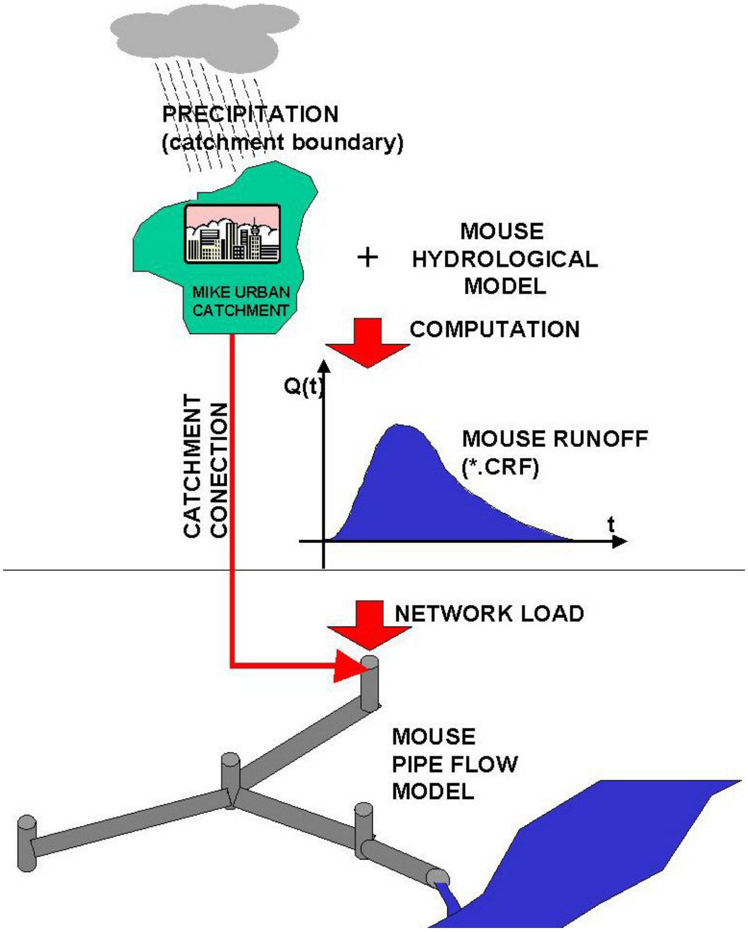

PRECIPITATION
(catchment boundary)
MIKE URBAN
CATCHMENT
+
MOUSE
HYDROLOGICAL
MODEL
COMPUTATION
Q(t)
MOUSE RUNOFF
(*.CRF)
t
CATCHMENT
CONECTION
NETWORK LOAD
MOUSE
PIPE FLOW
MODEL

Figure 4.1 Illustrated flow of information in hydrological modeling

4.1.1 MIKE URBAN Catchments

In the context of hydrological modeling, MIKE URBAN catchments are treated as hydrological units where storm runoff and infiltration (RDI) are generated on the basis of a single set of model parameters and input data. In fact, MIKE URBAN catchments represent the level of spatial discretization of the hydrological model.

The nature of MIKE URBAN catchments as geographical features, their creation, editing and management has been described in full detail in a separate chapter "MIKE URBAN Catchments". Here, only the aspects of relevance for the hydrological modeling will be repeated and further elaborated.

Catchments are essential for any hydrological model. In MIKE URBAN, the geographical extent of a catchment is determined by the catchment polygon perimeter. MIKE URBAN provides upon user request the total catchment area.

In some cases, geographical boundaries of a catchment do not coincide with the actual drainage area. E.g. a catchment extent may be defined on a basis of some administrative division, while the drainage network is present only in some parts of the catchment. In such cases, user may specify "Drainage area", which would simply overwrite the actual catchment area in all hydrological computations.

4.1.2 Connecting Catchments to the Network

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

Details about the principles and methods for catchment connections can be found in the chapter 13.4.

The information about the catchments connections is written in the MOUSE runoff result file (*.CRF), which can be utilized by the MOUSE pipe flow model.

4.1.3 Hydrological Models

Hydrological models for urban catchments include two distinct classes of models:

- Surface runoff model: These are the most common type in urban runoff analysis. The common characteristic of all the models in this class is that only surface runoff is computed. This implies discontinuous runoff hydrographs where flow starts as a result of rainfall and ceased back to zero again after the end of rainfall. As such, these models are suitable for relatively densely urbanized catchments with dominant amount of runoff generated on impervious surfaces, and for single-event analyses (e.g. design rainfall of certain recurrence interval). These models fail to provide realistic results in dominantly rural catchments and for long-term analyses involving multi-event rainfall series.

- Continuous hydrological models: These models treat the precipitation volume balance without any truncation, through more or less complex concepts. As a result, the generated runoff includes both the overland and sub-surface runoff components. Due to longer time scales involved, the runoff hydrographs appear practically continuous. An important property of continuous hydrological models is "hydrological memory", i.e. the ability to simulate the catchment reaction to certain rainfall dependent to previous rainfalls. This type of model is essential for any long-term analysis and for dominantly rural catchments. On the other hand, these models are usually incapable of simulating extremely fast response of heavily impermeable urban catchments.

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

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

The continuous hydrological model included is MOUSE RDI.

Any of these models can be used on its own, or in a combination of one surface runoff and MOUSE RDI.

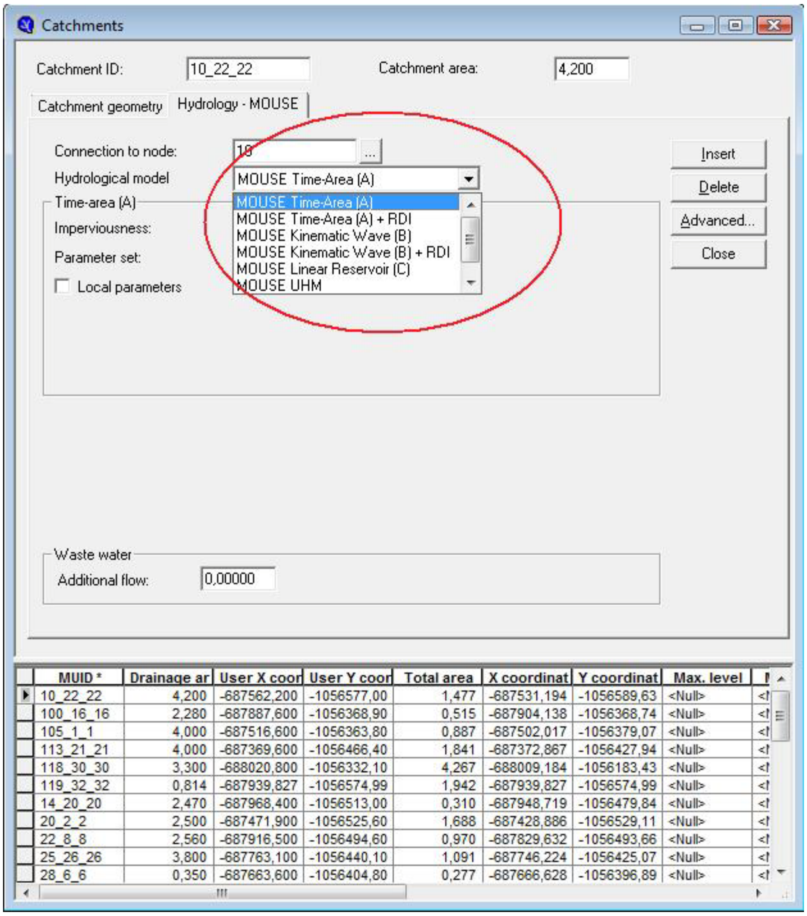
However, combining different models for individual catchments in one runoff computation is NOT possible.

Detailed description of the models is available in following paragraphs and in the technical reference documentation.

4.1.4 Creating Hydrological Models for a Catchment

Database records containing MIKE URBAN catchments do NOT include the information related to hydrological modeling. Such information is stored in separate tables - one table for each model. MIKE URBAN maintains the hydrological model tables automatically, i.e. new records are added and obsolete records deleted along with the catchment records.

Each time a catchment is inserted a record is also inserted under the hydrology tabs. Then the user can add hydrology data for each of the runoff methods available. See e.g. Figure 4.2.


Catchments
Catchment geometry
Hydrology - MOUSE
Time-area (A)
Imperviousness:
Parameter set:
☐ Local parameters
MOUSE Time-Area (A)
MOUSE Time-Area (A) + RDI
MOUSE Kinematic Wave (B)
MOUSE Kinematic Wave (B) + RDI
MOUSE Linear Reservoir (C)
MOUSE UHM
Insert
Delete
Advanced...
Close
Waste water

Catchment ID: 10_22_22

Catchment area: 4,200

Connection to node: 10

Hydrological model: MOUSE Time-Area (A)

Additional flow: 0,00000

MUID *	Drainage ar	User X coor	User Y coor	Total area	X coordinat	Y coordinat	Max. level
10_22_22	4,200	-687562,200	-1056577,00	1,477	-687531,194	-1056589,63	<Null>
100_16_16	2,280	-687887,600	-1056368,90	0,515	-687904,138	-1056368,74	<Null>
105_1_1	4,000	-687516,600	-1056363,80	0,887	-687502,017	-1056379,07	<Null>
113_21_21	4,000	-687369,600	-1056466,40	1,841	-687372,867	-1056427,94	<Null>
118_30_30	3,300	-688020,800	-1056332,10	4,267	-688009,184	-1056183,43	<Null>
119_32_32	0,814	-687939,827	-1056574,99	1,942	-687939,827	-1056574,99	<Null>
14_20_20	2,470	-687968,400	-1056513,00	0,310	-687948,719	-1056479,84	<Null>
20_2_2	2,500	-687471,900	-1056525,60	1,688	-687428,886	-1056529,11	<Null>
22_8_8	2,560	-687916,500	-1056494,60	0,970	-687829,632	-1056493,66	<Null>
25_26_26	3,800	-687763,100	-1056440,10	1,091	-687746,224	-1056425,07	<Null>
28_6_6	0,350	-687663,600	-1056404,80	0,277	-687666,628	-1056396,89	<Null>

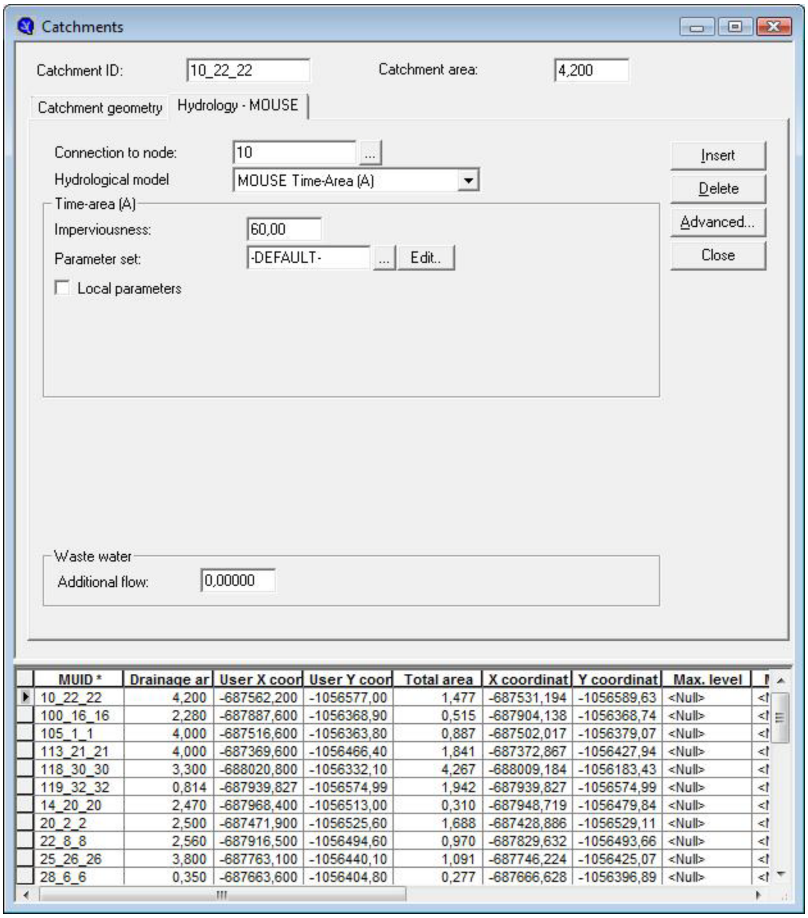
Figure 4.2 Choosing which runoff model records to insert

4.2 Time-Area Method (A)

The Time-Area Method (MOUSE model A) is a simple surface runoff model with minimum data requirements. The runoff computation is based on a simple treatment of hydrological losses and the runoff routing by the so called time-area curve. Details on technical background can be found in the technical reference documentation.

4.2.1 Model Data

The model data are stored in the database table msm_HmodA.



Catchment ID: 10_22_22

Catchment area: 4,200

Connection to node: 10

Hydrological model: MOUSE Time-Area (A)

Time-area (A)

Imperviousness: 60,00

Parameter set: -DEFAULT-

☐ Local parameters

Waste water

Additional flow: 0,00000

MUID *	Drainage ar	User X coor	User Y coor	Total area	X coordinat	Y coordinat	Max. level
10_22_22	4,200	-687562,200	-1056577,00	1,477	-687531,194	-1056589,63	<Null>
100_16_16	2,280	-687887,600	-1056368,90	0,515	-687904,138	-1056368,74	<Null>
105_1_1	4,000	-687516,600	-1056363,80	0,887	-687502,017	-1056379,07	<Null>
113_21_21	4,000	-687369,600	-1056466,40	1,841	-687372,867	-1056427,94	<Null>
118_30_30	3,300	-688020,800	-1056332,10	4,267	-688009,184	-1056183,43	<Null>
119_32_32	0,814	-687939,827	-1056574,99	1,942	-687939,827	-1056574,99	<Null>
14_20_20	2,470	-687968,400	-1056513,00	0,310	-687948,719	-1056479,84	<Null>
20_2_2	2,500	-687471,900	-1056525,60	1,688	-687428,886	-1056529,11	<Null>
22_8_8	2,560	-687916,500	-1056494,60	0,970	-687829,632	-1056493,66	<Null>
25_26_26	3,800	-687763,100	-1056440,10	1,091	-687746,224	-1056425,07	<Null>
28_6_6	0,350	-687663,600	-1056404,80	0,277	-687666,628	-1056396,89	<Null>

Figure 4.3 The Time-Area Model Editor (MOUSE|Runoff Models|Time-Area (A))

A full overview of the editor fields and corresponding database attributes is provided in the Table 4.1.

Table 4.1 Overview of the Time-Area Database Attributes (Table ‘msm_HModA’)

Edit field	Description	Used or required by simulations	Field name in datastructure
CatchmentID	Unique catchment identifier	Yes	CatchmentID
Catchment Area	Displays the actual model area (total area or drainage area)	Yes (calculated, read-only)	-

Table 4.1 Overview of the Time-Area Database Attributes (Table ‘msm_HModA’)

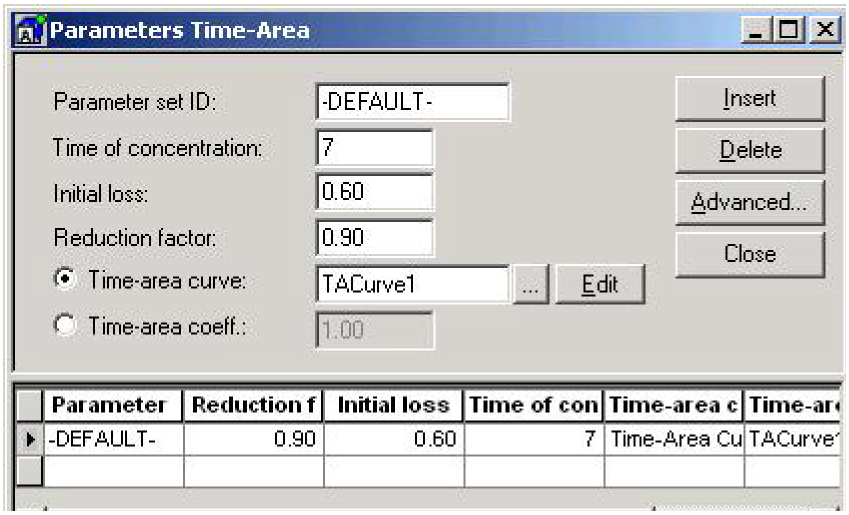
Edit field	Description	Used or required by simulations	Field name in datastructure
Impervious Area	Impervious catchment area, as percent of the actual model area	Yes	ImpArea
Parameter Set	Reference to a set of model parameters to be used for the current catchment	Yes	ParAID
Use Local Parameters	Allows for use of individual parameters locally	Optional	LocalNo
Reduction Factor	Local value of the hydrological reduction factor	Optional	Rfactor
Time of Concentration	Local value of the concentration time	Optional	ConcTime
Initial Loss	Local value of the initial loss	Optional	Iloss
Time-Area Curve/Coefficient radio buttons	Local switch for use of predefined tabulated T-A curves or analytically computed T-A relation	Optional	CoeffNo
Time Area Curve	Locally selected predefined T-A curve	Optional	ACurveID
Time-Area Coefficient	Local value of the analytical T-A curve coefficient	Optional	TACoeffID

4.2.2 Parameter Sets

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

New parameter sets can be inserted and values of individual parameters can be edited in the "Parameters Time-Area" editor. This can be activated at MOUSE|Runoff Models|Parameters Time-Area.

MIKE URBAN comes with a default parameter set (-DEFAULT-). User can insert any number of parameter sets and edit them according to the needs.



Parameter set ID: -DEFAULT-

Time of concentration: 7

Initial loss: 0.60

Reduction factor: 0.90

● Time-area curve: TACurve1

○ Time-area coeff.: 1.00

Parameter	Reduction f	Initial loss	Time of con	Time-area c	Time-ar
-DEFAULT-	0.90	0.60	7	Time-Area Cu	TACurve1

Figure 4.4 The Time-Area Parameter Sets Editor (MOUSE|Runoff Models|Parameters Time-Area

A full overview of the editor fields and corresponding database attributes is provided in the Table 4.2.

Table 4.2 Overview of the Time-Area Parameter Set Database Attributes (Table ‘msm_HParA’)

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

Table 4.2 Overview of the Time-Area Parameter Set Database Attributes (Table 'msm_HParA')

Edit field	Description	Used or required by simulations	Field name in datastructure
Time Area Curve	Predefined T-A curve ID	Optional, alternates with TACoeff	TACurveID
Time-Area Coefficient	Value of the analytical T-A curve coefficient	Optional, alternates with TACurveID	TACoeff

4.2.3 Time-Area Curve Editor

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

MIKE URBAN comes with three defaults time-area curves TACurve1, TACurve2 and TACurve3, applicable for rectangular, divergent and convergent catchment, respectively (see the figures below).

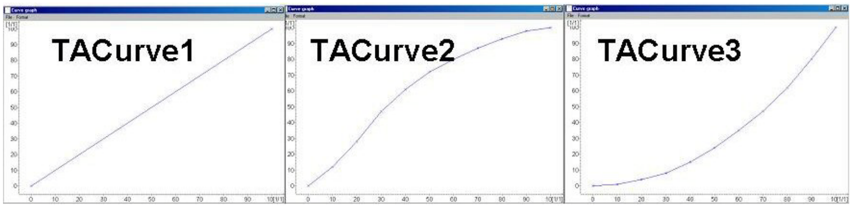
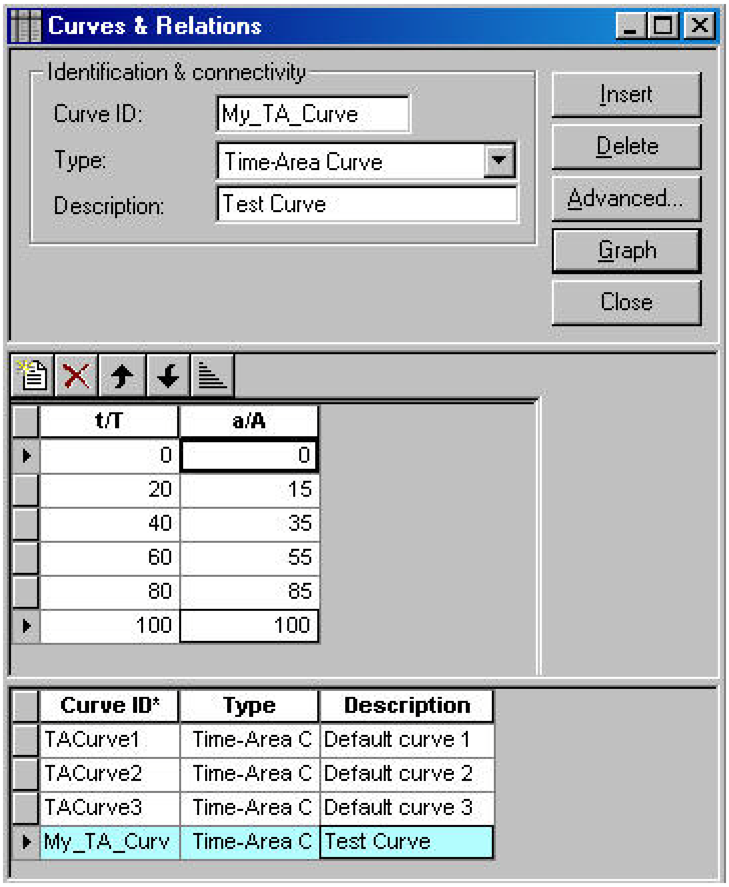


Figure 4.5 Three default time-area curves

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



Identification & connectivity

Curve ID: My_TA_Curve

Type: Time-Area Curve

Description: Test Curve

t/T	a/A
0	0
20	15
40	35
60	55
80	85
100	100

Curve ID*	Type	Description
TACurve1	Time-Area C	Default curve 1
TACurve2	Time-Area C	Default curve 2
TACurve3	Time-Area C	Default curve 3
My_TA_Curv	Time-Area C	Test Curve

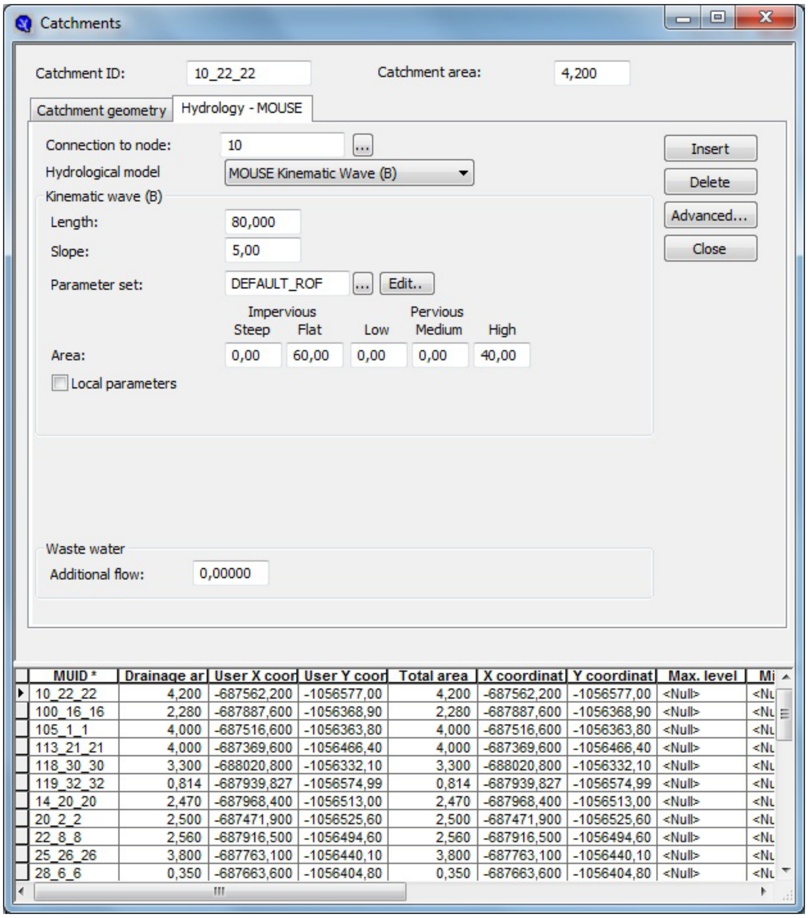
Figure 4.6 The Curves&Relations editor with one correct user defined time-area curve

4.3 Kinematic Wave (B)

The Kinematic Wave (MOUSE model B) is a surface runoff model with moderate data requirements. The runoff computation is based on a comprehensive treatment of hydrological losses (including infiltration) and the runoff routing by the kinematic wave (Manning) formula. Details on technical background can be found in the technical reference documentation.

4.3.1 Model Data

The model data are stored in the database table msm_HmodB.



Catchment ID: 10_22_22
Catchment area: 4,200

Connection to node: 10
Hydrological model: MOUSE Kinematic Wave (B)

Kinematic wave (B)
Length: 80,000
Slope: 5,00
Parameter set: DEFAULT_ROF

	Impervious Steep	Impervious Flat	Pervious Low	Pervious Medium	Pervious High
Area:	0,00	60,00	0,00	0,00	40,00

☐ Local parameters

Waste water
Additional flow: 0,00000

MUID *	Drainage ar	User X coor	User Y coor	Total area	X coordinat	Y coordinat	Max. level	Mi
10_22_22	4,200	-687562,200	-1056577,00	4,200	-687562,200	-1056577,00	<Null>	<Nu
100_16_16	2,280	-687887,600	-1056368,90	2,280	-687887,600	-1056368,90	<Null>	<Nu
105_1_1	4,000	-687516,600	-1056363,80	4,000	-687516,600	-1056363,80	<Null>	<Nu
113_21_21	4,000	-687369,600	-1056466,40	4,000	-687369,600	-1056466,40	<Null>	<Nu
118_30_30	3,300	-688020,800	-1056332,10	3,300	-688020,800	-1056332,10	<Null>	<Nu
119_32_32	0,814	-687939,827	-1056574,99	0,814	-687939,827	-1056574,99	<Null>	<Nu
14_20_20	2,470	-687968,400	-1056513,00	2,470	-687968,400	-1056513,00	<Null>	<Nu
20_2_2	2,500	-687471,900	-1056525,60	2,500	-687471,900	-1056525,60	<Null>	<Nu
22_8_8	2,560	-687916,500	-1056494,60	2,560	-687916,500	-1056494,60	<Null>	<Nu
25_26_26	3,800	-687763,100	-1056440,10	3,800	-687763,100	-1056440,10	<Null>	<Nu
28_6_6	0,350	-687663,600	-1056404,80	0,350	-687663,600	-1056404,80	<Null>	<Nu

Figure 4.7 The Kinematic Wave Model Editor (MOUSE|Runoff Models|Kinematic Wave (B)

A full overview of the editor fields and corresponding database attributes is provided in the Table 4.3.

Table 4.3 Overview of the Kinematic Wave Database Attributes (Table ‘msm_HModB)

Edit field	Description	Used or required by simulations	Field name in datastructure
CatchmentID	Unique catchment identifier	Yes	CatchmentID
Catchment Area	Displays the actual model area (total area or drainage area)	Yes, (calculated, read-only)	-

Table 4.3 Overview of the Kinematic Wave Database Attributes (Table ‘msm_HModB)

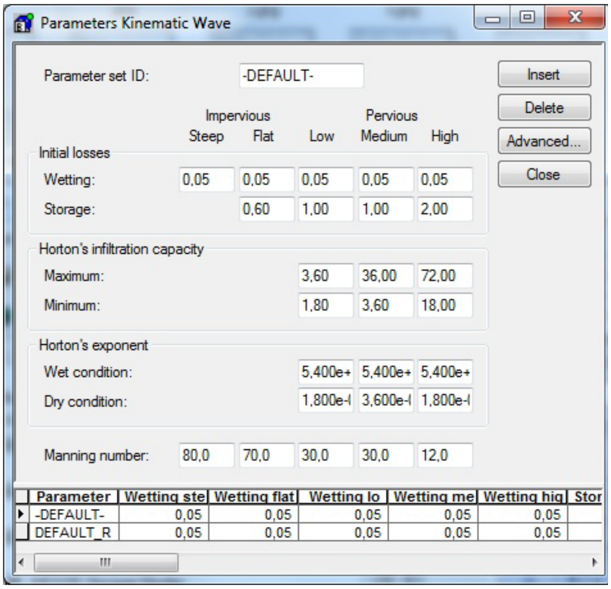
Edit field	Description	Used or required by simulations	Field name in datastructure
Length	Characteristic length of the catchment	Yes	Length
Slope	Representative slope of the catchment	Yes	Slope
Area (five fields)	Fraction of the actual model area for each surface category	Yes	AISteep, AIFlat, APSmall, APMedium, APLarge
Parameter Set	Reference to a set of model parameters to be used for the current catchment	Yes	ParBID
Use Local Parameters	Allows for use of individual parameters locally	Optional	LocalNo
Manning number (five fields)	Local value of the Manning number for various types of catchment surfaces	Optional	MISteep, MIFlat, MPSmall, MPMedium,MPLarge

4.3.2 Parameter Sets

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

New parameter sets can be inserted and values of individual parameters can be edited in the "Parameters Kinematic Wave" editor. This can be activated at MOUSE|Runoff Models|Parameters Kinematic Wave.

MIKE URBAN comes with a default parameter set (-DEFAULT-). User can insert any number of parameter sets and edit them according to the needs.



Parameter set ID: -DEFAULT-

	Impervious		Pervious		
	Steep	Flat	Low	Medium	High
Initial losses					
Wetting:	0,05	0,05	0,05	0,05	0,05
Storage:		0,60	1,00	1,00	2,00
Horton's infiltration capacity					
Maximum:			3,60	36,00	72,00
Minimum:			1,80	3,60	18,00
Horton's exponent					
Wet condition:			5,400e+	5,400e+	5,400e+
Dry condition:			1,800e-(	3,600e-(	1,800e-(
Manning number:	80,0	70,0	30,0	30,0	12,0

Parameter	Wetting ste	Wetting flat	Wetting lo	Wetting me	Wetting hig	Stor
-DEFAULT-	0,05	0,05	0,05	0,05	0,05	
DEFAULT_R	0,05	0,05	0,05	0,05	0,05	

Figure 4.8 The Kinematic Wave Parameter Sets Editor (MOUSE|Runoff Models|Parameters Kinematic Wave)

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

Table 4.4 Overview of the Kinematic Wave Parameter Set Database Attributes (Table 'msm_HParB'

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

Table 4.4 Overview of the Kinematic Wave Parameter Set Database Attributes (Table 'msm_HParB'

Edit field	Description	Used or required by simulations	Field name in datastructure
Minimum Infiltration Capacity (three fields)	Minimum Infiltration capacity on various types of surfaces	Yes	InfMinSmall, InfMinMedium, InfMinLarge
Horton Exponent Wet Conditions (three fields)	Horton Exponent for wet conditions on various types of surfaces	Yes	InfExpWetSmall, InfExpWetMedium, InfExpWetLarge
Horton Exponent Dry Conditions (three fields)	Horton Exponent for dry conditions on various types of surfaces	Yes	InfExpDrySmall, InfExpDryMedium, InfExpDryLarge
Manning number (five fields)	Manning number for various types of catchment surfaces	Yes	ManningSteep, ManningFlat, ManningSmall, ManningMedium, ManningLarge

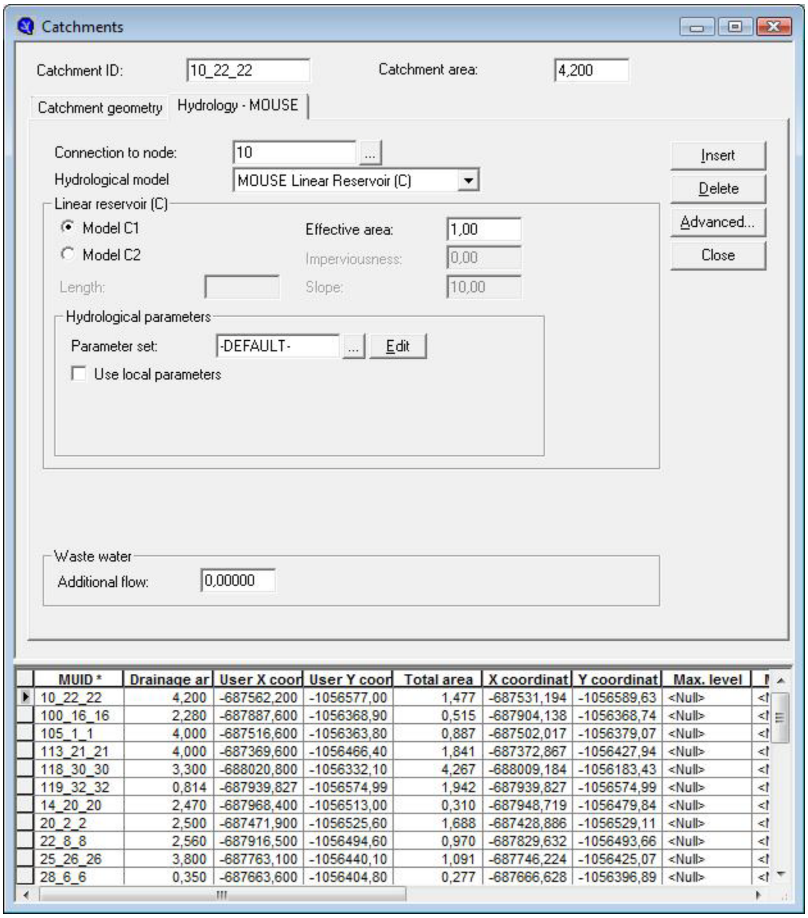
4.4 Linear Reservoir (C1 and C2)

The Linear Reservoir (MOUSE model C) is a surface runoff model with minimum data requirements. The runoff computation is based on a comprehensive treatment of hydrological losses (including infiltration) and the runoff routing by the linear reservoir principle. Details on technical background can be found in the technical reference documentation.

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

4.4.1 Model Data

The model data are stored in the database table msm_HmodC.



Catchment ID: 10_22_22

Catchment area: 4,200

Connection to node: 10

Hydrological model: MOUSE Linear Reservoir (C)

Linear reservoir (C)

● Model C1

○ Model C2

Effective area: 1,00

Imperviousness: 0,00

Length:

Slope: 10,00

Hydrological parameters

Parameter set: -DEFAULT-

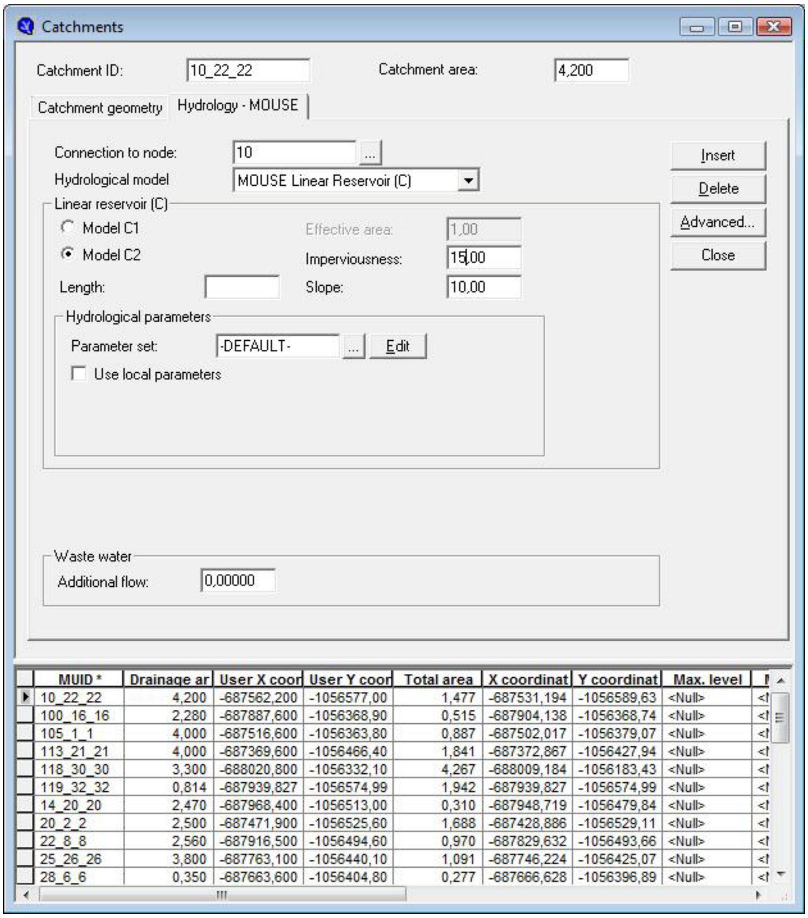
☐ Use local parameters

Waste water

Additional flow: 0,00000

MUID *	Drainage ar	User X coor	User Y coor	Total area	X coordinat	Y coordinat	Max. level
10_22_22	4,200	-687562,200	-1056577,00	1,477	-687531,194	-1056589,63	<Null>
100_16_16	2,280	-687887,600	-1056368,90	0,515	-687904,138	-1056368,74	<Null>
105_1_1	4,000	-687516,600	-1056363,80	0,887	-687502,017	-1056379,07	<Null>
113_21_21	4,000	-687369,600	-1056466,40	1,841	-687372,867	-1056427,94	<Null>
118_30_30	3,300	-688020,800	-1056332,10	4,267	-688009,184	-1056183,43	<Null>
119_32_32	0,814	-687939,827	-1056574,99	1,942	-687939,827	-1056574,99	<Null>
14_20_20	2,470	-687968,400	-1056513,00	0,310	-687948,719	-1056479,84	<Null>
20_2_2	2,500	-687471,900	-1056525,60	1,688	-687428,886	-1056529,11	<Null>
22_8_8	2,560	-687916,500	-1056494,60	0,970	-687829,632	-1056493,66	<Null>
25_26_26	3,800	-687763,100	-1056440,10	1,091	-687746,224	-1056425,07	<Null>
28_6_6	0,350	-687663,600	-1056404,80	0,277	-687666,628	-1056396,89	<Null>

Figure 4.9 The Linear Reservoir Model Editor (MOUSE|Runoff Models|Linear Reservoir (C)), set up for the work with model C1 (Dutch)



Catchment ID: 10_22_22
Catchment area: 4,200

Connection to node: 10
Hydrological model: MOUSE Linear Reservoir (C)

○ Model C1
● Model C2
Length:
Effective area: 1,00
Imperviousness: 15,00
Slope: 10,00

Parameter set: -DEFAULT-
☐ Use local parameters

Additional flow: 0,00000

	MUID *	Drainage ar	User X coor	User Y coor	Total area	X coordinat	Y coordinat	Max. level
▶	10_22_22	4,200	-687562,200	-1056577,00	1,477	-687531,194	-1056589,63	<Null>
	100_16_16	2,280	-687887,600	-1056368,90	0,515	-687904,138	-1056368,74	<Null>
	105_1_1	4,000	-687516,600	-1056363,80	0,887	-687502,017	-1056379,07	<Null>
	113_21_21	4,000	-687369,600	-1056466,40	1,841	-687372,867	-1056427,94	<Null>
	118_30_30	3,300	-688020,800	-1056332,10	4,267	-688009,184	-1056183,43	<Null>
	119_32_32	0,814	-687939,827	-1056574,99	1,942	-687939,827	-1056574,99	<Null>
	14_20_20	2,470	-687968,400	-1056513,00	0,310	-687948,719	-1056479,84	<Null>
	20_2_2	2,500	-687471,900	-1056525,60	1,688	-687428,886	-1056529,11	<Null>
	22_8_8	2,560	-687916,500	-1056494,60	0,970	-687829,632	-1056493,66	<Null>
	25_26_26	3,800	-687763,100	-1056440,10	1,091	-687746,224	-1056425,07	<Null>
	28_6_6	0,350	-687663,600	-1056404,80	0,277	-687666,628	-1056396,89	<Null>

Figure 4.10 The Linear Reservoir Model Editor (MOUSE|Runoff Models|Linear Reservoir (C)), set up for the work with model C2 (French)

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

Table 4.5 Overview of the Linear Reservoir Database Attributes (Table 'msm_HModC')

Edit field	Description	Used or required by simulations	Field name in datastructure
CatchmentID	Unique catchment identifier	Yes	CatchmentID
Catchment Area	Displays the actual model area (total area or drainage area)	Yes, (calculated, read-only)	-

Table 4.5 Overview of the Linear Reservoir Database Attributes (Table 'msm_HModC')

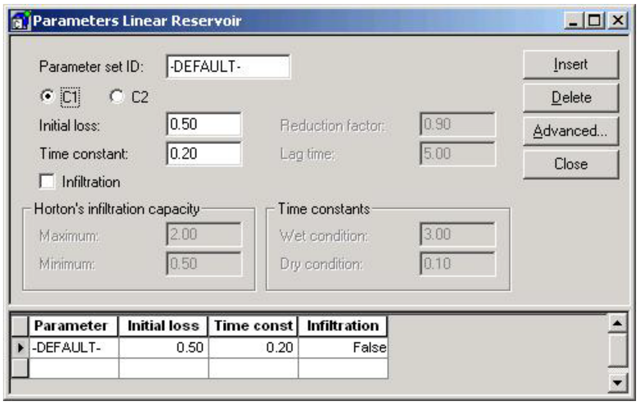
Edit field	Description	Used or required by simulations	Field name in datastructure
C1/C2	Toggle for editor setting for model C1 and C2, respectively	Yes	-
Length	Characteristic length of the catchment	Optional, model C2 only	Length
Slope	Representative slope of the catchment	Optional, model C2 only	Slope
Impervious Area	Impervious area as a fraction of the actual model area	Optional, model C2 only	C2Area
Contributing Area	Contributing area as a fraction of the actual model area	Optional, model C1 only	C1Area
Parameter Set	Reference to a set of model parameters to be used for the current catchment	Yes	ParCID
Use Local Parameters	Allows for use of individual parameters locally	Optional	LocalNo
Reduction Factor	Local value of the hydrological reduction factor	Optional, model C2 only	RedFactor
Lag Time	Local value of the lag time	Optional, model C2 only	LagTime
Time Constant	Local value of the time constant	Optional, model C1 only	TimeConst
Initial Loss	Local value of the initial loss	Optional, model C2 only	Iloss

4.4.2 Parameter Sets

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

New parameter sets can be inserted and values of individual parameters can be edited in the "Parameters Linear Reservoir" editor. This can be activated at MOUSE|Runoff Models|Parameters Linear Reservoir.

MIKE URBAN comes with a default parameter set (-DEFAULT-). User can insert any number of parameter sets and edit them according to the needs.



Parameter set ID: -DEFAULT-

● C1 ○ C2

Initial loss: 0.50

Time constant: 0.20

☐ Infiltration

Reduction factor: 0.90

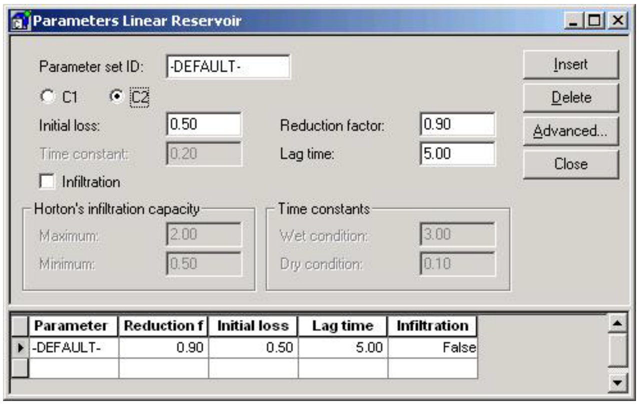
Lag time: 5.00

Horton's infiltration capacity
Maximum: 2.00
Minimum: 0.50

Time constants
Wet condition: 3.00
Dry condition: 0.10

Parameter	Initial loss	Time const	Infiltration
-DEFAULT-	0.50	0.20	False

Figure 4.11 The Linear Reservoir Parameter Sets Editor (MOUSE|Runoff Models|Parameters Linear Reservoir) - Dutch version



Parameter set ID: -DEFAULT-

○ C1 ● C2

Initial loss: 0.50

Time constant: 0.20

☐ Infiltration

Reduction factor: 0.90

Lag time: 5.00

Horton's infiltration capacity
Maximum: 2.00
Minimum: 0.50

Time constants
Wet condition: 3.00
Dry condition: 0.10

Parameter	Reduction f	Initial loss	Lag time	Infiltration
-DEFAULT-	0.90	0.50	5.00	False

Figure 4.12 The Linear Reservoir Parameter Sets Editor (MOUSE|Runoff Models|Parameters Linear Reservoir) - French version

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

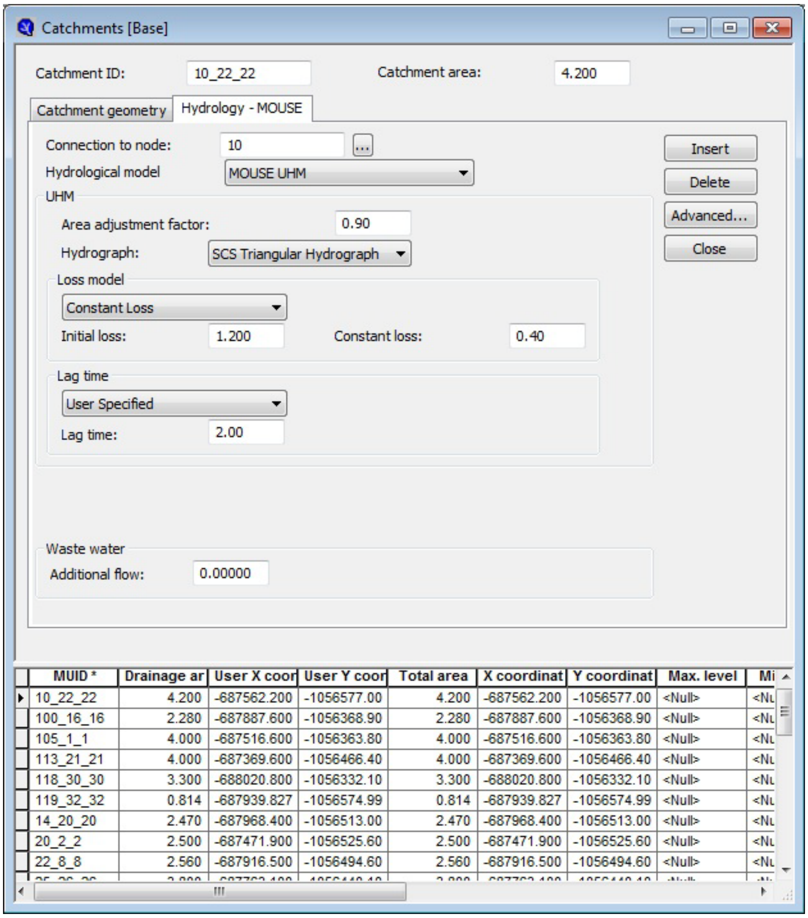
Table 4.6 Overview of the Linear Reservoir Parameter Set Database Attributes (Table ‘msm_HParC’)

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

4.5 Unit Hydrograph Method (UHM)

The Unit Hydrograph Method (MOUSE UHM) is a simple surface linear runoff model which can be used to derive hydrographs for any amount of excess precipitation. The runoff computation includes a comprehensive treatment of hydrological losses (i.e. calculation of excessive precipitation) and the runoff routing through creation of a composite hydrograph. Details on technical background can be found in the technical reference documentation.

The model data are stored in the database table msm_HModUHM.



Catchment ID: 10_22_22
Catchment area: 4.200

Connection to node: 10
Hydrological model: MOUSE UHM

UHM
Area adjustment factor: 0.90
Hydrograph: SCS Triangular Hydrograph

Loss model: Constant Loss
Initial loss: 1.200
Constant loss: 0.40

Lag time: User Specified
Lag time: 2.00

Waste water
Additional flow: 0.00000

MUID *	Drainage ar	User X coor	User Y coor	Total area	X coordinat	Y coordinat	Max. level	Mi
10_22_22	4.200	-687562.200	-1056577.00	4.200	-687562.200	-1056577.00	<Null>	<Nu
100_16_16	2.280	-687887.600	-1056368.90	2.280	-687887.600	-1056368.90	<Null>	<Nu
105_1_1	4.000	-687516.600	-1056363.80	4.000	-687516.600	-1056363.80	<Null>	<Nu
113_21_21	4.000	-687369.600	-1056466.40	4.000	-687369.600	-1056466.40	<Null>	<Nu
118_30_30	3.300	-688020.800	-1056332.10	3.300	-688020.800	-1056332.10	<Null>	<Nu
119_32_32	0.814	-687939.827	-1056574.99	0.814	-687939.827	-1056574.99	<Null>	<Nu
14_20_20	2.470	-687968.400	-1056513.00	2.470	-687968.400	-1056513.00	<Null>	<Nu
20_2_2	2.500	-687471.900	-1056525.60	2.500	-687471.900	-1056525.60	<Null>	<Nu
22_8_8	2.560	-687916.500	-1056494.60	2.560	-687916.500	-1056494.60	<Null>	<Nu

Figure 4.13 The Unit Hydroghraph Model Editor (MOUSE|Runoff Models|UHM)

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

Table 4.7 Overview of the UHM Database Attributes (Atble ‘msm_HModUHM’)

Edit field	Description	Used or required by simulations	Field name in datastructure
CatchmentID	Unique catchment identifier	Yes	CatchID
Catchment Area	Displays the actual model area (total area or drainage area)	Yes, (calculated, read-only)	-
Area Adjustment Factor	Relative size of contributing area	Yes	AreaFactor

Table 4.7 Overview of the UHM Database Attributes (Atble 'msm_HModUHM')

Edit field	Description	Used or required by simulations	Field name in datastructure
Hydrograph	Method for hydrograph computation	Yes	MethodNo
Peaking factor	Hydrograph peak factor	Optional, method SUH Standard only	Cp
Slope	Representative catchment slope	Optional, method SUH Alameda only	SuhSlope
Loss Model	Method for computing hydrological losses	Yes	LossModelNo
Initial Loss	Initial loss	Optional, "constant loss" method only	InitLoss
Constant Loss	Constant Loss	Optional, "constant loss" method only	ConstLoss
Runoff Coefficient	Runoff Coefficient	Optional, "proportional loss" method only	RunoffCoeff
Curve Number	Standard hydrograph curve number	Optional, "SCS Method" and "SCS Generalized" only	CurveNum
Initial AMC	Antecedent Moisture Condition	Optional, "SCS Method" only	AMC
Initial Abstraction Depth	Initial Abstraction Depth	Optional, "SCS generalized" only	InitAbstractDepth
Lag Time	Method for lag time	Yes	LagTimeMethodNo
Lag Time	User-specified lag time	Optional, "User specified" lag time method only	LagTime
Hydraulic Lengt	Hydraulic length of the catchment	Optional, "CSC Formula" lag time method only	HydraulicLength
LT Slope	Average catchment slope	Optional, "CSC Formula" lag time method only	Slope
LT Curve Number	CSC Curve number used for computing lag time	Optional, "CSC Formula" lag time method only	LagCurveNum

Table 4.7 Overview of the UHM Database Attributes (Atble ‘msm_HModUHM’)

Edit field	Description	Used or required by simulations	Field name in datastructure
L	Length of the main stream from outlet to the divide	Optional, "SUH Standard " lag time method only	SuhL
Lc	Length of the main stream from the outlet to the point closest to the catchment centroid	Optional, "SUH Standard" lag time method only	SuhLc
Ct	Watershed (catchment) coefficient	Optional, "SUH Standard" lag time method only	Ct
Stream Slope	Average Overland Slope	Optional, "SUH Alameda" lag time method only	StreamSlope
Basin factor	Basin Factor	Optional, "SUH Alameda" lag time method only	BasFactor

4.6 Additional Flow and RDI

Continuous runoff from MIKE URBAN catchments can be modelled at two distinct levels: either as a simple specification of a constant additional flow or as a MOUSE RDI computation.

The later option involves a continuous hydrological model MOUSE Rainfall Dependent Infiltration (RDI). RDI 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, unsaturated zone ("root-zone") and ground water. This enables continuous modeling of the runoff processes, which is particularly useful when long- term hydraulic and pollution load effects are analyzed.

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

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

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

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

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

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

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

Figure 4.14 shows an example illustrating the influence of previous hydrological conditions for the two components and their response to a rainfall.

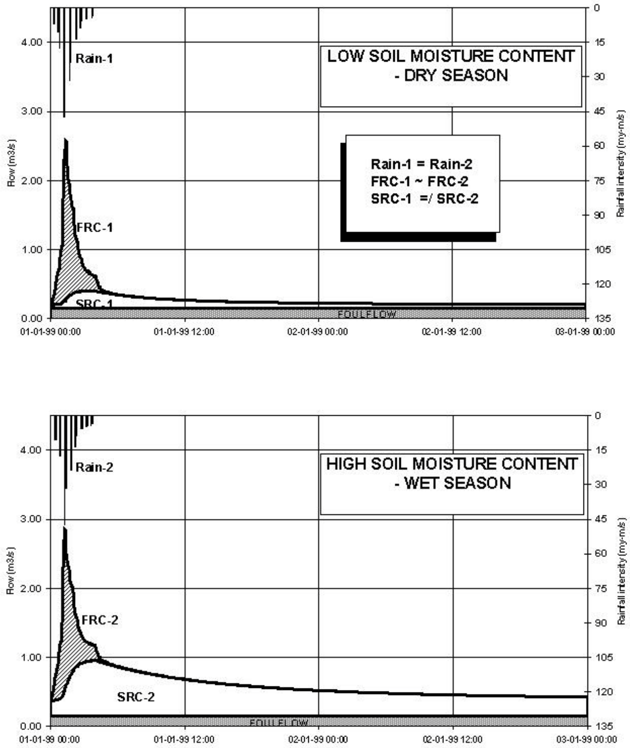


Figure 4.14 Different catchment response under the same rainfall, due to different soil moisture conditions at the beginning of the rainfall

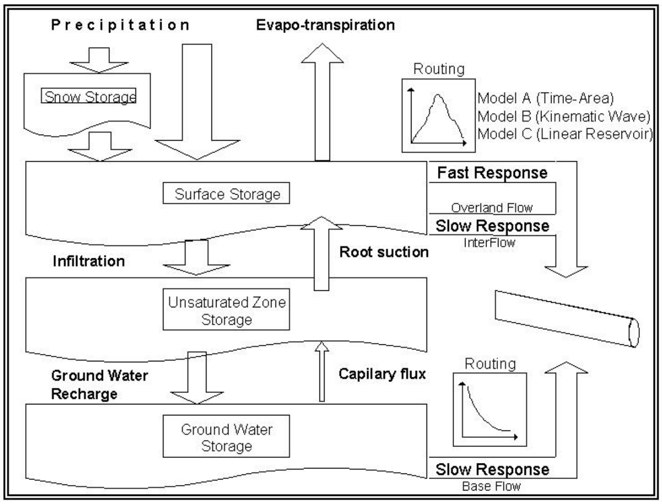
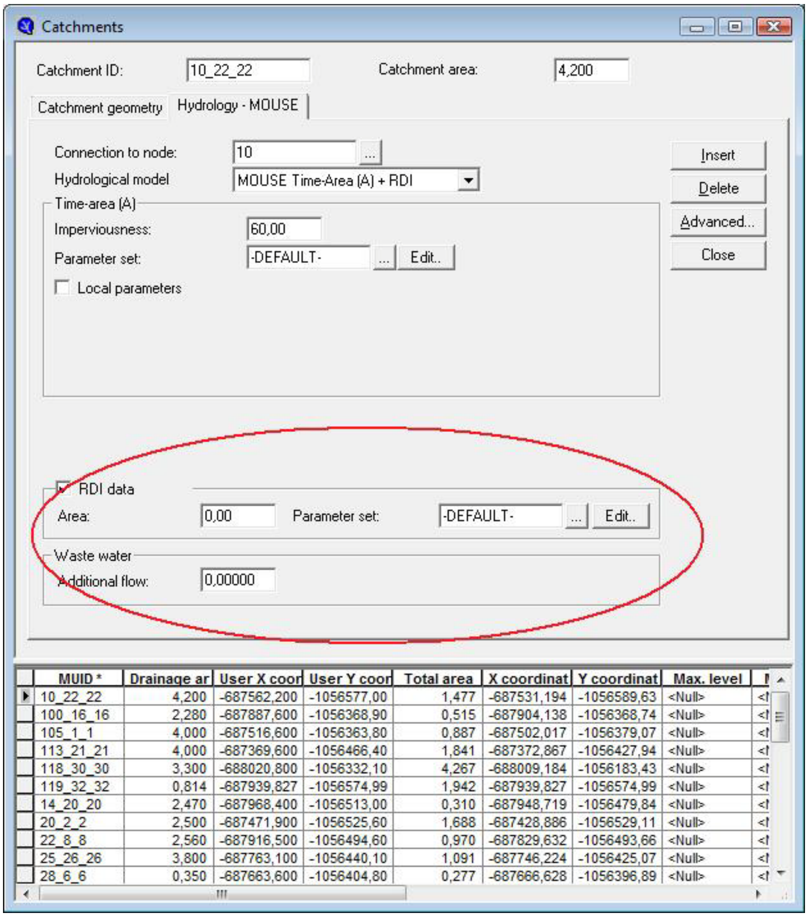

Precipitation
Evapo-transpiration
Snow Storage
Routing
Model A (Time-Area)
Model B (Kinematic Wave)
Model C (Linear Reservoir)
Surface Storage
Fast Response
Overland Flow
Slow Response
InterFlow
Infiltration
Root suction
Unsaturated Zone Storage
Ground Water Recharge
Capilary flux
Routing
Ground Water Storage
Slow Response
Base Flow

Figure 4.15 Schematics of the RDI Model

4.6.1 Model Data

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



Catchment ID: 10_22_22

Catchment area: 4,200

Connection to node: 10

Hydrological model: MOUSE Time-Area (A) + RDI

Imperviousness: 60,00

Parameter set: -DEFAULT-

☐ Local parameters

☑ RDI data

Area: 0,00

Parameter set: -DEFAULT-

Additional flow: 0,00000

	MUID *	Drainage ar	User X coor	User Y coor	Total area	X coordinat	Y coordinat	Max. level
▶	10_22_22	4,200	-687562,200	-1056577,00	1,477	-687531,194	-1056589,63	<Null>
	100_16_16	2,280	-687887,600	-1056368,90	0,515	-687904,138	-1056368,74	<Null>
	105_1_1	4,000	-687516,600	-1056363,80	0,887	-687502,017	-1056379,07	<Null>
	113_21_21	4,000	-687369,600	-1056466,40	1,841	-687372,867	-1056427,94	<Null>
	118_30_30	3,300	-688020,800	-1056332,10	4,267	-688009,184	-1056183,43	<Null>
	119_32_32	0,814	-687939,827	-1056574,99	1,942	-687939,827	-1056574,99	<Null>
	14_20_20	2,470	-687968,400	-1056513,00	0,310	-687948,719	-1056479,84	<Null>
	20_2_2	2,500	-687471,900	-1056525,60	1,688	-687428,886	-1056529,11	<Null>
	22_8_8	2,560	-687916,500	-1056494,60	0,970	-687829,632	-1056493,66	<Null>
	25_26_26	3,800	-687763,100	-1056440,10	1,091	-687746,224	-1056425,07	<Null>
	28_6_6	0,350	-687663,600	-1056404,80	0,277	-687666,628	-1056396,89	<Null>

Figure 4.16 The Additional Flow and RDI Model Editor (MOUSE|Runoff Models|Additional Flow and RDI

A full overview of the editor fields and corresponding database attributes is provided in the Table 4.8.

Table 4.8 Overview of the Additional Flow and RDI Database Attributes (Table 'msm_HModCRC')

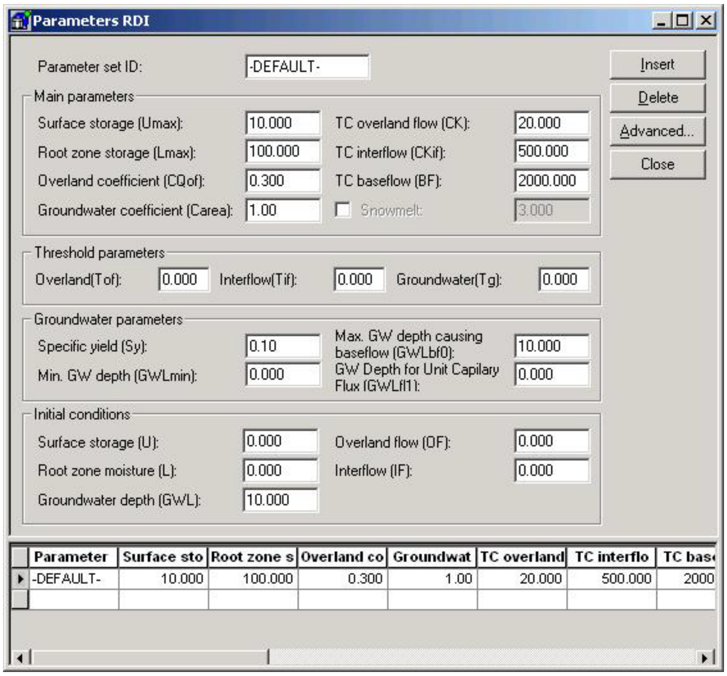
Edit field	Description	Used or required by simulations	Field name in datastructure
Catchment Area	Displays the actual model area (total area or drainage area)	Yes, (calculated, read-only)	-
Additional Flow	Amount of constant additional flow	Yes, default value = 0	AddFlow
Area checkbox	Toggle for activation of MOUSE RDI	Yes	RdiiNo
Area	RDI area as a fraction of the actual model area	Optional, if RDI activated	RdiiArea
Parameter Set	Reference to a set of RDI model parameters to be used for the current catchment	Optional, if RDI activated	ParRDIIID

4.6.2 MOUSE RDI Parameter Sets

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

New parameter sets can be inserted and values of individual parameters can be edited in the "Parameters RDI" editor. This can be activated at MOUSE|Runoff Models|Parameters RDI.

MIKE URBAN comes with a default RDI parameter set (-DEFAULT-). User can insert any number of parameter sets and edit them according to the needs.



Parameter set ID: -DEFAULT-

Main parameters

Surface storage (Umax): 10.000

Root zone storage (Lmax): 100.000

Overland coefficient (CQof): 0.300

Groundwater coefficient (Carea): 1.00

TC overland flow (CK): 20.000

TC interflow (CKif): 500.000

TC baseflow (BF): 2000.000

☐ Snowmelt: 3.000

Threshold parameters

Overland(Tof): 0.000

Interflow(Tif): 0.000

Groundwater(Tg): 0.000

Groundwater parameters

Specific yield (Sy): 0.10

Min. GW depth (GWLmin): 0.000

Max. GW depth causing baseflow (GWLbf0): 10.000

GW Depth for Unit Capilary Flux (GWLfl1): 0.000

Initial conditions

Surface storage (U): 0.000

Root zone moisture (L): 0.000

Groundwater depth (GWL): 10.000

Overland flow (OF): 0.000

Interflow (IF): 0.000

Parameter	Surface sto	Root zone s	Overland co	Groundwat	TC overland	TC interflo	TC bas
-DEFAULT-	10.000	100.000	0.300	1.00	20.000	500.000	2000

Figure 4.17 The MOUSE RDI Parameter Sets Editor (MOUSE|Runoff Models|Parameters RDI)

A full overview of the editor fields and corresponding database attributes is provided in the Table 4.9.

Table 4.9 Overview of the RDI Parameter Set Database Attributes (Table 'msm_HparRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
Parameter Set ID	Parameter set identifier	Yes	MUID
Surface Storage (Umax)	Capacity of surface storage	Yes	Umax
Root Zone Storage (Lmax)	Capacity of root zone ("lower") storage	Yes	Lmax
Overland Coeff. (CQOF)	Fraction of runoff going to overland flow	Yes	Cqof
Groundwater Coeff. (Carea)	Relative size of groundwater reservoir	Yes	GwCarea

Table 4.9 Overview of the RDI Parameter Set Database Attributes (Table 'msm_HparRDII')

Edit field	**Description**	**Used or required by simulations**	**Field name in datastructure**
TC Overland Flow	Time constant for overland runoff component	Yes	Ck
TC Interflow	Time constant for interflow runoff component	Yes	Ckif
TC baseflow	Time constant for baseflow runoff component	Yes	Ckbf
Snowmelt checkbox	Switch for activation of the snow storage/snowmelt processes	Yes	SnowmeltNo
Snowmelt	Snowmelt coefficient	Optional, if snowmelt activated	SnowmeltC
Overland flow threshold (Tof)	Fraction of surface storage capacity at which overland flow starts	Yes	Tof
Interflow threshold (Tif)	Fraction of surface storage capacity at which interflow starts	Yes	Tif
Baseflow threshold (Tg)	Fraction of surface storage capacity at which ground water recharge starts	Yes	Tg
Specific Yield	Specific yield of the groundwater reservoir (porosity)	Yes	GwSy
Min. GW Depth	Top of the groundwater storage (depth from surface)	Yes	Gwlmin
Max GW Depth	Bottom of the groundwater storage (depth from surface)	Yes	Gwlbf0
GW Depth for Unit Capillary Flux	Groundwater depth causing unit capillary flux	Yes	Gwlfl1

Table 4.9 Overview of the RDI Parameter Set Database Attributes (Table 'msm_HparRDII')

Edit field	Description	Used or required by simulations	Field name in datastructure
Initial Surface Storage	Water depth in the surface storage at the simulation start	Yes	InitU
Initial root zone moisture	Moisture contents in the root zone at the simulation start	Yes	InitL
Initial Groundwater Depth	Groundwater depth at the simulation start	Yes	InitGw
Initial Interflow	Interflow intensity at the simulation start	Yes	InitIf
Initial Overland Flow	Overland flow intensity at the simulation start	Yes	InitOf

4.7 Rainfall Data and Other Meteorological Variables - Boundary Conditions for Hydrological models

Any precipitation-runoff model is "driven" by the specified precipitation (rainfall) data. Rainfall data are usually provided as time series - either in the form of accumulated rainfall depths or as average rainfall intensities. In some cases, the rainfall can be given in a simple form as a "block rain" with constant intensity over the entire simulation period.

MOUSE RDI also uses air temperature (used for snowmelt process) and potential evapo-transpiration data. Air temperatures are given as time series of instantaneous values. Potential evapo-transpiration is provided as time series of accumulated evapo-transpiration (e.g. on monthly basis). Equally as rainfall, both air temperature and potential evapo-transpiration can be specified as constant for the entire simulation period.

MIKE URBAN MOUSE can utilize time series data stored in DHI proprietary format DFS0. Please refer to Chapter 4 for importing, reviewing and editing of time series data.

All three categories of meteorological variables act as boundary conditions for the hydrological model and as such must be specified through the

MOUSE model boundary system (MOUSE|Boundary Condition|Catchment Loads). As with any other boundary condition, this is done in two distinct steps:

1. Declaration of a catchment boundary (rainfall, air temperature or evapotranspiration) and its relation to the catchments

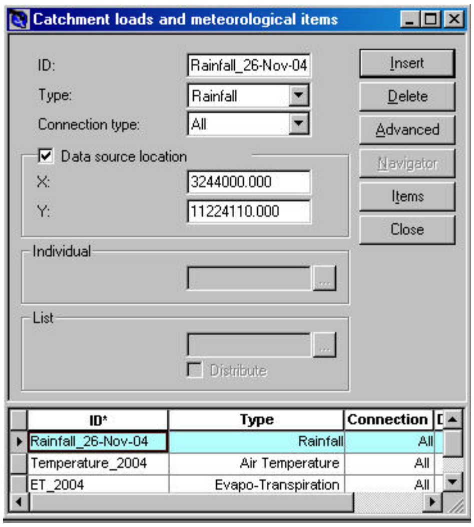
A new boundary is inserted by "Insert" button on the "Catchment Loads and meteorological items" editor (MOUSE|Boundary Condition|Catchment Loads). It is recommended to replace the default load identifier by a clearly descriptive string, which undoubtedly defines the actual contents of the boundary (e.g. Rainfall_26-Nov-04).

The current boundary will apply to the catchments according to the specified "Connection Type": All, List or Individual. If "All" is selected, the data source' (e.g. rain gauge or meteorological station) co-ordinates can optionally be specified.

The co-ordinates are of relevance only in cases where two or more boundaries of the same type have been specified with connection type "All": Instead of applying the boundary condition to the entire model, the system will apply the specified boundary conditions using the principle of geographical proximity to the catchment's centrepoint.

Some data sources (e.g. dfs0 time series) include the information about the data source co-ordinates. By activating the data source co-ordinates group on this dialog, the specified co-ordinates would overwrite the original co-ordinates from the time series file.

If the connection type is set to "List" or "Individual", the current boundary condition will be applied to the specified catchment(s) independently on the existence of co-ordinate information.



ID: Rainfall_26-Nov-04
Type: Rainfall
Connection type: All
☑ Data source location
X: 3244000.000
Y: 11224110.000
☐ Distribute

ID*	Type	Connection
Rainfall_26-Nov-04	Rainfall	All
Temperature_2004	Air Temperature	All
ET_2004	Evapo-Transpiration	All

Figure 4.18 The "Catchment Loads…" editor with a rainfall boundary condition applied for all catchments in the model. The co-ordinates specified under the "Data Source Location" represent the rain gauge location

At this point, the declared catchment boundary is just a "placeholder" for the actual rainfall data. These have to be specified in the second step.

2. Specification of the load items

Upon completed declaration of the catchment boundary, user must add the actual "substance" to it. This is done through the specification of "items" associated with the current boundary. Each catchment boundary condition must include at least one item. By the nature of the meteorological boundaries, they can include only the "default" item types.

The items can be created and edited in the "Boundary Items" editor. This can be open by "Items" button on the "Catchment Loads…" editor (recommended), or through the menu at MOUSE|Boundary Conditions|Items.

If opened from the "Boundary Loads…" editor, the "Boundary Items" editor displays only the items associated with the current boundary. Initially, for a newly declared boundary, the list of items will be empty. A new item is created by the "Insert" button.

For any item of the catchment meteorological boundary "Temporal Variation" can specified either as "Constant" or as "Time series/Result File". If "Constant" is selected, user must specify the constant value. If "Time series" is selected, user must select "DFS0" a source type. The actual DFS0 file to be applied ("Path") can be specified by typing the file name (and a full path if the file is NOT located in the project directory) or by the help of file browser.

Per default, the specified boundary will be applied throughout the entire duration of the simulation or during the period covered by the time series. Optionally, the user can limit the temporal validity of the specified file to any period within the DFS0 file's duration, by specifying the temporal validity. For the rest of the simulation period, the boundary conditions must be "extended" by some other time series or constant value. This is achieved by inserting and specifying another boundary item.

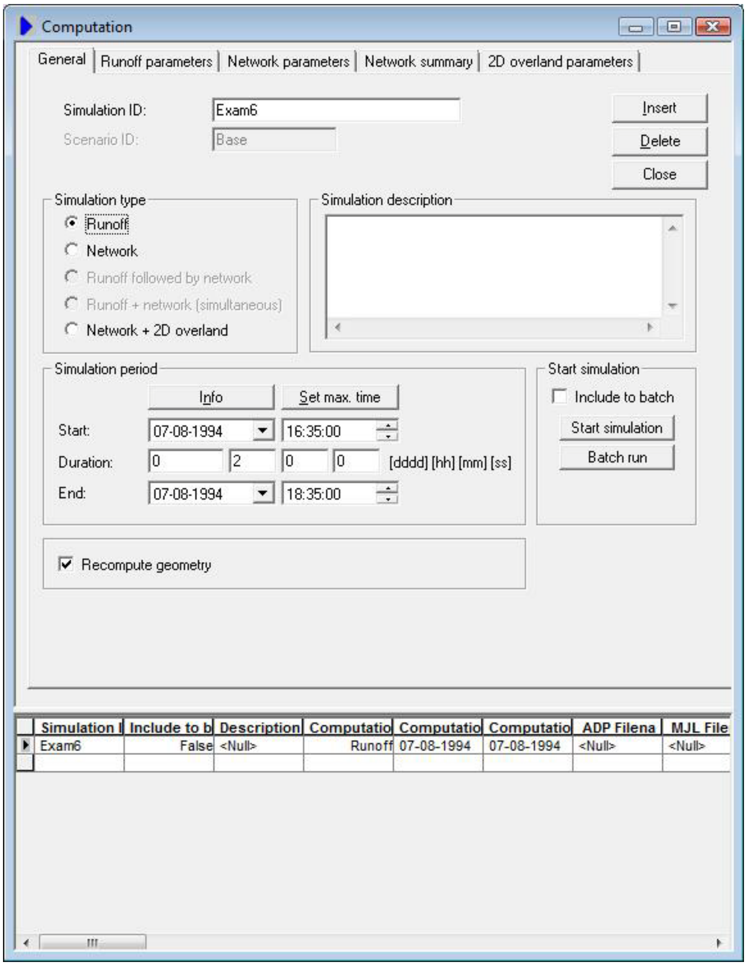
A full reference on the functionality of the MOUSE boundary system is available in Chapter 6.

4.8 Running the Runoff Computations

Upon the definition of model data for all catchments, defining the catchment connections (if the computed runoff is to be loaded into the network model) and the definition of the boundary conditions, the model is ready for the runoff computation.

The computation is set-up and initiated from the MOUSE Computation dialog (Simulation|Run MOUSE) - see Chapter 7 for full detail).

On the "General" TAB, the user must specify the essential information about the simulation (see example in Figure 4.19), and the runoff-specific parameters on the "Runoff parameters" TAB (see example in Figure 4.20).


Computation
General
Runoff parameters
Network parameters
Network summary
2D overland parameters
Insert
Delete
Close
Info
Set max. time
Start simulation
Batch run

Simulation ID: Exam6

Scenario ID: Base

Simulation type
● Runoff
○ Network
○ Runoff followed by network
○ Runoff + network (simultaneous)
○ Network + 2D overland

Simulation description

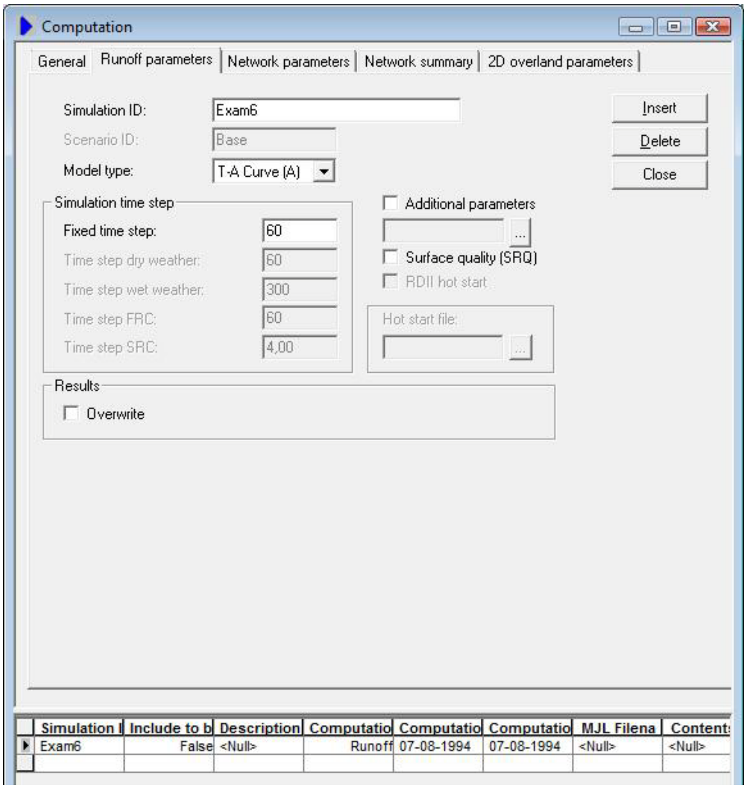
Simulation period
Start: 07-08-1994 16:35:00
Duration: 0 2 0 0 [dddd] [hh] [mm] [ss]
End: 07-08-1994 18:35:00

Start simulation
☐ Include to batch

☑ Recompute geometry

Simulation I	Include to b	Description	Computatio	Computatio	Computatio	ADP Filena	MJL File
Exam6	False	<Null>	Runoff	07-08-1994	07-08-1994	<Null>	<Null>

Figure 4.19 The "MOUSE Computation dialog (Simulation|Run MOUSE), with general computation parameters



Simulation ID: Exam6
Scenario ID: Base
Model type: T-A Curve (A)

Simulation time step
Fixed time step: 60
Time step dry weather: 60
Time step wet weather: 300
Time step FRC: 60
Time step SRC: 4,00

☐ Additional parameters
☐ Surface quality (SRQ)
☐ RDII hot start
Hot start file:

Results
☐ Overwrite

Simulation I	Include to b	Description	Computatio	Computatio	Computatio	MJL Filena	Content:
Exam6	False	<Null>	Runoff	07-08-1994	07-08-1994	<Null>	<Null>

Figure 4.20 The "MOUSE Computation dialog (Simulation|Run MOUSE), with runoff-specific (T-A Curve) computation parameters

4.9 MOUSE RDI - Guidelines for application

4.9.1 Choice of calculation time step

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

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

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

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

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

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

4.9.2 The RDI hotstart

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

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

4.9.3 The RDI result files

Two result files are generated by a MOUSE RDI calculation. These are:

- *.CRF file, containing maximum five time series for each sub-catchment, namely:
    - discharge, calculated with the Surface Runoff Model (the FRC component),
    - discharge, calculated with the RDI model (the SRC component),
    - total discharge,
    - variation of water content in the surface storage for the Surface Runoff Model,
    - variation of water content in the snow storage for the Surface Runoff Model.

The *.CRF file is used as input data for a MOUSE Hydrodynamic calculation.

- *.NOF file (optional), containing detailed information about the processes treated by a RDI- model, e.g.:
    - different flow components in the RDI model,
    - variation of water content in the different storage in the RDI model.

The *.NOF file is used for calibration of the SRC component and for RDI hotstart.

In the *.CRF file the time series are saved with two various intervals, the shorter one for the periods when the Surface Runoff Model is used, and a larger one in the remaining periods. In the *.NOF file the time series are saved with the larger time interval, which is equal to the time step used for the RDI calculation.

4.9.4 MOUSE RDI Validation

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

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

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

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

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

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

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

MOUSE supports visual comparison of the calculated time series with any time series of the same type contained in the time series database. E.g., when validating the model, the calculated discharge can be plotted on the same graph with the measured discharge and compared.

In the present version of MOUSE RDI there is no automatic calculation or evaluation of specific numeric validation criteria as mentioned above. If appropriate, analysis of that type can be conducted so that the calculated time series are exported to a spreadsheet or some other program for further processing and comparison with measured time series.

In the example related to the illustrations, overflow occurs within the model area. MOUSE RDI can not describe this kind of processes, which complicates the choice of validation criteria.

Surface runoff model

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

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

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

General hydrological model - RDI

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

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

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

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

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

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

The following sequence of action is recommended:

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

  The total volume generally also contains the runoff from impervious areas (Surface Runoff Model) - see Chapter 4.

- Next, the overland flow coefficient CQOF is adjusted to obtain a correct distribution of volume between overland flow (peak flows) and baseflow. This is done after wet periods and preferably for a period with low evaporation.

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

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

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

  An adjustment of CKBF does not influence the size of the discharged volume studied for a longer period, but displaces the volumes in time.

- CKOF is adjusted against the response, i.e. the shape of the peak flows. This is done during periods with heavy rainfall, preferably after a wet period.

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

- A reduction of Umax reduces the actual evapo-transpiration, the process responsible for reduced discharges during period with high potential evaporation. The effect of reducing Umax will be largest for periods preceded by a wet period. Additionally, an increased overland flow is obtained, as well as more water transported to the groundwater storage resulting in an delayed effect of increased baseflow, because of the long response time of baseflow.

  An important behaviour of the RDI model is that the surface storage must be filled-up before overland flow and infiltration, respectively, occur. Therefore, during dry periods with high potential evaporation, Umax can be estimated from how much rainfall is required for filling-up the surface storage, i.e. generating overland flow. The same methodology can also be used for the periods with low potential evaporation, but only if the rain event is preceded by a long dry period.

- CKIF is adjusted against the response of interflow during periods with low potential evaporation. A reduction of CKIF will result in a small increase in volume during these periods.
- The relative water content in the unsaturated zone (i.e. root-zone), L/Lmax controls several of the different water transports in the RDI model. Since the storage capacity, Lmax, influences the velocity of the filling of L towards Lmax, Lmax is adjusted during periods of heavy filling of the root zone storage. This usually occurs during periods with low potential evaporation preferably in combination with a wet period.

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

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

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

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

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

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

TIF is adjusted after a dry period when filling of the root zone storage, preferably in combination with low potential evaporation. However, TIF is one of the less important parameters.

- The degree-day-coefficient, Csnow can be estimated from analysis of the relation between temperature, water content in the snow storage and measured discharge. When temperature is below zero, the precipitation is stored in the snow storage. When temperature is above zero the content in the snow storage is emptied into the surface storage, where the velocity of emptying is controlled by Csnow. An increase of Csnow increases the emptying procedure.

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

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

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

  Figure 8 shows an example of the build-up of the snow cover, followed by the snow-melting process. The calculated and measured flow reactions during the same period are shown. The example is from the treatment plant at Duvbacken, Gävle, Sweden. Considering the complexity of the snow melting process within urban areas, a fairly good description was obtained with the RDI model.

Since the variation of water contents in the surface and root zone storage controls many of the other processes, they should be studied continuously throughout the calibration procedure. Figure 9 shows an example of the variation of water content in the surface storage, root zone storage and groundwater storage. The example comes from the catchment of Rya treatment plant, Göteborg, Sweden. It appears that the root zone storage is emptied only during the summer period, because the evaporation during the rest of the year is almost non-existent. Discharge from the groundwater storage exists continuously all year around. Drawing of the surface storage is faster during summer period since the evaporation is high, and is therefore the dominating effect on the surface storage. During periods with low evaporation, drawing of the surface storage is controlled by the given time constant for interflow, CKIF.

The example also shows that filling of the root zone and groundwater storage only occurs when the surface storage is completely filled, i.e. when precipitation has filled-up the surface storage. A larger surface storage, i.e. a larger Umax, will therefore imply that this happens more rarely and at a smaller extent, allowing a larger part of the precipitation to evaporate.

A smaller root zone storage, i.e. a smaller Lmax, would have led to an increased relative variation in the storage. Furthermore, the actual evaporation will decrease in case of smaller root zone storage, because less water is available for the vegetation to draw water for transpiration, mainly during summer period.

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

4.9.5 Overflow within the model area

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

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

4.9.6 Non-precipitation dependent flow components

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

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

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

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

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

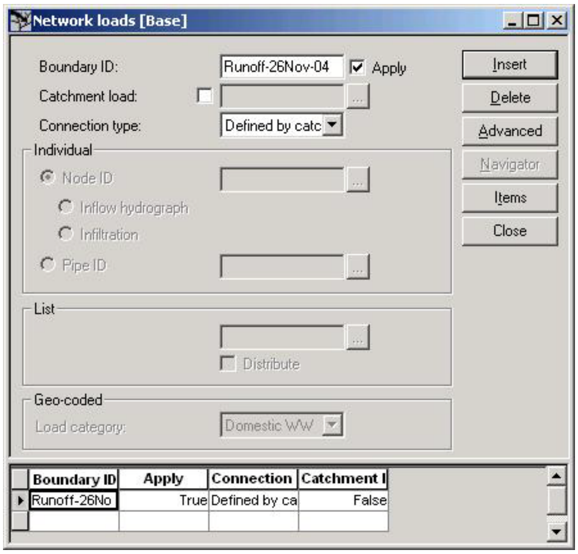
4.10 Using the Computed Runoff as Network Hydraulic Load

The computed runoff is frequently applied as a hydraulic load to the collection network. This is achieved through the MOUSE boundary system (MOUSE|Boundary Condition|Network Loads). As with any other network load, this is done in two distinct steps:

1. Declaration of the network load and its connectivity to the network

A new network load is inserted by "Insert" button on the "Network Loads" editor (MOUSE|Boundary Condition|Network Loads). It is recommended to replace the default load identifier by a clearly descriptive string, which undoubtedly defines the actual contents of the load (e.g. Runoff_event 26-Nov-04).

Concerning the network connectivity of the storm runoff, the option "Defined by Catchment Connections" must be selected. This selection means that the model will utilize the catchment connection information from the runoff result file (*.CRF) used. I.e. no explicit declaration of the runoff connection point is needed.



Boundary ID: Runoff-26Nov-04 ☑ Apply

Catchment load: ☐

Connection type: Defined by catc

Individual
● Node ID
○ Inflow hydrograph
○ Infiltration
○ Pipe ID

List
☐ Distribute

Geo-coded
Load category: Domestic WW

Boundary ID	Apply	Connection	Catchment l
Runoff-26No	True	Defined by ca	False

Figure 4.21 The "Network Loads" editor with runoff defined as a load. "Defined by catchment connections" implies that the MOUSE runoff result file (*.CRF) will be used as a source of connection information

At this point, the declared network load is just a "placeholder" for the actual data. These have to be specified in the second step.

2. Specification of the load items

Upon completed declaration of the network load, user must add the actual "substance" to it. This is done through the specification of "items" associated with the current load. Per default, each load must include the "flow item". This carries the information about the actual water amount and its temporal variation. Optionally, items containing similar information about pollutants, sediments or water temperature associated with this load may be added.

The items can be created and edited in the "Boundary Items" editor. This can be open by "Items" button on the "Network Loads" editor (recommended), or through the menu at MOUSE|Boundary Conditions|Items.

If opened from the "Network Loads" editor, the "Boundary Items" editor displays only the items associated with the current load. Initially, for a newly declared load, the list of items will be empty.

"Insert" creates a new item. Automatically, each new item is categorized as "Default", i.e. as flow item. If the current MOUSE model includes any pollutants of sediments, user may change the item type to be "Pollutant" or "Sediment". In any case, at least one flow item must be present on the list.

For the default item, "Temporal Variation" is specified as "Time series/Result File". In the "Time series" group, "MOUSE" (i.e. MOUSE result file) is selected as data source type. The actual file to be applied ("Path") can be specified by typing the file name (and a full path if the file is NOT located in the project directory) or by the help of file browser.

Per default, the specified runoff result file will load the network throughout its entire duration, corresponding to the runoff computation Start and End times. Optionally, the user can limit the temporal validity of the specified file to any period within the file's duration, by specifying the temporal validity. For the rest of the simulation period, the network can be loaded by some other runoff result file. This is achieved by inserting and specifying another flow item.

A full reference on the functionality of the MOUSE boundary system is available in the Chapter 6.

4.11 Low Impact Development (LID)

Water sensitive urban design (WSUD) represents an approach to land development (or re-development) that works with nature to manage storm water as close to its source as possible, it is known also as Low Impact Development (LID).

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

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

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

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

This document describes the implementation and development of model support of various LID structures in MIKE URBAN.

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

- Modelling of LID at screening level - catchment-based approach
- Detailed hydraulic modelling of individual LID structures - drainage network based approach.
  This method will provide the user with the option of detailed modelling of individual LID structures connected hydraulically to the storm water pipe network. This approach is based on the concept of soakaway node explained in section 3.3.5 Model Concept of Soakaway *(p. 29)*.

In this document, the MIKE URBAN implementation of the screening level - the catchment-based approach - is explained.

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

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

Please be aware that modelling of LID practices in MIKE URBAN is allowed for the Kinematic Wave runoff model (Model B) and runs with the MIKE 1D engine, exclusively.

4.11.1 LID Controls

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

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

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

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

Bio Retention Cells

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

The different structural layers of this feature are presented in Figure 4.22. There are one or several relevant hydrological processes associated with each layer.

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

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

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

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

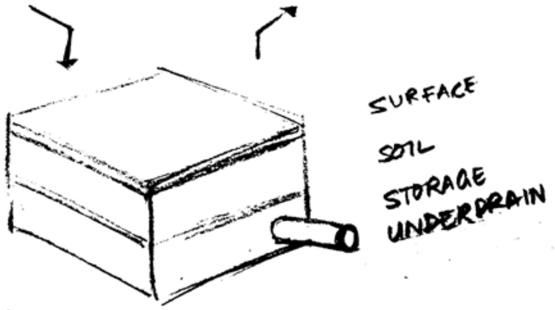


Figure 4.22 Bio retention cell structure layers

Infiltration Trenches

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

In Figure 4.23 a schematic drawing of the infiltration trench structure is presented. This type of LID control includes surface, storage and, optionally, underdrain.

The processes simulated for the infiltration trench are similar as for the bioretention cell, except for the missing soil layer: storm water from the surface enters the storage directly, i.e. without detention in the soil layer.

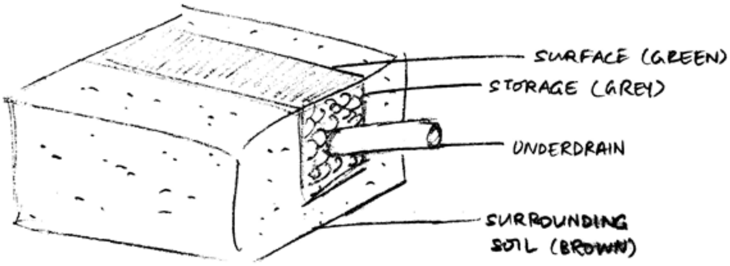


Figure 4.23 Infiltration trench structure layers

Porous Pavement

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

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

In Figure 4.24 it is sketched the porous pavement structure comprehending surface, pavement material, storage and underdrain.

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

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

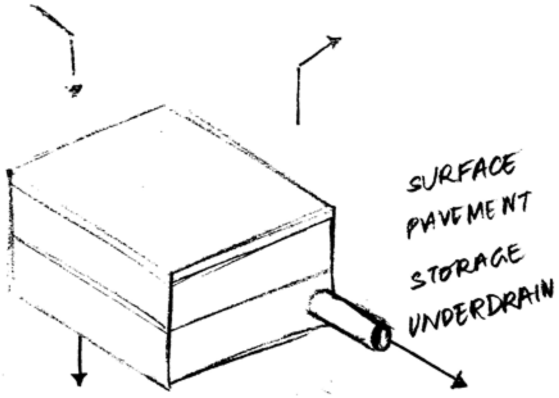


Figure 4.24 Porous Pavement structure layers

Rain Barrels

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

A rain barrel includes storage (with total porosity, i.e. empty volume) and underdrain.

The processes simulated include water detention in the storage, overflow and drainage (i.e. storage recovery). The action of the underdrain is timed: it can be specified the time offset between the end of rain and the drain activation.

This feature allows for the simulation of rainwater re-use for garden watering, which normally takes place several hours or days after the rain has stopped.

A conceptual sketch of a rain barrel is presented in Figure 4.25.

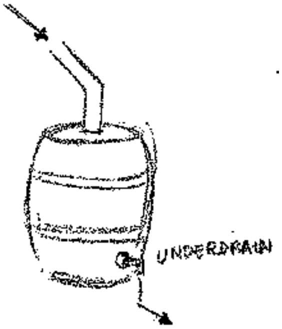


Figure 4.25 Rain Barrel

Vegetative Swales

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

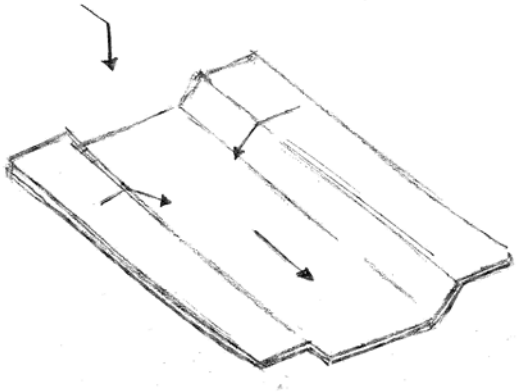


Figure 4.26 Vegetative Swale layer structure

Rain Garden

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

The different layers comprising rain gardens are presented in Figure 4.27. These include surface and soil layer

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

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

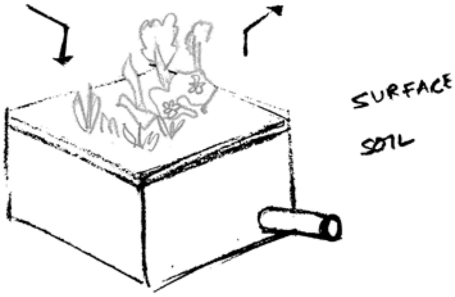


Figure 4.27 Rain garden process layers

Green Roof

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

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

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

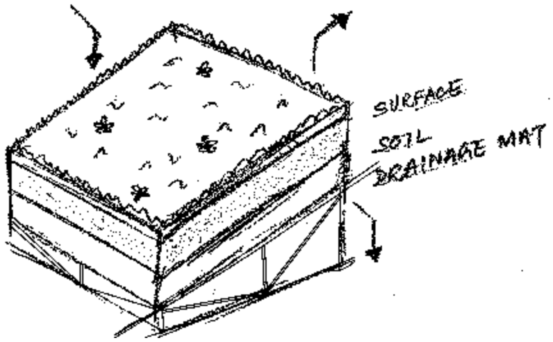


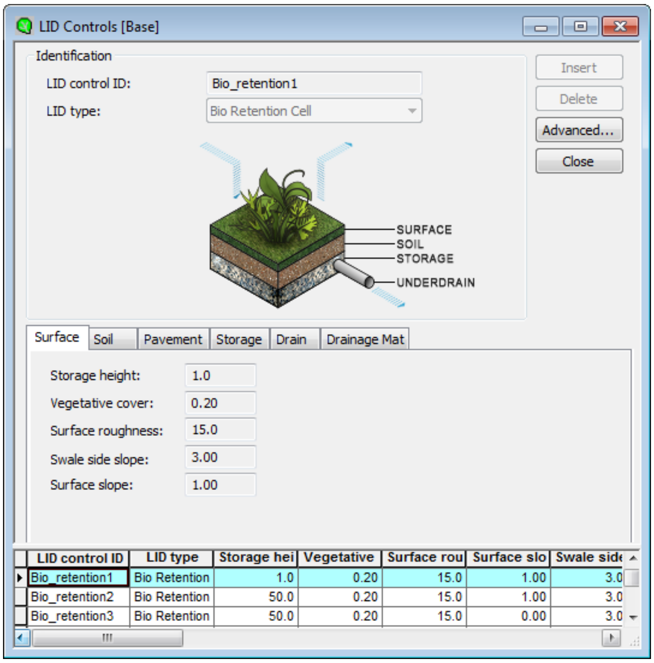
Figure 4.28 Green roof process layers

4.11.2 The LID Controls Editor

The MIKE URBAN LID Controls editor organizes the data input for the different layers and functional elements of Low Impact Development structures. The input data is organized into following group and tabs:

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

Figure 4.29 through Figure 4.34 illustrate the six tabs containing the data for various functional elements of the LID structure. Table 4.10 and Table 4.11 provide a full reference for the attributes of the msm_LIDControl database table.


LID Controls [Base]
Identification
Insert
Delete
Advanced...
Close
SURFACE
SOIL
STORAGE
UNDERDRAIN
Surface
Soil
Pavement
Storage
Drain
Drainage Mat

LID control ID: Bio_retention1

LID type: Bio Retention Cell

Storage height: 1.0

Vegetative cover: 0.20

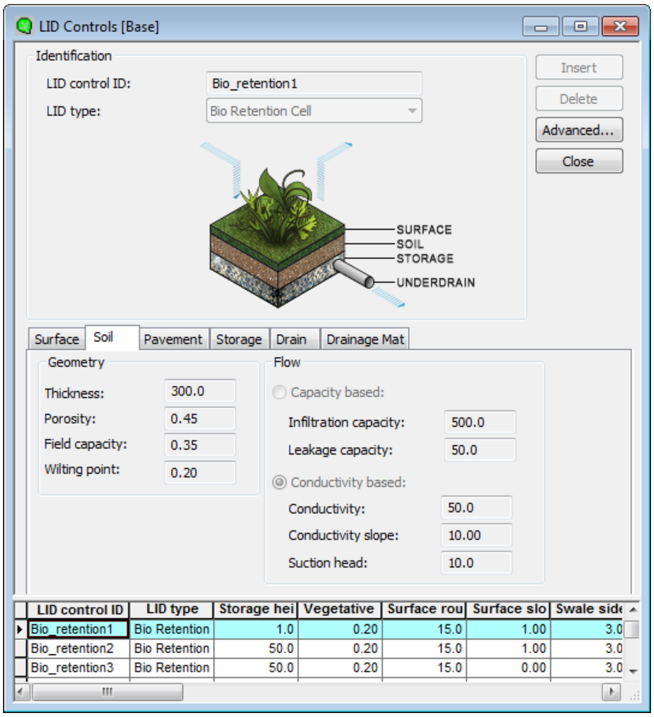
Surface roughness: 15.0

Swale side slope: 3.00

Surface slope: 1.00

LID control ID	LID type	Storage hei	Vegetative	Surface rou	Surface slo	Swale side
Bio_retention1	Bio Retention	1.0	0.20	15.0	1.00	3.0
Bio_retention2	Bio Retention	50.0	0.20	15.0	1.00	3.0
Bio_retention3	Bio Retention	50.0	0.20	15.0	0.00	3.0

Figure 4.29 The MIKE URBAN LID Control Editor - Surface Tab


LID Controls [Base]
Identification
Insert
Delete
Advanced...
Close
SURFACE
SOIL
STORAGE
UNDERDRAIN
Surface
Soil
Pavement
Storage
Drain
Drainage Mat

LID control ID: Bio_retention1

LID type: Bio Retention Cell

Geometry

Thickness: 300.0

Porosity: 0.45

Field capacity: 0.35

Wilting point: 0.20

Flow

○ Capacity based:

Infiltration capacity: 500.0

Leakage capacity: 50.0

● Conductivity based:

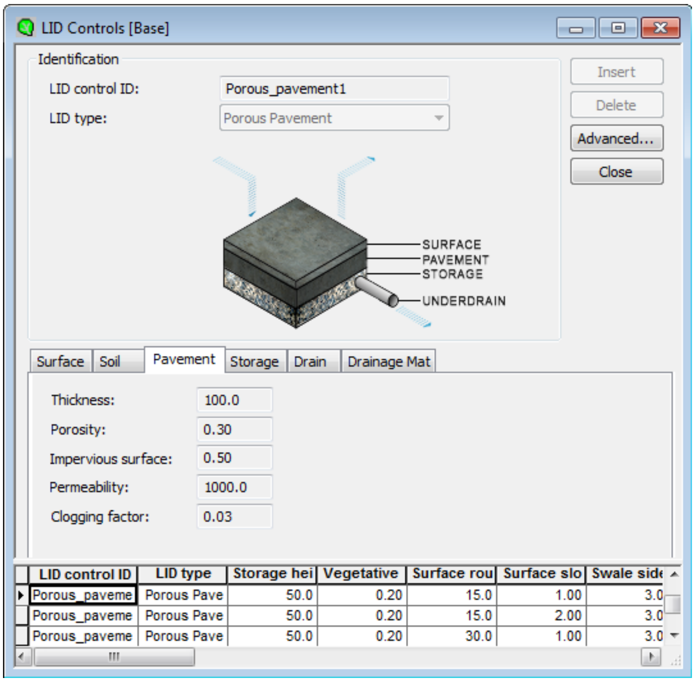
Conductivity: 50.0

Conductivity slope: 10.00

Suction head: 10.0

LID control ID	LID type	Storage hei	Vegetative	Surface rou	Surface slo	Swale side
Bio_retention1	Bio Retention	1.0	0.20	15.0	1.00	3.0
Bio_retention2	Bio Retention	50.0	0.20	15.0	1.00	3.0
Bio_retention3	Bio Retention	50.0	0.20	15.0	0.00	3.0

Figure 4.30 The MIKE URBAN LID Control Editor - Soil Tab


LID Controls [Base]
Identification
Insert
Delete
Advanced...
Close
SURFACE
PAVEMENT
STORAGE
UNDERDRAIN
Surface
Soil
Pavement
Storage
Drain
Drainage Mat

LID control ID: Porous_pavement1

LID type: Porous Pavement

Thickness: 100.0

Porosity: 0.30

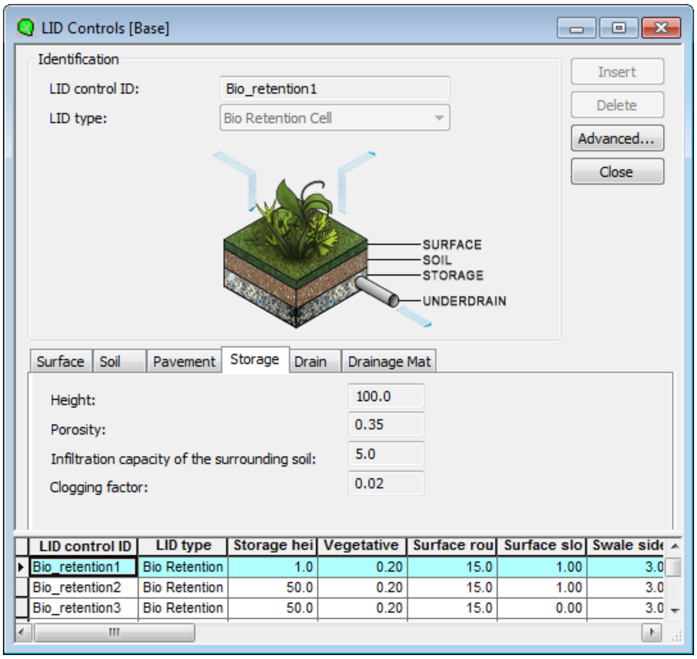
Impervious surface: 0.50

Permeability: 1000.0

Clogging factor: 0.03

LID control ID	LID type	Storage hei	Vegetative	Surface rou	Surface slo	Swale side
Porous_paveme	Porous Pave	50.0	0.20	15.0	1.00	3.0
Porous_paveme	Porous Pave	50.0	0.20	15.0	2.00	3.0
Porous_paveme	Porous Pave	50.0	0.20	30.0	1.00	3.0

Figure 4.31 The MIKE URBAN LID Control Editor - Pavement Tab


LID Controls [Base]
Identification
Insert
Delete
Advanced...
Close
SURFACE
SOIL
STORAGE
UNDERDRAIN
Surface
Soil
Pavement
Storage
Drain
Drainage Mat

LID control ID: Bio_retention1

LID type: Bio Retention Cell

Height: 100.0

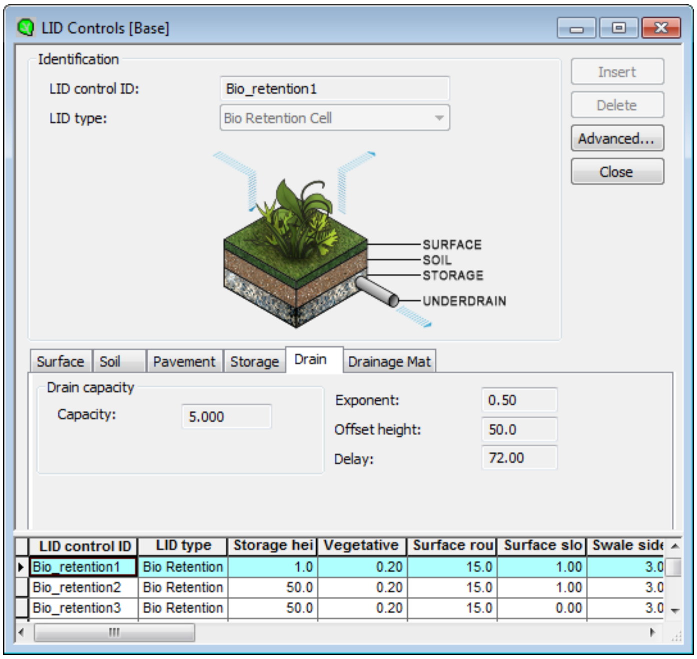
Porosity: 0.35

Infiltration capacity of the surrounding soil: 5.0

Clogging factor: 0.02

LID control ID	LID type	Storage hei	Vegetative	Surface rou	Surface slo	Swale side
Bio_retention1	Bio Retention	1.0	0.20	15.0	1.00	3.0
Bio_retention2	Bio Retention	50.0	0.20	15.0	1.00	3.0
Bio_retention3	Bio Retention	50.0	0.20	15.0	0.00	3.0

Figure 4.32 The MIKE URBAN LID Control Editor - Storage Tab


LID Controls [Base]
Insert
Delete
Advanced...
Close
SURFACE
SOIL
STORAGE
UNDERDRAIN
Surface
Soil
Pavement
Storage
Drain
Drainage Mat

Identification

LID control ID: Bio_retention1

LID type: Bio Retention Cell

Drain capacity

Capacity: 5.000

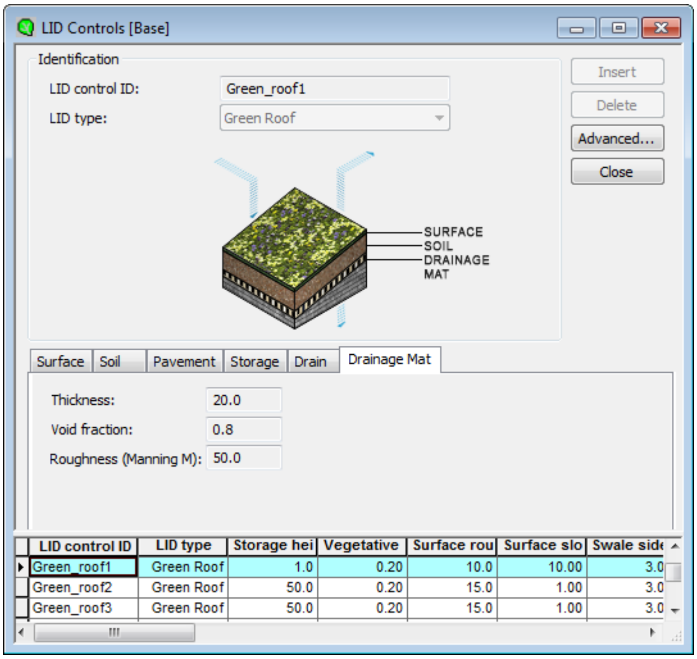
Exponent: 0.50

Offset height: 50.0

Delay: 72.00

LID control ID	LID type	Storage hei	Vegetative	Surface rou	Surface slo	Swale side
Bio_retention1	Bio Retention	1.0	0.20	15.0	1.00	3.0
Bio_retention2	Bio Retention	50.0	0.20	15.0	1.00	3.0
Bio_retention3	Bio Retention	50.0	0.20	15.0	0.00	3.0

Figure 4.33 The MIKE URBAN LID Control Editor - Drain Tab


LID Controls [Base]
Insert
Delete
Advanced...
Close
SURFACE
SOIL
DRAINAGE MAT
Surface
Soil
Pavement
Storage
Drain
Drainage Mat

Identification

LID control ID: Green_roof1

LID type: Green Roof

Thickness: 20.0

Void fraction: 0.8

Roughness (Manning M): 50.0

LID control ID	LID type	Storage hei	Vegetative	Surface rou	Surface slo	Swale side
Green_roof1	Green Roof	1.0	0.20	10.0	10.00	3.0
Green_roof2	Green Roof	50.0	0.20	15.0	1.00	3.0
Green_roof3	Green Roof	50.0	0.20	15.0	1.00	3.0

Figure 4.34 The MIKE URBAN LID Control Editor - Drainage Mat Tab

Identification

Table 4.10 The LID Control Identification Group

Edit field	Description	Used or required by simulations	Field name in data structure
LID Control ID	ID of low impact development, LID	Yes	MUID
LID Type	Type of LID 1:Bio Retention Cell 2:Porous Pavement 3:Infiltration trench 4:Rain Barrel 5: Vegetative Swale 6:Rain Garden 7:Green Roof	Yes	LIDTypeNo

LID Control data Specification

Table 4.11 The LID data specification

Tab/Edit field	Description	Used or required by simulations	Field name in data structure
Surface/ Storage Depth	When confining walls or berms are present, this is the maximum depth to which water can pond above the surface before overflow occurs. For LIDs that experience ponding it is the height of any surface depression storage. For swales, it is the height of its trapezoidal cross section.	Yes If LIDTypeNo = (1,2,3,5)	StorHt
Surface/ Vegetative Cover	The fraction of the storage area above the surface that is filled with vegetation (0 = no vegetation, 1 = no storage available). NOTE: for infiltration trench, vegetative cover is typically non-existent	Yes If LIDTypeNo = (1,2,3,5)	VegFrac
Surface/ Surface Roughness	Manning´s n or M (used in combination with surface slope and width) for routing of overflow from the surface of bio-retention cell, rain garden, infiltration trench, porous pavement and for flow routing in vegetative swale (see Table 4.12 for typical values). If specified zero (for all types except vegetative swale), no routing of the overflow is applied	Yes If LIDTypeNo = (1,2,3,5 and 6)	Rough
Surface/ Surface Slope	Slope (used in combination with surface roughness and width) for routing of overflow from the surface of bio-retention cell, rain garden, infiltration trench, porous pavement and for flow routing in vegetative swale. If specified zero (for all types except vegetative swale), no routing of the overflow is applied	Yes If LIDTypeNo = (1,2,3,5 and 6)	Slope
Surface/ Swale Side Slope	Slope (run over rise) of the sidewalls of a vegetative swale´s cross section. Used for the calculation of the stored volume and wetted width. This value is ignored for other types of LIDs	Yes If LIDTypeNo=5	Xslope
Soil/ Thickness	Thickness of the soil layer. Typical values range from 450 to 900 mm for rain gardens, street planters and other types of land-based bio-retention units, but only 75 to 150 mm for green roofs	Yes If LIDTypeNo = (1, 6, 7)	SThick
Soil/ Porosity	Volume of pore space relative to total volume of soil (as a fraction)	Yes If LIDTypeNo = (1, 6, 7)	Por

Table 4.11 The LID data specification

Tab/Edit field	Description	Used or required by simulations	Field name in data structure
Soil/ Field Capacity	Volume of pore water relative to total volume held in the soil after excess water has drained away and the rate of downward movement has decreased (as a fraction). Vertical percolation of water through the soil occurs only when field capacity level is reached or exceeded.	Yes If LIDTypeNo = (1, 6, 7)	FC
Soil/ Wilting Point	Volume of pore water relative to total volume for a well-dried soil where only bound water remains (as a fraction). The moisture content of the soil cannot fall below this limit and wilting point cannot be higher than the Field Capacity level.	Yes If LIDTypeNo = (1, 6, 7)	WP
Soil/ Infiltration capacity	The rate of water moving from surface into the soil. It is characteristics of the surface and the uppermost soil layer (equivalent to Horton´s initial infiltration capacity). Infiltration takes place at all times during a rain event and after, as long as there is water available as rainfall, as run on or as water stored in the surface.	Yes If LIDTypeNo = (1, 6, 7)	InfiltrationCapacity
Soil/ Leakage capacity	The rate of water leaving the soil layer into storage. It is characteristic of the soil layer. Leakage starts when soil storage exceeds field capacity, continues at all times while soil storage is above the field capacity.	Yes If LIDTypeNo = (1, 6, 7) AND FlowMethod = 1 (capacity based))	LeakageCapacity
Soil/ Conductivity	Hydraulic conductivity for the fully saturated soil. This is equivalent to leakage capacity	Yes If LIDTypeNo = (1, 6, 7) AND FlowMethod = 2 (conductivity based))	Ksat
Soil/ Conductivity slope	Slope of the curve of log (conductivity) versus soil moisture content (dimensionless). Typical values range from 5 for sands to 15 for silty clay	Yes If LIDTypeNo = (1, 6, 7) AND FlowMethod = 2 (conductivity based)	Kcoeff
Soil/ Suction Head	The average value of soil capillary suction along the wetting front. This is the same parameter as used in the Green-Ampt infiltration model	Yes If LIDTypeNo = (1, 6, 7) AND FlowMethod = 2 (conductivity based)	Suct
Pavement/ Thickness	Thickness of the pavement. Typical values are 100 to 150 mm	Yes If LIDTypeNo = 2	PThick

Table 4.11 The LID data specification

Tab/Edit field	Description	Used or required by simulations	Field name in data structure
Pavement/ Porosity	The ratio (expressed as a fraction) of the volume of the pores or interstices of a material to the total volume of the pavement. Typical values range from 0.11 to 0.17 for pavements Note that porosity = void ratio / (1 + void ratio).	Yes If LIDTypeNo =2	Pvratio
Pavement/ Impervious Surface	Ratio of impervious paver material to total area for modular systems; 0 for continuous porous pavement systems	Yes If LIDTypeNo = 2	FracImp
Pavement/ Permeability	Permeability of concrete or asphalt used in continuous systems or hydraulic conductivity of the fill material (gravel or sand) used in modular systems. Permeability of new porous concrete or asphalt is high (>2450 mm/h), but over time the fine particles in the runoff tend to clog the pavement, reducing the permeability of the structure.	Yes If LIDTypeNo = 2	Perm
Pavement/ Clogging Factor	Voids that are clogged due to fine particles accumulation, as a fraction of total voids area. Use a value of 0 to ignore clogging. Max. value = 1.	Yes If LIDTypeNo = 2	PVClog
Storage/ Height	The height of the storage layer in case of bio retention cell, porous pavement and infiltration trench; the height of a rain barrel (mm or inches). Crushed stone and gravel layers are usually 150 to 450 mm thick while rain barrels vary in height from 600 mm upwards.	Yes If LIDTypeNo = (1,2,3,4)	Height
Storage/ Porosity	The ratio of the volume of the pores or interstices of a material to the total volume of the layer. Typical values range from 0.30 to 0.45 for gravel beds. Note that porosity = void ratio / (1 +void ratio).	Yes If LIDTypeNo = (1,2,3)	Svratio
Storage/ Conductivity	The maximum rate at which water infiltrates to the surrounding soil through the bottom of the freshly constructed storage layer.	Yes If LIDTypeNo = (1, 2, 3)	Filt
Storage/ Clogging Factor	Volume of voids that are clogged due to fine particles accumulation. Use a value of 0 to ignore clogging.	Yes If LIDTypeNo = (1,2,3)	SVclog
Drain/ Capacity	Coefficient C that determines the rate of flow through the underdrain as a function of height of stored water above the drain bottom.	Yes If LIDTypeNo = (1,2,3,4)	Coeff
Drain/ Exponent	Exponent n that determines the rate of flow through the underdrain as a function of height of stored water above the drain height.	Yes If LIDTypeNo = (1,2,3,4)	Expon

Table 4.11 The LID data specification

Tab/Edit field	Description	Used or required by simulations	Field name in data structure
Drain/ Offset Height	Height of any underdrain piping above the bottom of a storage layer or rain barrel	Yes If LIDTypeNo = (1,2,3,4)	Offset
Drain/ Delay	The number of dry weather hours that must elapse before the drain in a rain barrel is activated	Yes If LIDTypeNo = 4	Delay
Drainage Mat/ Thickness	The thickness of the mat or plate. It typically ranges between 25 to 50 mm	Yes If LIDTypeNo = 7	DMThick
Void Fraction	The ratio of void volume to total volume in the mat. It typically ranges from 0.5 to 0.6	Yes If LIDTypeNo = 7	DMVFraction
Roughness	Manning´s number, used to compute the horizontal flow rate of drained water through the mat. In absence of standard product specifications provided by manufacturers, the roughness must be estimated. Usage of n-values from 0.1 to 0.4 (M = 2.5 - 10) is suggested.	Yes If LIDTypeNo = 7	DMRough

In Table 4.12 are presented examples of Manning' values (M) for different types of surfaces. Table 4.13 shows hydraulic conductivity properties for various porous media.

Table 4.12 Manning´s M of surface for porous pavement or vegetative swale [a]

Surface type	Manning M
Smooth asphalt	91
Smooth concrete	83
Ordinary concrete lining	77
Good wood	71
Brick with cement mortar	71
Vitrified clay	67
Cast Iron	67
Corrugated metal pipes	42
Cement rubble surface	42
Fallow soils (no residue)	20
Cultivated soils	50 to 20
Residue cover < 20%	17
Residue cover > 20%	6
Range (natural)	8
Short, prairie	7
Dense	4

Table 4.12 Manning´s M of surface for porous pavement or vegetative swale [a]

Surface type	**Manning M**
Bermuda grass	2
Woods	10
Light underbrush	2,5
Dense underbrush	1,25

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

Table 4.13 Hydraulic conductivity and porosity of unconsolidated porous media [a]

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

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

4.11.3 LID Deployment

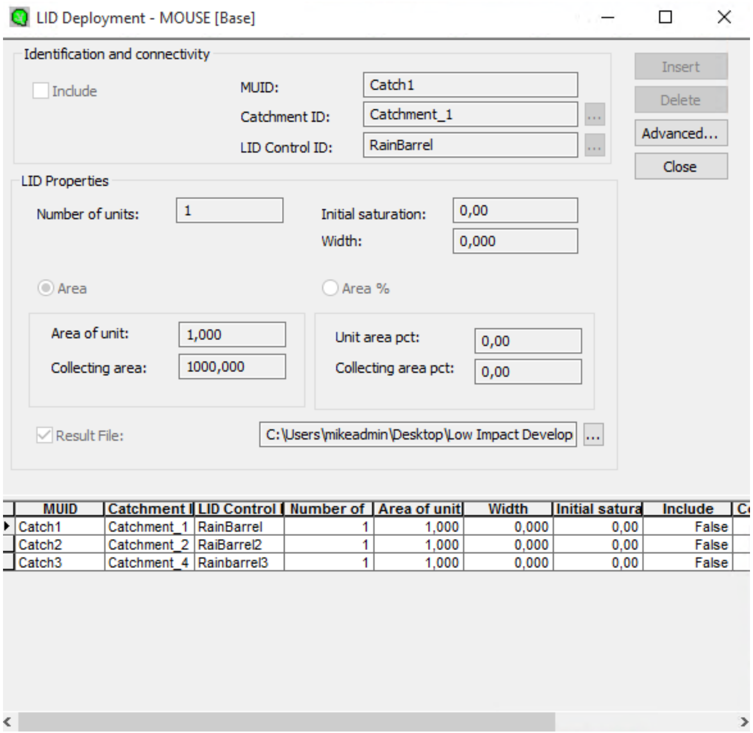
The LID Controls are assigned to a catchment by means of the LID deployment editor. In this editor, it is possible to specify the size (i.e. area) of the individual control, the size of the catchment´s impervious area that is funneled into it and the number of the current LID control units.

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

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

The input data is organized into following groups:

- Identification and Connectivity
- LID Deployment Properties


LID Deployment - MOUSE [Base]
Identification and connectivity
LID Properties
Insert
Delete
Advanced...
Close
Area
Area %

☐ Include

MUID: Catch1

Catchment ID: Catchment_1

LID Control ID: RainBarrel

Number of units: 1

Initial saturation: 0,00

Width: 0,000

Area of unit: 1,000

Collecting area: 1000,000

Unit area pct: 0,00

Collecting area pct: 0,00

☑ Result File: C:\Users\mikeadmin\Desktop\Low Impact Develop

MUID	Catchment I	LID Control I	Number of	Area of unit	Width	Initial satura	Include	C
Catch1	Catchment_1	RainBarrel	1	1,000	0,000	0,00	False	
Catch2	Catchment_2	RaiBarrel2	1	1,000	0,000	0,00	False	
Catch3	Catchment_4	Rainbarrel3	1	1,000	0,000	0,00	False	

Figure 4.35 MIKE 1D LID deployment editor

Identification and Connectivity

Each deployed control LID is linked to "LID Deployment MUID". I.e. each deployment represents a LID control deployed on a single catchment in a specified number of units. For each LID deployment, a result file can be generated, containing time series of relevant variables (fluxes and storages). This result file is in the dfs0 time series format.

A full reference of the attributes related to LID deployment in shown in Table 4.14 and Table 4.15.

Table 4.14 The LID Deployment Identification and Connectivity Group

Edit Field	Description	Used or required by simulations	Field name in data structure
MU ID	MIKE URBAN ID	Yes	MUID
Catchment ID	Identification of the catchment	Yes	CatchID
LID Control ID	Identification of the LID control to be deployed	Yes	LidID

The possibility of excluding a control is handled by the "Include" checkbox. By default, this checkbox is un-checked.

LID Deployment Properties

Table 4.15 The LID Deployment Properties

Edit Field	Description	Used or required by simulations	Field name in data structure
Number of units	Number of replicate LID units deployed within the catchment	Yes	ReplicateNumber
Area of unit	The surface area of each replicate LID unit	Yes	UnitArea
Unit area pct	The surface area of each replicate LID unit, expressed as a percentage of the total catchment area	Yes (not if the Area of unit is defined)	UnitAreaPercent
Collecting area	A LID practice is connected to a tributary area whose runoff is treated by the unit. This area includes the LID practice area itself..	Yes	CollectingArea
Collecting area pct	The tributary area connected to the LID unit presented as a percentage of the catchment	Yes (not if the Collecting area is defined)	CollectingAreaPercent
Initial saturation	For Bio-Retention Cells, rain gardens, and green roofs this is the degree to which the unit´s soil is initially filled with water (0% saturation corresponds to the wilting point moisture.	Yes	InitSat
Width	The width of the outflow face of each identical LID unit (meter or feet). This parameter is applied for all control units that use overland flow to transport surface runoff off the unit, being these roofs, pavement, trenches, and swales for the other control types this parameter can be set as 0	Yes (for porous pavement, swales, roofs and trenches)	Width
Result File	Checkbox for creating a result file specific for the current deployment (dfs0 file). A further detailed description of this file is presented in the Result File section	Yes	RptFileNo
File name	The name of an optional file where detailed time series for the LID are presented as dfs0 result file	Yes	RptFileName

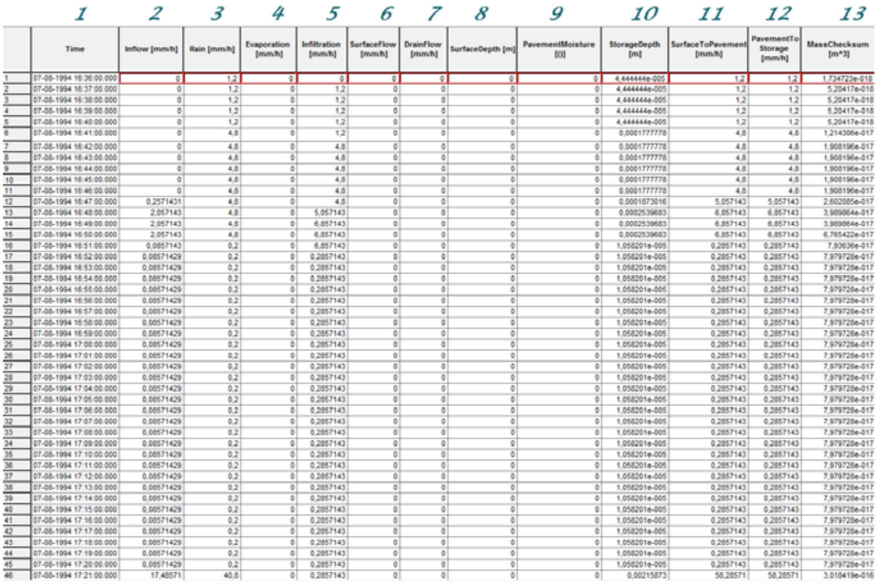
The LID deployment result file

Optionally, for each LID control deployment, a dfs0 time series file can be created. This file includes time series of relevant variables inside the LID control in terms of inflow, flow between layers, storage levels in various layers and output from the control to the native soil.

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

The contents of the dfs0 file depends on the actual LID control type. In Figure 4.36, an example of the dfs0 file for porous pavement is presented. Thirteen columns contain time series for the processes occurring inside the porous pavement.

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



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

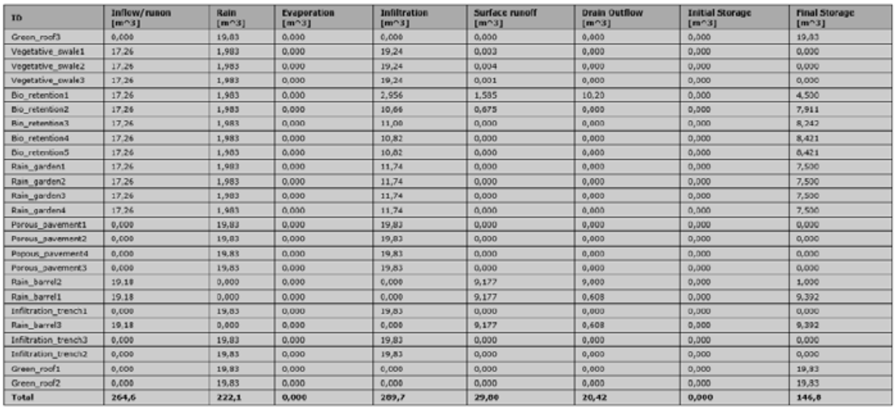
Figure 4.36 Example of a result report dfs0 file per deployment (a porous pavement control)

1. **Time:** calendar time for the simulated time steps..
2. **Inflow (mm/h):** inflow to the LID unit given as a multiple of the rain intensity and the collecting area outside LID/LID area. It is the run-on from the collecting area outside the LID and rain on top of the LID unit. The run on represents the net rain on the impervious collecting area, i.e. with initial losses subtracted.

3. **Rain (mm/h):** input rain for the catchment containing the LID control unit. This rain loads the LID area directly.
4. **Evaporation (mm/h):** this is a given parameter specified by the user by means of boundary conditions valid for the catchment as well as for the LID unit.
5. **Infiltration (mm/h):** infiltration from the LID unit to the surrounding native soil.
6. **Surface flow (mm/h):** surface water flow. Represents overland flow for vegetative swale; overflow for other LID controls
7. **Drain flow (mm/h):** water flow through the drain.
8. **Surface depth (m):** accumulated water on the surface expressed as the water height on the surface. Maximum value is the specified surface storage height.
9. **Soil moisture ():** presented as a fraction, relative to the total volume of the soil layer. Corresponds to the water that is held in the spaces between soil particles. Maximum soil moisture (i.e. full saturation) is equivalent to the specified soil porosity.
10. **Pavement moisture():** presented as a fraction, relative to the total volume of the porous pavement layer. Corresponds to the water that is held in the pavement's voids. Maximum pavement moisture (i.e. full saturation) is equivalent to the specified pavement porosity.
11. **Storage depth (m):** accumulated water in the storage layer expressed as the water height in the storage. Maximum value is the storage height.
12. **Surface to soil (mm/h):** infiltration from the surface layer to the soil layer expressed in mm/h.
13. **Surface to pavement (mm/h):** infiltration from the surface layer to the pavement layer expressed in mm/h.
14. **Soil to storage (mm/h):**): flow from the soil layer to the storage layer expressed in mm/h.
15. **Pavement to storage (mm/h):** flow from the pavement layer to the storage layer mm/h.
16. **MassChecksum (m^3):** regards to the error check in the mass water balance per time step computed, a low value in the order of 1*E-015 indicates an accurate internal flow estimation.

The LID Simulation Summary

The LID simulation summary is provided as part of the overall simulation summary html file. The summary table reports the volume balance for each LID deployment (see example in Figure 4.37).



ID	Inflow/runon [m^3]	Rain [m^3]	Evaporation [m^3]	Infiltration [m^3]	Surface runoff [m^3]	Drain Outflow [m^3]	Initial Storage [m^3]	Final Storage [m^3]
Green_roof3	0,000	19,83	0,000	0,000	0,000	0,000	0,000	19,83
Vegetative_swale1	17,26	1,983	0,000	19,24	0,003	0,000	0,000	0,000
Vegetative_swale2	17,26	1,983	0,000	19,24	0,004	0,000	0,000	0,000
Vegetative_swale3	17,26	1,983	0,000	19,24	0,001	0,000	0,000	0,000
Bio_retention1	17.26	1,983	0,000	2,956	1.585	10.20	0,000	4,500
Bio_retention2	17,26	1,983	0,000	10,66	0,675	0,000	0,000	7,911
Bio_retention3	17,26	1,983	0,000	11,00	0,000	0,000	0,000	8,242
Bio_retention4	17,26	1,983	0,000	10,82	0,000	0,000	0,000	8,421
Bio_retention5	17,26	1,983	0,000	10,82	0,000	0,000	0,000	8,421
Rain_garden1	17,26	1,983	0,000	11,74	0,000	0,000	0,000	7,500
Rain_garden2	17,26	1,983	0,000	11,74	0,000	0,000	0,000	7,500
Rain_garden3	17,26	1,983	0,000	11,74	0,000	0,000	0,000	7,500
Rain_garden4	17,26	1,983	0,000	11,74	0,000	0,000	0,000	7,500
Porous_pavement1	0,000	19,83	0,000	19,83	0,000	0,000	0,000	0,000
Porous_pavement2	0,000	19,83	0,000	19,83	0,000	0,000	0,000	0,000
Popous_pavement4	0,000	19,83	0,000	19,83	0,000	0,000	0,000	0,000
Porous_pavement3	0,000	19,83	0,000	19,83	0,000	0,000	0,000	0,000
Rain_barrel2	19,18	0,000	0,000	0,000	9,177	9,000	0,000	1,000
Rain_barrel1	19.18	0,000	0,000	0,000	9.177	0,608	0,000	9,392
Infiltration_trench1	0,000	19,83	0,000	19,83	0,000	0,000	0,000	0,000
Rain_barrel3	19,18	0,000	0,000	0,000	9,177	0,608	0,000	9,392
Infiltration_trench3	0,000	19,83	0,000	19,83	0,000	0,000	0,000	0,000
Infiltration_trench2	0,000	19,83	0,000	19,83	0,000	0,000	0,000	0,000
Green_roof1	0,000	19,83	0,000	0,000	0,000	0,000	0,000	19,83
Green_roof2	0,000	19,83	0,000	0,000	0,000	0,000	0,000	19,83
Total	**264,6**	**222,1**	**0,000**	**209,7**	**29,80**	**20,42**	**0,000**	**146,8**

Figure 4.37 Example of a LID simulation summary

4 Time Series

Time series can represent both input and result time series. Examples of input time series are e.g. rain event, a varying water level at an outlet etc.

These input time series can be entered in MIKE URBAN through the time series editor. Time series in the time series editor are saved in dfs0 format. In the time series editor it is possible so save several time series with the same time axis in different *time series items*, these various items are then saved in one *time series object*. When using the time series as tidal variations of e.g. a specific load in the simulation, reference to the individual time series items within the time series objects are made.

4.1 Inserting New Time Series

When inserting a new time series (TS) go to Edit|Time Series. This will bring up the dialog shown in Figure 4.1. It is normally not expected that a massive TS data input would be done 'manually' by typing data. If the data are already available in a digital format, they can readily be imported into MIKE URBAN as described later in this section.

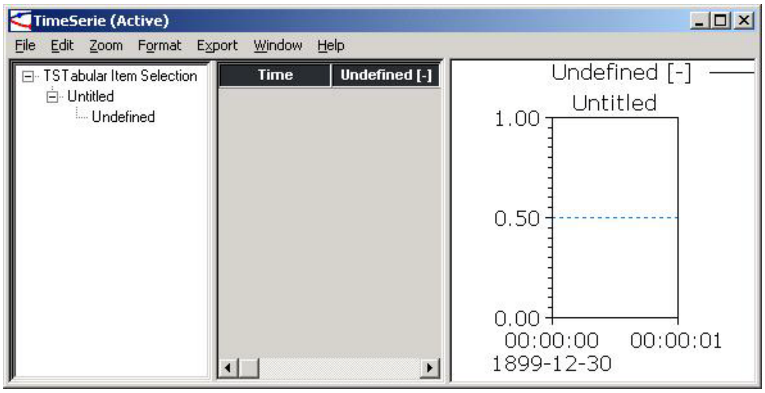

TimeSerie (Active)
File Edit Zoom Format Export Window Help
TSTabular Item Selection
Untitled
Undefined
Time
Undefined [-]
Undefined [-]
Untitled
1.00
0.50
0.00
00:00:00
00:00:01
1899-12-30

Figure 4.1 The empty time series dialog

If the data are to be typed in anyway, the TS data input starts by specifying the basic TS properties: number of values, start date and time, time interval between successive values (assuming a uniform time resolution of the TS) etc. When data are imported it may also be desirable to change some of the properties, please refer to the example later in this section.

In order to set the properties for the time series object:

- Right-click on the TS object

- Choose ‘TSObject Properties’ on the local menu

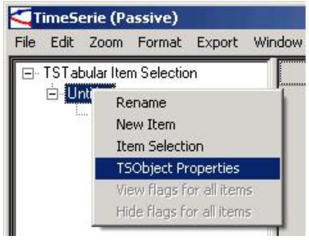

TimeSerie (Passive)
File Edit Zoom Format Export Window
TSTabular Item Selection
Unt
Rename
New Item
Item Selection
TSObject Properties
View flags for all items
Hide flags for all items

Figure 4.2 Right-click on the TS object to get to the local dialog

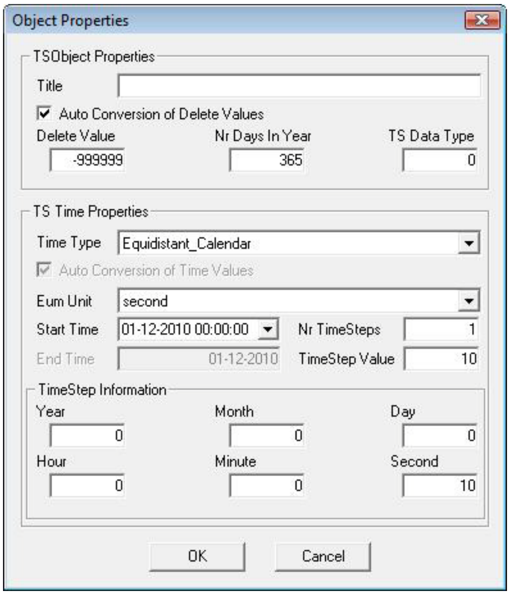
After specifying the properties for the TS object, you can now continue to specify the properties for the TS items. One item is per default already available.

A new TS item is created this way:

- Right-click on the TS object
- Choose ‘New Item’ on the local menu

4.1.1 Properties of time series object

The properties of the time series object can be set in the dialog seen in Figure 4.3. A description of the input fields is found in Table 4.1.



Object Properties

TSObject Properties

Title

☑ Auto Conversion of Delete Values

Delete Value: -999999

Nr Days In Year: 365

TS Data Type: 0

TS Time Properties

Time Type: Equidistant_Calendar

☑ Auto Conversion of Time Values

Eum Unit: second

Start Time: 01-12-2010 00:00:00

Nr TimeSteps: 1

End Time: 01-12-2010

TimeStep Value: 10

TimeStep Information

Year: 0

Month: 0

Day: 0

Hour: 0

Minute: 0

Second: 10

Figure 4.3 The TSObject Properties dialog

Table 4.1 The TSObject properties

Field	Description
Title	Title of the time series object
Auto Conversion of Delete Values	When tick mark is set then the values that are in 'Delete Value' will be converted on the fly if the unit specified in the 'EUM Unit' field is changed. Only conversion between units of the same type can be done. If the tick mark is not set then it is possible to change the unit without the delete value typed in being converted.
Delete Value	Threshold for values that are so small that they should be disregarded.
Nr Days in Year	If a time step of months is used, the number of days is used to compute the number of days in a month. Otherwise not used
TS Data Type	Not relevant

Table 4.1 The TSObject properties

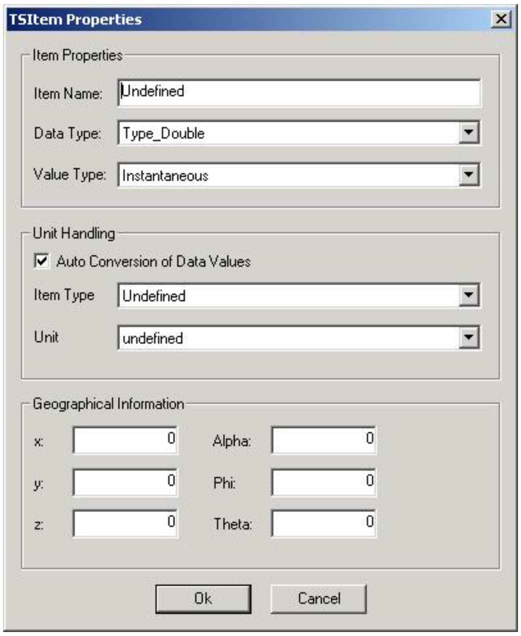
Field	Description
Time Type	A time definition can be equidistant, meaning that the time values difference between consecutive Timesteps is constant through the entire time or non-equidistant, meaning that the time values difference between consecutive Timesteps is not constant through the entire time. Besides these two classifications, a time definition can be Calendar type, meaning that the time values are specified as dates or Relative type, meaning that time values are specified as numbers. **Equidistant_Calendar** - time values difference between consecutive Timesteps is constant through the entire time and time values are specified as dates **Non_Equidistant_Calendar** - time values difference between consecutive Timesteps is not constant through the entire time and time values are specified as dates. **Equidistant_Relative** - time values difference between consecutive Timesteps is constant through the entire time and time values are specified as doubles. **Non_Equidistant_Relative**- time values difference between consecutive Timesteps is not constant through the entire time and time values are specified as doubles. **Undefined_Time** - time type is not specified
Auto Conversion of Time Values	When tick mark is set then the values specified in the timer series will be converted on the fly if the unit specified in the 'EUM Unit' field is changed. Only conversion between units of the same type can be done. If the tick mark is not set then it is possible to change the unit without the delete value typed in being converted.
EUM Unit	The unit applied for the time axis
Start Time	Specifies the start of the time series within the TS object
End Time	Specifies the end of the time series within the TS object
Nr TimeSteps	Specifies the number of time steps within the TS object. This number needs to be larger than 0.

Table 4.1 The TSObject properties

Field	Description
Time Step Value	Specifies the time step value within the TS object. If you specify the time steps in the fields in the ‘TimeStep Information’ group box then the time step value will be computed automatically in the unit specified in the ‘EUM Unit’ field.
Year, Month, Day, Hour, Minute, Second	Specification of time step. If e.g. a time step of 1 day and 1 hour is appropriate then type ‘1’ in the ‘Day’ field and ‘1’ in the ’Hour’ field

4.1.2 Properties of time series item

The properties of the time series item can be set in the dialog seen in Figure 4.4. A description of the input fields is found in Table 4.2.



Item Name: Undefined
Data Type: Type_Double
Value Type: Instantaneous

☑ Auto Conversion of Data Values
Item Type: Undefined
Unit: undefined

x: 0
Alpha: 0
y: 0
Phi: 0
z: 0
Theta: 0

Figure 4.4 The TSItem Properties dialog

Table 4.2 The TSItem properties

Field	Description
Item Name	Name of the item
Data Type	Type_Float: 4 byte floating point number, Type_Double: 8 byte floating point number , Type_Char: 1 byte signed integer char, Type_Int: 4 byte signed integer int, Type_Unsigned: 4 byte unsigned integer, Type_Short: 2 byte signed integer short Type_Unsigned_Short: 2 byte unsigned integer When writing dfs0 files please remember that most of the applications that deal with this file types use always and exclusively Type_Float.

Table 4.2 The TSItem properties

Field	Description
Value Type	**Instantaneous**: means that the values are representative at one precise instant. **Accumulated**: means that the values are representative of one successive accumulation over the time and always relative to the start of the event to register values from. For example, the rainfall accumulated over the year if we have monthly rainfall values. **Step_Accumulated**: means that values are representative of an accumulation over a Timestep. For example, rainfall is a step-accumulated value in the following context. Let’s say that we start measuring rainfall at 10:00:00. At 11:00:00 someone looks at tipping bucket and registers the value of 10 and then empties the tipping bucket. At 12:00:00 the same process but with a value of 15 and so on. So, in a timeseries we shall have the value 10 at Timestep 11:00:00 and the value 15 at Timestep 12:00:00 and so on. Values represent the timespan between the previous Timestep and the current Timestep **Mean_Step_Accumulated**: means that values are representative of an average accumulation per Timestep. Picking the previous example, a mean step accumulated type could be used to register the average rainfall per 15 minutes. Which would be 2.5 (we are working now with 15 minutes and registering in 1 hour Timestep) for 11:00:00 and 3.75 for 12:00:00 and so on. Values represent the timespan between the previous Timestep and the current Timestep **Reverse_Mean_Step_Accumulated**: is equal to Mean_Step_Accumulated type, but values represent the timespan between the current Timestep and the next Timestep. Used for forecasting purposes.

Table 4.2 The TSItem properties

Field	Description
Auto Conversion of Data Values	If tick mark is set then the values that are entered will be converted on the fly if the unit specified in the 'Unit' field is changed. Only conversion between units of the same type can be done. If the tick mark is not set then it is possible to change the unit without the values typed in being converted.
Item Type	The type of time series that is entered. E.g.if you choose Water Level the units in the 'Unit' field will only display units that are relevant for that type of time series. If 'undefined' is chosen all the units displayed in 'Unit' will not be filtered but will display all possible units.
Unit	Displays units relevant for the time series type chosen in 'Unit'
x,y	The x and y co-ordinates used to connect a time series of rainfall, evaporation and temperature to a catchment. If x and y co-ordinates are specified in the "Catchment loads and meterological... " dialog- then these co-ordinates will overwrite the coordinates typed in with the time series.
z, Alpha, Phi, Theta	Not used in MIKE URBAN

4.1.3 Time series plot properties

The properties on the time series plot can be accessed by right-clicking on the plot. The local menu see in Figure 4.5. If you have more than one TS object you need to select the relevant TS object before the TS belonging to this object are displayed on the plot.

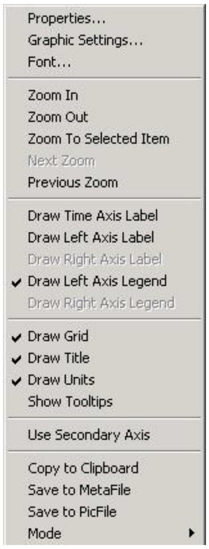

Properties...
Graphic Settings...
Font...
Zoom In
Zoom Out
Zoom To Selected Item
Next Zoom
Previous Zoom
Draw Time Axis Label
Draw Left Axis Label
Draw Right Axis Label
✓ Draw Left Axis Legend
Draw Right Axis Legend
✓ Draw Grid
✓ Draw Title
✓ Draw Units
Show Tooltips
Use Secondary Axis
Copy to Clipboard
Save to MetaFile
Save to PicFile
Mode

Figure 4.5 Local menu on the TS plot

Properties

Allows to change properties on the x-axis, y-axis and text annotations, e.g. display and placement of legends.

Graphic Settings

Allows to change the display of the graphs, e.g. line thickness, line colour, line type (solid, dotted etc.).

Font

Allows to change the font used for the title, legends and the axis.

Zoom in, Zoom Out, Zoom to Selected Item, Next Zoom, Previous Zoom

Allows to access various zoom functionality.

Draw Time Axis Label, Draw Left Axis Label, Draw Right Axis Label, Draw Left Legend Label, Draw Right Legend Label

Controls the settings of the axis and legends related to the axis.

Draw Grid, Draw Title, Draw Units, Show Tooltips

Controls the display of the grid, title, units and tooltips.

Use Secondary Axis

Toggles the secondary axis on/off.

Copy to Clipboard, Save to MetaFile, Save to PicFile

Allows to copy the plot to the clipboard as well as saving to either MetaFile or PicFile.

Mode

Delete Points

When this mode is selected, when the mouse pointer is near a data value, the pointer becomes a rubber and clicking on the left button of the mouse, deletes the data value. The data value is not really deleted from the data set but its value is set to delete value.

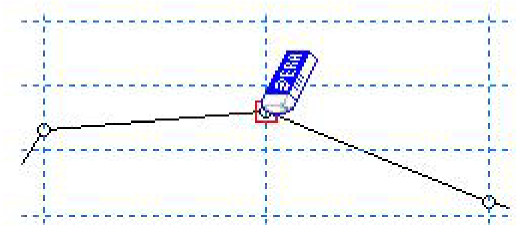


Figure 4.6 Delete points mode

Move Points

When the mouse pointer is near a data value, the mouse pointer becomes a moving cross and you can move the data value by moving the mouse while keeping the left button of the mouse down (drag). Before the data value drag begins, the data value is selected. When moving data values, if the selected item is also being shown in a Timeseries Tabular, the correspondent cell value in Timeseries Tabular changes accordingly.

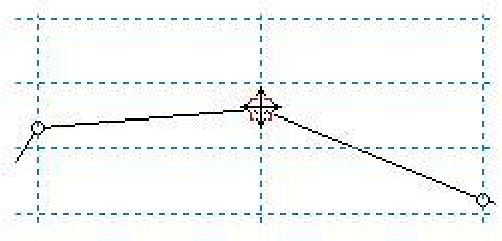


Figure 4.7 Move points mode

Select Points

When clicking in a data value that data value is selected. The current Timestep is set to the selected data value timestep. A red square around the data value appears and the correspondent cell in the Timeseries Tabular (if the selected item is also being shown in a Timeseries Tabular) is selected

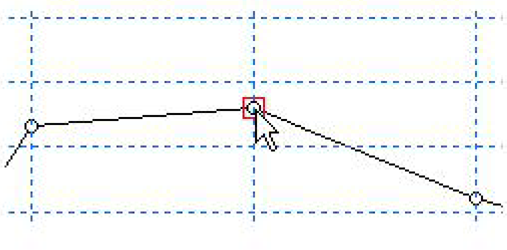


Figure 4.8 Select points mode

Set Bookmarks

When the mouse pointer is near a data value not bookmarked, the pointer becomes a pushpin. Clicking on the left button of the mouse bookmarks the data value.

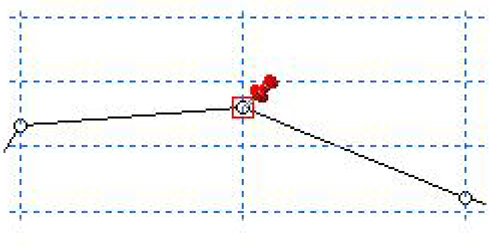


Figure 4.9 Set bookmarks mode

Clear Bookmarks

When the mouse pointer is near a bookmarked data value, the pointer becomes a rubber. Clicking on the left button of the mouse removes the bookmark for the data value

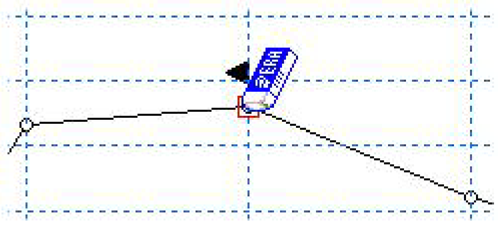


Figure 4.10 Clear bookmarks mode

Add Lines, Delete Lines, Move Lines

Allows to add, delete and move a vertical line. If the TS is a result time series the placement of the line is synchronized with the horizontal and longitudinal profile plot, e.g. if you place the line at a specific time, the horizontal plot (if it displays results layers) will display the results for the same time.

4.2 Example: How to enter a rain time series

So the procedure is:

1. Go to Edit|Time Series
2. Right-click on the ‘Untitled’ and choose TSObject Properties, see Figure 4.11

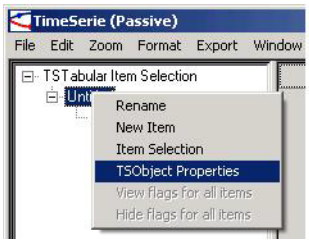
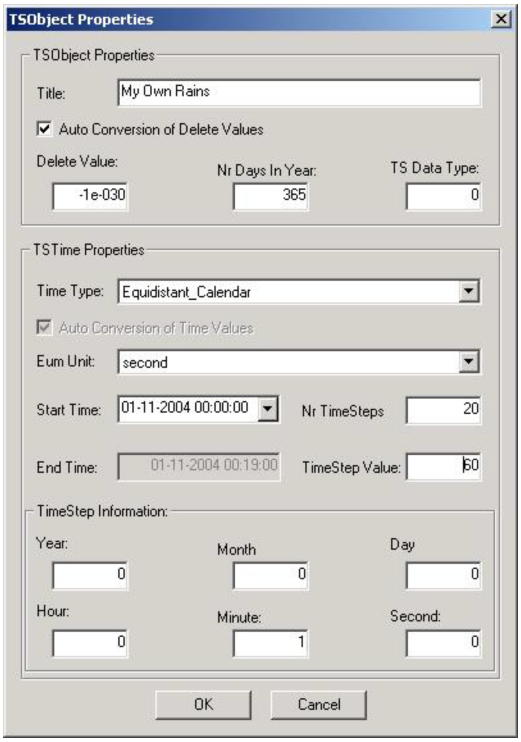

TimeSerie (Passive)
File Edit Zoom Format Export Window
TSTabular Item Selection
Unti
Rename
New Item
Item Selection
TSObject Properties
View flags for all items
Hide flags for all items

Figure 4.11 Right-Click to set the properties for the time series

3. On the dialog that comes up your are able to specify the various properties. The properties for this specific example are seen in Figure 4.12.

In this example we are specifying that the rain starts at the 1st of November 2004, that we would like 20 time steps that are a minute apart.


TSObject Properties
OK
Cancel

TSObject Properties

Title: My Own Rains

☑ Auto Conversion of Delete Values

Delete Value: -1e-030

Nr Days In Year: 365

TS Data Type: 0

TSTime Properties

Time Type: Equidistant_Calendar

☑ Auto Conversion of Time Values

Eum Unit: second

Start Time: 01-11-2004 00:00:00

Nr TimeSteps: 20

End Time: 01-11-2004 00:19:00

TimeStep Value: 60

TimeStep Information:

Year: 0

Month: 0

Day: 0

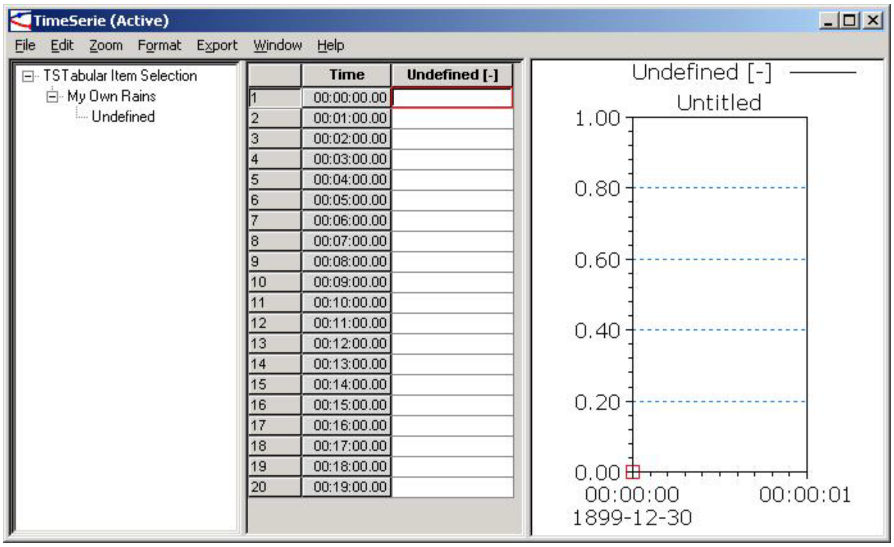
Hour: 0

Minute: 1

Second: 0

Figure 4.12 Example of TSbject Properties

4. And then it will look like seen in Figure 4.13


TimeSerie (Active)
File
Edit
Zoom
Format
Export
Window
Help
Undefined [-]
Untitled
1.00
0.80
0.60
0.40
0.20
0.00
00:00:00
1899-12-30
00:00:01

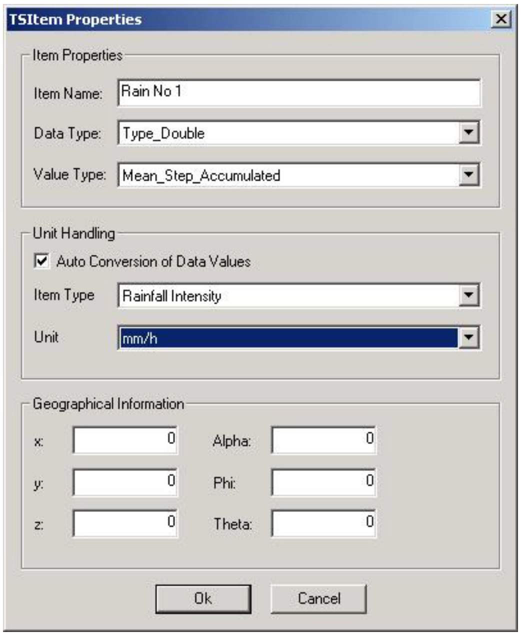
TSTabular Item Selection
- My Own Rains
  - Undefined

	Time	Undefined [-]
1	00:00:00.00	
2	00:01:00.00	
3	00:02:00.00	
4	00:03:00.00	
5	00:04:00.00	
6	00:05:00.00	
7	00:06:00.00	
8	00:07:00.00	
9	00:08:00.00	
10	00:09:00.00	
11	00:10:00.00	
12	00:11:00.00	
13	00:12:00.00	
14	00:13:00.00	
15	00:14:00.00	
16	00:15:00.00	
17	00:16:00.00	
18	00:17:00.00	
19	00:18:00.00	
20	00:19:00.00	

Figure 4.13 Example of how it looks after specifying the TSObject Properties

Now right-click on the TS item (per default named ‘Undefined’ before renaming) and specify the properties for the TS item. This is done as shown in Figure 4.14.

Note how the type ‘Value Type’ is set to ‘mean_step_accumulated’. This is done as time series of rain are always plotted as histograms. The unit in this example is chosen to be ‘mm/h’, but could equally be any of the other units applicable for rainfall intensities.



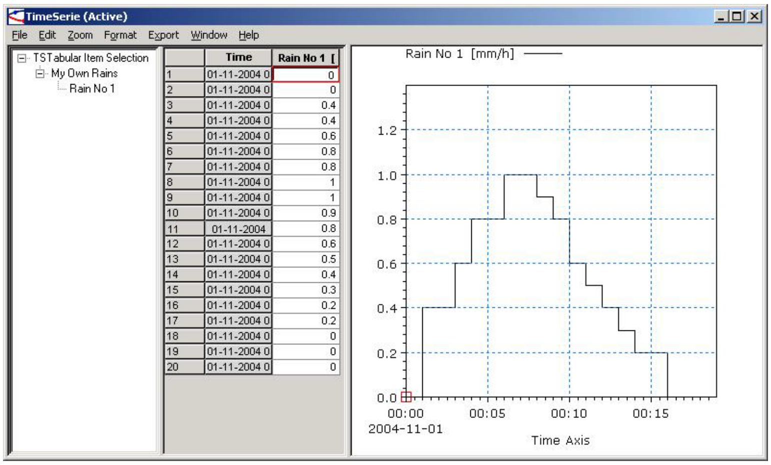
Item Properties
Item Name: Rain No 1
Data Type: Type_Double
Value Type: Mean_Step_Accumulated

Unit Handling
☑ Auto Conversion of Data Values
Item Type: Rainfall Intensity
Unit: mm/h

Geographical Information
x: 0
y: 0
z: 0
Alpha: 0
Phi: 0
Theta: 0

Figure 4.14 Example of TSItem properties

5. After the specification of the TSItem properties, you can start to type in the various values. A result of this is seen in Figure 4.15

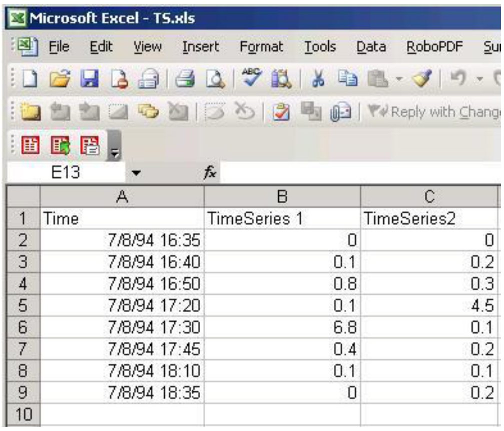

TimeSerie (Active)
File
Edit
Zoom
Format
Export
Window
Help
TSTabular Item Selection
My Own Rains
Rain No 1
Rain No 1 [mm/h]
1.2
1.0
0.8
0.6
0.4
0.2
0.0
00:00
2004-11-01
00:05
00:10
00:15
Time Axis

	Time	Rain No 1 [
1	01-11-2004 0	0
2	01-11-2004 0	0
3	01-11-2004 0	0.4
4	01-11-2004 0	0.4
5	01-11-2004 0	0.6
6	01-11-2004 0	0.8
7	01-11-2004 0	0.8
8	01-11-2004 0	1
9	01-11-2004 0	1
10	01-11-2004 0	0.9
11	01-11-2004	0.8
12	01-11-2004 0	0.6
13	01-11-2004 0	0.5
14	01-11-2004 0	0.4
15	01-11-2004 0	0.3
16	01-11-2004 0	0.2
17	01-11-2004 0	0.2
18	01-11-2004 0	0
19	01-11-2004 0	0
20	01-11-2004 0	0

Figure 4.15 Example of how the rainfall is displayed after typing in the values

4.3 Example: How to Import a Time Series from Excel

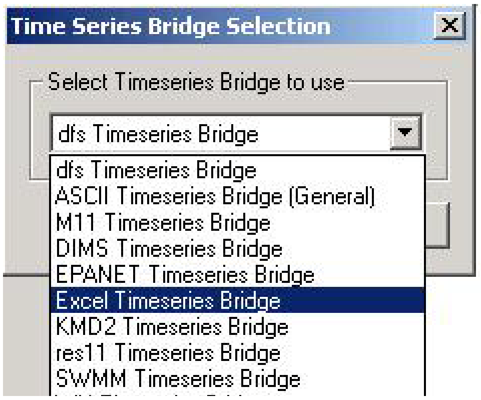
Assume that you have two time series in Excel that look as seen in Figure 4.16.


Microsoft Excel - TS.xls
File
Edit
View
Insert
Format
Tools
Data
RoboPDF
Su
Reply with Chang
E13
fx

	A	B	C
1	Time	TimeSeries 1	TimeSeries2
2	7/8/94 16:35	0	0
3	7/8/94 16:40	0.1	0.2
4	7/8/94 16:50	0.8	0.3
5	7/8/94 17:20	0.1	4.5
6	7/8/94 17:30	6.8	0.1
7	7/8/94 17:45	0.4	0.2
8	7/8/94 18:10	0.1	0.1
9	7/8/94 18:35	0	0.2
10			

Figure 4.16 The TS used in the example

1. Go to Edit|Time Series
2. Choose File|Open
3. On the dialog that comes up choose between available bridges for time series, see e.g. Figure 4.17



Time Series Bridge Selection

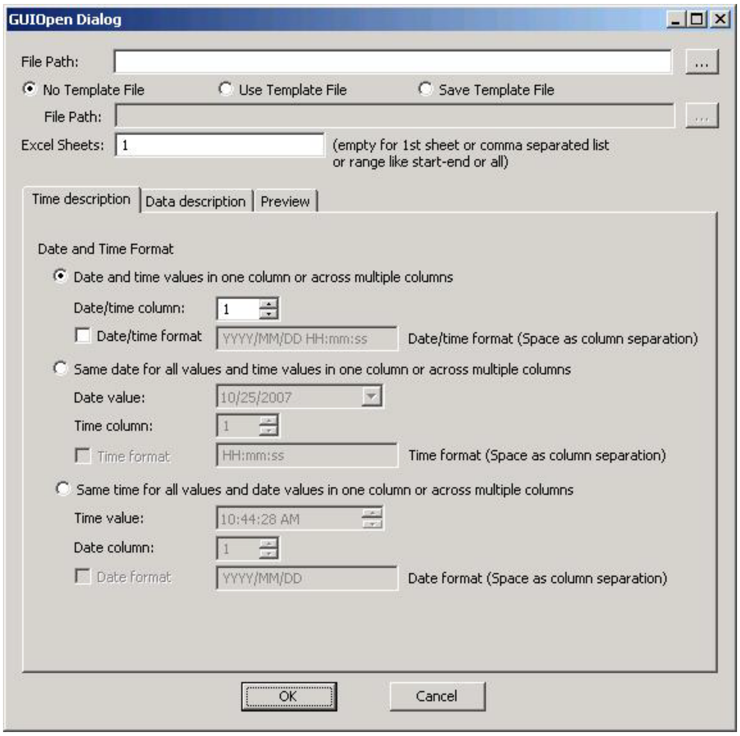
Select Timeseries Bridge to use

dfs Timeseries Bridge

- dfs Timeseries Bridge
- ASCII Timeseries Bridge (General)
- M11 Timeseries Bridge
- DIMS Timeseries Bridge
- EPANET Timeseries Bridge
- Excel Timeseries Bridge
- KMD2 Timeseries Bridge
- res11 Timeseries Bridge
- SWMM Timeseries Bridge

Figure 4.17 Time Series Bridge Selection dialog

4. Choose the ‘Excel Timeseries Bridge’ and press ‘OK’
5. The dialog seen in Figure 4.18 appears



GUIOpen Dialog

File Path:

● No Template File ○ Use Template File ○ Save Template File

File Path:

Excel Sheets: 1 (empty for 1st sheet or comma separated list or range like start-end or all)

Time description | Data description | Preview

Date and Time Format

● Date and time values in one column or across multiple columns

Date/time column: 1

☐ Date/time format YYYY/MM/DD HH:mm:ss Date/time format (Space as column separation)

○ Same date for all values and time values in one column or across multiple columns

Date value: 10/25/2007

Time column: 1

☐ Time format HH:mm:ss Time format (Space as column separation)

○ Same time for all values and date values in one column or across multiple columns

Time value: 10:44:28 AM

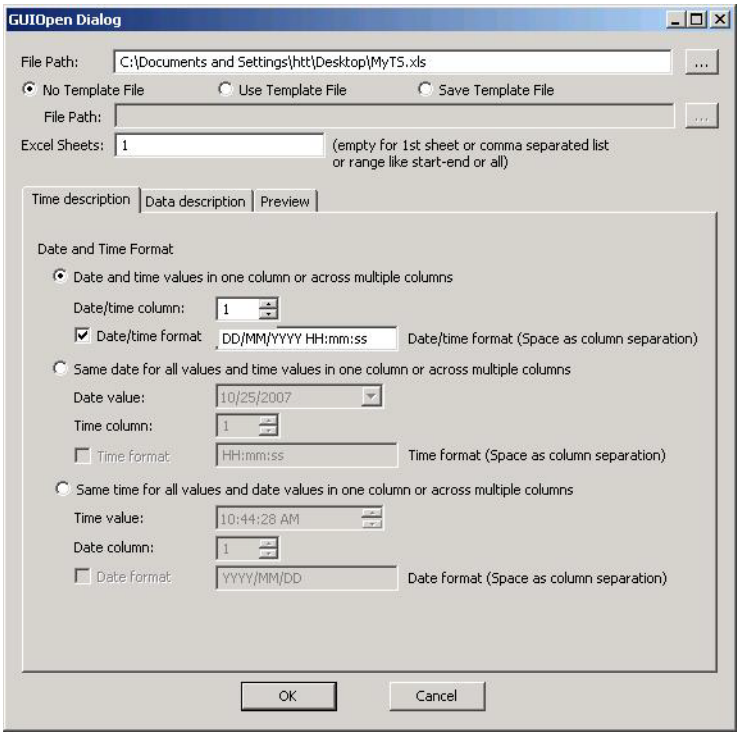
Date column: 1

☐ Date format YYYY/MM/DD Date format (Space as column separation)

Figure 4.18 Dialog for specifying the import from Excel and ASCII files

6. Now you need to specify how data should be interpreted, e.g. is data a rainfall in mm - or an intensity. First, choose the file and then specify how time should be interpreted. In this case the time and date column is

found in one column. Hence the time description tab page will look as seen in Figure 4.19.



GUIOpen Dialog

File Path: C:\Documents and Settings\htt\Desktop\MyTS.xls

● No Template File ○ Use Template File ○ Save Template File

File Path:

Excel Sheets: 1 (empty for 1st sheet or comma separated list or range like start-end or all)

Date and Time Format

● Date and time values in one column or across multiple columns

Date/time column: 1

☑ Date/time format DD/MM/YYYY HH:mm:ss Date/time format (Space as column separation)

○ Same date for all values and time values in one column or across multiple columns

Date value: 10/25/2007

Time column: 1

☐ Time format HH:mm:ss Time format (Space as column separation)

○ Same time for all values and date values in one column or across multiple columns

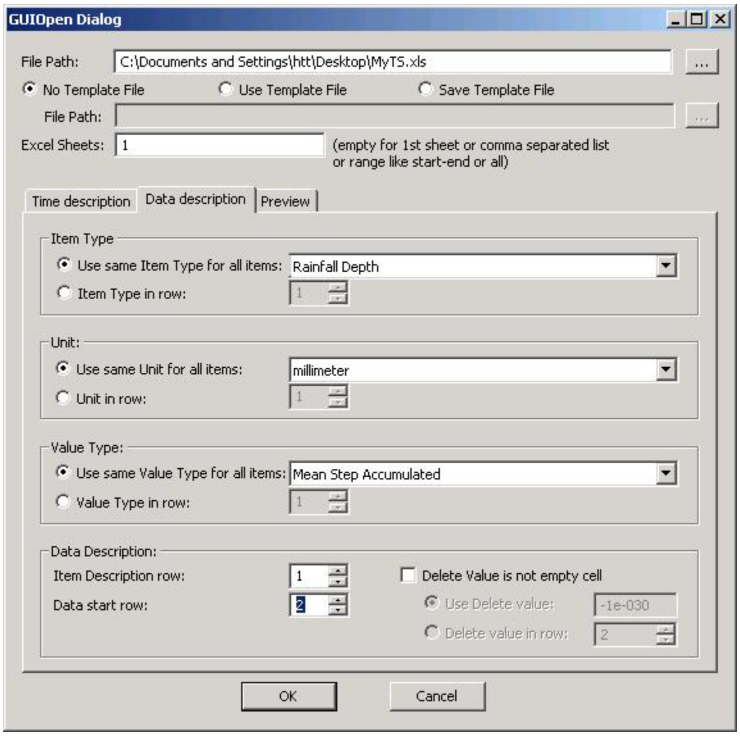
Time value: 10:44:28 AM

Date column: 1

☐ Date format YYYY/MM/DD Date format (Space as column separation)

Figure 4.19 The time description for the excel file in this example

7. In this example both time series are rainfall depth measured in mm. The values are mean step accumulated values (see previous section). As the first row in the Excel sheet is used for descriptive headings, data will firs start in row two. All this is then specified on the data description tab page, see Figure 4.20.


GUIOpen Dialog
Time description
Data description
Preview
OK
Cancel

File Path: C:\Documents and Settings\htt\Desktop\MyTS.xls

● No Template File ○ Use Template File ○ Save Template File

File Path:

Excel Sheets: 1 (empty for 1st sheet or comma separated list or range like start-end or all)

Item Type

● Use same Item Type for all items: Rainfall Depth

○ Item Type in row: 1

Unit:

● Use same Unit for all items: millimeter

○ Unit in row: 1

Value Type:

● Use same Value Type for all items: Mean Step Accumulated

○ Value Type in row: 1

Data Description:

Item Description row: 1

Data start row: 2

☐ Delete Value is not empty cell

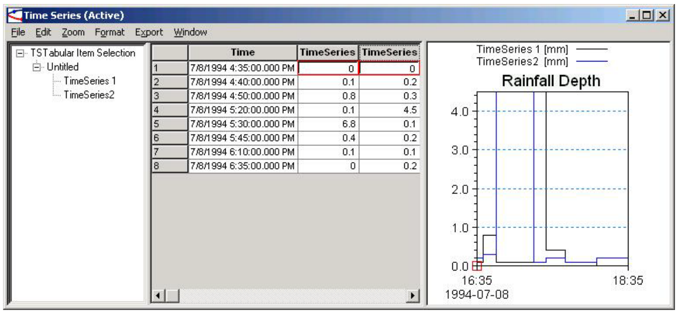
● Use Delete value: -1e-030

○ Delete value in row: 2

Figure 4.20 The data description for the excel file in this example

8. Then go to the Preview tab page. Here you can see how your time series is interpreted before you import the data into MIKE URBAN. If an error has been made on the previous pages, e.g. you have specified that data will start in row one instead of row two you will now receive an error message and data cannot be previewed. You can now choose to either import the data straight ahead or to save a template file first that you can reuse when importing data of the same format.

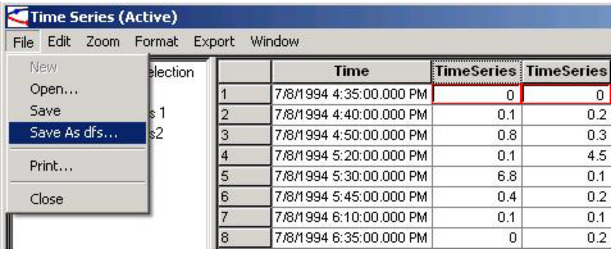
9. The result of the import then looks like seen in Figure 4.21.


Time Series (Active)
File
Edit
Zoom
Format
Export
Window
TSTabular Item Selection
Untitled
TimeSeries 1
TimeSeries2
TimeSeries 1 [mm]
TimeSeries2 [mm]
Rainfall Depth
4.0
3.0
2.0
1.0
0.0
16:35
18:35
1994-07-08

	Time	TimeSeries	TimeSeries
1	7/8/1994 4:35:00.000 PM	0	0
2	7/8/1994 4:40:00.000 PM	0.1	0.2
3	7/8/1994 4:50:00.000 PM	0.8	0.3
4	7/8/1994 5:20:00.000 PM	0.1	4.5
5	7/8/1994 5:30:00.000 PM	6.8	0.1
6	7/8/1994 5:45:00.000 PM	0.4	0.2
7	7/8/1994 6:10:00.000 PM	0.1	0.1
8	7/8/1994 6:35:00.000 PM	0	0.2

Figure 4.21 The TS after import

10. The save the data in the native storage format used in MIKE URBAN for time series, i.e. the dfs0 format. This is done from the file menu of the time series dialog. See Figure 4.22.


Time Series (Active)
File
Edit
Zoom
Format
Export
Window
New
Open...
Save
Save As dfs...
Print...
Close
election
s 1
s2

	Time	TimeSeries	TimeSeries
1	7/8/1994 4:35:00.000 PM	0	0
2	7/8/1994 4:40:00.000 PM	0.1	0.2
3	7/8/1994 4:50:00.000 PM	0.8	0.3
4	7/8/1994 5:20:00.000 PM	0.1	4.5
5	7/8/1994 5:30:00.000 PM	6.8	0.1
6	7/8/1994 5:45:00.000 PM	0.4	0.2
7	7/8/1994 6:10:00.000 PM	0.1	0.1
8	7/8/1994 6:35:00.000 PM	0	0.2

Figure 4.22 Saving data into the dfs0 format

You can change the properties for the display of the time series. See Figure 4.23.

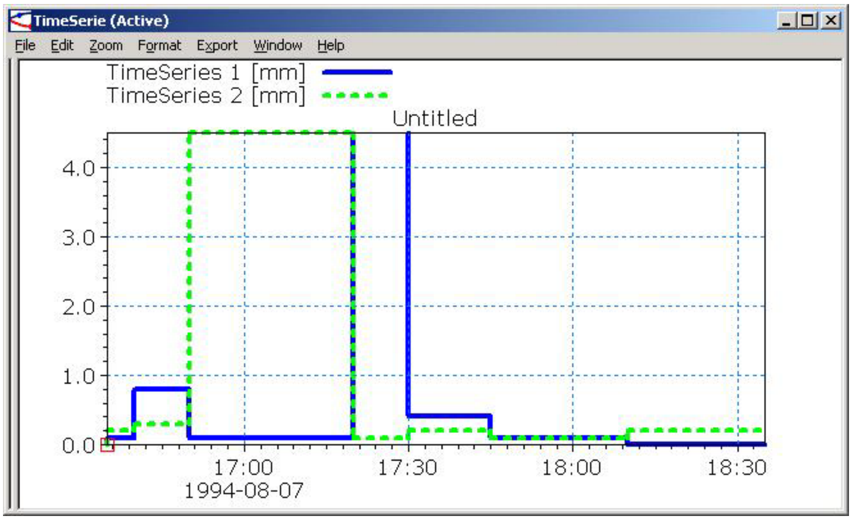

TimeSerie (Active)
File
Edit
Zoom
Format
Export
Window
Help
TimeSeries 1 [mm]
TimeSeries 2 [mm]
Untitled
4.0
3.0
2.0
1.0
0.0
17:00
1994-08-07
17:30
18:00
18:30

Figure 4.23 In the plot the graphic settings have also been changed to make it easier to see the difference.

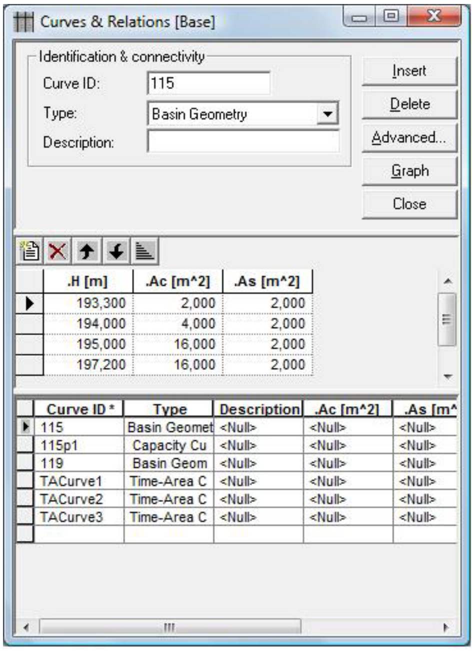
5 Curves and Relations

5.1 Introduction

In MOUSE|Curves and Relations (Figure 5.1) a number of tabular data used in other data dialogs are specified.

There are a number of different types of tabular data:

- Capacity Curve QH (used for Pumps)
- Capacity Curve QdH (used for Pumps)
- Pump Acceleration Curve (used for Pumps)
- Regulation Qmax(H)
- Regulation Qmax(dH)
- QH Relation (when specifying a QH relation for a node)
- Time-Area Curve (used in Time Area runoff model)
- Removal Efficiency (used for the efficiency curve for removal weirs)
- Basin Geometry (used for Basins)
- Valve Rating Curve (used for Valves)
- DQ Relation (used for Curb Inlets)
- QQ Relation (used for Curb Inlets)
- Capacity Curve QdH & Power
- Undefined (used for surface runoff pollutants - SRC, FRC and TOT dataset)



Curve ID: 115
Type: Basin Geometry
Description:

.H [m]	.Ac [m^2]	.As [m^2]
193,300	2,000	2,000
194,000	4,000	2,000
195,000	16,000	2,000
197,200	16,000	2,000

Curve ID *	Type	Description	.Ac [m^2]	.As [m^
115	Basin Geomet	<Null>	<Null>	<Null>
115p1	Capacity Cu	<Null>	<Null>	<Null>
119	Basin Geom	<Null>	<Null>	<Null>
TACurve1	Time-Area C	<Null>	<Null>	<Null>
TACurve2	Time-Area C	<Null>	<Null>	<Null>
TACurve3	Time-Area C	<Null>	<Null>	<Null>

Figure 5.1 Curves & Relations dialog

5.1.1 Capacity curves

It is possible to define two different capacity curves - both used for specifying pump operation. The capacity curve can be specified either as a 'Capacity Curve QH' relation (for screw pumps) or as 'Capacity Curve QdH' relation (for differential head pumps), where 'H' is the absolute water level in the pump's wet well (at 'Location'), and 'dH' is the water level difference between the 'To' and the 'Location' nodes. A pump type with a 'Capacity Curve QH' relation is named a screw pump, while a pump type with a 'Capacity Curve Q-dH' relation is named a differential head pump.

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

Please also note that you can specify a pump capacity curve with the energy consumption as well - for more on this type please refer to section 5.1.10.

5.1.2 Pump acceleration curves

Pumps can be RTC controlled. For those that are PID controlled, the acceleration of a pump can be specified as dependent on the actual flow. This pump acceleration curve is then specified as a number of ‘Q, dQ/dt’ data sets.

5.1.3 Regulation curves Qmax(H) and Qmax(dH)

The regulation curves Qmax(H) and Qmax(dH) are used in the regulation of the maximum discharge in links. The regulation can be either a maximum discharge as a function of the water level in user specified node or a maximum discharge as a function of the water level difference between two user specified nodes. Please refer to section 3.4.43.4 for regulation in links.

5.1.4 QH relation

QH relations can be used for various types of manholes, outlets, basins and storage nodes.

Manholes, Basins

When using a QH relation in either a manhole or a basin it is possible to specify an infiltration (Q) based on the water level in the manhole/basin.

Outlets

Using a QH relation in an outlet means that you specify the discharge out of the outlet based on the water level in the outlet.

Storage node

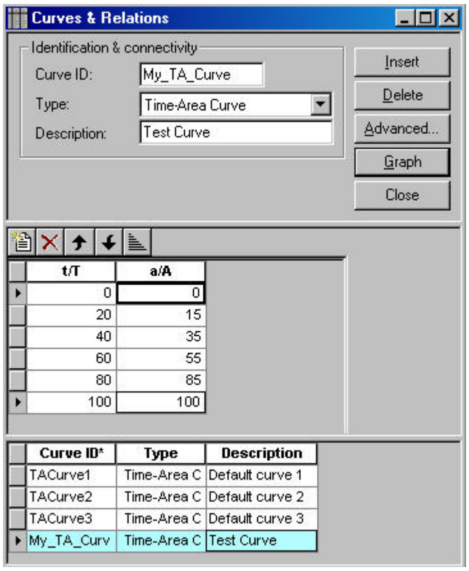
Return of water from storage basins back into the MOUSE network is specified in the dialog "Emptying Storage Nodes". An emptying function is a QH relation, where 'Q' is the flow from the storage node into a 'receiving node', and 'H' is the water level in a 'control node'. The flow according to the emptying function is possible only until the storage node is completely empty.

5.1.5 Time-Area curve

The Time-Area curve is used in the Time-Area runoff model (see section 4.2). A Time-Area curve represents the contributing part of the catchment surface as function of time.

MIKE URBAN comes with three defaults time-area curves TACurve1, TACurve2 and TACurve3, applicable for rectangular, divergent and convergent catchment, respectively (see more in 5.1.5).

You can define any number of custom time-area curves. Each Time-area table must start with a pair of values (0,0) and must end with a pair of values representing the whole (per default, MIKE URBAN maintains T-A curves in percent (%), i.e. the last pair of values in the table must be (100,100)).



Curve ID: My_TA_Curve
Type: Time-Area Curve
Description: Test Curve

t/T	a/A
0	0
20	15
40	35
60	55
80	85
100	100

Curve ID*	Type	Description
TACurve1	Time-Area C	Default curve 1
TACurve2	Time-Area C	Default curve 2
TACurve3	Time-Area C	Default curve 3
My_TA_Curv	Time-Area C	Test Curve

Figure 5.2 Example of user defined Time-Area curve

5.1.6 Removal efficiency

There are three methods available for removal of sediments in weirs (see also section 12.6). In one of these methods you specify the relation between discharge towards the weir and the effiency, i.e. the efficiency curve. The removal efficency is hence a function of Q and the efficiency (unit less 1/1).

5.1.7 Basin geometry

The basin geometry is described as a tabulated area-elevation function. You specify a set of (H, A_c, A_s) values. The A_c is the cross section area perpendicular to the main flow direction in the basin (used to calculate the velocity). The A_s is the surface area of the basin (used to calculate the volume). Both parameters are specified as functions of the water level, H, in the basin.

The H-column for the basin geometry can start at any value, e.g. 0.0 for interpretation of H as depth in the basin. The MOUSE Engine will associate the first H-value to the bottom level of the node. This means that the same geometry can be reused several places in the model. If we let H_{max} indicate the maximum level before flooding the highest value of the top of the specified basin or the ground level is applied. Therefore if the top of the basin geometry is below the ground level the specified basin geometry will be extended with additional points to allow for flooding.

5.1.8 Valve rating curve

A valve is a functional relation which connects two nodes of a MOUSE network.

A rating curve is specified to define the relation between the valve opening (%) and resistance (k).

5.1.9 DQ and QQ relations

Two curve types can be specified for the two different types of Curb Inlets (see 3.7):

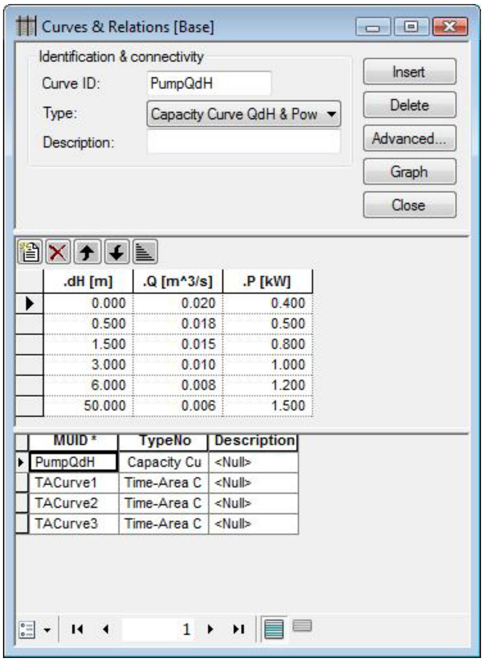
- Capacity Curve, DQ (depth/discharge relation specified in the Curb Inlets dialog)
- Capacity Curve, QQ ($Q_{approach,}Q_{capture}$ relation specified in the On-grade capture dialog).

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

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

5.1.10 Capacity curve QdH & Power

If you have the knowledge of the power consumption of the pump at various pump levels it is possible to add this to the pump capacity curve by using the ‘QdH & Power’ capacity curve (see Figure 5.3).



Curve ID: PumpQdH

Type: Capacity Curve QdH & Pow

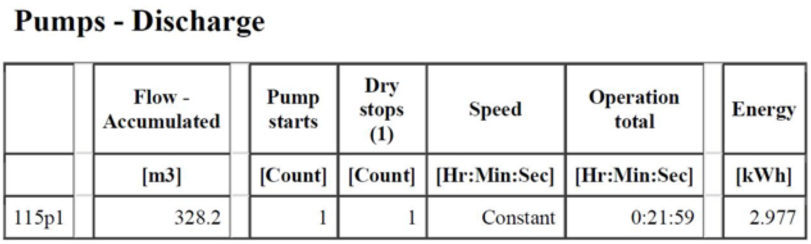
Description:

.dH [m]	.Q [m^3/s]	.P [kW]
0.000	0.020	0.400
0.500	0.018	0.500
1.500	0.015	0.800
3.000	0.010	1.000
6.000	0.008	1.200
50.000	0.006	1.500

MUID *	TypeNo	Description
PumpQdH	Capacity Cu	<Null>
TACurve1	Time-Area C	<Null>
TACurve2	Time-Area C	<Null>
TACurve3	Time-Area C	<Null>

Figure 5.3 Pump capacity curve including power consumption

After the simulation with a ‘QdH & Power’ the summary will contain information on the power consumption during the simulation period.



Pumps - Discharge

	Flow - Accumulated	Pump starts	Dry stops (1)	Speed	Operation total	Energy
	[m3]	[Count]	[Count]	[Hr:Min:Sec]	[Hr:Min:Sec]	[kWh]
115p1	328.2	1	1	Constant	0:21:59	2.977

Figure 5.4 Snapshot from the html-summary with example of output when using the QdH & Power capacity curve

5.1.11 Undefined type

The undefined type is used for surface runoff pollutants - SRC, FRC and TOT dataset.

The concentrations of a given pollutant can be given for each specific catchment. For one catchment it is possible to give a concentration on the slow runoff component (SRC) and the fast runoff component (FRC) or override these settings stating a concentration for the total runoff (TOT). Instead of fixed concentrations it is possible to state a name of a tabular data set (the undefined type). The tabular data set should then hold a table with concentrations versus normalised flow, i.e. the runoff divided by the total catchment area. See more in section 12.7.

6 Boundary Conditions

In broad terms, a model boundary condition can be defined as an external interference, which forces the behaviour of the computed variables within the model domain.

A great part of the boundary conditions represent various types of water loads (rainfall, infiltration, wastewater…). The main characteristic of these "load" boundary conditions is that they contain a "transport medium" - water. Water can transport other material (dissolved pollutants, oxygen, sediments), heat (temperature) and organisms (bacteria and para-sites), as well as various other properties, such as pH, conductivity, turbidity, etc. Properties of "water load" boundary conditions (expressed as concentrations, mass flow, temperature, etc.) are also, in strict terms, boundary conditions for e.g. pollution transport model (Advection-Dispersion). However, these are not treated as separate boundary conditions, but items inseparably associated with the water loads. E.g. a lateral inflow is a water load boundary condition for the hydrodynamic process in the network (defined as discharge item), while the temperature, concentration, etc. associated to this particular inflow are specified as pollutant items of that “load” boundary condition. Effectively, pollutant items of that “load” boundary conditions to the AD process in the network.

Boundary conditions other than water loads are "simple" in a sense that they are fully defined by the boundary condition variable itself, i.e. no additional item is possible to be associated with them. These are water levels, air temperature, evapo-transpiration, etc.

In the following the terminology used is:

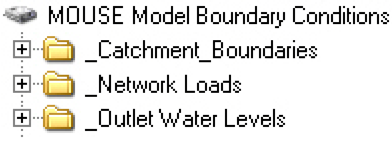
- **Boundary condition**: it is defined by its type and model connectivity MOUSE distinguishes three groups of boundary conditions:
- Catchment Loads and Meteorological Boundary Conditions
- Network Loads
- External Water Levels
- **Boundary item**: per definition, it is a property inseparably associated with a boundary condition. A boundary item is fully defined by its type, temporal variation and quantity/quality. Through the link to a boundary condition to which it belongs, connection of a boundary item to a model is ensured.

The variable which defines a boundary condition is a default item of the boundary condition.

The structure of boundary conditions and relations between boundary conditions and associated items can best be illustrated by a tree-like structure, similar to archive directories and files.

On the top level, there are three groups of boundary conditions (see figure….below). Each of these three groups is served by a separate editor, described in full detail further below.

Depending on the model, individual boundary conditions may be added.



- MOUSE Model Boundary Conditions
  - _Catchment_Boundaries
  - _Network Loads
  - _Outlet Water Levels

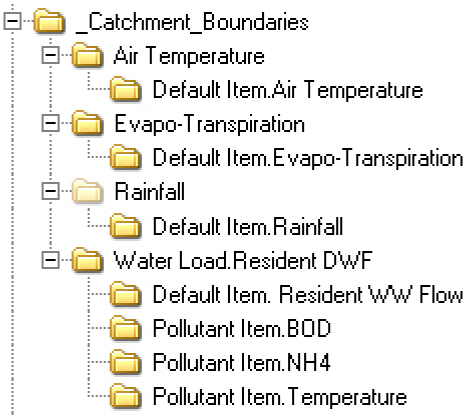
Figure 6.1 Basic frame of MOUSE Boundary Conditions with three groups

6.1 Catchment Boundary Condition

Types of variables associated with catchment boundary conditions are:

- Air temperature
- Evapo-transpiration
- Rainfall
- Flow - including WQ, sediment and temperature properties

An example illustrating the catchment boundary conditions is presented in below figure



- _Catchment_Boundaries
  - Air Temperature
    - Default Item.Air Temperature
  - Evapo-Transpiration
    - Default Item.Evapo-Transpiration
  - Rainfall
    - Default Item.Rainfall
  - Water Load.Resident DWF
    - Default Item. Resident WW Flow
    - Pollutant Item.BOD
    - Pollutant Item.NH4
    - Pollutant Item.Temperature

Figure 6.2 The Catchment Boundary Conditions - Example

Any number of each boundary conditions types can be specified.

In this example, the model includes four catchment boundary conditions:

"Air Temperature" is a boundary condition of the type "air temperature", and always has a temperature (constant or a time series) associated as a default item. This type of boundary condition is used for snow melt calculations.

"Evapo-Transpiration" is a boundary condition of the type "evapo-transpiration ", and always has potential evapo-transpiration (constant or a time series) associated as a default item. This type of boundary is used for RDI hydrological calculations.The dfs0 file needs to be of the type ‘Evapo-Transpiration’. Delete values will be considered equal to 0.0.

"Rainfall" is a boundary condition of the type "rainfall", and always has rainfall (constant or a time series) associated as a default item. This type of boundary is used for precipitation-runoff hydrological calculations. Association of WQ properties (e.g. temperature, acidity, etc.) with rainfall is not supported. When using a time series the dfs0-file needs to be one of the following Item Types: ‘Rainfall Intensity’, ‘Rainfall’ or ‘Rainfall Depth’. Delete values will be considered equal to 0.0.

"Water Load.Resident DWF" is a boundary condition of the type "catchment load", and always has discharge (constant, cyclic or a time series) associated as a default item. This type of boundary represents various kinds of hydraulic loads, such as area-based or PE-based dry weather loads, area-based infiltration, etc. The discharge can be associated with any pollutant, sediment or temperature item (constant, cyclic or a time series). In the example above, the default item (Resident WW flow) is associated with pollutants BOD, NH4 and Temperature.

Note that the first three boundary conditions are used by a runoff-precipitation model, while the water load boundary condition can either be appended to the computed runoff (only flow) or passed directly to the network connected to the catchments.

6.1.1 Visualization of catchment boundaries

The catchment loads are visualized by toogling the display of ‘MOUSE Catchment Boundaries’ on in the TOC.

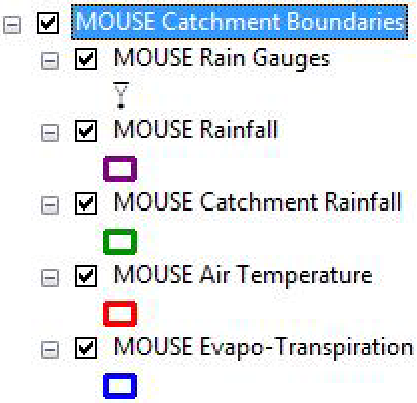

☑ MOUSE Catchment Boundaries
☑ MOUSE Rain Gauges
☑ MOUSE Rainfall
☑ MOUSE Catchment Rainfall
☑ MOUSE Air Temperature
☑ MOUSE Evapo-Transpiration

Figure 6.3 MOUSE Catchment Boundaries

The following items can be toggled on/off under ‘MOUSE Catchment Boundaries’ in the TOC:

- MOUSE Rain Gauges (represents the rain gauges, as defined by the ‘Data source location’)
- MOUSE Rainfall (shows catchments with a catchment load of type ‘Rainfall’)
- MOUSE Catchment Rainfall (shows catchments with a catchment load of type ‘Catchment Discharge’)
- MOUSE Air Temperature (shows catchments with a catchment load of type ‘Air Temperature’)
- MOUSE Evapo-Transpiration (shows catchments with a catchment load of type ‘Evapo-Transpiration’)

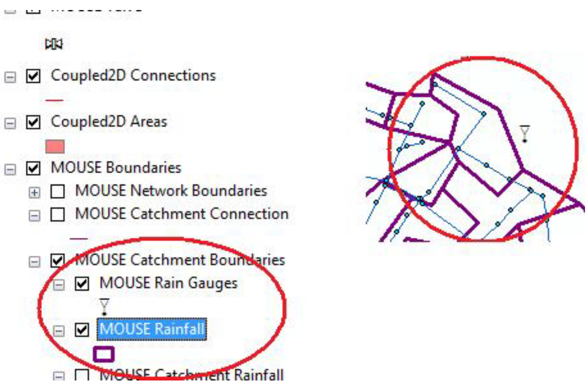

☑ Coupled2D Connections
☑ Coupled2D Areas
☑ MOUSE Boundaries
☐ MOUSE Network Boundaries
☐ MOUSE Catchment Connection
☑ MOUSE Catchment Boundaries
☑ MOUSE Rain Gauges
☑ MOUSE Rainfall
☐ MOUSE Catchment Rainfall

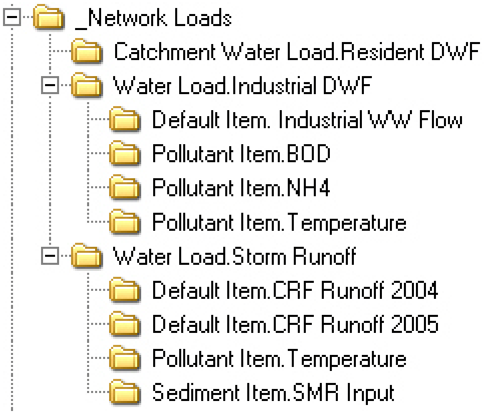
Figure 6.4 Example of the visualization of catchment boundaries

6.2 Network Boundary Conditions

Types of variables associated with MOUSE network (i.e node/grid point) boundary conditions are:

- Water level
- Flow (discharge) - incl. WQ and sediment properties

The discharge boundary conditions are termed as "Network Loads". An example illustrating the network loads is presented (see illustration below)



- _Network Loads
  - Catchment Water Load.Resident DWF
  - Water Load.Industrial DWF
    - Default Item. Industrial WW Flow
    - Pollutant Item.BOD
    - Pollutant Item.NH4
    - Pollutant Item.Temperature
  - Water Load.Storm Runoff
    - Default Item.CRF Runoff 2004
    - Default Item.CRF Runoff 2005
    - Pollutant Item.Temperature
    - Sediment Item.SMR Input

Figure 6.5 MOUSE Network Loads - Example

Network Loads represent all kinds of hydraulic loads, such as DWF point-inflows, individual lateral inflows, infiltration, storm runoff, etc. This type of boundary condition always has discharge (constant, cyclic or a time series) associated as a default item. The discharge can be associated with any pollutant, sediment or temperature item (constant, cyclic or a time series).

Any number of network loads can be specified, independently on the location point.

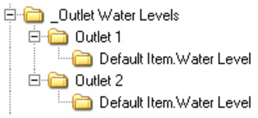
In this example, the model includes three network loads:

"Water Load.Industrial DWF" represent some industrial loads. In the example above, the default item (Resident WW flow) is associated with pollutants BOD and NH4 and with Temperature.

"Water Load.Storm Runoff" represent calculated storm runoff loads. Note that in the example above, two default items are specified, each covering different time period. The default items (i.e. storm runoff itself) are associated with temperature and pollutants BOD and NH4.

"Catchment Water Load.Resident DWF" is an example of a special type of network load definition. It is actually only a reference to already existing "catchment load" (in this case "Water Load.Resident DWF") and a signal to the model to pass the catchment load to the network model. This type of load definition does not need any item. Please note that this is only valid for the MOUSE engine. The MIKE 1D engine does not require this step.

The downstream water level boundary conditions are termed as "External water levels" (see illustration below).



- _Outlet Water Levels
  - Outlet 1
    - Default Item.Water Level
  - Outlet 2
    - Default Item.Water Level

Figure 6.6 External Water levels - Example

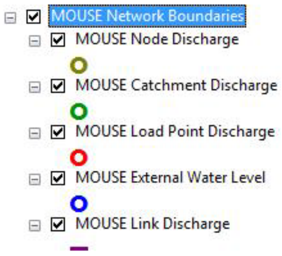
Only one water level boundary condition is allowed at each network outlet. If nothing is specified for an outlet, the model assumes a constant water level equal to the outlet invert level. I.e. a free outlet is assumed.

This type of boundary condition always has water level (constant or a time series) associated as a default item.

In the example above, two external water level boundary conditions have been defined.

6.2.1 Visualization of network boundaries

The network loads are visualized by toogling the display of ‘MOUSE Network Boundaries’ on in the TOC.



- ☑ MOUSE Network Boundaries
  - ☑ MOUSE Node Discharge
  - ☑ MOUSE Catchment Discharge
  - ☑ MOUSE Load Point Discharge
  - ☑ MOUSE External Water Level
  - ☑ MOUSE Link Discharge

Figure 6.7 MOUSE Network Boundaries

The following items can be toggled on/off under ‘MOUSE Network Boundaries’ in the TOC:

- MOUSE Node Discharge (shows nodes with a network load of type Inflow Hydrograph or Infiltration Loss assigned to)

- MOUSE Catchment Discharge (reflects the network load using a Catchment Load. In this specific case, the boundary condition will be displayed even if no Boundary Item is created)
- MOUSE Load Point Discharge (shows the nodes that have a Load point assigned)
- MOUSE External Water Level (shows the outlets that have an External Water Level assigned as a boundary condition)
- MOUSE Link Discharge (shows links with a network load of type Inflow Hydrograph or Infiltration Loss assigned to)

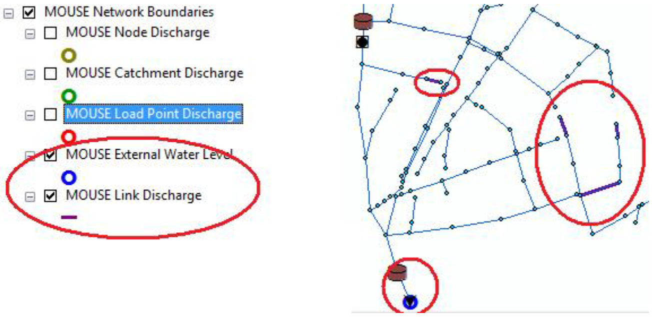

☑ MOUSE Network Boundaries
☐ MOUSE Node Discharge
☐ MOUSE Catchment Discharge
☐ MOUSE Load Point Discharge
☑ MOUSE External Water Level
☑ MOUSE Link Discharge

Figure 6.8 Example of the visualization of network boundaries

6.3 Boundary Condition Editors

Three editors are available for specifying boundary conditions corresponding to three boundary condition groups and one common editor for boundary items:

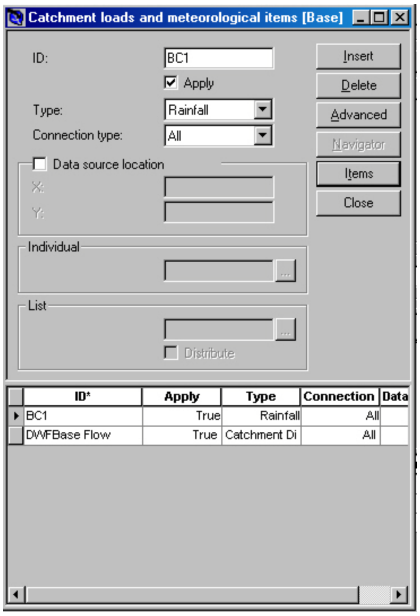
- Catchments Loads and Meteorological Items
- Network Loads
- External Water levels

Common for all three editors is the specification of the Boundary condition identifier and controlling if the specified boundary is actually applied to the model or not (per default, each new boundary condition is applied).

To sort the different loads (catchment loads, network loads and external water levels) in each editor, use the right click option "Sort" on top of the "Boundary ID" column. Double clicking on the column will display all boundaries conditions loaded in the database indistinctly of the editor type where the user is positioned.

Additionally, there is a common editor for boundary items.

6.3.1 Catchments Loads and Meteorological Items Editor



Catchment loads and meteorological items [Base]

ID: BC1

☑ Apply

Type: Rainfall

Connection type: All

☐ Data source location

☐ Distribute

	ID*	Apply	Type	Connection	Data
▸	BC1	True	Rainfall	All	
	DWFBase Flow	True	Catchment Di	All	

Figure 6.9 The catchment loads editor

The following describes how to define an boundary condition for a catchment.

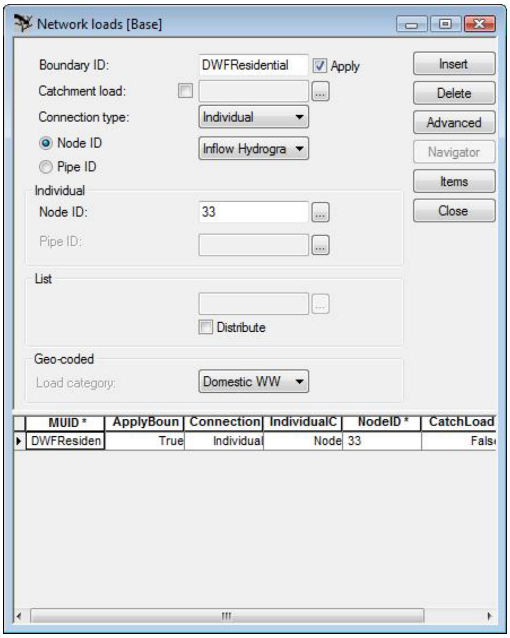
As the first action after inserting a new boundary conditions, a proper name (ID) must be specified. We recommend that you use a descriptive ID.

Next, a type of boundary connection is selected, among Rainfall, Air temperature, Evapo-transpiration and Catchment Discharge. For the first three types, co-ordinates of a data source (e.g. raingauge) may be specified (Data Source location). The co-ordinates are used for the simulation of spatially distributed rainfall (snowmelt, evapo-transpiration), in cases when the data file itself (i.e. time series file) does not contain this information. Checking the "Data source Location" overrides the coordinates from the TS file.

The "Connection Type" determines the mode of identification of the catchments for which this boundary condition applies. "Individual" applies for a single catchment, and requires a specification of the catchment ID. "List" applies for a set of catchments, contained in the catchment selection file (*.CSE or *.MUS). "All" applies to all catchments. In case of two or more meteorological boundary conditions of the same type specified, the co-ordinate information is used to distribute the boundaries accordingly. This latter only applies when the "Connection Type" is set to "All". It is only possible to have one rain used as a boundary condition per catchment.

To sort the different loads use the right click option "Sort" on top of the "Boundary ID" column. Double clicking on the column will display all boundaries conditions loaded in the database.

6.3.2 Network Loads Editor



Boundary ID: DWFResidential ☑ Apply

Catchment load: ☐

Connection type: Individual

● Node ID

○ Pipe ID

Inflow Hydrogra

Node ID: 33

Pipe ID:

☐ Distribute

Load category: Domestic WW

MUID *	ApplyBoun	Connection	IndividualC	NodeID *	CatchLoad
DWFResiden	True	Individual	Node	33	Fals

Figure 6.10 The network loads editor

The following describes how to define an boundary condition for the network. This could e.g. be a discharge entering a manhole or pipe infiltration.

As the first action after inserting a new boundary condition, a proper name (ID) must be specified. We recommend that you use a descriptive ID.

The "Connection Type" determines the mode of definition of the network location(s) to which this boundary applies. "Individual" applies for a single node or pipe, and requires a specification of the Node ID or Link ID.

If a load is connected to a link, it will be uniformly distributed over all computational grid H-points along the link.

"List" applies for a set of node or links, contained in the selection files (*.NSE, LSE or *.MUS). "Defined by Catchment Connections" applies to the storm runoff loads from MOUSE *.CRF file.

For a node - either an inflow hydrograph can be specified or an infiltration boundary can be specified. If the 'Infiltration' option is chosen the boundary items page allows to specify a positive or negative rate. So either infiltration to the node or loss of water from the node to the surrounding ground can be used.

The loss can be defined for any type of node, but it is primarily intended for use with basins or nodes with geometry representing small ponds. The infiltration can be defined as a constant rate (velocity) which is multiplied by the actual wet area in the node. Note that the rate value should be given as a negative value for specifying a loss of water from the network model. Applying a positive value will result in water being added to the node.

Check-box "Distribute" is of relevance for network loads connected to a list of nodes or links. If checked, the specified load will be uniformly distributed over the listed nodes or links. Otherwise, each location specified in the list will receive the specified load.

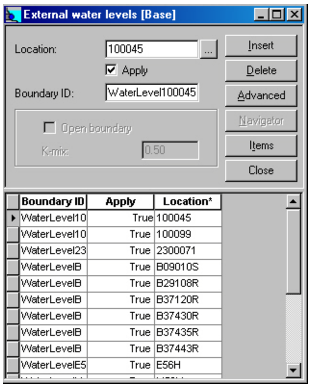
"Geo-coded" applies for point-loads (typically DWF) geo-coded to the network nodes. User can select which one of the geo-coded load categories applies for this boundary condition.

Please note that when you use the boundary type “Defined by catchment” and afterwards specify a time series ID in the boundary items dialog then this item is always used regardless of type (water, component) or extension (dfs0, CRF). If you have not specified a time series ID then the catchment name will be used. This is only applicable for the boundary type “Defined by catchment” for other types the time series ID is always used.

To sort the different loads use the right click option "Sort" on top of the "Boundary ID" column. Double clicking on the column will display all boundaries conditions loaded in the database.

6.3.3 External Water Levels Editor

This editor is dedicated to specification of external water levels interacting with the collection system.



Location: 100045

☑ Apply

Boundary ID: WaterLevel100045

☐ Open boundary

K-mix: 0.50

Boundary ID	Apply	Location*
WaterLevel10	True	100045
WaterLevel10	True	100099
WaterLevel23	True	2300071
WaterLevelB	True	B09010S
WaterLevelB	True	B29108R
WaterLevelB	True	B37120R
WaterLevelB	True	B37430R
WaterLevelB	True	B37435R
WaterLevelB	True	B37443R
WaterLevelE5	True	E56H

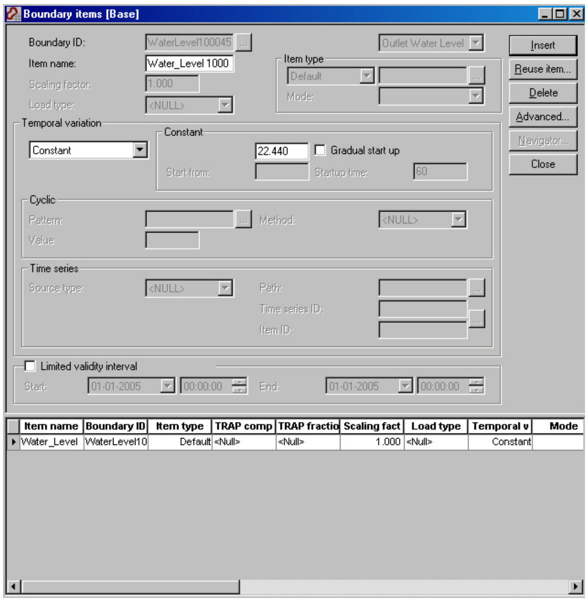
Figure 6.11 External water levels editor

In addition to the specification of the boundary condition ID and the location (outlet), user can define an open boundary here (relevant for pollution transport simulations).

To sort the different loads use the right click option "Sort" on top of the "Boundary ID" column. Double clicking on the column will display all boundaries conditions loaded in the database.

6.3.4 Boundary Items Editor

Boundary item editor allows for a full definition of a boundary item in terms of type, temporal variation, quantity/quality and temporal validity



Boundary ID: WaterLevel100045
Outlet Water Level
Item name: Water_Level 1000
Scaling factor: 1.000
Load type: <NULL>
Item type: Default
Mode:
Temporal variation: Constant
Constant: 22.440
☐ Gradual start up
Start from:
Startup time: 60
Pattern:
Value
Method: <NULL>
Source type: <NULL>
Path:
Time series ID:
Item ID:
☐ Limited validity interval
Start: 01-01-2005 00:00:00
End: 01-01-2005 00:00:00

Item name	Boundary ID	Item type	TRAP comp	TRAP fractio	Scaling fact	Load type	Temporal v	Mode
Water_Level	WaterLevel10	Default	<Null>	<Null>	1.000	<Null>	Constant	

Figure 6.12 The Boundary Items Editor

The 'Boundary Items' editor has been designed to operate in conjunction with the three boundary condition editors. When open simultaneously with one of these, this editor displays only the boundary items associated with the currently active boundary condition. Otherwise, displays all boundary items.

'Boundary Items' editor allows for a full definition of a boundary item in terms of type, temporal variation, quantity/quality and temporal validity.

A new boundary item is created by pressing "Insert". The new item is automatically associated with the currently active boundary condition and is appended to the list of associated items. Alternatively, already existing item can be re-used and associates it with the currently active boundary. "Re-use Item" actually makes a copy of a selected item and adds it on the list of items for the current boundary condition. By these means user can assign the same item (e.g. concentration of some pollutant in wastewater) to many boundary conditions quickly.

A new item can also be inserted while none of the boundary condition dialogs is open. In that case, user must "manually" connect the item to a boundary condition. There is no automatic control over the match between the boundary condition type and the actual item definition, so due care is needed.

Each new item is set as "default". For some types of boundary conditions, this can be changed to "Pollutant" or Sediment". In the later case, a pollutant component or a sediment fraction must be selected from the respective lists. The pollutant items can be defined in terms of concentrations or loads.

The scaling factor determines the fraction of the specified load which is actually loaded into the model. Per default, the scaling factor is set to 1.

Temporal variations

Temporal variation of boundary items may take one of the following forms:

- Constant - any boundary item can be defined as a constant
- Cyclic (Diurnal) - all hydraulic loads items can be defined as diurnal
- Time series

A constant boundary item is the simplest form of boundary. In case of large constant network loads, the HD model may display some stability problems. This can be avoided by allowing a gradual increase of the load over a specified start-up time.

Cyclic (diurnal) boundary items are typically associated with wastewater loads. Cyclic loads may be quantified in several ways: as area-based or person equivalent-based loads (associated with catchment loads), as average load or geo-coded loads (associated with network loads).

Time series may come from various sources, including previous MOUSE result file (this latter option includes MOUSE files CRF, SMR and PMR).

Temporal validity

Each boundary item may be specified with limited temporal validity.

Per default, each item is considered valid during the entire simulation period. Indeed, in case of TS items, the validity is also limited by the length of TS.

If multiple items are specified they may all be with default temporal validity (i.e. over the entire simulation period), or with overlapping validity periods (e.g. when mixing de-fault and limited validity) or with periods with missing coverage.

6.4 Examples

In the following paragraphs some examples on setting up boundary conditions are shown step-by-step

6.4.1 How to add a varying water level at an outlet?

This boundary condition falls into the category 'External Water Levels'. The definition involves the specification of a boundary condition itself and the water level (default) item.

- Enter Edit mode: Go to Edit|Start Editing

- Go to MOUSE|Boundary Conditions|External Water Levels
- Press Insert button
- Boundary ID is then given automatically (hint: you can change this ID if you wish to)
- Choose the list button next to the location field (hint: you can also type in the Location directly - note that you can only add External Water Levels to MOUSE outlets)
- Select the outlet from the list press OK button (hint: you can just double-click)

Press the Items button to open the Boundary items dialog to connect the time variation for the water level. Note that the Boundary ID is automatically filled in on the Boundary items dialog.

- After having pressed Insert an item ID will automatically be given (hint: you can change this ID if you wish to)

Now the temporal variation can be specified:

- Choose “Time Series/Result File” in order to specify an irregular varying water level. This activates the Time Series part of the editor
- Choose DFS0 as the Source Type, click the list button next to the Path field and choose the dfs0 file with the water level (item ID is only necessary to fill out if the dfs0 file contains more than one item, i.e. time series)

After completing this, the time-varying water level has been successfully added to the outlet.

6.4.2 How to add infiltration in a pipe?

This boundary condition falls into the category 'Network Loads'. The definition involves the specification of a boundary condition itself and the infiltration item.

- Enter Edit mode: Go to Edit|Start Editing
- Go to MOUSE|Boundary Conditions|Network Loads
- Press Insert button
- Boundary ID is then given automatically (hint: you can change this ID if you wish to)
- Choose the connection type to ‘Individual’ and set the radio button in ‘Pipe ID’
- Select the pipe from the list and press OK button (hint: you can just double-click)

Press the Items button to open the Boundary items dialog to add the infiltration value. Note that the Boundary ID is automatically filled in on the Boundary items dialog.

- After having pressed Insert an item ID will automatically be given (hint: you can change this ID if you wish to)

Now the infiltration can be specified. In this case we choose a constant infiltration:

- Choose “Constant” in order to specify a constant infiltration. This activates the ‘Constant’ part of the editor
- Type in an infiltration value in the active field. The value is provided as an inflow per length unit of the pipe and is hence also kept constant along the pipe.

After completing this, the constant infiltration to the pipe has been successfully added.

6.4.3 How to add a rainfall as a boundary condition to the catchments?

This boundary condition falls into the category 'Catchment Loads And Meteorological Items'. The definition involves the specification of a boundary condition itself and the rainfall (default) item.

- Enter Edit mode: Go to Edit|Start Editing
- Go to MOUSE|Boundary Conditions|Catchment Loads
- Press Insert button
- ID is then given automatically (hint: you can change this ID if you wish to)
- Set Type = Rainfall
- Set Connection type, choose = All (All = all catchments, List = a group of catchments, Individual = a single catchment)

Note: Now a rainfall is assigned to all catchments. By checkmarking the data source location field you can specify where the rain gauge is placed

- Press the Items button to open the "Items" editor.
- Press Insert button on the Boundary Items dialog to insert a rainfall item
- After having pressed Insert an item ID will automatically be given (hint: you can change this ID if you wish to)

Now the temporal variation can be specified:

- Choose Time Series/Result File in order to choose a time series for the rain event
- This activates the Time Series part of the dialog

- Choose DFS0 as the Source Type, and select the dfs0 file with the rain event (item ID is only necessary to fill out if the dfs0 file contains more than one item, i.e. time series)

Now the rain event has successfully been connected to the catchments!

6.4.4 How to add a discharge to a node?

This boundary condition falls into the category 'Network Loads'. The definition involves the specification of a boundary condition itself and the discharge (default) item.

- Enter Edit mode: Go to Edit|Start Editing
- Go to MOUSE|Boundary Conditions|Network Loads
- Press Insert
- A Load ID is automatically assigned (you can change this)
- Set the Connection type to 'Individual'
- Choose the relevant Node ID
- Now press the Items button. This will open the Boundary Items dialog where the discharge variation can be specified
- Press Insert button on the Boundary Items dialog. The Boundary ID field is automatically filled in by the current boundary ID. This ensures a bond between the Boundary condition and this item.
- After having pressed Insert an item ID will automatically be given (hint: you can change this ID if you wish to)
- Specify that the load type is 'Other'. This will insure a correct categorisation of this load in the simulation summary statistics and in the job-list generation for LTS simulations.

Now the temporal variation can be specified:

- Choose Time Series/Result File in order to specify a irregular varying discharge (note: constant as well as cyclic variation can also be chosen)
- This activates the Time Series part
- Choose DFS0 as the Source Type, click next to the Path field and choose the dfs0 file with the water level (item ID is only necessary to fill out if the dfs0 file contains more than one item, i.e. time series)

Now the discharge is successfully connected to your node!

6.4.5 How to add runoff results as input for the network computation?

The runoff results are treated as network loads in MIKE URBAN, and can be specified following the same principle as any other load (see the example above). Alternatively, a 'shortcut' specification can be used.

The 'standard' definition of runoff input to the network model is described in the following example. Further below, the "shortcut" hint is provided.

- Enter Edit mode: Go to Edit|Start Editing
- Go to MOUSE|Boundary Conditions|Network Loads
- Press Insert
- A Load ID is automatically assigned (you can change this)
- Set the Connection type to 'Defined by catchment connection'. No further specification of the location is required, since this is controlled by the catchment connections.

The specification of the load type will allow the computation to take into account where the load came from initially - and hence also report this in e.g. the summary after the simulation

- Now press the Items button. This will open the Boundary Items dialog where the runoff result file to be used can be specified
- Press Insert button on the Boundary Items dialog. The Boundary ID field is automatically filled in by the current boundary ID. This ensures a bond between the Boundary condition and this item.
- After having pressed Insert an item ID will automatically be given (hint: you can change this ID if you wish to).
- Specify that the load type is 'Storm Runoff'. This will insure a correct categorisation of this load in the simulation summary statistics and in the job-list generation for LTS simulations.

Now the temporal variation can be specified.

- Choose Time Series/Result File in order to choose the result file from the catchment simulation
- This activates the Time Series part of the dialog

Choose MOUSE as the Source Type, click the list button next to the Path field and select the *.CRF file containing the results from the runoff computation.

Now the runoff results are used as input for the network computation!

Alternatively, a 'shortcut' definition can be used. On the MOUSE Computations dialog ('Network Parameters') user can checkmark the 'Runoff' in the Runoff Input group, and select the runoff file from the list. The effect of this specification is identical as if the runoff input is specified through the boundary conditions dialogs. Please note, that this way it is NOT possible to assign any water quality properties to the runoff input. If this is required, the boundary conditions editors must me used.

6.4.6 How do I add DWF in my network dependent on number of inhabitants?

Please note that the steps are different depending on your choice of numerical engine (MOUSE or MIKE1D).

The PE-based and catchment area-based DWF fall into the category of catchment loads. This is because both the input parameters and the results of the computation are catchment-related. However, in most applications, this load is supposed to be passed into the network through the catchment connection points. The following example describes the entire definition process.

The overall procedure involves the following major steps:

- Define the diurnal patterns (note: not a part of this tutorial)
- Define the catchment load. Note that inhabitants (PEs) are associated with catchments, so the DWF load which they generate is also associated with catchments
- Define the boundary item for the catchment DWF load
- This step only applies for the MOUSE engine (for the MIKE 1D engine this step can be skipped): Define a network load and declare it as a "carrier" of the catchment load defined previously. This allows for a direct passage of the catchment load into the network. The water is loaded into the network at the catchment connection points. Note that for this load no item definition is required.

The detailed steps:

- Enter Edit mode: Go to Edit|Start Editing
- Go to MOUSE|Boundary Conditions|Catchment Loads
- Press Insert button
- ID is then given automatically (hint: you can change this ID if you wish to)
- Set Type = Catchment Discharge
- Set Connection type, choose = Individual (All = all catchments, List = a group of catchments, Individual = a single catchment)
- Choose the list button next to the Individual field and select the catchment (hint: you can also type in the catchment ID directly)

The Catchment load and its connectivity to the catchment(s) is specified. The following step is the definition of the boundary item, i.e. temporal variation, quantity and temporal validity of the load.

- Press the Items button. This opens the Items editor.
- Press Insert button on the Boundary Items dialog

Note that the boundary ID is automatically filled in on the Boundary items dialog.

After having pressed Insert an item ID will automatically be given (hint: you can change this ID if you wish to)

The scaling factor and item type are per default set properly, so no action is required. Now the temporal variation can be specified:

- Choose "Cyclic" in order to specify a diurnal load variation. This activates the Cyclic part of the dialog.
- Select a predefined pattern from the list.
- Specify the Method as "PE Based". This indicates that the DWF contribution is computed taking the number of inhabitants PE on the catchment into account
- Specify the water amount per PE/day

Note: "Average" means that an average diurnal value will be used (taken from the field 'Value'), while 'Area based' will compute the total flow taking the catchment area into account.

This completes a full definition of the catchment DWF load based on PEs. The remaining step (which is only needed in case of using the MOUSE engine) is to allow this load be directly applied to the network model.

- Go to MOUSE|Boundary Conditions|Network Loads
- Press Insert
- A Load ID is automatically assigned (you can change this)
- Set the tickmark in 'Catchment Load' and select the boundary ID from the list for the catchment load just created.

The specification of the 'Load Type' will allow the computation to take into account where the load came from initially - and hence also report this in e.g. the summary after the simulation.

6.4.7 How to attach a pollutant concentration to a network load?

Attaching a pollutant concentration involves adding a pollutant item to already existing network load.

- Enter Edit mode: Go to Edit|Start Editing
- Go to MOUSE|Boundary Conditions|Network Loads
- Scroll through the list of loads and place cursor on the wanted load record.

- Press the 'Items' button. This will open the Boundary Items dialog where items associated with the actual load will be displayed. There will be at least one discharge item and, possibly, some pollutant and sediment items.
- Press Insert button on the Boundary Items dialog. The Boundary ID field is automatically filled in by the current boundary ID. This ensures a bond between the Boundary condition and the new item.
- After having pressed Insert an item ID will automatically be given (hint: you should change this ID in order to be able to identify it easier)
- Specify the new item type as 'Pollutant'. This will activate the part of the editor where a pollutant must be selected from the list. Note that pollutant can be specified as concentration (i.e. mass per unit volume of water) or as load (i.e. mass per unit of time). This specification is not effective for the cases where actual pollutant data include the information of actual unit (e.g. dfs0 time series).
- Now the temporal variation can be specified. Note that specification of cyclic variation in a combination with "concentration" affects the mass balance, i.e. the simulated pollutant mass changes with changes in cyclic pattern. This must be compensated by appropriate adjustment of the pollutant concentration.

Now a pollutant (concentration or load) has been successfully connected to your network load!

6.5 Repetitive Profile Editors

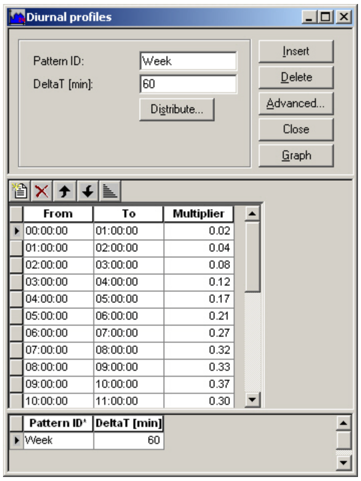
The 'Repetitive Profile Editors' can be used for generating dimensionless, cyclic time series ('repetitive profiles') with a fixed time resolution of one hour. E.g., it can be applied for defining diurnal profiles that can describe the Dry Weather Flow (DWF) from a specific catchment. An unlimited number of repetitive profiles can be applied to different groups of catchments in order to reflect that e.g. an industrial area will have a different DWF flow description than a rural or residential area. The DWF may, however, also vary according to the time of week or year and holidays.

The 'Repetitive Profile Editor' consists of four sections:

- Diurnal Patterns- used for specifying the diurnal patterns
- Profiles Calendar - used for specifying when the diurnal patterns are to be used
- Cyclic Profiles - used for coupling of individual diurnal patterns with profiles calendar definition
- Special Days - used for specifying days that are to be considered as exceptions to the calendar (e.g. the 1st of January).

6.6 Diurnal Patterns

A diurnal pattern consists of the 'PatternID', a DeltaT [min] and the non-dimensional coefficients. The dimensionless coefficients may be given any appropriate value, as long as their relative size reflects the diurnal variation. Prior entering the multiplier coefficients, the DelataT is set for the pattern and clicking on 'Distribute' fills the two time columns with the appropriate numbers of values for having a daily pattern.



Pattern ID: Week
DeltaT [min]: 60

From	To	Multiplier
00:00:00	01:00:00	0.02
01:00:00	02:00:00	0.04
02:00:00	03:00:00	0.08
03:00:00	04:00:00	0.12
04:00:00	05:00:00	0.17
05:00:00	06:00:00	0.21
06:00:00	07:00:00	0.27
07:00:00	08:00:00	0.32
08:00:00	09:00:00	0.33
09:00:00	10:00:00	0.37
10:00:00	11:00:00	0.30

Pattern ID'	DeltaT [min]
Week	60

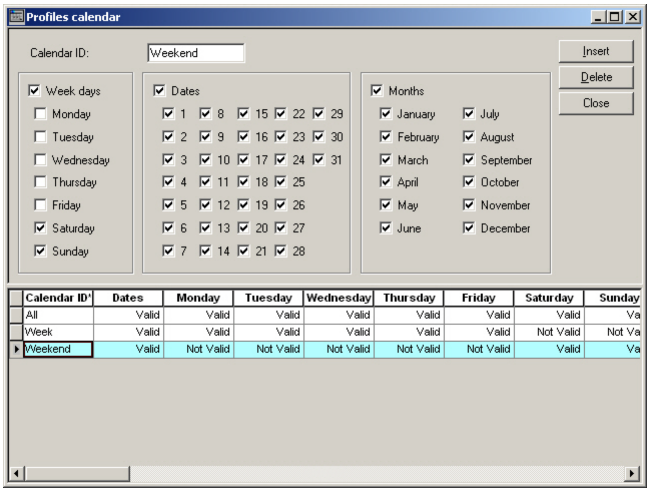
Figure 6.13 Diurnal Pattern Editor

An unlimited number of diurnal patterns can be entered. E.g. one for weekdays and one for weekends, but also one for weekdays belonging to one group of catchments and one for weekdays belonging to a different group of catchments. Clicking on 'Graph' gives a graphical display of the profile.

6.7 Profiles Calendar

The purpose of the profiles calendar is to specify when the diurnal patterns are to be applied, e.g. only during the summer, only in February, only on weekdays, only on each first in the month, etc.

Three checkboxes 'Week Days', 'Dates' and 'Months' are available for the activation of three different, mutually inclusive modes of the calendar definition. This means that if any day is to be included in the current calendar definition, it must be included in all three definition modes. If any of the definition modes is not activated (i.e. the checkbox not ticked), it does not affect the calendar definition. If only a calendar's ID is given, but no tick marks are set, the diurnal pattern that the profiles calendar is assigned to would be unconditionally used.



Calendar ID: Weekend

☑ Week days
☐ Monday
☐ Tuesday
☐ Wednesday
☐ Thursday
☐ Friday
☑ Saturday
☑ Sunday

☑ Dates
☑ 1 ☑ 8 ☑ 15 ☑ 22 ☑ 29
☑ 2 ☑ 9 ☑ 16 ☑ 23 ☑ 30
☑ 3 ☑ 10 ☑ 17 ☑ 24 ☑ 31
☑ 4 ☑ 11 ☑ 18 ☑ 25
☑ 5 ☑ 12 ☑ 19 ☑ 26
☑ 6 ☑ 13 ☑ 20 ☑ 27
☑ 7 ☑ 14 ☑ 21 ☑ 28

☑ Months
☑ January ☑ July
☑ February ☑ August
☑ March ☑ September
☑ April ☑ October
☑ May ☑ November
☑ June ☑ December

Calendar ID	Dates	Monday	Tuesday	Wednesday	Thursday	Friday	Saturday	Sunday
All	Valid	Valid	Valid	Valid	Valid	Valid	Valid	Va
Week	Valid	Valid	Valid	Valid	Valid	Valid	Not Valid	Not Va
Weekend	Valid	Not Valid	Not Valid	Not Valid	Not Valid	Not Valid	Valid	Va

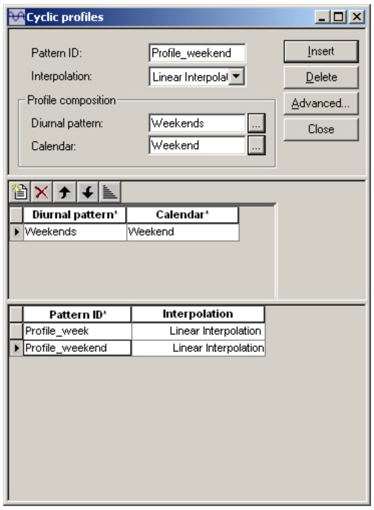
Figure 6.14 Profiles Calendar Editor

6.8 Cyclic Profiles

A 'Cyclic Profile' (defined by its 'Profile ID' string max 40.characters) links the diurnal pattern with the relevant calendar definitions.

Each pattern can contain an arbitrary number of profile-calendar links. For inserting a new link, use the "Insert new line" icon. For deleting a selected link use the "Delete selected line" icon. If the calendar definitions for two or more links are covering the same period, the links positioned highest on the list will have the highest priority. In other words, the definition given by some link covers only the days which are not included into any of the higher positioned links. In order to control the priorities, it is possible to change the order of sets in the pattern with the "Move line up" and "Move line down" icons.

For each pattern the 'Interpolation' method is chosen to either 'Linear Interpolation' or 'No Interpolation'. The 'Linear Interpolation' performs a linear interpolation between the values given in the diurnal profile while the 'No Interpolation' will apply a step function. For use with DWF, the 'Linear Interpolation' is recommended



Pattern ID: Profile_weekend
Interpolation: Linear Interpolat
Diurnal pattern: Weekends
Calendar: Weekend

Diurnal pattern*	Calendar*
Weekends	Weekend

Pattern ID*	Interpolation
Profile_week	Linear Interpolation
Profile_weekend	Linear Interpolation

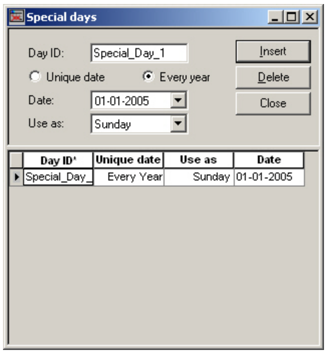
Figure 6.15 Cyclic Profiles Editor

6.9 Special Days

The 'Special days' is used for the specification of individual days that should be considered differently than it is given by the profiles calendar.

E.g. the 1st of January each year (the New year holiday) even though it comes every year on a different day of the week could be coupled with the diurnal profile of e.g. Sunday, since the diurnal pattern of a holiday matches better with Sunday than by any other week day.

The special days are divided in two categories: Special days that are to be considered every year and special days that are only considered once (Unique date). An example of this could be the Easter holiday which will not be on the same date every year.


Special days
Insert
Delete
Close

Day ID: Special_Day_1

○ Unique date ● Every year

Date: 01-01-2005

Use as: Sunday

Day ID	Unique date	Use as	Date
Special_Day_	Every Year	Sunday	01-01-2005

Figure 6.16 Special Days Editor

7 MOUSE Simulations

MOUSE simulations may be started from ‘Simulation | Run MOUSE’. Simulation options for MOUSE are also defined here.

The editor is split into following five tabsheets:

- General
- Runoff parameters
- Network parameters
- Network Summary
- 2D overland parameters

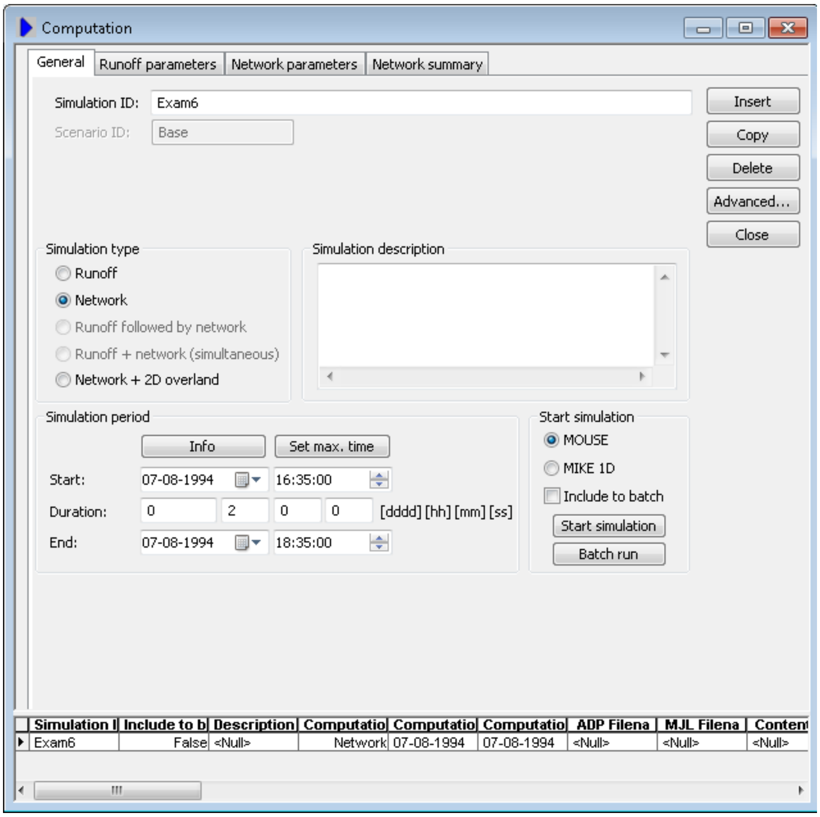
The ‘Simulation ID’ and ‘Scenario ID’ are found on all four tabsheets as a common reference.

7.1 The General Simulation Settings

On the General page the following can e.g. be specified:

- Name of simulation (equal to the name of the result file)
- Simulation period
- Simulation type

The General page includes a button in the upper right ‘corner’ that allows to copy simulation settings (to be re-used in another simulation, e.g. for another scenario).


Computation
General
Runoff parameters
Network parameters
Network summary
Insert
Copy
Delete
Advanced...
Close
Info
Set max. time
Start simulation
Batch run

Simulation ID: Exam6

Scenario ID: Base

Simulation type
○ Runoff
● Network
○ Runoff followed by network
○ Runoff + network (simultaneous)
○ Network + 2D overland

Simulation description

Simulation period
Start: 07-08-1994 16:35:00
Duration: 0 2 0 0 [dddd] [hh] [mm] [ss]
End: 07-08-1994 18:35:00

Start simulation
● MOUSE
○ MIKE 1D
☐ Include to batch

Simulation I	Include to b	Description	Computatio	Computatio	Computatio	ADP Filena	MJL Filena	Content
Exam6	False	<Null>	Network	07-08-1994	07-08-1994	<Null>	<Null>	<Null>

Figure 7.1 The MOUSE Simulation General Editor dialog

Edit fields on the General page

A full overview of the editor fields and corresponding database attributes is provided in the Table 7.1.

Table 7.1 Overview of the general simulation settings (Table 'msm_Project')

Edit field	Description	Used or required by simulations	Field name in datastructure
Simulation ID	ID of simulation - user specified. ID will be reflected in the name of project and result files	Yes	MUID
Scenario ID	ID of selected Scenario - Activating scenario is done in the Scenario Manager dialog	Yes	ScenarioID
Runoff/Network radio buttons	Switch for starting runoff, network or network combined with 2D simulation	Yes	ComputationTypeNo

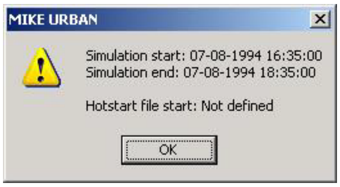
Table 7.1 Overview of the general simulation settings (Table 'msm_Project')

Edit field	Description	Used or required by simulations	Field name in datastructure
Simulation Description	User's descriptive information	No	Description
Start	Start date and time for the simulation	Yes	ComputationBegin
Duration	Allows to specify the duration of the simulaion. Entering the duration will automatically fill in the end date and time of the simulation. If the start and end time are specified the duration will be filled automatically	Optional (will be automatically filled)	-
End	End date and time for the simulation	Yes	ComputationEnd

'Info' button

The 'Info' button opens a dialog containing:

1. Start and end time of the simulation similar to the values found under max time
2. Start and end time of a hotstart file (if specified) (Table: msm_Project, Attribute: PRFHotstartFileName)



MIKE URBAN

Simulation start: 07-08-1994 16:35:00
Simulation end: 07-08-1994 18:35:00

Hotstart file start: Not defined

Figure 7.2 Example of the dialog that comes up, when pressing the 'Info' button

'Set max. time' button

The 'Set max. time' button will set the maximum simulation time by filling in the start time and the end time of the simulation. The start time of a simulation is considered the latest start time of all boundaries. Likewise the end time for the simulation is considered the earliest end time of all boundaries.

Each boundary contains a number of items which can cover different parts of the simulation.

If a validity is specified for an item this specifies the start and end time. If a validity is not specified only items specified as timeseries have a start and end time. If either a constant or cyclic value is given without validity the item is not included in the evaluation.

7.1.1 Choice of numerical engine

For each simulation you can choose to use either the classic MOUSE engine or the MIKE 1D engine. Please refer to seperate documentation on the difference between these two engines and the possibilities provided with them.

It is important to note that when running the numerical engine of you choice you need to keep the input for the simulation within the same numerical engine. I.e. when using results from a runoff simulation as input to a pipe flow simulation then both the runoff and the pipeflow simulation needs to be done with the same numerical engine. If runoff result file is computed with e.g. the MOUSE engine and you try to use it as input for MIKE 1D pipeflow simulation you will hence get an error message. You can, however, create a number of simulation records and then choose to have some done with MOUSE and some with MIKE 1D.

The MIKE 1D result files all have the same extension (*res1d), while the MOUSE result files have different extensions based on simulation type. Several places you will hence see both result types appear as a filter for file types you can choose between:

- Computation dialog - RDII hotstart file. The filter says: "Hotstart Files (*.nof, *.res1d)
- Computation dialog - runoff input. The filter says: "Runoff Files (*.crf, *.res1d)
- Computation dialog - hotstart file. The filter says: "Hotstart files (*.prf, res1d)"
- Boundary Items - result file/time series. Both MOUSE Result Files and res1d files can be used
- LTS initial conditions - hotstart file. The filter says: "Hotstart Files (*.prf, res1d)"

7.1.2 Hotstart files

What is hot start?

Hot start is a facility that enables application of initial conditions for a simulation, other than the default initial conditions, i.e. empty network and uniform concentrations of pollutants in the pipe flow model, or initial values of hydrological state variables in the RDII hydrological model.

What is a hot start file?

The hot start initial conditions represent the state of the system as recorded in a previously generated result file at a specified time - the so-called hot start file.

Relevant information from a hot start file for a network simulation includes:

- Water levels and pollutants' concentrations in each node of the system;
- Water levels, discharges and pollutants' concentrations in all grid points of the network;
- Pumps, weirs, orifices (gates) and valves flows and pollutants concentrations in these flows;
- Position of controllable weirs, gates and valves (if RTC is applied).

For the RDII model, relevant information in the hot start file includes all relevant hydrological state variables, such as surface storage, groundwater depth, etc.

A hot start file for a network simulation must be generated with exactly the same model setup as the model used in a hot start simulation run. A result file used as a hot start file must be a complete ("standard") result file. I.e. user-specified result file with reduced contents cannot be used as a hot start file.

Why and when is hot start relevant?

Hot start files can be used for the following purposes:

- To avoid potential numerical instabilities that occasionally take place under Dynamic Wave computations during the initial phase of the simulation.
- To create a realistic initial conditions for a productive simulation. This typically implies levying a set of constant base flows (for a natural channel network) or a set of dry weather sanitary flows (for a sewer network) over some startup period of time. The resulting hot start file from this run is then used to initialize a subsequent run where the inflows of real interest are imposed.
- To divide long continuous simulations into shorter simulation periods. Here are the initial conditions for each subsequent simulation period set as the end of the previous simulation period.

Specification of the hot start

Hot start options are set in the Simulation dialog (runoff parameters tab and network parameters tab)

For the RDII runoff model, the only parameter is the Hot start Filename. The system automatically detects the values of the relevant variables at the simulation start time and initiates the actual simulation run accordingly.

For the pipe flow model, the following parameters are needed:

- Hot start Filename: This is the name of an existing result file from which initial conditions for the simulation will be loaded.
- Hot start Date and Time: The date and time at which the initial conditions are taken from the hot start file. The hot start time must be within the hot start file time coverage.

- AD Hotstart (checkbox): If checked, the AD model is initiated from the AD hot start file (with identical file name as the HD hot start file), at the date and time specified. Otherwise, AD model is initiated uniformly by the specified initial values for the AD components .

If the Hot Start Filename specified is identical with the current simulation result file name (concatenated by simulation MUID, ScenarioID and the extension .PRF),

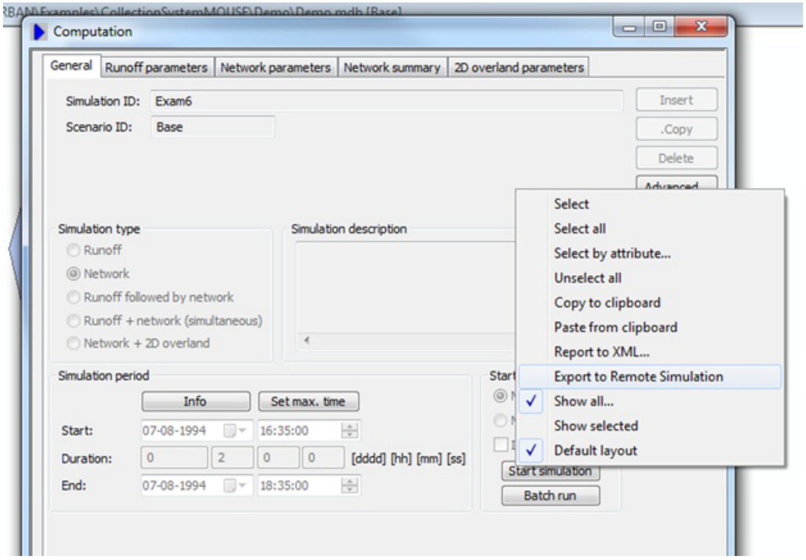
The results of the current simulation will be appended to the hot start file, starting at the simulation start time. In this case, hot start date and time are ignored, as the simulation hot starts at the simulation start time. Any information in the hot start file beyond the simulation start date and time (the actual hot start date and time) will be lost. This part of the file will be replaced by the new simulation results.

7.1.3 Export to remote simulation

It is possible to run computations on a remote computer via the DHI Remote Simulation Console (see the DHI Remote Simulation documentation for details).

Before running a Remote Simulation, an engine specific input file must be created. These engine specific input files have the extension *.m1dx for MIKE 1D and *.mex for MOUSE.

To create an engine specific input file, open the ‘General’ tab of the ‘Computation’ dialog and click ‘Advanced’ . This opens a menu where there is an option called ‘Export to Remote Simulation’:


RBAN\Examples\CollectionSystemMOUSE\Demo\Demo.mdb [Base]
Computation
General
Runoff parameters
Network parameters
Network summary
2D overland parameters
Insert
.Copy
Delete
Advanced
Simulation type
Simulation description
Simulation period
Info
Set max. time
Start
Start simulation
Batch run
Select
Select all
Select by attribute...
Unselect all
Copy to clipboard
Paste from clipboard
Report to XML...
Export to Remote Simulation
✓ Show all...
Show selected
✓ Default layout

Simulation ID: Exam6
Scenario ID: Base

Simulation type:
○ Runoff
● Network
○ Runoff followed by network
○ Runoff + network (simultaneous)
○ Network + 2D overland

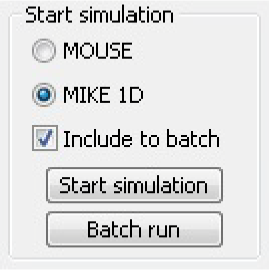
Start: 07-08-1994 16:35:00
Duration: 0 2 0 0 [dddd] [hh] [mm] [ss]
End: 07-08-1994 18:35:00

Figure 7.3 Export to Remote Simulation option

Selecting ‘Export to Remote Simulation’ will export the setup to a *.m1dx file or a *.mex file, depending on the choice of simulation engine. The full name of the exported file will be composed of the Simulation ID, the Scenario ID and the extension.

7.1.4 Batch simulations

If you need to run more simulations sequentially you can choose to do so by including these to a batch run. This is done by setting the tick mark in ‘Include to batch’ for each of the simulations that you wish to include to a batch run. Once all simulations have been included pressing the ‘Batch run’ button will execute all simulations that have the ‘include to batch’ flag set in the sequence that they are specified in the simulation dialog. This means that multiple scenarios can be simulated in batch without user interaction. Each scenario will be activated automatically before the simulation is executed.


Start simulation
Start simulation
Batch run

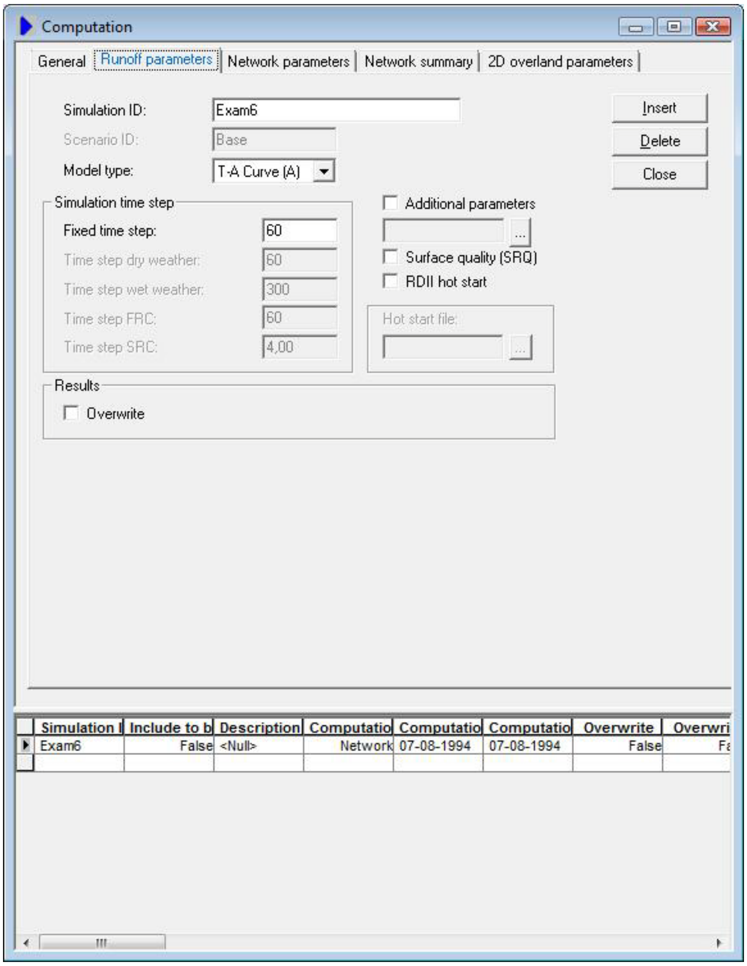
○ MOUSE
● MIKE 1D
☑ Include to batch

Figure 7.4 Each simulation can be included to a batch simulation

7.2 The Runoff Simulation Settings

On the Runoff parameters page the following can e.g. be specified:

- Name of simulation (equal to the name of the result file)
- Simulation type
- Simulation description
- Simulation period
- Start simulation


Computation
General
Runoff parameters
Network parameters
Network summary
2D overland parameters
Insert
Delete
Close

Simulation ID: Exam6

Scenario ID: Base

Model type: T-A Curve (A)

Simulation time step

Fixed time step: 60

Time step dry weather: 60

Time step wet weather: 300

Time step FRC: 60

Time step SRC: 4,00

☐ Additional parameters

☐ Surface quality (SRQ)

☐ RDII hot start

Hot start file:

Results

☐ Overwrite

	Simulation I	Include to b	Description	Computatio	Computatio	Computatio	Overwrite	Overwri
▶	Exam6	False	<Null>	Network	07-08-1994	07-08-1994	False	Fa

Figure 7.5 The MOUSE Simulation Runoff Editor Dialog

Table 7.2 The Simulation Runoff Identification

Edit field	Description	Used or required by simulations	Field name in datastructure
Simulation ID	ID of simulation - user specified. ID will be reflected in the name of project and result files	Yes	MUID
Scenario ID	ID of selected Scenario - Activating scenario is done in the Scenario Manager dialog	Yes	ScenarioID
Model Type	Selection among available runoff models and combinations: T-A Curve (A), Kinematic Wave (B), Model C1, Model C2, RDI (solo), RDI + B, UHM	Yes	RunoffModel-TypeNo
Fixed time step		Yes, for runoff models A, B and UHM	DtFixedSec
Time step dry weather	Time step during dry periods	Yes, for runoff model C1 and C2	DtDryPeriodSec
Time step wet weather	Time step during wet periods	Yes, for runoff model C1 and C2	DtWetPeriodSec
Time step FRC	RDII Fast Runoff Component time step	Yes for RDI + either model A or B for the fast runoff component	RDIIDtFRCSec
Time step SRC	RDII Slow Runoff Component time step	Yes, when RDI modeling is selected	RDIIDtSRCSec
Additional parameters - checkbox	If this checkbox is set to true an file for the runoff simulation can be chosen.	Optional	ADPRunoff-FileNo
Surface quality (SRQ) checkbox	Flag for including the surface quality process in the simulation	Optional	SRQComputa-tionNo

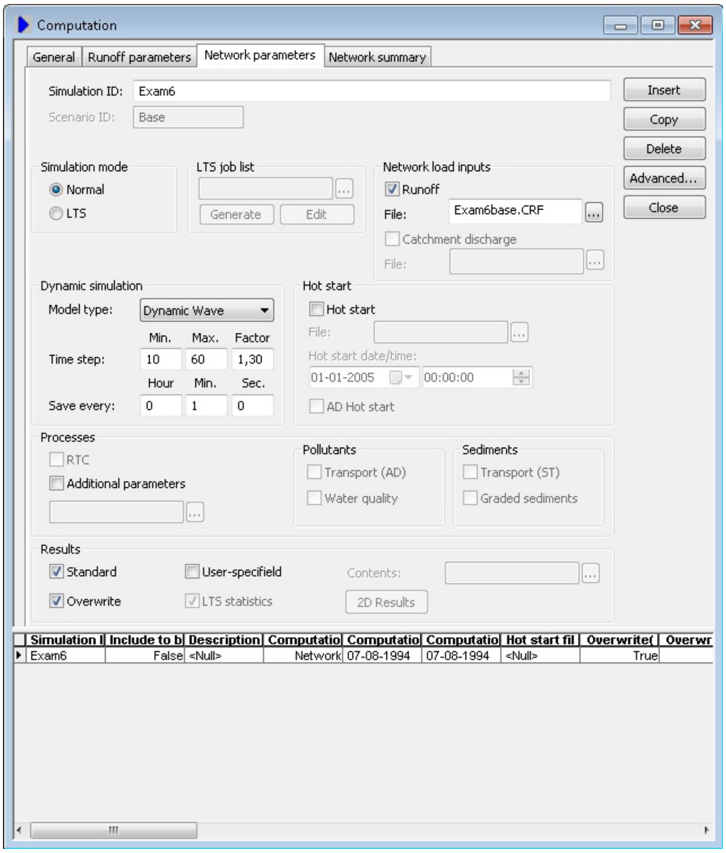
Table 7.2 The Simulation Runoff Identification

Edit field	Description	Used or required by simulations	Field name in datastructure
RDII hot start checkbox	If this checkbox is set to true a hotstart file for the RDII part can be chosen.	Optional	RDIIHotStartNo
Overwrite checkbox	Flag which controls if existing runoff result files can be overwritten by a new one created in this simulation under the same name.	Optional	

7.3 The Network Simulation Settings

The Network page is organized into following groups:

- Identification of simulation
- Simulation mode
- LTS Job list
- Runoff Input
- Dynamic simulation
- Hot Start
- Processes
- Results



Simulation ID: Exam6

Scenario ID: Base

Simulation mode: ● Normal ○ LTS

LTS job list:

Network load inputs: ☑ Runoff, File: Exam6base.CRF; ☐ Catchment discharge, File:

Dynamic simulation: Model type: Dynamic Wave

	Min.	Max.	Factor
Time step:	10	60	1,30

	Hour	Min.	Sec.
Save every:	0	1	0

Hot start: ☐ Hot start; File: ; Hot start date/time: 01-01-2005 00:00:00; ☐ AD Hot start

Processes: ☐ RTC; ☐ Additional parameters

Pollutants: ☐ Transport (AD); ☐ Water quality

Sediments: ☐ Transport (ST); ☐ Graded sediments

Results: ☑ Standard; ☑ Overwrite; ☐ User-specifield; ☑ LTS statistics; Contents:

Simulation I	Include to b	Description	Computatio	Computatio	Computatio	Hot start fil	Overwrite(	Overwr
Exam6	False	<Null>	Network	07-08-1994	07-08-1994	<Null>	True	

Figure 7.6 The MOUSE Simulation Network Editor Dialog

Table 7.3 The Simulation Network Identification

Edit field	Description	Used or required by simulations	Field name in datastructure
Simulation ID	ID of simulation - user specified. ID will be reflected in the name of project and result files	Yes	MUID
Scenario ID	ID of selected Scenario - Activating scenario is done in the Scenario Manager dialog	Yes	ScenarioID
Simulation mode - radio buttons ‘Normal’ and ‘LTS’	Choice between Normal (“one event”) simulation or LTS (Long Term Simulation) simulation	Yes	Computation-ModeNo

Table 7.3 The Simulation Network Identification

Edit field	Description	Used or required by simulations	Field name in datastructure
	Reference to a MJL file (LST job list) to be used in the LST simulation	Optional	MJLFileName
Runoff checkbox	Reference to a CRF file I(runoff hydrographs) to be used in network simulation	Yes if runoff hydrographs should be included as boundary	CRFFileName
Model type	The model to apply for the network simulation: Dynamic Wave, Diffusive Wave, Kinematic Wave	Yes	NetworkModelTypeNo
Time step	Time step data for network model. Minnimum time step, maximum time step and increase factor	Yes	DtMinSec, DtMaxSec, DtIncreaseFactor
Save every	Save result frequency for network model	Optional	DtSaveHour, DtSaveMin, DtSaveSec
Hot start	Seleting a hotstart file for the simulation	Optional	HDHotStartNo
File	Reference to HD hot start result file	Yes if Hotstart is selected	PRFHotStartFileName
Hot start date/time	Specification of start date/time in the Hot start file	Yes, if Hotstart is selected	HotStartDateTime
AD Hot start	Reference to AD hot start result file	Optional	ADHotStartNo
RTC	Real Time Control simulation	Optional	RTCComputationNO
Additional parameters	Selecting the optional ADP file for the simulation	Optional	ADPNetworkFileNo
File	Reference to ADP file (additional parameter file), please refer to seperate documentation on the content of this file	Yes, if additional parameters is chosen	ADPFileName
Transport(AD)	Advection Dispersion simulation	Optional	ADComputationNo
Water quality	Water quality simulation	Optional	WQComputationNo
Transport (ST)	Simulation of sediment transport	Optional	STComputationNo
Graded Sediment	Simulation of graded sediments	Optional	GradedSedimentNo

Table 7.3 The Simulation Network Identification

Edit field	Description	Used or required by simulations	Field name in datastructure
Standard	Standard result file	Optional	StandardResultNo
User-specified	User specified result file	Optional	UserSpecifiedResultNo
File	Reference to result selection file	Yes, if User-specified is chosen	RSFDataID
Overwrite	Overwrite existing result file	Optional	NAllowOwerwriteNo
LTS Statistices	Calculation of LTS statistics	Optional	LTSStatisticsNo
2D Results	Definition of 2D outputs	Yes, if Network + 2D overland simulation is chosen	-

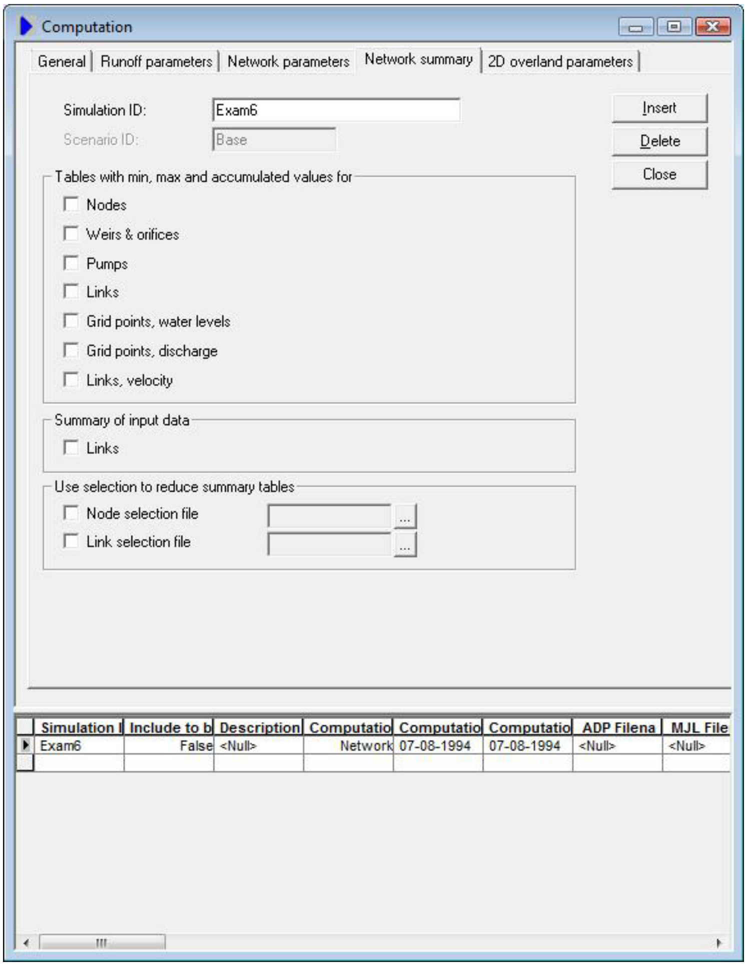
For the 2D overland parameters, please refer to separate chapter on this.

The additional parameter file (the ADP-file) is a separate file with additional settings for the simulation. Please refer to the seperate documentation on this file for further information.

7.4 The Summary Simulation Settings

On the Summary page you specify content of the summary after a MOUSE engine simulation. The MIKE 1D engine produces a different summary. The Summary page is organized into following groups:

- Identification of simulation
- Tables with min, max and accumulated values for
- Summary of input data
- Use selection to reduce summary tables


Computation
General
Runoff parameters
Network parameters
Network summary
2D overland parameters
Insert
Delete
Close
...
...

Simulation ID: Exam6

Scenario ID: Base

Tables with min, max and accumulated values for

☐ Nodes

☐ Weirs & orifices

☐ Pumps

☐ Links

☐ Grid points, water levels

☐ Grid points, discharge

☐ Links, velocity

Summary of input data

☐ Links

Use selection to reduce summary tables

☐ Node selection file

☐ Link selection file

Simulation I	Include to b	Description	Computatio	Computatio	Computatio	ADP Filena	MJL File
Exam6	False	<Null>	Network	07-08-1994	07-08-1994	<Null>	<Null>

Figure 7.7 The MOUSE Simulation Summary Editor Dialog

Table 7.4 The Simulation Summary identification

Edit field	Description	Used or required by simulations	Field name in datastructure
Simulation ID	ID of simulation - user specified. ID will be reflected in the name of project and result files	Yes	MUID
Scenario ID	ID of selected Scenario - Activating scenario is done in the Scenario Manager dialog	Yes	ScenarioID
Nodes	Tables with min, max and accumulated values will be included in the summary if checkbox is true.	Optional	SummaryNodeNo

Table 7.4 The Simulation Summary identification

Edit field	Description	Used or required by simulations	Field name in datastructure
Weirs & orfices	Tables with min, max and accumulated values will be included in the summary if checkbox is true.	Optional	Summary-WeirNo
Pumps	Tables with min, max and accumulated values will be included in the summary if checkbox is true.	Optional	Summary-PumpNo
Links	Tables with min, max and accumulated values will be included in the summary if checkbox is true.	Optional	SummaryLinkNo
Grid points, water levels	Tables with min, max and accumulated values will be included in the summary if checkbox is true.	Optional	SummaryLinkLevelNo
Grid points, discharge	Tables with min, max and accumulated values will be included in the summary if checkbox is true.	Optional	SummaryLink-DischargeNo
Links, velocity	Tables with min, max and accumulated values will be included in the summary if checkbox is true.	Optional	SummaryLinkVelocityNo
Summary of input data, Links	Tables with min, max and accumulated values will be included in the summary if checkbox is true.	Optional	SummaryLinkInputNo
Node selection file checkbox	Tables with min, max and accumulated values will be included in the summary if checkbox is true.	Optional	NodeSelectionNo
Node selection file	Tables with min, max and accumulated values will be included in the summary if checkbox is true.	Yes, if NodeSelectionNo = True	NodeSelection-Name
Link selection file checkbox	Tables with min, max and accumulated values will be included in the summary if checkbox is true.	Optional	LinkSelectionNo
Link selection file	Tables with min, max and accumulated values will be included in the summary if checkbox is true.	Yes, if LinkSelectionNo = True	LinkSelection-Name

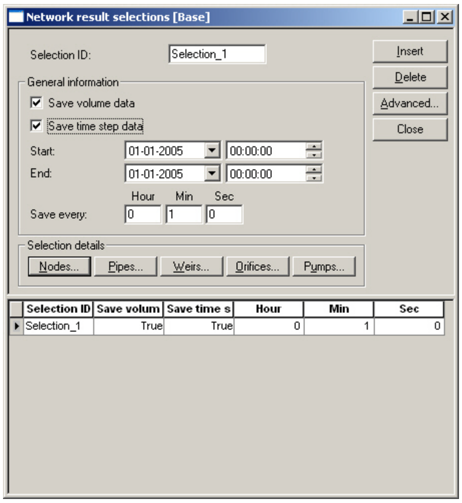
7.5 The 2D Overland Parameters

This tab page specifies the parameters used when setting up a combined pipe flow and overland flow simulation. Please refer to the chapter on 2D Overland Flow for more on these settings.

7.6 MOUSE Result Selections

It is possible to control the output from a MOUSE pipe flow simulation by defining a user specified result file. A user specified result file contains only the simulated time series according to the specification.

The definition of an user specified result selection contents is achieved through MIKE URBAN menu option "Simulation | MOUSE Result selections…"



Selection ID: Selection_1

☑ Save volume data

☑ Save time step data

Start: 01-01-2005 00:00:00

End: 01-01-2005 00:00:00

Save every: Hour 0 Min 1 Sec 0

Selection ID	Save volum	Save time s	Hour	Min	Sec
Selection_1	True	True	0	1	0

Figure 7.8 Network Result Selection Editor

By ticking the 'Save volume data' checkbox, data regarding mass balance in the system will be saved in the result file. If ticked, 'Save time step data' will save information about the used time steps in the simulation.

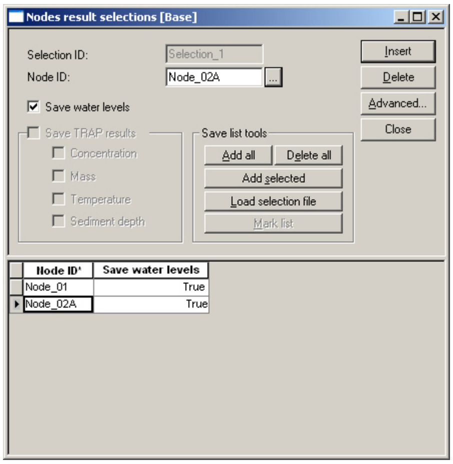
It is possible to save data in a time period that does not necessarily cover the whole period of the simulation. Make sure that the interval specified by 'start' and 'end' dates is at least partly covered by the simulation period. Saving frequency specified for the 'Reduced' result file is not dependent on the time interval chosen for the complete result file, if such a file has been specified.

Results can be saved from four categories; Nodes, Pipes, Weirs, Orifices and Pumps. Consequently, the result save specification is divided into four separate tables. By pressing the appropriate button a specific editor can be accessed.

It should be noted that if a set of nodes is selected, and the selection (from the horizontal plot or from the selection file) is loaded into the pumps or weirs list, only nodes where pumps or weirs do actually exist would turn up on the list.

In cases where several pumps or weirs are associated with one node, the results will be saved for all these pumps or weirs.

Presence of a certain item on the list is not sufficient for it to be included into the user specified result file. This is first ensured after a result type is chosen e.g. water level. In figure Figure 7.9 is displayed the Editor for ‘Nodes result selections’



Selection ID: Selection_1
Node ID: Node_02A

☑ Save water levels

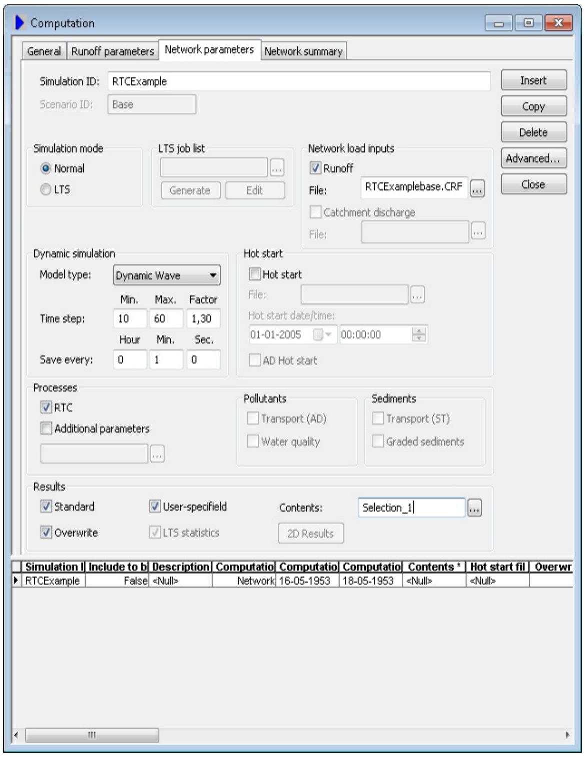
☐ Save TRAP results
☐ Concentration
☐ Mass
☐ Temperature
☐ Sediment depth

Node ID'	Save water levels
Node_01	True
Node_02A	True

Figure 7.9 Nodes Result Selection Editor

The 'Mark List' option helps clearing or checking all items on the displayed list. If a queried list is displayed, 'Mark List' will operate only on the currently displayed list. The remaining part of the list will retain earlier settings.

When a result selection file has been specified then it is activated in the ‘Run MOUSE | Network parameters’ editor by selecting the Selection ID in the ‘Results’group in the field labeled ‘Contents:’ - see Figure 7.10


Computation
General
Runoff parameters
Network parameters
Network summary
Insert
Copy
Delete
Advanced...
Close
Generate
Edit
2D Results

Simulation ID: RTCExample
Scenario ID: Base

Simulation mode
● Normal
○ LTS

LTS job list

Network load inputs
☑ Runoff
File: RTCExamplebase.CRF
☐ Catchment discharge
File:

Dynamic simulation
Model type: Dynamic Wave

	Min.	Max.	Factor
Time step:	10	60	1,30
	Hour	**Min.**	**Sec.**
Save every:	0	1	0

Hot start
☐ Hot start
File:
Hot start date/time:
01-01-2005 00:00:00
☐ AD Hot start

Processes
☑ RTC
☐ Additional parameters

Pollutants
☐ Transport (AD)
☐ Water quality

Sediments
☐ Transport (ST)
☐ Graded sediments

Results
☑ Standard
☑ User-specifield
Contents: Selection_1
☑ Overwrite
☑ LTS statistics

Simulation I	Include to b	Description	Computatio	Computatio	Computatio	Contents *	Hot start fil	Overwr
RTCExample	False	<Null>	Network	16-05-1953	18-05-1953	<Null>	<Null>	

Figure 7.10 Selection of ‘Selection_1’ - result selection file

8 2D Overland Flow

8.1 Introduction

Surface flooding can be simulated with MIKE URBAN using the following two approaches:

1. Combined 1D and 2D model: The flow in the subsurface pipes is simulated using the MOUSE 1D engine and the overland flow is simulated using the MIKE 21 2D overland flow model. This combination of models is called MIKE FLOOD. The main input required for the 2D overland flow model is a digital elevation model. Based on this, the 2D model is capable of simulating overland flow paths and velocities. The main advantages of using this approach are:
   - Leads to more reliable and more accurate modelling
   - Requires less engineering judgement
   - Requires less engineering hours
   - Provides better result visualization

   The main disadvantage of this approach is the longer simulation run times.

2. Two-layer 1D model: The flow in subsurface pipes as well as the overland flow are both simulated using the MOUSE 1D engine in MIKE URBAN. This approach requires that the overland flow paths (streets, etc.) are known or estimated beforehand and each flow path is then defined as an open channel in MIKE URBAN. Flow exchange between the pipes and the overland channels is accounted for using MIKE URBAN model components, such as weirs, orifices or curb inlets. The engineering time and judgement required in order to predefine the overland flow paths can be substantial, and this is one of the main disadvantages of this approach. It also adds uncertainties and inaccuracies to the modelling, as the model will only allow overland flow to take place along paths anticipated by the modeller. The main advantages of this approach is shorter simulation run times.

   The remaining part of this chapter explains how to apply the combined 1D and 2D modelling approach.

8.2 Input Required - Overview

The main input required for combined 1D and 2D flow modelling is the following:

1. A working pipe flow model.
2. A digital elevation model (DEM). MIKE URBAN 2D currently accepts DEMs specified as raster data sets which have to be stored in ESRI Grid

format or DEMs provided in the .dfs2 file format. The latter is the DHI file format for raster data sets.

3. A number of couplings between the 1D model and the 2D model. Such couplings are connections between the two models allowing for exchange of water between the two model domains. Couplings can, for example, represent manholes or curb inlets.

8.3 Input Required - Details

This section of the manual describes the details of the steps it takes to define a 2D overland flow model and to combine it with an existing 1D pipe flow model.

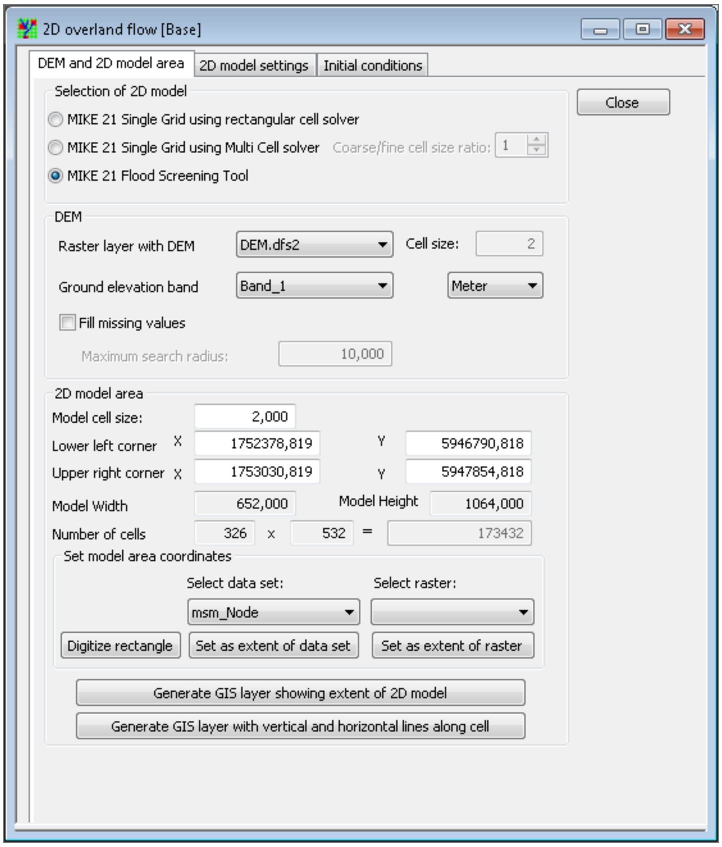
8.3.1 Choosing 2D model

Select MOUSE | 2D Overland Flow | 2D Model Settings ... to open the dialog shown in Figure 8.1.

The underlying 2D overland flow model, MIKE 21, offers the following different types of 2D model solvers:

1. Single Grid using a rectangular cell solver. The 2D model area is covered by square calculation cells.
2. Single Grid using a rectangular multi-cell solver. Again, the 2D model area is covered by a grid of square calculation cells. This is referred to as the coarse grid. Each coarse grid cell is further subdivided into N by N cells called the fine grid. N is an integer number - the ratio between the linear cell dimension of the coarse and the fine grid.
3. MIKE 21 Flood Screening Tool (FST) is designed for inland flooding applications. By means of FST the user can run 2D overland flood simulations faster than the regular MIKE 21 overland flow simulations. The Flood Screening Tool (FST) provides a simple hydrodynamic solver for computations regarding inland flooding, is based on MIKE 21 Flow Model but uses simpler equations with some parameters from MIKE 21 Flow Model being excluded. In FST solver the convective terms are omitted from the model, the viscous terms are omitted in the governing equations, and the influence of wind, atmospheric pressure variation, Coriolis and wave radiation stresses cannot be included.

For further information on the pros and cons of each of the 2D solvers, please refer to the MIKE 21 documentation. For more information regarding the equations employed in MIKE 21 FST please consult MIKE 21 Flow Model & MIKE 21 Flood Screening Tool Hydrodynamic Module - Scientific Documentation.


2D overland flow [Base]
DEM and 2D model area
2D model settings
Initial conditions
Close
Digitize rectangle
Set as extent of data set
Set as extent of raster
Generate GIS layer showing extent of 2D model
Generate GIS layer with vertical and horizontal lines along cell

Selection of 2D model

○ MIKE 21 Single Grid using rectangular cell solver
○ MIKE 21 Single Grid using Multi Cell solver Coarse/fine cell size ratio: 1
● MIKE 21 Flood Screening Tool

DEM

Raster layer with DEM: DEM.dfs2
Cell size: 2
Ground elevation band: Band_1
Meter
☐ Fill missing values
Maximum search radius: 10,000

2D model area

Model cell size: 2,000
Lower left corner X 1752378,819 Y 5946790,818
Upper right corner X 1753030,819 Y 5947854,818
Model Width 652,000 Model Height 1064,000
Number of cells 326 x 532 = 173432

Set model area coordinates

Select data set: msm_Node
Select raster:

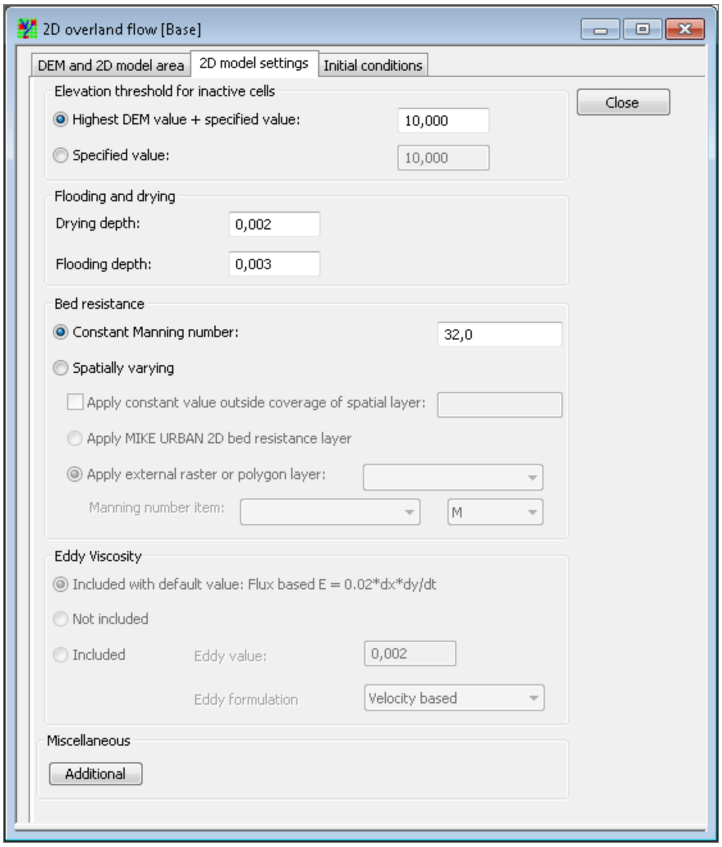
Figure 8.1 Selection of DEM and 2D model area

Flood Screening Tool in MIKE URBAN

In MIKE URBAN the Flood Screening Tool is included in the 2D overland flow editor. In the editor it is possible to enable FST by selecting this module radio button in the "DEM and 2D model area" tab, see Figure 8.1.

Enabling MIKE 21 Flood Screening Tool automatically disable the Eddy Viscosity settings which are consequently greyed out in the "2D model settings", see Figure 8.2.

Some advantages of FST compared to MIKE URBAN Flood is that it allows to run faster simulations and requires less information than normal MIKE URBAN Flood simulations.



2D overland flow [Base]

DEM and 2D model area | 2D model settings | Initial conditions

Close

Elevation threshold for inactive cells

● Highest DEM value + specified value: 10,000

○ Specified value: 10,000

Flooding and drying

Drying depth: 0,002

Flooding depth: 0,003

Bed resistance

● Constant Manning number: 32,0

○ Spatially varying

☐ Apply constant value outside coverage of spatial layer:

○ Apply MIKE URBAN 2D bed resistance layer

● Apply external raster or polygon layer:

Manning number item: M

Eddy Viscosity

● Included with default value: Flux based E = 0.02*dx*dy/dt

○ Not included

○ Included Eddy value: 0,002

Eddy formulation Velocity based

Miscellaneous

Additional

Figure 8.2 2D model settings while selecting FST.

8.3.2 Adding the DEM to the map view

The DEM to be applied is loaded to the map view using the Insert Layer button in the Layer Management toolbar or by choosing Insert Layer in Background Layers within the TOC. Both these options will accept DEM raster data stored in ESRI Grid format or in DHI dfs2 format. Depending on your GIS installation other raster formats could be available.

8.3.3 Defining the 2D model domain and the resolution

The following steps should be followed:

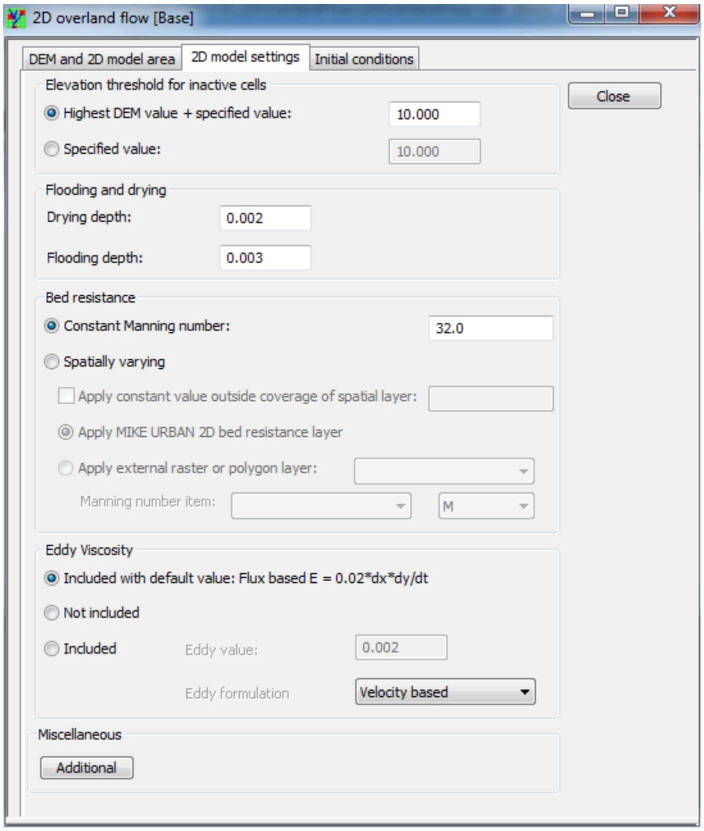
1. Select the desired raster layer as the DEM to be applied in the model.
2. Enter the X- and Y-coordinates for the lower left hand and upper right hand corner of the rectangular area to be modelled with the 2D model. Instead of manually entering the coordinates the buttons ‘Digitize rectangle’, ‘Set as extent of data set’ or ‘Set as extent of raster’ can be used to

set the coordinates corresponding to the rectangular extent of either a manually digitized polygon, a MIKE URBAN data set or a raster.

3. Enter the desired 2D model resolution. The model resolution does not have to be the same as the resolution of the DEM. However, if chosen to be the same and if the lower left hand corner of the model area coincides with the lower left hand corner of a cell in the DEM, then the DEM data used in the 2D model will be an exact copy of the input DEM data. Otherwise, the DEM data used in the 2D model will be interpolated from the DEM data provided as input.

8.3.4 2D model settings

Select MOUSE | 2D Overland Flow | 2D Model Settings ... followed by clicking on the tab '2D model settings' to open the dialog shown in Figure 8.3.



2D overland flow [Base]

Elevation threshold for inactive cells

● Highest DEM value + specified value: 10.000

○ Specified value: 10.000

Flooding and drying

Drying depth: 0.002

Flooding depth: 0.003

Bed resistance

● Constant Manning number: 32.0

○ Spatially varying

☐ Apply constant value outside coverage of spatial layer:

● Apply MIKE URBAN 2D bed resistance layer

○ Apply external raster or polygon layer:

Manning number item: M

Eddy Viscosity

● Included with default value: Flux based E = 0.02*dx*dy/dt

○ Not included

○ Included

Eddy value: 0.002

Eddy formulation: Velocity based

Miscellaneous

Figure 8.3 Editing 2D model settings

This dialog allows for the editing of the following 2D model settings:

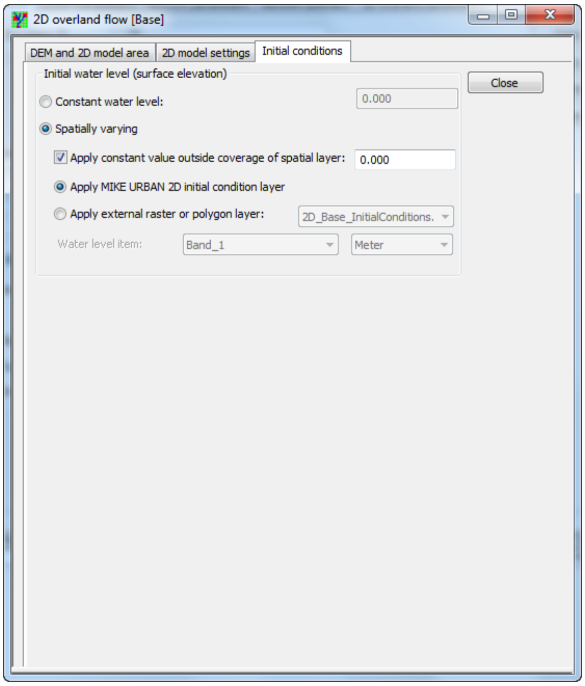
- **Elevation threshold for inactive cells**. The threshold value is applied for two purposes:
    - The threshold value for inactive cells is the DEM elevation above which flooding is assumed not to take place. For overland flow simulations, the threshold value is often set equal to the highest DEM value within the model area. However, if significant parts of the DEM within the model area are at high elevations, which will never get flooded, a lower value can be specified for the threshold value. Any calculation cell in the 2D model area with an elevation higher than the threshold value will remain dry (called true land in MIKE 21 terminology) throughout the simulation.
    - The first row/column of cells along the perimeter of the 2D model area DEM data will automatically be assigned the elevation threshold value as elevation. The only exception from this is where 2D boundaries are defined. See 8.3.9 for further details. The value will also be assigned to empty cells or areas of empty cells adjacent to the first row/column of cells along the perimeter.
- **Flooding and drying**. For 2D overland flow simulations, flooding and drying is an important mechanism as it reduces the simulation runtime by not having to spend computational efforts on simulating flows in dry cells. The drying and flooding depths are used for deciding when to exclude and to reintroduce cells in the computations. The drying depth must be less than the flooding depth.
- **Bed resistance**. The bed resistance coefficient in terms of the Manning (M) number ($m^{1/3}/s$) can be specified as a constant value applied throughout the 2D model area or as spatially varying for which two options are available:
    - Apply the MIKE URBAN 2D resistance layer: This is a polygon layer stored in the MIKE URBAN database. See 8.3.7 for further details on working with the 2D resistance layer in MIKE URBAN.
    - Apply external raster or polygon layer: If data on spatially varying Manning number is available either as a raster layer or as a polygon layer then the raster/polygon can be referenced and used directly after inserting the data to the map view as a background layer.

  In both cases a raster data set with the same extent and resolution as the 2D model will automatically be interpolated from the specified polygon or raster data. If these data do not cover the entire 2D model area MIKE URBAN will apply the specified constant value outside the coverage if the option “Apply constant value outside the coverage of the spatial layer” is selected. If coverage is incomplete and the option is not selected an error message will appear at the time of starting the simulation.

- **Eddy viscosity**. The eddy viscosity influences how eddies are simulated as a result of spatial velocity or flux gradients. Higher values of the eddy viscosity may also dampen numerical instabilities, and increasing the eddy viscosity is thus sometimes used as a method for stabilizing models. The eddy viscosity can be specified in three ways:
  - It can be included in terms of an automatically calculated, flux based value equal to $0.2 \cdot \Delta x \cdot \Delta y / \Delta t$, where Δx and Δy are the model cell size and Δt is the simulation time step.
  - It can be excluded.
  - It can be included as a user defined value, using either a flux based or velocity based equation.
- **Miscellaneous**. Read Section 8.3.9.

8.3.5 2D initial conditions

Select MOUSE | 2D Overland Flow | 2D Model Settings … followed by clicking on the tab ‘Initial Conditions’ to open the dialog shown in Figure 8.4



Initial water level (surface elevation)

○ Constant water level: 0.000

● Spatially varying

☑ Apply constant value outside coverage of spatial layer: 0.000

● Apply MIKE URBAN 2D initial condition layer

○ Apply external raster or polygon layer: 2D_Base_InitialConditions.

Water level item: Band_1 Meter

Figure 8.4 Specification of initial conditions for the 2D model

The initial water level (water surface elevation) applied in the 2D model can be specified as a constant value applied throughout the 2D model area or as spatially varying for which two options are available:

1. Apply the MIKE URBAN 2D initial conditions layer: This is a polygon layer stored in the MIKE URBAN database. See 8.3.6 for further details on working with the 2D initial conditions layer in MIKE URBAN.
2. Apply external raster or polygon layer: If data on spatially varying initial water level is available either as a raster layer or as a polygon layer then the raster/polygon can be referenced and used directly after inserting the data to the map view as a background layer.

In both cases a raster data set with the same extent and resolution as the 2D model will automatically be interpolated from the specified polygon or raster data. If these data do not cover the entire 2D model area MIKE URBAN will apply the specified constant value outside the coverage if the option “Apply constant value outside the coverage of the spatial layer” is selected. If coverage is incomplete and the option is not selected an error message will appear at the time of starting the simulation.

The specified initial water level (spatially constant or varying) will only apply to 2D cells with a ground elevation less than the initial water level. 2D cells with higher ground elevation will initially be dry.

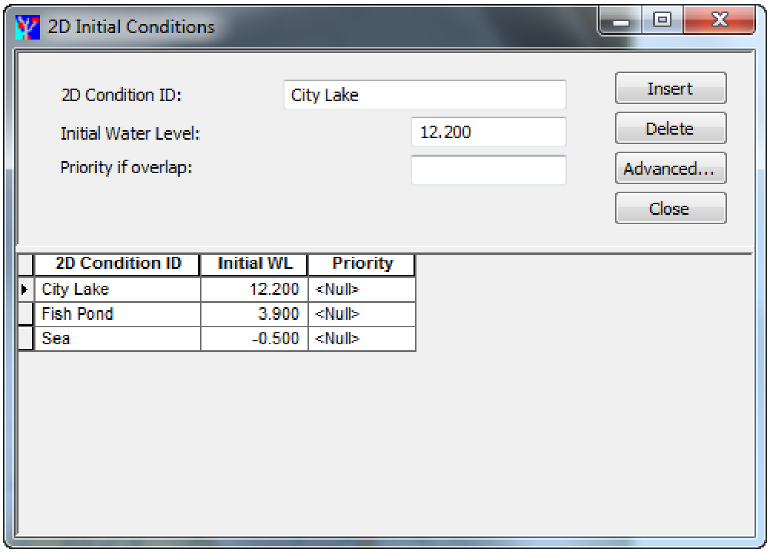
8.3.6 2D initial condition polygon layer

Spatially varying initial conditions for the 2D model are supported in two ways:

1. Using the polygons stored in the MIKE URBAN database and
2. Using external raster or polygon data.

The choice between these options is done in the Initial Conditions tab page on MOUSE | 2D Overland Flow | 2D Model Settings ... This section explains how to work with the 2D initial condition polygons stored in the MIKE URBAN database.

The form and grid view dialog for the table holding the polygons opens when choosing MOUSE | 2D Overland Flow | 2D Initial Conditions … as shown in Figure 8.5.



2D Condition ID: City Lake
Initial Water Level: 12.200
Priority if overlap:

2D Condition ID	Initial WL	Priority
City Lake	12.200	<Null>
Fish Pond	3.900	<Null>
Sea	-0.500	<Null>

Figure 8.5 Form and grid view for 2D Initial Conditions dialog

Initial condition polygons typically apply to areas such as lakes, ponds or strips of coastal waters located within the 2D model and which each have a more or less horizontal water surface initially.

If an area is covered by two or more polygons a priority can be specified for each polygon. The polygon with the highest priority (lowest numerical value) will be used in the simulation. If the priority is unspecified the polygon will have the lowest priority.

Similar to catchment polygons the initial condition polygons can be inserted and edited graphically with the Feature Edit toolbar.

Table 8.1 shows a full overview of the editor fields:

Table 8.1 Editor fields in form and grid view for 2D Initial Conditions dialog

Edit field	Description	Used or required by simulation	Field name in data structure
2D Condition ID	Name or ID of area	No	MUID

Table 8.1 Editor fields in form and grid view for 2D Initial Conditions dialog

Edit field	Description	Used or required by simulation	Field name in data structure
Initial Water Level	Initial 2D water level in the polygon area	Yes	InitialWaterLevel
Priority if overlap	Used to prioritise between polygons with an overlapping area. Low numerical value represents high priority.	Yes	Priority

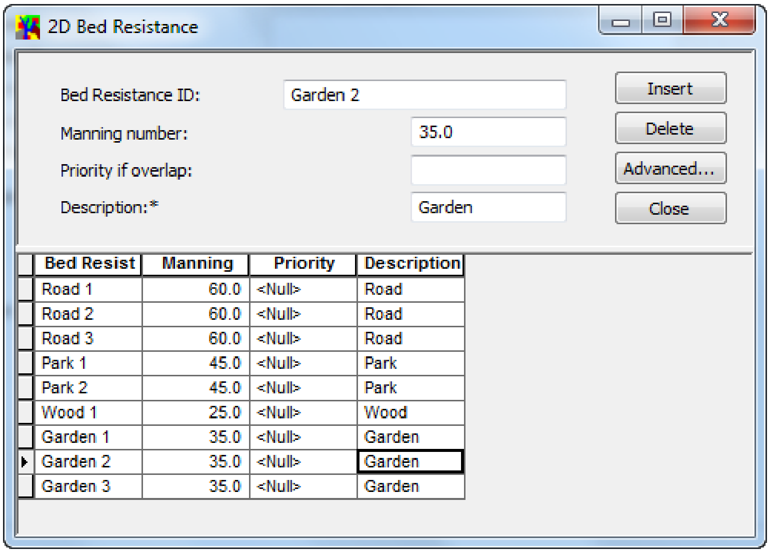
8.3.7 2D bed resistance polygon layer

Spatially varying bed resistance for the 2D model are supported in two ways:

1. Using the polygons stored in the MIKE URBAN database and
2. using external raster or polygon data.

The choice between these options is done in the 2D Model Settings tab page on MOUSE | 2D Overland Flow | 2D Model Settings ... This section explains how to work with the 2D bed resistance polygons stored in the MIKE URBAN database.

The form and grid view dialog for the table holding the polygons opens when choosing MOUSE | 2D Overland Flow | 2D Bed Resistance … as shown in Figure 8.6.



Bed Resistance ID: Garden 2
Manning number: 35.0
Priority if overlap:
Description:* Garden

Bed Resist	Manning	Priority	Description
Road 1	60.0	<Null>	Road
Road 2	60.0	<Null>	Road
Road 3	60.0	<Null>	Road
Park 1	45.0	<Null>	Park
Park 2	45.0	<Null>	Park
Wood 1	25.0	<Null>	Wood
Garden 1	35.0	<Null>	Garden
Garden 2	35.0	<Null>	Garden
Garden 3	35.0	<Null>	Garden

Figure 8.6 Form and grid view for 2D Bed Resistance dialog

Bed resistance polygons are applied to represent land use or other ground surface classification with a strong relation to bed resistance. The polygon table is typically populated by importing land use polygons for instance from a number of Shape files each holding multiple polygons with the same land use. The description field in the table can be used to store the land use and thereby, with the selection and field calculator tools, ease bulk changes of the bed resistance for all polygons of the same land use.

If an area is covered by two or more polygons a priority can be specified for each polygon. The polygon with the highest priority (lowest numerical value) will be used in the simulation. If the priority is unspecified the polygon will have the lowest priority.

Similar to catchment polygons the bed resistance polygons can be inserted and edited graphically with the Feature Edit toolbar.

Table 8.2 shows a full overview of the editor fields:

Table 8.2 Editor fields in form and grid view for 2D Bed Resistance dialog

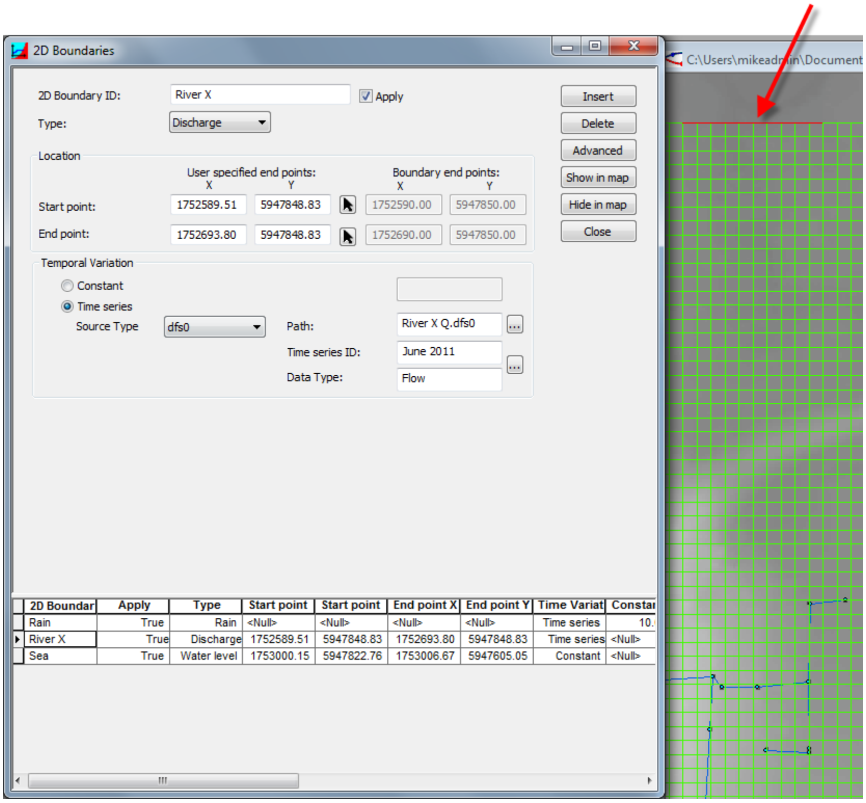
Edit field	Description	Used or required by simulation	Field name in data structure
Bed Resistance ID	Name or ID of area	No	MUID
Manning number	Manning number applied as bed resistance within the polygon	Yes	ManningNO
Priority if overlap	Used to prioritise between polygons with an overlapping area. Low numerical value represents high priority.	Yes	Priority
Description	Description, land use or other classification of the ground surface within the polygon	No	Description

8.3.8 2D boundaries

By default the 2D model has a closed boundary and water will not enter or leave the 2D model area across the perimeter of the 2D model areas. However, it is possible to define the following types of 2D boundaries:

- **Rain**: This applies in cases where rainfall should enter each 2D computational cell at each time step. Both constant rain and rain with spatial and temporal variation are supported. Rain directly on 2D cells is used as an alternative to using one of the rainfall-runoff models in MIKE URBAN.
- **Water level**: This applies to a stretch of the perimeter of the 2D model area and is typically applied where the 2D model perimeter crosses a larger water body such as a lake or the sea where the water level is known (constant or time varying). However, a water level boundary can also be applied as a free outlet allowing water to leave the 2D model area rather than backing up against the closed boundary.
- **Discharge**: This applies to a stretch of the perimeter of the 2D model area and is typically applied where the 2D model perimeter crosses a river or similar where the discharge is known (constant or time varying).

Boundaries for the 2D model is defined with the form and grid view which opens when selecting MOUSE | 2D Overland Flow | 2D Boundaries ... , see Figure 8.7.



2D Boundaries

2D Boundary ID: River X ☑ Apply

Type: Discharge

Location

	User specified end points: X	Y	Boundary end points: X	Y
Start point:	1752589.51	5947848.83	1752590.00	5947850.00
End point:	1752693.80	5947848.83	1752690.00	5947850.00

Temporal Variation

○ Constant

● Time series

Source Type: dfs0

Path: River X Q.dfs0

Time series ID: June 2011

Data Type: Flow

2D Boundar	Apply	Type	Start point	Start point	End point X	End point Y	Time Variat	Constar
Rain	True	Rain	<Null>	<Null>	<Null>	<Null>	Time series	10.
River X	True	Discharge	1752589.51	5947848.83	1752693.80	5947848.83	Time series	<Null>
Sea	True	Water level	1753000.15	5947822.76	1753006.67	5947605.05	Constant	<Null>

Figure 8.7 Form and grid view for 2D boundaries dialog. The arrow points to the graphical line showing the location of the boundary

There are three primary choices to make for each boundary:

1. **Type**: Choose between rain, water level and discharge
2. **Location**: This only applies for water level and discharge boundaries. The location of the boundary is defined by the x- and y-coordinate of the start and end point. The coordinates can either be typed in or defined by pointing and clicking at the start and end locations in the map view. The latter is done with the arrow-button shown to the right of the x-y edit fields. When clicking one of the arrow-buttons the 2D boundary dialog will disappear and the user is supposed to point and click at the intended start or end location (and the dialog will then reappear). MIKE URBAN will automatically snap to the nearest corner of a 2D cell along the perimeter of the 2D model, and the x- and y-coordinate of the snapped point will appear in the disabled X and Y edit fields named “Boundary end

points". If the user manually types in the coordinates then snapping will occur when choosing Advanced | Recompute, when the boundary is shown on the map or when a simulation is run. The location of 2D boundaries can be visualized in the map view by means of a red line between the snapped boundary end points. See Figure 8.7.

3. **Value**: The value of the boundary condition can be defined as constant or time varying with a .dfs0 file. Rain boundaries can additionally have a spatial variation in which case the data has to be provided in a .dfs2 file.

The discharge boundary is to be specified with positive numbers for flow in the same direction as the X and Y axis. This means that flow into the model area on the northern and eastern boundaries has to be specified as a negative number.

It should further be noted that 2D boundary files (dfs0 and dfs2) should be specified as floats and with an equidistant calendar axis see Chapter 4 'Time Series' for configuration. For a rainfall boundary the item type should be set to "Rainfall rate" and must only include one item in the file. For a spatially varying rain specified with a dfs2 file the file must have the same number of cells and cell size as the DEM.

Table 8.3 Editor fields in form and grid view for 2D Boundaries dialog

Edit field	Description	Used or required by simulation	Field name in data structure
2D Boundary ID	Name or ID of boundary	No	MUID
User defined end points	X- and Y- coordinate of the user defined start and end point of the boundary. Does not apply for rainfall.	No	BCStartX BCStartY BCEndX BCEndY
Boundary end points	X- and Y- coordinate of the snapped start and end point of the boundary. Does not apply for rainfall.	Yes	BCStartX_C BCStartY_C BCEndX_C BCEndY_C
Temporal variation	Choice of constant or temporal variation of boundary data.	Yes	VariationNo

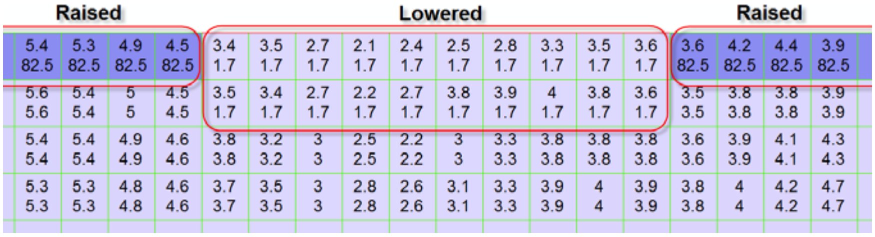
Table 8.3 Editor fields in form and grid view for 2D Boundaries dialog

Edit field	Description	Used or required by simulation	Field name in data structure
Source type	Choice of spatially constant or varying boundary data.	Yes	BridgeTypeNo
Path	Name and path of file with boundary data.	Yes	TSConnection
Time series ID	Name of selected item in boundary data file.	Yes	TimeSeriesName
Data type	Type of selected item in boundary data file.	Yes	DataTypeName

8.3.9 Automatic model adjustments along water level and discharge boundaries

Adjustment of DEM

Along the 2D model perimeter the ground elevation in the first row/column of 2D cells is automatically raised to a high elevation (elevation threshold for active cells) to ensure zero flow across the perimeter. However, this does not apply along stretches where a 2D water level or discharge boundary is defined. Instead, the ground elevation is lowered automatically to ensure suitable numerics of the simulation at the boundary, because a boundary must be wet all the time. By default the lowering is 0.5 meter and it applies to the first two rows/columns parallel to the 2D boundary. All 2D cells are lowered to the same value which equals the lowering value subtracted from the lowest cell in the first non-adjusted row/column (by default the 3rd) along the 2D boundary. The lowering and the width of the indentation can be adjusted by clicking the “Additional” button in MOUSE | 2D Overland Flow | 2D Model Settings ... Figure 8.8 shows an example of how the DEM is adjusted along a boundary.



Raised				Lowered										Raised			
5.4 82.5	5.3 82.5	4.9 82.5	4.5 82.5	3.4 1.7	3.5 1.7	2.7 1.7	2.1 1.7	2.4 1.7	2.5 1.7	2.8 1.7	3.3 1.7	3.5 1.7	3.6 1.7	3.6 82.5	4.2 82.5	4.4 82.5	3.9 82.5
5.6 5.6	5.4 5.4	5 5	4.5 4.5	3.5 1.7	3.4 1.7	2.7 1.7	2.2 1.7	2.7 1.7	3.8 1.7	3.9 1.7	4 1.7	3.8 1.7	3.6 1.7	3.5 3.5	3.8 3.8	3.8 3.8	3.9 3.9
5.4 5.4	5.4 5.4	4.9 4.9	4.6 4.6	3.8 3.8	3.2 3.2	3 3	2.5 2.5	2.2 2.2	3 3	3.3 3.3	3.8 3.8	3.8 3.8	3.8 3.8	3.6 3.6	3.9 3.9	4.1 4.1	4.3 4.3
5.3 5.3	5.3 5.3	4.8 4.8	4.6 4.6	3.7 3.7	3.5 3.5	3 3	2.8 2.8	2.6 2.6	3.1 3.1	3.3 3.3	3.9 3.9	4 4	3.9 3.9	3.8 3.8	4 4	4.2 4.2	4.7 4.7

Figure 8.8 Example of automatic DEM adjustment along an open 2D boundary spanning 10 2D cells. In each 2D cell the original DEM (upper) and adjusted DEM (lower) is shown

Adjustment of initial conditions

The lowered indentation explained above must be wet initially and MIKE URBAN ensures that by automatically adjusting the initial conditions such that the indentation is initially filled with water. This requires a spatially varying initial condition which is automatically generated by MIKE URBAN. Hence, although a constant initial water level is selected in MOUSE | 2D Overland Flow | 2D Model Settings ... then the 2D initial conditions that are actually applied will be spatially varying if there any open boundaries are defined. The user specified value will still be applied for the entire model area where an adjustment of the DEM has not occurred.

8.3.10 Defining couplings

Defining the couplings between the pipe flow model and the 2D overland flow model involves the following steps:

1. Selection of the pipe flow components to be coupled. These are typically mostly manholes, but it is also possible to establish couplings for basins, weirs, pumps and outlets.
2. Selection of the specific calculation cells in the 2D overland flow model, which the selected pipe flow model components should be coupled to. This selection of 2D cells is done automatically based on number of cells or a search radius (see coupling parameters below for details).
3. For manholes and basins a flow formula is applied for calculating the exchange of flow between the pipe flow model and the overland flow model. The preferred flow formula and the associated parameters have to be selected.

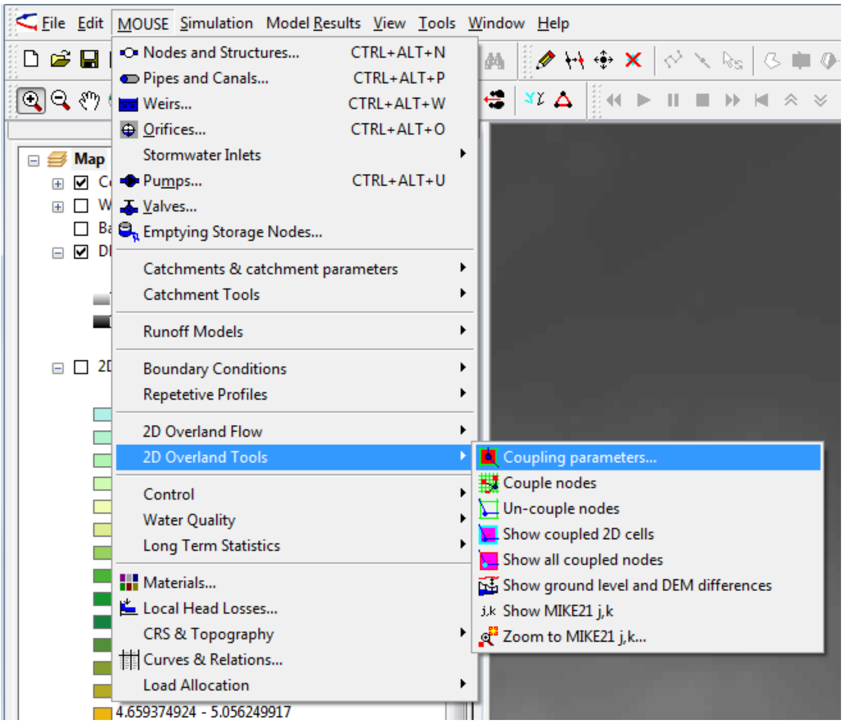
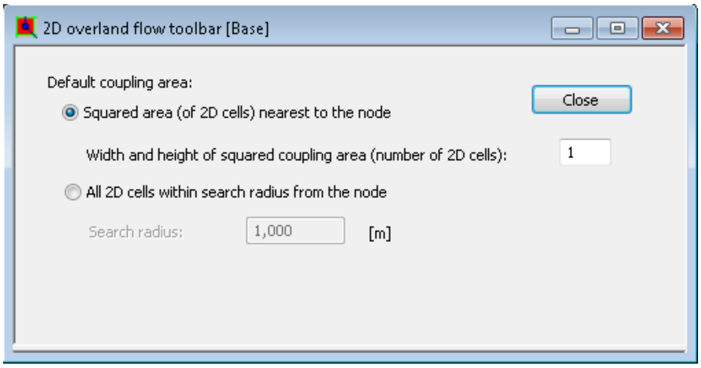

File Edit MOUSE Simulation Model Results View Tools Window Help
Nodes and Structures... CTRL+ALT+N
Pipes and Canals... CTRL+ALT+P
Weirs... CTRL+ALT+W
Orifices... CTRL+ALT+O
Stormwater Inlets
Pumps... CTRL+ALT+U
Valves...
Emptying Storage Nodes...
Catchments & catchment parameters
Catchment Tools
Runoff Models
Boundary Conditions
Repetetive Profiles
2D Overland Flow
2D Overland Tools
Control
Water Quality
Long Term Statistics
Materials...
Local Head Losses...
CRS & Topography
Curves & Relations...
Load Allocation
Coupling parameters...
Couple nodes
Un-couple nodes
Show coupled 2D cells
Show all coupled nodes
Show ground level and DEM differences
Show MIKE21 j,k
Zoom to MIKE21 j,k...
Map
4.659374924 - 5.056249917

Figure 8.9 Sub-menu for 2D Overland Land Tools

The following set of tools found in the sub-menu MOUSE|2D Overland Tools (see Figure 8.9) are available for defining the couplings:

- **Coupling parameters**
  Selecting this will open the dialog shown in Figure 8.10.

  When coupling a pipe model node to the 2D model, a coupling area covering the spatial extent of the 2D cells involved in the coupling has to be defined. The coupling area will automatically be defined by the tool "Couple nodes". The selection in this dialog defines how many 2D model cells the coupling area will span. There are two choices. 1) Either a squared coupling area with the width/height as the specified number of cells. 2) A coupling area defined by all 2D cells within the specified search radius from the node in question. If no 2D cells will have a cell centre within the specified coupling area, then the 2D cell containing the node in question will be selected as the coupling area.


2D overland flow toolbar [Base]

Default coupling area:

● Squared area (of 2D cells) nearest to the node

Width and height of squared coupling area (number of 2D cells): 1

○ All 2D cells within search radius from the node

Search radius: 1,000 [m]

Figure 8.10 Dialog for parameters for the 2D overland flow tools

- Couple nodes
  This tool applies to manholes, basins and outlets that are not already coupled. If any nodes are selected the tool will apply only to those selected nodes - otherwise to all. Once a node is coupled the following will happen:
  - The node will be marked as coupled and it will show-up as such in the tab '2D Overland' in the dialog MOUSE|Nodes.
  - A coupling area polygon will be defined and an entry will be added to the table 'Coupled 2D Areas'. If selected for drawing the coupling area will show up in the map view.
  - A line between the node and the centre of the coupling area will be defined and an entry will be added to the table 'Coupled 2D Connections'. If selected for drawing the connection line will show up in the map view.

  Figure 8.11 shows an example of three nodes which have been coupled to respectively a single 2D cell, to 3x3 2D cells and to the 2D cells within a circular search radius.

  Once a coupling area has been defined automatically, its shape and location can be manually edited using the tools in the Feature Edit toolbar.

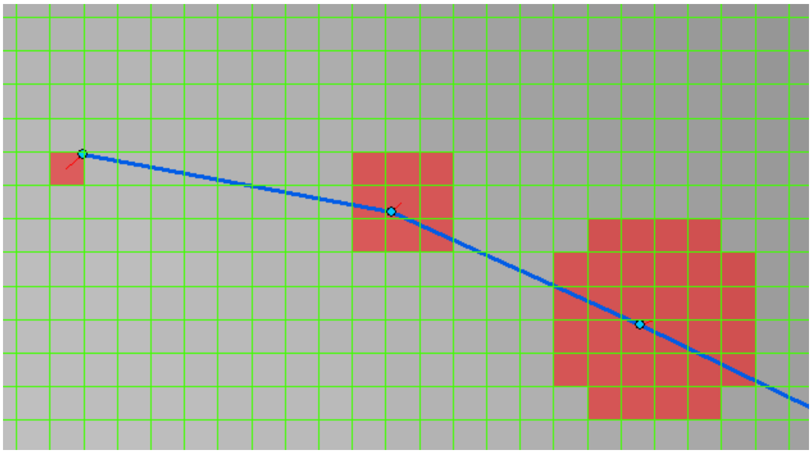


Figure 8.11 Nodes coupled to varying numbers of 2D cells

- **Un-coupled nodes**
  If any coupled nodes are selected this tool will apply only to those - otherwise to all coupled nodes. The tool will un-couple nodes by taking the following actions:
  - The node will be marked as un-coupled and show-up as such in the tab '2D Overland' in the dialog MOUSE|Nodes.
  - The associated coupling area polygon will be deleted.
  - The associated coupling connection will be deleted.

- **Show coupled 2D cells**
  This tool will select all coupling area polygons in the table "Coupled 2D Areas". If any nodes are selected, the tool will only select the coupling area polygons associated to the selected nodes.

- **Show all coupled nodes**
  This tool will select all coupled nodes.

- Show ground level and DEM differences
  If any coupled nodes are selected, this tool will apply only to those - otherwise to all coupled nodes. This tool will calculate the difference between the ground level in each node and the elevation in the raster layer selected as the DEM. The differences will be added to the map view as a point layer.

- Show MIKE 21 j,k
  Mostly there is no reason to be concerned about the so-called j and k coordinates in MIKE 21. These are the horizontal and vertical indices of the 2D cells in the MIKE 21 model grid. The cell in the lower left hand corner of the 2D model area is defined as (j,k)=(0,0).

  After clicking this menu item, the MIKE 21 j,k coordinates at the position of the cursor, when pointing in the map view, will appear in the status bar alongside with the X,Y coordinates and the map scale. See Figure 8.12.

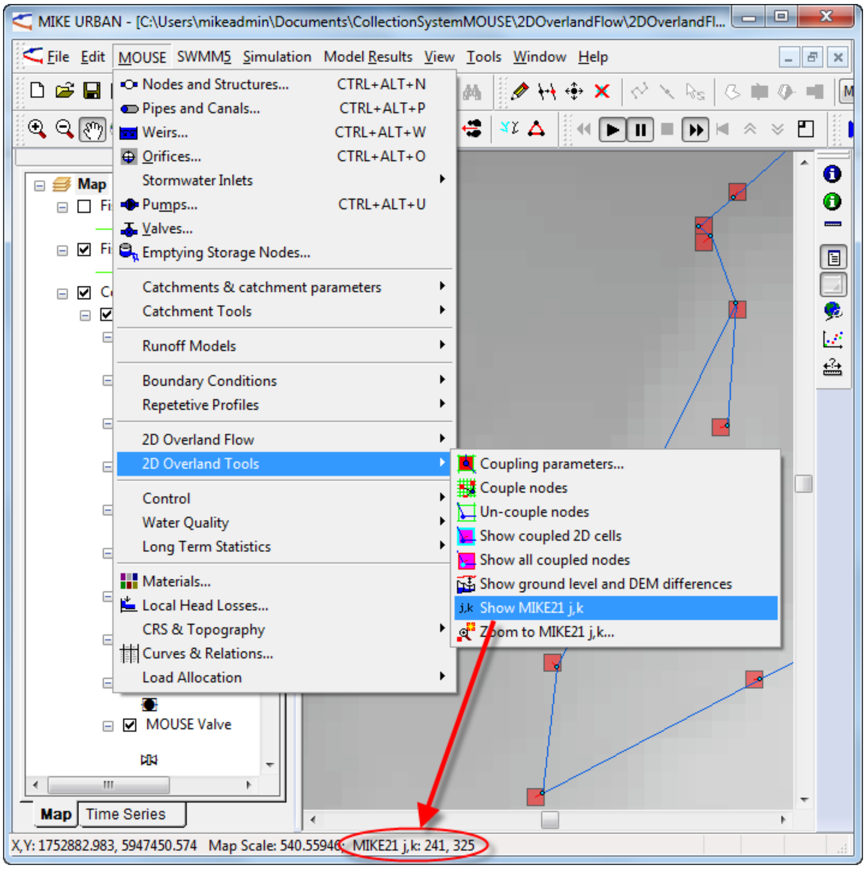

MIKE URBAN - [C:\Users\mikeadmin\Documents\CollectionSystemMOUSE\2DOverlandFlow\2DOverlandFl...
File Edit MOUSE SWMM5 Simulation Model Results View Tools Window Help
Nodes and Structures... CTRL+ALT+N
Pipes and Canals... CTRL+ALT+P
Weirs... CTRL+ALT+W
Orifices... CTRL+ALT+O
Stormwater Inlets
Pumps... CTRL+ALT+U
Valves...
Emptying Storage Nodes...
Catchments & catchment parameters
Catchment Tools
Runoff Models
Boundary Conditions
Repetetive Profiles
2D Overland Flow
2D Overland Tools
Control
Water Quality
Long Term Statistics
Materials...
Local Head Losses...
CRS & Topography
Curves & Relations...
Load Allocation
Coupling parameters...
Couple nodes
Un-couple nodes
Show coupled 2D cells
Show all coupled nodes
Show ground level and DEM differences
Show MIKE21 j,k
Zoom to MIKE21 j,k...
Map
MOUSE Valve
Map Time Series
X,Y: 1752882.983, 5947450.574 Map Scale: 540.55946 MIKE21 j,k: 241, 325

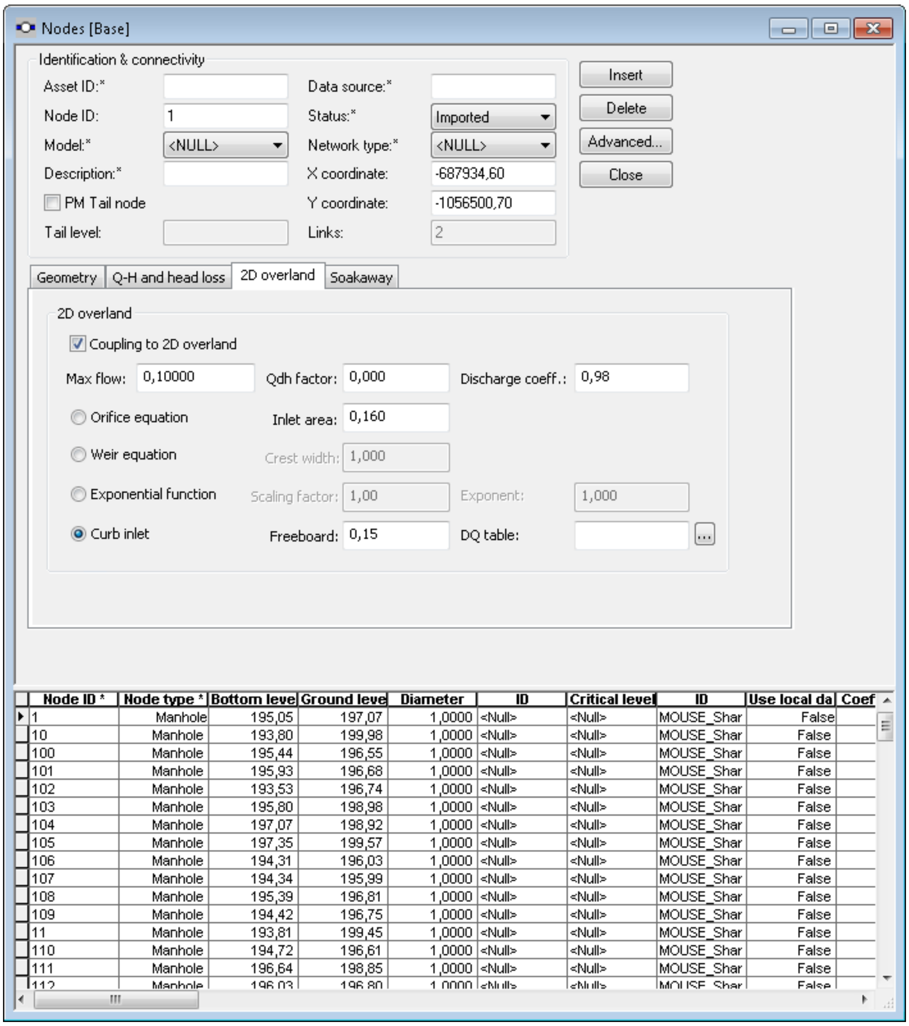
Figure 8.12 Showing MIKE 21 j,k coordinates in status bar

- Zoom to MIKE 21 j,k
  This tool will prompt the user for a j,k coordinate set and subsequently the map view will zoom to the specified location with a zoom level such that approximately 10x10 2D cells are visible.

8.3.11 Flow parameters at manholes and basins

Urban sewer network models can be coupled to MIKE 21 two dimensional models to simulate urban flood patterns in MIKE FLOOD, the unidimensional model can now be simulated using the MIKE 1D engine whilst the 2D model can be run using the MIKE 21 engine.

Once a manhole or basin has been coupled, the node attribute ‘Coupling to 2D Overland’ will be selected. See Figure 8.13. On this tab the user can set various parameters for how the exchange flow between the pipe flow model and the overland flow model is calculated. For more information, please refer to MIKE FLOOD User Manual.



Identification & connectivity

Asset ID:*
Node ID: 1
Model:* <NULL>
Description:*
☐ PM Tail node
Tail level:
Data source:*
Status:* Imported
Network type:* <NULL>
X coordinate: -687934,60
Y coordinate: -1056500,70
Links: 2

2D overland

☑ Coupling to 2D overland
Max flow: 0,10000
Qdh factor: 0,000
Discharge coeff.: 0,98
○ Orifice equation
Inlet area: 0,160
○ Weir equation
Crest width: 1,000
○ Exponential function
Scaling factor: 1,00
Exponent: 1,000
● Curb inlet
Freeboard: 0,15
DQ table:

Node ID *	Node type *	Bottom leve	Ground leve	Diameter	ID	Critical level	ID	Use local da	Coef
1	Manhole	195,05	197,07	1,0000	<Null>	<Null>	MOUSE_Shar	False	
10	Manhole	193,80	199,98	1,0000	<Null>	<Null>	MOUSE_Shar	False	
100	Manhole	195,44	196,55	1,0000	<Null>	<Null>	MOUSE_Shar	False	
101	Manhole	195,93	196,68	1,0000	<Null>	<Null>	MOUSE_Shar	False	
102	Manhole	193,53	196,74	1,0000	<Null>	<Null>	MOUSE_Shar	False	
103	Manhole	195,80	198,98	1,0000	<Null>	<Null>	MOUSE_Shar	False	
104	Manhole	197,07	198,92	1,0000	<Null>	<Null>	MOUSE_Shar	False	
105	Manhole	197,35	199,57	1,0000	<Null>	<Null>	MOUSE_Shar	False	
106	Manhole	194,31	196,03	1,0000	<Null>	<Null>	MOUSE_Shar	False	
107	Manhole	194,34	195,99	1,0000	<Null>	<Null>	MOUSE_Shar	False	
108	Manhole	195,39	196,81	1,0000	<Null>	<Null>	MOUSE_Shar	False	
109	Manhole	194,42	196,75	1,0000	<Null>	<Null>	MOUSE_Shar	False	
11	Manhole	193,81	199,45	1,0000	<Null>	<Null>	MOUSE_Shar	False	
110	Manhole	194,72	196,61	1,0000	<Null>	<Null>	MOUSE_Shar	False	
111	Manhole	196,64	198,85	1,0000	<Null>	<Null>	MOUSE_Shar	False	
112	Manhole	196,03	196,80	1,0000	<Null>	<Null>	MOUSE_Shar	False	

Figure 8.13 2D Overland flow tab in the nodes dialog

- **Max flow**
  This parameters gives an upper limit to the discharge that can flow through the link. If the discharge calculated from the flow equation exceeds this value, then the discharge is reduced to this value. It determines how much flow can be exchanged between 2D and 1D models.
- **Qdh factor**
  A parameter indicating the water level at which the calculated discharge should be surpressed. May be used for stabilizing setups where the instability is caused by a small difference in the water level in the two coupled models. The suppression is calculated as the water level difference divided by the Qdh factor to the power of 2.

Calculation method

The exchange flow between the 1D and 2D models may be calculated using 4 different methods:

- Orifice equation: The flow is governed by a standard orifice equation.
- Weir equation: The flow is described through a weir equation.
- Exponential function: The flow is governed by a simple exponential function.
- Curb Inlet: The flow between MIKE URBAN and MIKE 21 is transferred at a grate or inlet from a surface overland flow network to the sub-surface pipe network. The transfer capacity of the connection is specified as a DQ-relation (tabular data type).
  When water is transferred to the surface from the sewer pipe network the flow is calculated with the orifice equation

Depending on the calculation method used, additional information is required.

- **Inlet area**
  The inlet area is only used when applying the orifice equation for describing the flow exchange between the 1D model and the 2D model. The greater the cross sectional area the greater the conveyance capacity of the coupling. This parameter corresponds physically to the area of the manhole cover. For a manhole the inlet area used in the calculation will be the smallest value of the specified inlet area and the inlet area calculated from the manhole diameter. For a basin the specified inlet area will always be used.

- **Orifice coefficient**
  The orifice coefficient is a non-dimensional factor that may be used to scale the orifice flow.

- **Crest width**
  The crest width is used when the flow exchange is described through a weir equation. The weir crest should typically be the circumference of the manhole cover.

- **Scaling factor**
  This factor is only applicable if the simple exponential expression is used to describe the flow exchange. The scaling factor has a linear effect on the discharge. Thus, an increase in this value will generate a larger flow for a certain water level difference between the 1D and 2D model.

- **Exponent**
  Only applicable if the simple exponential expression is used to describe the flow exchange. An increase in the exponent factor has a strong impact on the discharge. An increase in this value will generate a larger flow for a specific water level difference between the 1D and 2D model.

- **Freeboard**
  The freeboard (see Figure 8.14) defines a critical water level (Invert level- Freeboard) at the connection node in the pipe network below which the defined DQ relation apply. For submerged and reverse flow (surcharge), the transfer capacity of the connection reverts to a standard orifice relationship.

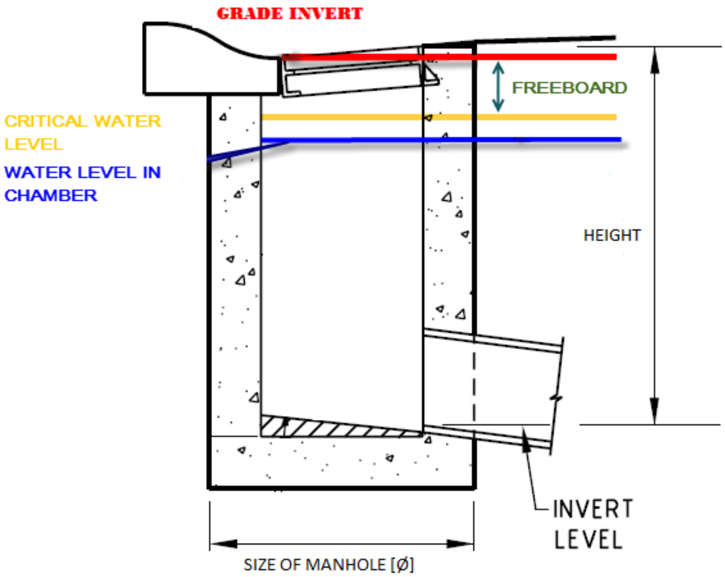


Figure 8.14 Freeboard detail in a Curb Inlet

The freeboard describes the distance (depth) from the Grade Invert Level to the Critical Water Level (i.e. Freeboard = Grade Invert Level - Critical Water Level). The Critical Water Level is the level above which the basin is considered to be 'drowned' such that the DQ relationship for transferring water from the surface to the node no longer applies. For submerged and reverse flow conditions the exchange of water across the coupling reverts to a standard orifice equation.

- **Capacity curve DQ Relation**
  The DQ relation specifies the depth based capacity curve for the curb inlet. Values must be monotonously increasing in depth and discharge starting at (0,0). For depths in excess of the maximum value specified in the last row of the table, the last corresponding discharge is used. Positive discharge is considered flow from MIKE 21 into MIKE URBAN (overland to sewer network).

8.3.12 Outlets

An outlet coupled to the 2D overland flow model does not require other parameters than the definition of the coupling area itself. The average water levels in the 2D cells to which the coupling is made will then be applied as a water level boundary. The same algorithm will apply as when a water level boundary is specified in an outlet in the pipe flow model. This will result in an outflow through the outlet (positive or negative) which will correspondingly apply as a point inflow (positive or negative) distributed to the 2D cells in the coupling area. For more information, please refer to MIKE FLOOD User Manual.

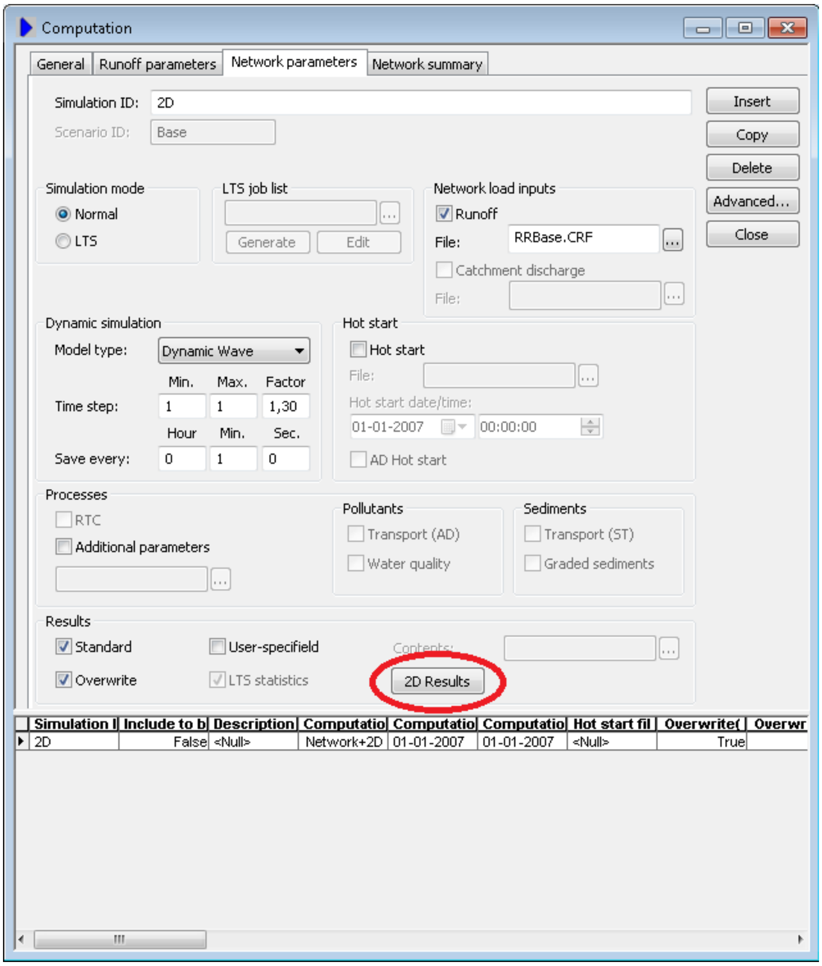
8.3.13 Pumps and weirs

A pump or a weir can be coupled to the 2D model if the To-node is not defined. When coupled to the 2D model, the weir/pump discharge will be added to the 2D model as a point inflow distributed to the 2D cells in the coupling area. The pump/weir discharge will always be independent of the water level in the 2D model and the flow direction in weir/pump couplings will always be from pipe flow model to 2D model. For more information, please refer to MIKE FLOOD User Manual.

8.4 Running the Combined 1D and 2D Simulations

8.4.1 Setting the simulation type and requesting 2D results

To run the combined pipe flow and overland flow simulation three issues needs to be addressed in the simulation dialog as shown in Figure 8.15. Firstly, the simulation type needs to be "Network + 2D overland". Secondly, the time step in the 2D model and the request for output from the 2D model have to be defined in the dialog "2D Overland Results" as shown in Figure 8.16. This dialog can be opened from the "2D Results" button located in the "Network parameters" tab.


Computation
General
Runoff parameters
Network parameters
Network summary
Insert
Copy
Delete
Advanced...
Close
Generate
Edit
2D Results

Simulation ID: 2D

Scenario ID: Base

Simulation mode

● Normal

○ LTS

LTS job list

Network load inputs

☑ Runoff

File: RRBase.CRF

☐ Catchment discharge

File:

Dynamic simulation

Model type: Dynamic Wave

	Min.	Max.	Factor
Time step:	1	1	1,30
	Hour	**Min.**	**Sec.**
Save every:	0	1	0

Hot start

☐ Hot start

File:

Hot start date/time: 01-01-2007 00:00:00

☐ AD Hot start

Processes

☐ RTC

☐ Additional parameters

Pollutants

☐ Transport (AD)

☐ Water quality

Sediments

☐ Transport (ST)

☐ Graded sediments

Results

☑ Standard

☐ User-specifield

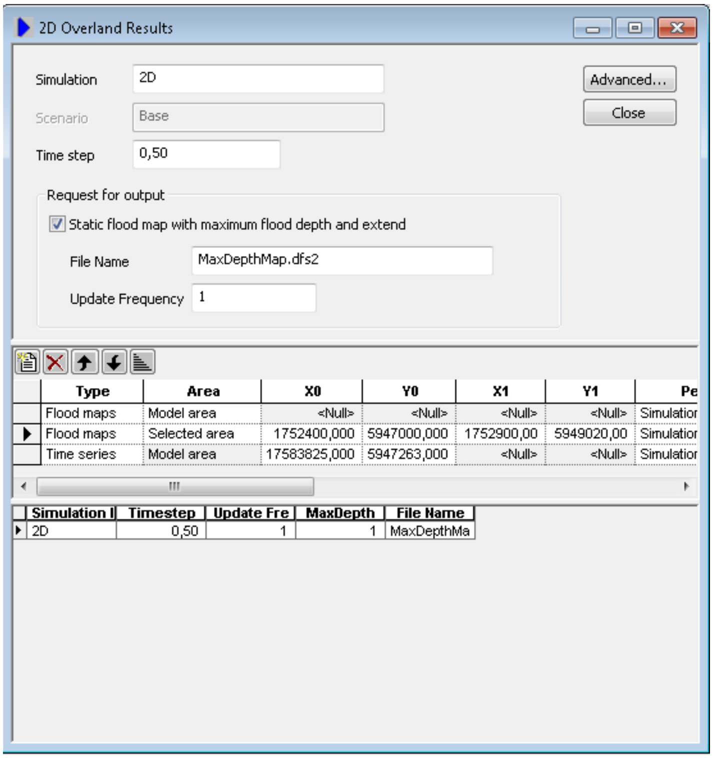
☑ Overwrite

☑ LTS statistics

Contents:

Simulation I	Include to b	Description	Computatio	Computatio	Computatio	Hot start fil	Overwrite(	Overwr
2D	False	<Null>	Network+2D	01-01-2007	01-01-2007	<Null>	True	

Figure 8.15 2D Overland flow options in the computation dialog



Simulation: 2D

Scenario: Base

Time step: 0,50

☑ Static flood map with maximum flood depth and extend

File Name: MaxDepthMap.dfs2

Update Frequency: 1

Type	Area	X0	Y0	X1	Y1	Pe
Flood maps	Model area	<Null>	<Null>	<Null>	<Null>	Simulatior
Flood maps	Selected area	1752400,000	5947000,000	1752900,00	5949020,00	Simulatior
Time series	Model area	17583825,000	5947263,000	<Null>	<Null>	Simulatior

Simulation I	Timestep	Update Fre	MaxDepth	File Name
2D	0,50	1	1	MaxDepthMa

Figure 8.16 Selection of time step and request for output in 2D model

In the tab ‘2D Overland Results’ the 2D time step is specified. The request for output is specified as a single 2D output file with maximum flood depth and in the table below, a number of output files with the flood depth at multiple time steps through the simulation. For the latter the following has to be specified for each requested result file:

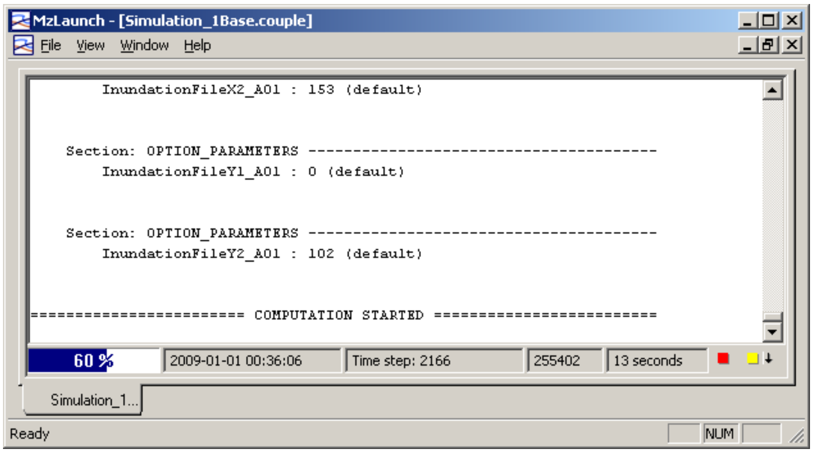
Table 8.4 Specifying the content for the result files

Item name	Explanation
Type	The choice between the following output files both containing multiple time steps: Flood maps: A 2D output file in .dfs2 raster format. Time series: A time series output file in .dfs0 format.
Model area	This is only enabled for flood map output files. The user can choose be-tween: Model area: The output file will then cover the full 2D model area. Selected area: The output file will then cover only a sub-region of the full 2D model area. The sub-region is a rectangle defined by the following two set of X,Y coordinates:
X0, Y0 X1, Y1	Type = Flood Maps: Only enabled if the model area is chosen as selected area. The lower left and upper right corners of the sub-region from which 2D output will be generated is defined by the two coordinate sets (X0, Y0) and (X1, Y1) Type = Time series: Results are saved from the location (X0, Y0). (X1, Y1) is not used and thus disabled.
Period	Two choices are available: Simulation period: Output will be saved for the entire simulation period Used defined; User wants output to be saved for a period shorter than the simulation period. The period is specified in the following two items
Start End	Only enabled if the Period is selected as User Defined. Start and End is the date and time for desired period for saving results
NoSteps	An integer value specifying the number of time steps at which results are saved
FileName	Name of the result file to be saved. The file name cannot be changed by the user.
Title	User specified title which is saved as header information in the result file
Depth, Level, Velocity & Flux	Option for saving water depth, level (water surface elevation), velocity and flux (P, Q) respectively

The standard MIKE 21 results (H,P,Q) will be generated when Depth and Flux are chosen. If you would like more information on the result type flux (P and Q), you can check the MIKE 21 documentation.

8.4.2 Starting simulation

After pushing the button Start Simulation in the dialog shown in Figure 8.15 MIKE URBAN data will be exported as for a regular MOUSE simulation. Following that export, the MIKE FLOOD simulation progress window shown in Figure 8.17 will appear.



```
        InundationFileX2_A01 : 153 (default)


    Section: OPTION_PARAMETERS ---------------------------------------
        InundationFileY1_A01 : 0 (default)


    Section: OPTION_PARAMETERS ---------------------------------------
        InundationFileY2_A01 : 102 (default)


======================== COMPUTATION STARTED =========================
```

60 % | 2009-01-01 00:36:06 | Time step: 2166 | 255402 | 13 seconds

Figure 8.17 Simulation progress window for MIKE FLOOD

8.5 Visualising Simulation Results

The result files generated by the 2D overland flow simulations are time series saved in .dfs0 files and raster data saved in .dfs2 files with water depth or velocities. 2D results in time series files are shown as any other time series file. This section explains how to visualize results in .dfs2 files.

The .dfs2 raster files may contain a single time step or multiple time steps. The former is the case for files containing the maximum flood extend. Such should be added as a raster layer to the map view using the Insert Layer button in the Layer Management toolbar. Dfs2 files with multiple time steps are visualized as explained in the following.

In the main menu bar, choose Model Result > Load Results > Overland Results in order to select the 2D overland result file. This will add information about the file content to the MIKE URBAN session, but as .dfs2 files are potentially very large the data from the individual time steps are not loaded to memory. Thus, nothing will be added to the TOC or the map view when a

.dfs2 files is selected. There are three ways to subsequently visualize the content of a .dfs2 file:

1. As a raster layer in the map view. To do so, select Model Results > Add Results Layer. Subsequently, select the desired.dfs2 file and data type which then will be added to the TOC and map view. The map view will show data from the time step selected with the Animation toolbar. See Figure 8.18.
2. As a time series plot extracted at a particular location. To do so, in the Simulation toolbar click on the Time Series button and then select the desired .dfs2 file and data type. Subsequently, move the cursor to the location in the map view from where a time series should be extracted from the .dfs2 file. See Figure 8.18.
3. As a water surface elevation line in a longitudinal profile view. To do so, firstly a longitudinal profile view should be opened. In here right-click and choose Properties. In the Properties dialog the Raster tab allows for adding raster data to the profile view. See Figure 8.19.

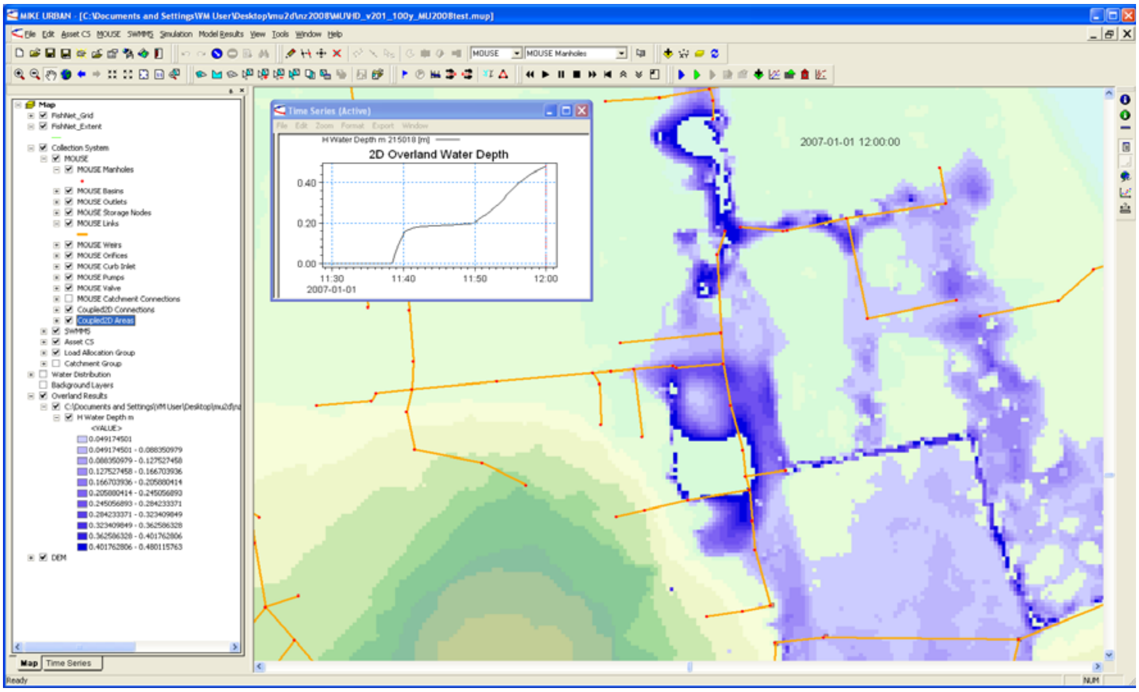

MIKE URBAN - [C:\Documents and Settings\VM User\Desktop\mu2d\nz2008\MUVID_v201_100y_MU2008test.mup]
File Edit Asset CS MOUSE SWMM5 Simulation Model Results View Tools Window Help
MOUSE
MOUSE Manholes
Map
FishNet_Grid
FishNet_Extent
Collection System
MOUSE
MOUSE Manholes
MOUSE Basins
MOUSE Outlets
MOUSE Storage Nodes
MOUSE Links
MOUSE Weirs
MOUSE Orifices
MOUSE Curb Inlet
MOUSE Pumps
MOUSE Valve
MOUSE Catchment Connections
Coupled2D Connections
Coupled2D Areas
SWMM5
Asset CS
Load Allocation Group
Catchment Group
Water Distribution
Background Layers
Overland Results
C:\Documents and Settings\VM User\Desktop\mu2d\n
H Water Depth m
<VALUE>
0.0491174501
0.0491174501 - 0.0883350979
0.0883350979 - 0.127527458
0.127527458 - 0.166703936
0.166703936 - 0.205880414
0.205880414 - 0.245056893
0.245056893 - 0.284233371
0.284233371 - 0.323409849
0.323409849 - 0.362586328
0.362586328 - 0.401762806
0.401762806 - 0.480115763
DEM
Map
Time Series
Ready
Time Series (Active)
File Edit Zoom Format Export Window
H Water Depth m 215018 [m]
2D Overland Water Depth
0.40
0.20
0.00
11:30
2007-01-01
11:40
11:50
12:00
2007-01-01 12:00:00
NUM

Figure 8.18   2D flood results shown as raster data and time series

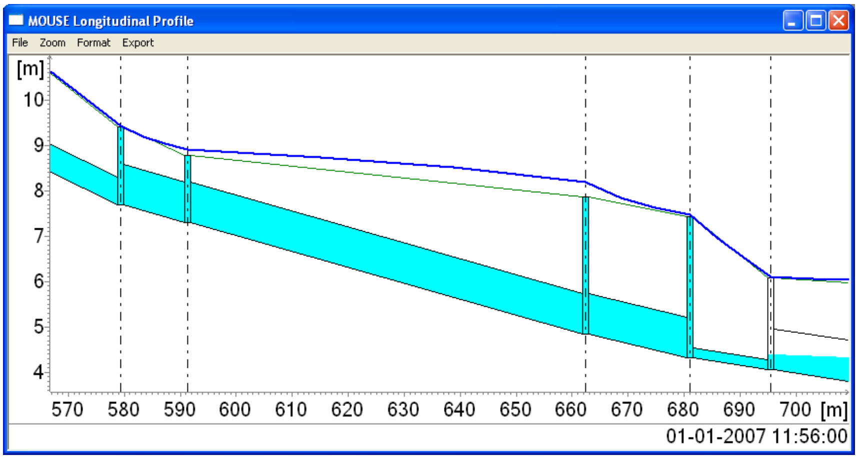

MOUSE Longitudinal Profile
File Zoom Format Export
[m]
10
9
8
7
6
5
4
570 580 590 600 610 620 630 640 650 660 670 680 690 700 [m]
01-01-2007 11:56:00

Figure 8.19 2D flood results shown as water surface profile in longitudinal profile

8.6 Understanding the Files Generated

When starting a combined 1D and 2D simulation a number of input files will be generated for MIKE FLOOD. In most cases these files do not need to be altered. However, in some cases the files generated may be edited further with MIKE Zero e.g. if a MIKE 11 component should be added or some of the MIKE 21 parameters need further adjustment. The following provides an explanation of the files being generated.

Table 8.5 Files generated

Filename	**Filetype**	**Explanation**
<Simulation ID> <ScenarioID> .couple	MIKE FLOOD input file	This file is an input file for the MIKE FLOOD simulation and it can be edited further with the MIKE FLOOD editor in MIKE ZERO. In short, the file contains reference to the MIKE 21 input file and the MIKE URBAN .mex file as well as how these two models are coupled.
<Simulation ID> <ScenarioID> .m21	MIKE 21 input file	This file is an input file for the MIKE 21 component of the MIKE FLOOD simulation and it can be edited further with the MIKE 21 editor in MIKE ZERO. In short, the file contains the parameters for MIKE 21 as specified in the MIKE URBAN dialog for 2D Overland Flow.

Table 8.5 Files generated

Filename	Filetype	Explanation
<Simulation ID> <ScenarioID> Coarse .dfs2	Raster file in dfs2 format	This file contains the DEM data for the 2D model area selected. Data in the file is extracted from the original DEM specified in MIKE URBAN at the requested model resolution.
<Simulation ID> <ScenarioID> Fine .dfs2	Raster file in dfs2 format	This file contains the DEM data for the 2D model area selected. This file is only generated if the Multi Cell 2D model has been chosen. Data in the file is extracted from the original DEM specified in MIKE URBAN at the requested model resolution divided by the fine/coarse ratio.
<Simulation ID> <Scenario ID> InitialConditions.dfs2	Raster file in dfs2 format	This file contains spatially varying 2D initial conditions.
<Simulation ID> <Scenario ID> Bed_Resist.dfs2	Raster file in dfs2 format	This file contains spatially varying 2D bed resistance

9 MOUSE Control module

The MOUSE Control module features advanced Real-Time Control (RTC) simulation capabilities for urban drainage and sewer 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:

- Pump
- Weir
- Orifice with gate
- Orifice with weir
- Valve

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

The process of selecting an appropriate control function under the current operational situations relies on the evaluation of system state conditions including measurable and derived hydraulic and WQ variables (e.g. water level, flow, pollutant concentration, level difference), device status (e.g. gate blade position, pump ON/OFF) and the current control function.

The control functions range from the simplest constants for the operational variables (e.g. constant weir crest setting or constant flow set-point) to dynamic controlled variables, set in a continuous functional relation with any of the measurable variables in the system (e.g. CSO discharge set-point as a function of flow concentration or a pump START/STOP levels as functions of water level at strategic location in the system).

9.1 RTC in Urban Drainage and Sewer Systems

Real time control (RTC) is an active control and operation of flow regulators based on real-time information about the system state.

RTC is feasible where it proves that flexible redistribution of water in space and time contributes to the fulfilment of the specified operational objectives, based on economically and technically sound solutions. Accordingly, application of RTC to urban drainage and sewer systems may be relevant:

- where the system has substantial transport, storage or treatment capacity which is not effectively used under passive system operation;

- where typical rainfall patterns over the catchment area exhibit high degrees of spatial variability resulting in some parts of the system becoming overloaded whilst others are under utilised;
- where the urban wastewater system includes treatment processes whose performance is amenable to active, short term control;
- where the assimilative capacity of the receiving waters is variable over time.

Usually, RTC is implemented as an integral part of a rehabilitation/upgrade scheme also involving significant civil upgrading works to increase the transport, treatment or assimilative capacity of the urban wastewater system. In such circumstances, the role of RTC is to optimise the operation of both the new and the existing facilities, thereby maximising the benefit in performance terms. Where the overall objective is to achieve compliance with specified performance targets, RTC serves to minimise the scale and extent of the necessary works.

9.2 Architecture of RTC Systems

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

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

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

In the latter case, the control decisions are derived by evaluating (comparing) the current value of the controlled flow variable and the pre-defined **set-point** value. The control algorithm is based on the numerical solution of the “continuous control problem” equation and is usually termed as **PID** (**P**roportional**-I**ntegral**-D**ifferential) **control**. The actuation signal for the regulator is generated by a PID controller, which usually appears as part of the operational strategy programmed in a **P**rogrammable **L**ogical **C**ontroller (PLC).

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

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

If the operational strategy is based on conditions local to the regulated device (for example the ON/OFF-control of a pump based on the water level in a wet well) it is called **local control**. A PLC receives signals (measurements) from local sensors and sends the control decisions (actuation signals) to the regulators. The usual situation for a sewer system is to have a number of local controllers associated with pumps.

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

Additional component needed is then a **data transmission system** (UHF radio, leased or dialled telephone lines, GSM, etc.) to transfer data between sensors, controllers and the global controller. In connection with the global controller function, an RTC system is usually equipped with the **data management and storage** facilities (data bases) and the **user interface**. This is usually termed as **SCADA** (**S**upervisory **C**ontrol **A**nd **D**ata **A**cquisition) system.

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

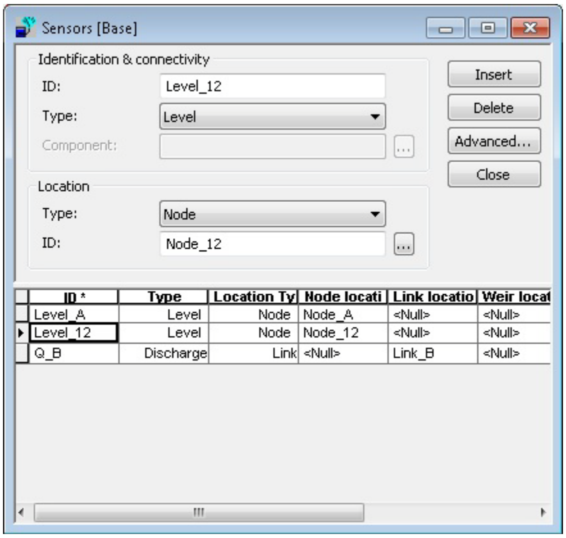
9.3 MOUSE Control vs. Real Life

MOUSE RTC simulates reactive local and global RTC systems in urban drainage and sewer networks. The software implementation is inherently a conceptualisation of real life, of which the user must be fully aware. Some conceptualisations applied in MOUSE RTC are listed below.

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

9.4 Sensors

A sensor is a physical device positioned somewhere in the system, which provides information of the actual value of a monitored variable. A sensor can only monitor one variable – if more variables are measured at the same locality a corresponding number of sensors has to be described.



Identification & connectivity

ID: Level_12

Type: Level

Component:

Location

Type: Node

ID: Node_12

ID *	Type	Location Ty	Node locati	Link locatio	Weir locat
Level_A	Level	Node	Node_A	<Null>	<Null>
Level_12	Level	Node	Node_12	<Null>	<Null>
Q_B	Discharge	Link	<Null>	Link_B	<Null>

Figure 9.1 The sensor dialog

Sensor ID

Each sensor needs a unique ID, which can be used to access the sensor information from other dialogs.

Type

The type of the sensor defines which variable the sensor measures.

Location Type and Location

Depending on the sensor type, there could be one or more location types. Accordingly, the actual location can be specified be selecting from the comprehensive list.

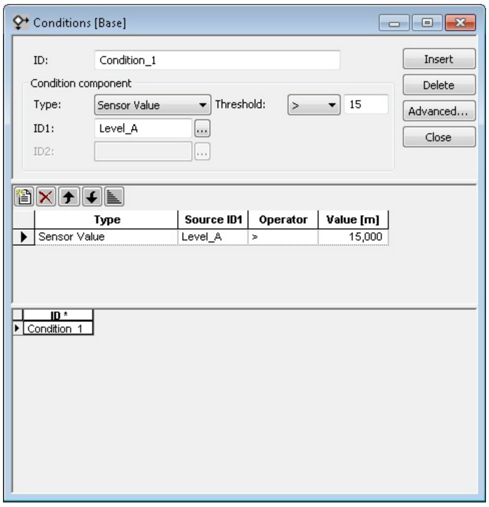
Component

For the "Concentration" sensor type, a measured WQ component must be specified.

9.5 Logical Conditions

A logical condition stands as a frame which demarcates the boundaries of a certain operational situation in the controlled system. This frame consists of an arbitrary number of independent logical tests on the various operational variables, where the relation of the actual value (or state) of the variable (provided by a sensor) is tested against the specified threshold (limit) value (or state). The individual tests are evaluated as “TRUE” or “FALSE”, with the outcome depending on the actual variable value (or state), the threshold and the specified operator.

A logical condition is evaluated as "TRUE", only if all of its constitutive tests are evaluated as "TRUE". If only one of the tests is "FALSE", the logical condition is rejected.



ID: Condition_1

Condition component

Type: Sensor Value

Threshold: > 15

ID1: Level_A

ID2:

	Type	Source ID1	Operator	Value [m]
▶	Sensor Value	Level_A	>	15,000

	ID *
▶	Condition_1

Figure 9.2 Conditions Editor

Condition ID

Each Logical Condition needs a unique ID, which can be used to access the logical condition information from other dialogs.

For each of the tests included in a logical condition the following should be specified.

Type and ID

The type and identification of the variable which should be evaluated. Depending on the type, it might be necessary with two IDs –e.g. for the evaluation of difference between two sensor values.

For the type "Diff of 2 Sensor Values" the sensors specified under ID1 and ID2 must be of the same type and sensors measuring either level, discharge or concentration.

For the type "Sum of 2 Sensor Values" the sensors specified under ID1 and ID2 must both be sensors measuring discharge.

For the type "Change In Sensor Value" the sensor specified under ID1 must be a sensor measuring level. The threshold will correspond to a speed of change (dH/dt).

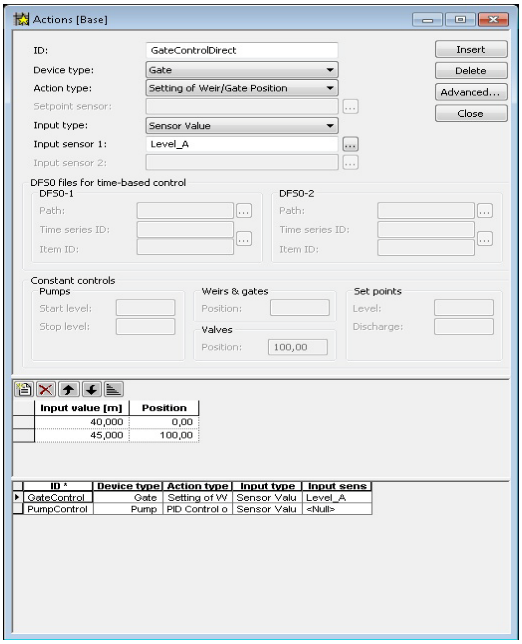
For the type "Accumulated Value" the sensor specified under ID1 must be a sensor measuring mass flux.

Operator and Limit

This part of a test defines the condition which must be fulfilled for the test to be evaluated as "TRUE". In cases with the tested continuous variables, it consists of an algebraic operator (< or >) and a limit value. In cases of logical evaluation (e.g. pump ON/OFF), the test is defined simply by setting the desired device state as "TRUE" or "FALSE"

9.6 Control Actions

A control action is a functional relation between input from one or two sensors and the set-point or the setting for the controlled device.


Actions [Base]
Insert
Delete
Advanced...
Close
Setpoint sensor:
Input sensor 2:
DFS0 files for time-based control
DFS0-1
Path:
Time series ID:
Item ID:
DFS0-2
Path:
Time series ID:
Item ID:
Constant controls
Pumps
Start level:
Stop level:
Weirs & gates
Position:
Set points
Level:
Discharge:

ID: GateControlDirect
Device type: Gate
Action type: Setting of Weir/Gate Position
Input type: Sensor Value
Input sensor 1: Level_A
Valves Position: 100,00

Input value [m]	Position
40,000	0,00
45,000	100,00

ID *	Device type	Action type	Input type	Input sens
GateControl	Gate	Setting of W	Sensor Valu	Level_A
PumpControl	Pump	PID Control o	Sensor Valu	<Null>

Figure 9.3 Control Actions Editor

Action ID

Each control action needs a unique ID, which can be used to access the actions from other dialogs.

For each of the actions the following should be specified.

Device Type and Action Type

These two fields hold the information about the applicability of the action (device type) and which type of control it describes. The devices which can be controlled are pumps weirs and gates, and each of these types can be controlled directly or by a PID control.

If a direct control is used, the action specifies information of START- and STOP levels for pumps, or of the wanted position for gates or weirs. If a PID-control is applied, the action specifies a set-point (flow or level) anywhere in the system.

Setpoint Sensor

For actions of PID-type, it must be specified where in the system the set-point is located. For this purpose a Sensor ID must be specified.

Input Type and Input Sensor

The type and identification of the parameter which should be evaluated. Depending on the type it might be necessary with two sensor IDs –e.g. if the flow is regulated as a function of the difference between two level sensor values. If the "Constant" type is selected, a constant value is specified, instead of the sensor(s) and the action.

Input and Output Values

The functional relation between the actual input value (sensor reading or a combination of two sensors) and the set-point value (or setting). The tabulated values are linearly interpolated.

9.7 PID parameter sets

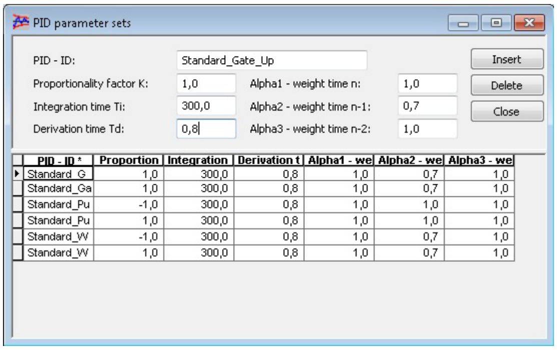
MOUSE RTC includes the possibilities for PID-control of weirs, gates, pumps and valves. Independently on the choice of the controlled variable, the PID algorithm adjusts the settings of the regulator (or outputs in case of pumps), according to the current error between the specified set-point and the actual value of the controlled variable. The following settings/outputs can be used as means of flow control:

- **Weirs**: weir crest level setting
- **Gates**: level of the bottom lip of the gate setting
- **Pumps**: pump discharge
- **Valves**: opening of the valve

Note: When using a weir as a PID regulator, the weir crest level will not be used. On the other hand, the START and STOP levels for pumps will be used to start a PID-controlled pump.

The following types of controlled variables/set-points are available:

- H, water level in a node.
- Q, flow in a pipe.



PID - ID: Standard_Gate_Up

Proportionality factor K: 1,0

Integration time Ti: 300,0

Derivation time Td: 0,8

Alpha1 - weight time n: 1,0

Alpha2 - weight time n-1: 0,7

Alpha3 - weight time n-2: 1,0

PID - ID *	Proportion	Integration	Derivation t	Alpha1 - we	Alpha2 - we	Alpha3 - we
Standard_G	1,0	300,0	0,8	1,0	0,7	1,0
Standard_Ga	1,0	300,0	0,8	1,0	0,7	1,0
Standard_Pu	-1,0	300,0	0,8	1,0	1,0	1,0
Standard_Pu	1,0	300,0	0,8	1,0	1,0	1,0
Standard_W	-1,0	300,0	0,8	1,0	0,7	1,0
Standard_W	1,0	300,0	0,8	1,0	0,7	1,0

Figure 9.4 PID Parameter Sets

PID ID

Each set of PID parameter sets is identified with a unique ID which can be used to access the information from other dialogs.

Proportionality Factor, Integration Time and Derivation Time

These are the 3 main parameters for the PID control.

Alpha-1, Alpha-2 and Alpha-3

Weighting factors for time level n, n-1 and n-2.

9.7.1 Calibration of the PID constants

Tuning of the PID constants (Ti, Td and Kd) is not a straightforward task. Understanding of the theoretical background and the numerical solution of the control equation would be beneficial in this process. The following may be used as an elementary guideline.

Typical values of the PID constants and weighting factors

T_i: 300 sec.

T_d: 0.8 sec.

K:Pumps and gates:

-setpoint downstream:**1.0**

-setpoint upstream:-**1.0**

Weirs

-setpoint downstream:-**1.0**

-setpoint upstream:**1.0**

1.0

Gates and weirs:**0.7**

Pumps:**1.0**

1.0

NOTE: The sign on the K-factor is very important - if it is wrong it will cause the control function not to work at all since the device will typically move to one of the extreme positions and stay there till the end of the simulation.

Figure 9.5, Figure 9.6 and Figure 9.7 show examples of how the actual variable (flow or water level) can fluctuate around the set-point as consequence of various choices of the PID constants. Each figure has three different graphs depending on whether the constant is too high, too low or adequate.

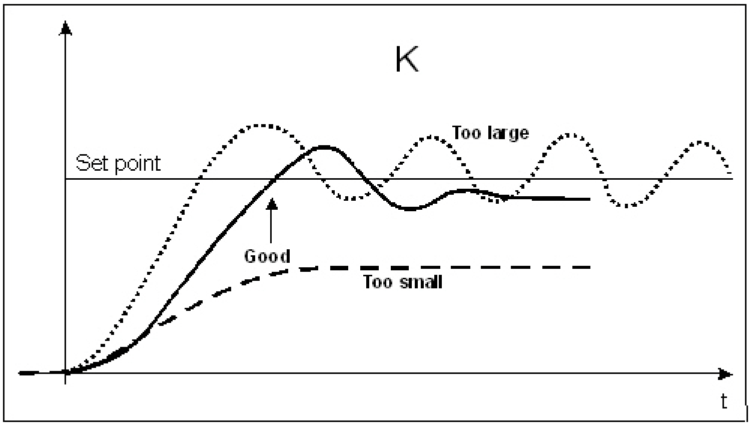


Figure 9.5 Fluctuations around the set point depending on the size of the proportionality factor, K

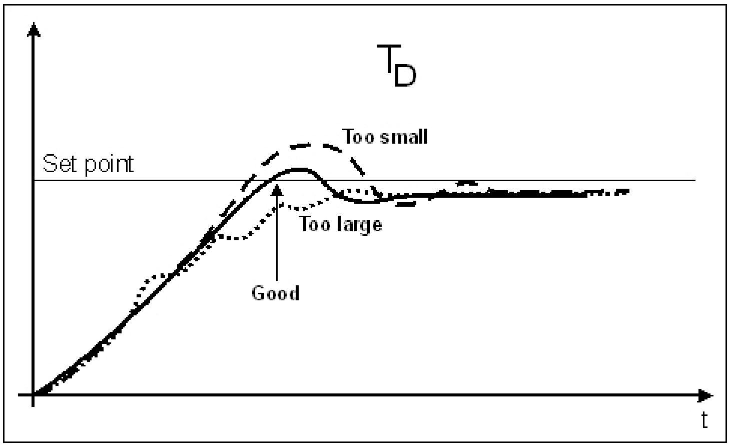


Figure 9.6 Fluctuations around the set point depending on the size of the derivation time, TD.

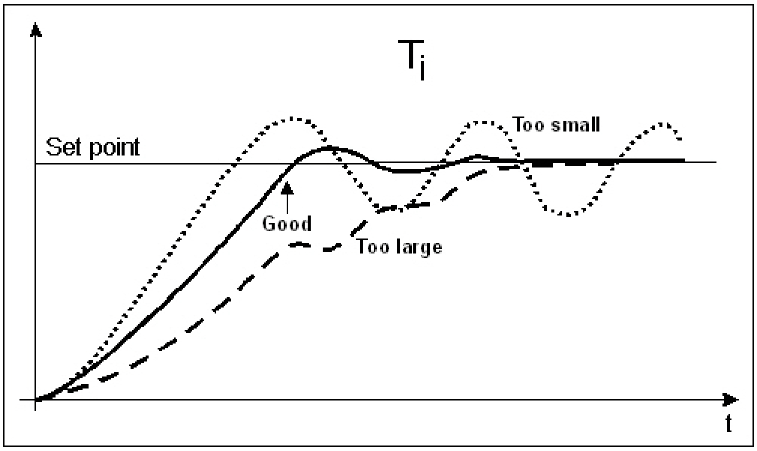
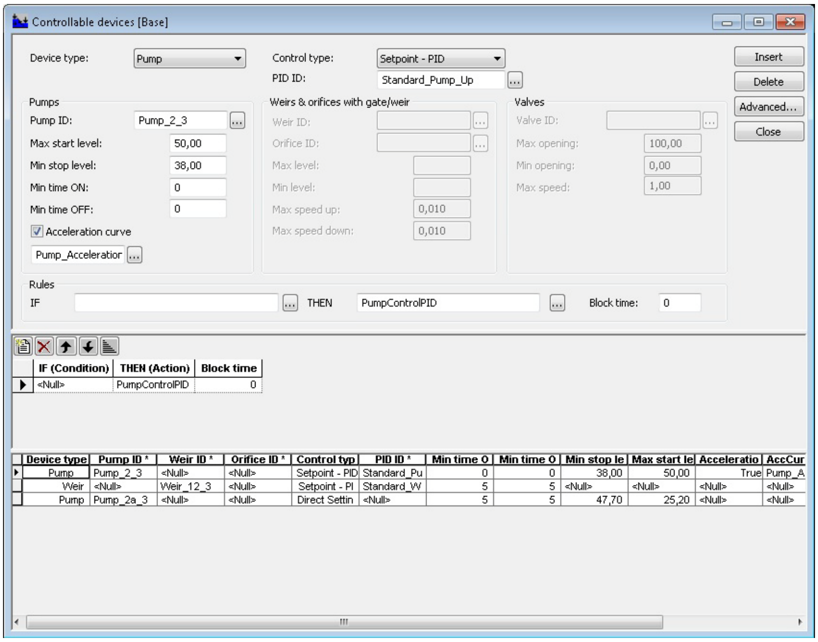


Figure 9.7 Fluctuations around the set point depending on the size of the integration time, Ti

9.8 Controllable Devices

In this dialog the operational control logic for controllable devices is specified. The control is specified as a set of rules, linking the logical conditions and the control actions. The rules are evaluated sequentially following the rules list sequence.


Controllable devices [Base]
Insert
Delete
Advanced...
Close
Pumps
Weirs & orifices with gate/weir
Valves
Rules

Device type: Pump
Control type: Setpoint - PID
PID ID: Standard_Pump_Up
Pump ID: Pump_2_3
Max start level: 50,00
Min stop level: 38,00
Min time ON: 0
Min time OFF: 0
☑ Acceleration curve
Pump_Acceleratior
Weir ID:
Orifice ID:
Max level:
Min level:
Max speed up: 0,010
Max speed down: 0,010
Valve ID:
Max opening: 100,00
Min opening: 0,00
Max speed: 1,00
IF
THEN PumpControlPID
Block time: 0

IF (Condition)	THEN (Action)	Block time
<Null>	PumpControlPID	0

Device type	Pump ID *	Weir ID *	Orifice ID *	Control typ	PID ID *	Min time O	Min time O	Min stop le	Max start le	Acceleratio	AccCur
Pump	Pump_2_3	<Null>	<Null>	Setpoint - PID	Standard_Pu	0	0	38,00	50,00	True	Pump_A
Weir	<Null>	Weir_12_3	<Null>	Setpoint - PI	Standard_W	5	5	<Null>	<Null>	<Null>	<Null>
Pump	Pump_2a_3	<Null>	<Null>	Direct Settin	<Null>	5	5	47,70	25,20	<Null>	<Null>

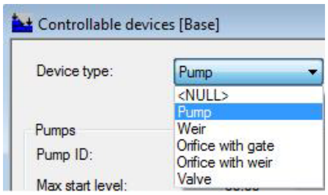
Figure 9.8 Dialog for controllable devices

Device Type and Device ID

Device Type and ID identifies the device which should be controlled. The device types available are 'Pump', 'Weir', 'Orifice with gate’, ‘Orifice with weir’ and 'Valve' (see Figure 9.9). The three devices 'Pump, 'Weir' and 'Valve' require that you have defined the devices first in the relevant dialogs for pumps, weirs or valves.

For 'Orifice with gate' and 'Orifice with weir' you need to define an orifice first. Then the 'Controllable devices' dialog is used for adding either a movable gate or a movable weir to the orifice.

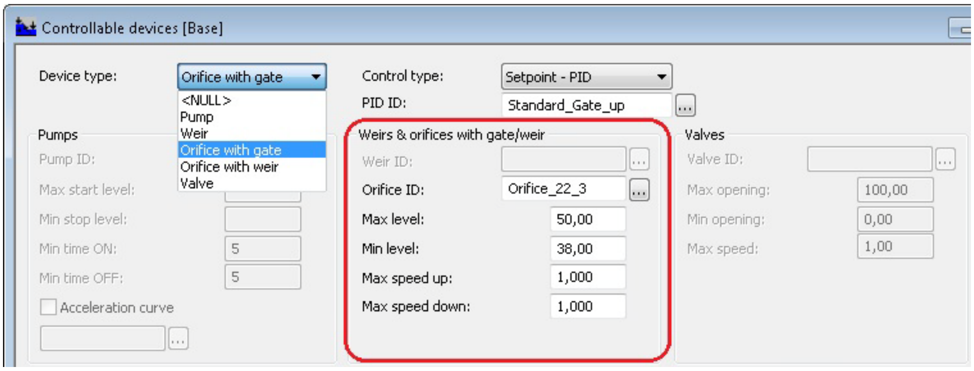
For 'Orifice with weir' you only need to make the selection in this dialog. You should not add the weir in the 'Weir' dialog (please refer later section on the difference between an ‘ordinary’ weir and a weir in a orifice).


Controllable devices [Base]
Pumps
Pump ID:
Max start level:
<NULL>
Pump
Weir
Orifice with gate
Orifice with weir
Valve

Device type: Pump

Figure 9.9 The different device types

When the 'Device type' has been selected the specific parameters for the configuration of the device can be entered. For this purpose the dialog has three separate sections. The section to the left covers pumps while the section to the right covers valves.The center section covers three different device types: 'Weir', 'Orifice with gate' and 'Orifice with weir'. See Figure 9.10.


Controllable devices [Base]

Device type: Orifice with gate

Control type: Setpoint - PID

PID ID: Standard_Gate_up

Pumps

Pump ID:

Max start level:

Min stop level:

Min time ON: 5

Min time OFF: 5

☐ Acceleration curve

Weirs & orifices with gate/weir

Weir ID:

Orifice ID: Orifice_22_3

Max level: 50,00

Min level: 38,00

Max speed up: 1,000

Max speed down: 1,000

Valves

Valve ID:

Max opening: 100,00

Min opening: 0,00

Max speed: 1,00

Figure 9.10 Input for the various device types differs

If e.g. the device type is 'Weir' then you have access to the 'Weir ID' field for selecting the weir to be controlled.

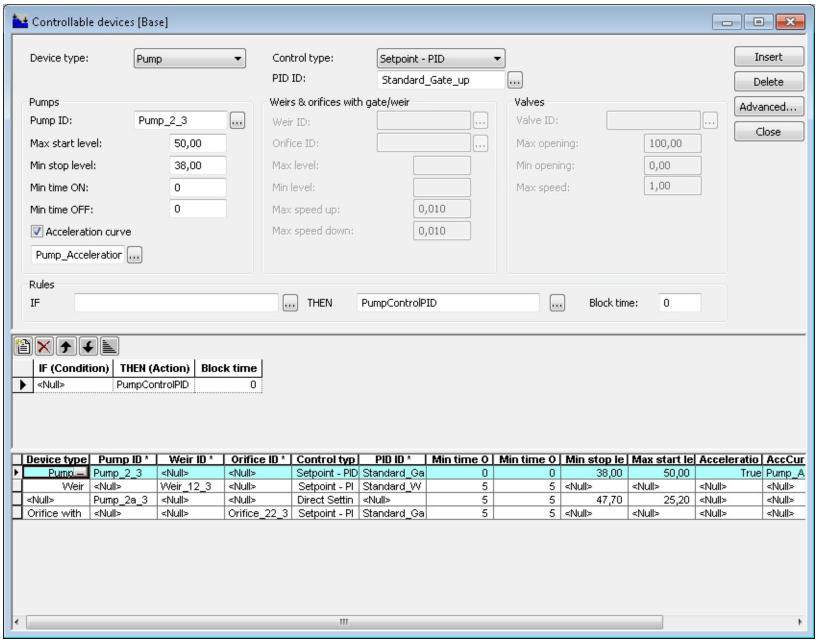
For the device types 'Orifice with gate' and 'Orifice with weir' you will have access to the 'Orifice ID' field where you can select the orifice for which you want to add either a gate or a weir as defined by the device type selection.

The input for each for these types is described in later sections.

Control Type and PID-ID

Once the device type is specified the control type needs to be specified. The control type of the controlled devices can be either direct setting, PID-control of set-point or no control. If PID-control is used a set of PID-parameters should be selected by the PID-ID.

9.8.1 Pumps



Device type: Pump
Control type: Setpoint - PID
PID ID: Standard_Gate_up

Pumps
Pump ID: Pump_2_3
Max start level: 50,00
Min stop level: 38,00
Min time ON: 0
Min time OFF: 0
☑ Acceleration curve
Pump_Acceleratior

Weirs & orifices with gate/weir
Weir ID:
Orifice ID:
Max level:
Min level:
Max speed up: 0,010
Max speed down: 0,010

Valves
Valve ID:
Max opening: 100,00
Min opening: 0,00
Max speed: 1,00

Rules
IF
THEN PumpControlPID
Block time: 0

IF (Condition)	THEN (Action)	Block time
<Null>	PumpControlPID	0

Device type	Pump ID *	Weir ID *	Orifice ID *	Control typ	PID ID *	Min time O	Min time O	Min stop le	Max start le	Acceleratio	AccCur
Pump	Pump_2_3	<Null>	<Null>	Setpoint - PID	Standard_Ga	0	0	38,00	50,00	True	Pump_A
Weir	<Null>	Weir_12_3	<Null>	Setpoint - PI	Standard_W	5	5	<Null>	<Null>	<Null>	<Null>
<Null>	Pump_2a_3	<Null>	<Null>	Direct Settin	<Null>	5	5	47,70	25,20	<Null>	<Null>
Orifice with	<Null>	<Null>	Orifice_22_3	Setpoint - PI	Standard_Ga	5	5	<Null>	<Null>	<Null>	<Null>

Figure 9.11 Dialog with control data specific for pumps

Max Start Level

The maximum START level for the pump. If exceeded, the pump is unconditionally switched ON.

Min Stop Level

The minimum STOP level for the pump. If a lower level occurs, the pump is unconditionally switched OFF.

Min time Pump ON

The minimum time the pump has to be ON before it can stop.

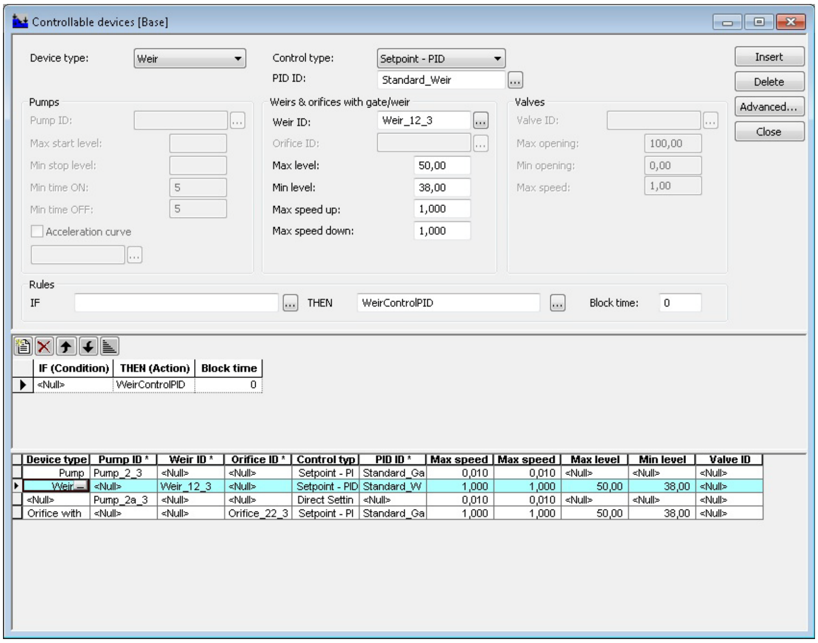
Min time Pump OFF

The minimum time the pump has to be OFF before it can start again.

Acceleration Curve (for PID control only)

For PID-controls, the acceleration of the pump can be specified as dependent on the actual flow. The acceleration curve is specified in tabular data and referred to by the ID of the data set.

9.8.2 Weirs



Device type: Weir

Control type: Setpoint - PID

PID ID: Standard_Weir

Pump ID:

Max start level:

Min stop level:

Min time ON: 5

Min time OFF: 5

☐ Acceleration curve

Weir ID: Weir_12_3

Orifice ID:

Max level: 50,00

Min level: 38,00

Max speed up: 1,000

Max speed down: 1,000

Valve ID:

Max opening: 100,00

Min opening: 0,00

Max speed: 1,00

IF

THEN WeirControlPID

Block time: 0

IF (Condition)	THEN (Action)	Block time
<Null>	WeirControlPID	0

Device type	Pump ID *	Weir ID *	Orifice ID *	Control typ	PID ID *	Max speed	Max speed	Max level	Min level	Valve ID
Pump	Pump_2_3	<Null>	<Null>	Setpoint - PI	Standard_Ga	0,010	0,010	<Null>	<Null>	<Null>
Weir	<Null>	Weir_12_3	<Null>	Setpoint - PID	Standard_W	1,000	1,000	50,00	38,00	<Null>
<Null>	Pump_2a_3	<Null>	<Null>	Direct Settin	<Null>	0,010	0,010	<Null>	<Null>	<Null>
Orifice with	<Null>	<Null>	Orifice_22_3	Setpoint - PI	Standard_Ga	1,000	1,000	50,00	38,00	<Null>

Figure 9.12 Dialog with control data specific for weirs

Max Level

The maximum elevation of the movable weir crest.

Min Level

The minimum elevation of the movable weir crest.

The fixed weir crest level is not used for a RTC weir.

Max Speed Up

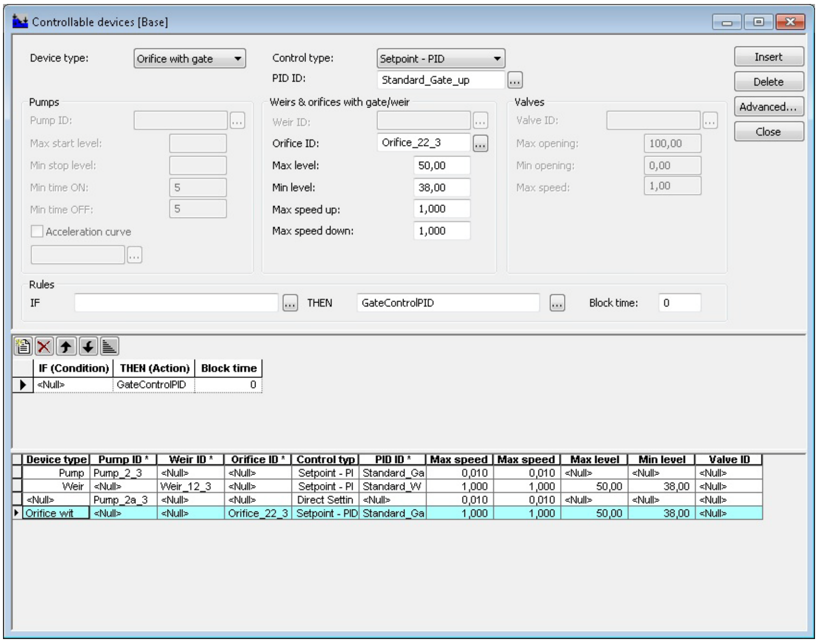
The maximum velocity for movement of the weir in upward direction.

Max Speed Down

The maximum velocity for movement of the weir in downward direction.

9.8.3 Orifices with weirs and gates

If the device type is either ‘Orifice with gate’ or ‘Orifice with weir’ the mid-section of the dialog becomes active.


Controllable devices [Base]
Insert
Delete
Advanced...
Close
Pumps
Pump ID:
Max start level:
Min stop level:
Acceleration curve
Weirs & orifices with gate/weir
Weir ID:
Valves
Valve ID:

Device type: Orifice with gate
Control type: Setpoint - PID
PID ID: Standard_Gate_up
Min time ON: 5
Min time OFF: 5
Orifice ID: Orifice_22_3
Max level: 50,00
Min level: 38,00
Max speed up: 1,000
Max speed down: 1,000
Max opening: 100,00
Min opening: 0,00
Max speed: 1,00
Rules
IF
THEN GateControlPID
Block time: 0

IF (Condition)	THEN (Action)	Block time
<Null>	GateControlPID	0

Device type	Pump ID *	Weir ID *	Orifice ID *	Control typ	PID ID *	Max speed	Max speed	Max level	Min level	Valve ID
Pump	Pump_2_3	<Null>	<Null>	Setpoint - PI	Standard_Ga	0,010	0,010	<Null>	<Null>	<Null>
Weir	<Null>	Weir_12_3	<Null>	Setpoint - PI	Standard_W	1,000	1,000	50,00	38,00	<Null>
<Null>	Pump_2a_3	<Null>	<Null>	Direct Settin	<Null>	0,010	0,010	<Null>	<Null>	<Null>
Orifice wit	<Null>	<Null>	Orifice_22_3	Setpoint - PID	Standard_Ga	1,000	1,000	50,00	38,00	<Null>

Figure 9.13 Dialog with control data specific for orifices

Max Level

The maximum elevation of the movable gate/weir.

Min Level

The minimum elevation of the movable gate/weir.

Max Speed Up

The maximum velocity for movement of the gate/weir in upward direction.

Max Speed Down

The maximum velocity for movement of the gate/weir in downward direction.

9.8.4 Difference between weir and orifice with weir

It is possible to define weirs in two different ways. Either through the 'Weir' dialog alone which we will refer to as an 'ordinary weir'. The second way is to use the combination of an orifice and the controllable device type 'Orifice with weir'.

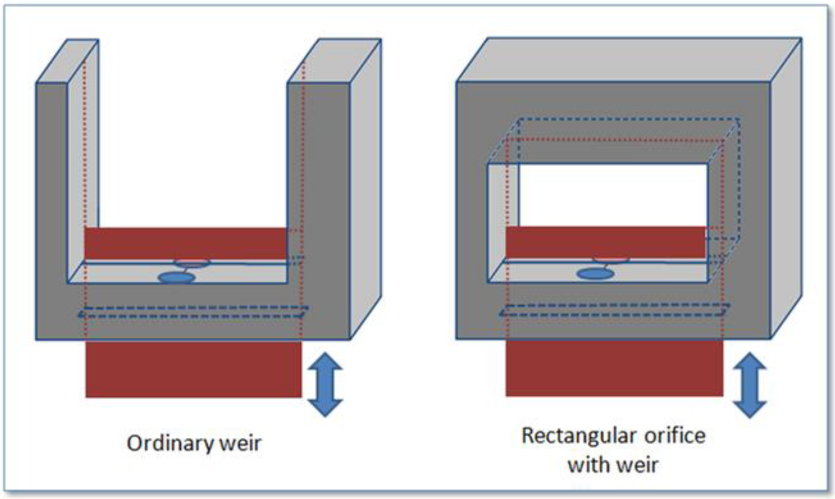
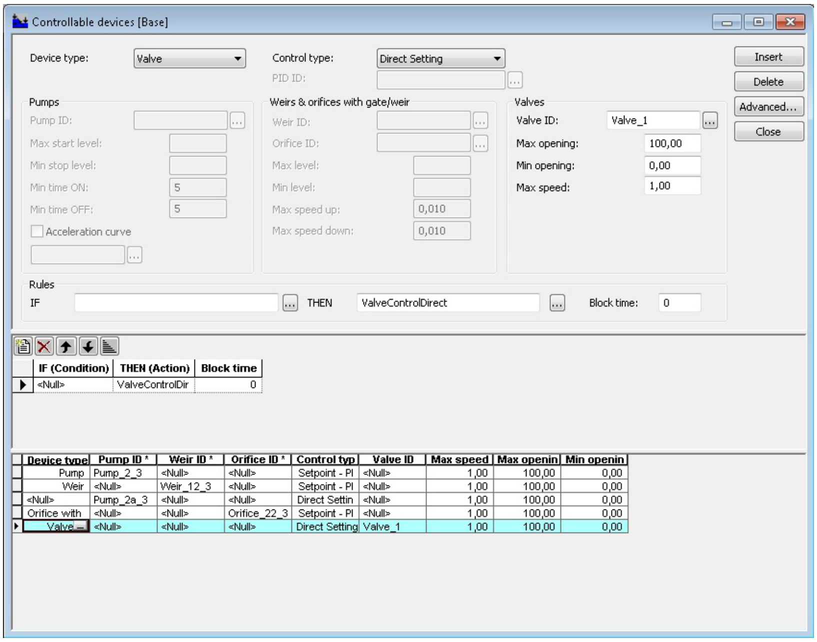


Figure 9.14 Difference between a ordinary weir and an orifice with a weir.

The main difference is that the 'Orifice with weir' can close the orifice completely while the ordinary weir is always open upwards. It has no ceiling and in principle the flow can always pass over the weir if the water level is higher than the weir crest level.

For the 'Orifice with weir' once the weir is fully closing the orifice then no flow will pass the weir even if the water level is above the crest level.

9.8.5 Valves



Device type: Valve
Control type: Direct Setting
PID ID:

Pump ID:
Max start level:
Min stop level:
Min time ON: 5
Min time OFF: 5
☐ Acceleration curve

Weir ID:
Orifice ID:
Max level:
Min level:
Max speed up: 0,010
Max speed down: 0,010

Valve ID: Valve_1
Max opening: 100,00
Min opening: 0,00
Max speed: 1,00

IF THEN ValveControlDirect Block time: 0

IF (Condition)	THEN (Action)	Block time
<Null>	ValveControlDir	0

Device type	Pump ID *	Weir ID *	Orifice ID *	Control typ	Valve ID	Max speed	Max openin	Min openin
Pump	Pump_2_3	<Null>	<Null>	Setpoint - PI	<Null>	1,00	100,00	0,00
Weir	<Null>	Weir_12_3	<Null>	Setpoint - PI	<Null>	1,00	100,00	0,00
<Null>	Pump_2a_3	<Null>	<Null>	Direct Settin	<Null>	1,00	100,00	0,00
Orifice with	<Null>	<Null>	Orifice_22_3	Setpoint - PI	<Null>	1,00	100,00	0,00
Valve	<Null>	<Null>	<Null>	Direct Setting	Valve_1	1,00	100,00	0,00

Figure 9.15 Dialog with control data specific for valves

Max Opening

The maximum opening of the valve in percentage.

Min Opening

The minimum opening of the valve in percentage.

Max Speed

The maximum velocity for movement of the valve in percentages/sec.

9.8.6 Control rules

Any number of rules can be specified to control the device. The statements are evaluated sequentially starting from the top. This means that appropriate sequence of rules is essential for the achievement of the desired control logic. Evaluation of a logical condition belonging to a rule as "TRUE", leads to the selection of the specified control action. If a logical conditions is "FALSE", the evaluation proceeds to the next rule on the list.

If no logical condition is specified, the rule is unconditionally evaluated as "TRUE". This implies that the last rule in the sequence must not include any logical conditions in order to ensure a selection of a "default" control action if all specified conditions are found "FALSE".

The blocking time ensures that a certain rule is applied for a minimum period. This means that the system is locked to certain rule in a period equal to the blocking period after it was activated.

When specifying the rules a few constraints should be notified:

All control actions used to control a specific device must be of the same function type – corresponding to the specified Device Type and Control Type.

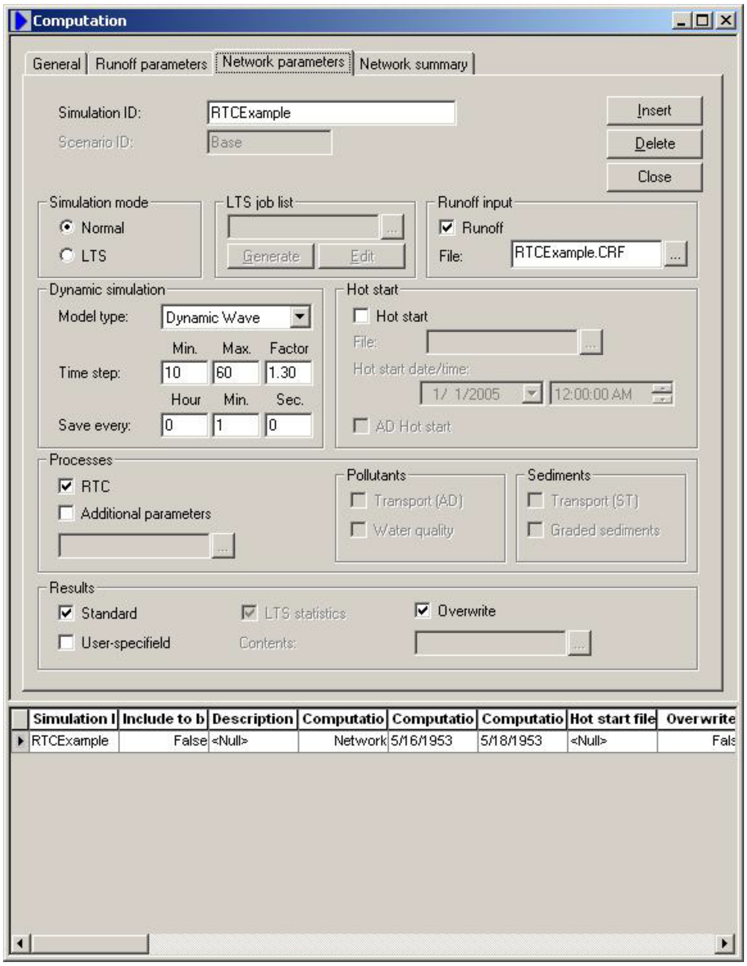
For PID-control, all control actions must refer to the same set-point sensor – change of the set-point sensor during simulation is not allowed

For time dependent control (Input sensor of type “Time since start of simulation”) only ONE rule can be specified. The menu allows more functions to be inserted, but only the first one will be used during the simulation.

When starting a simulation, the system checks if these conditions are fulfilled and in the case of any violation, the simulation will not start.

9.9 MOUSE Control Computations

A MOUSE Control computation is started from the usual simulation dialog and will be performed if the checkbox “RTC” is marked, see Figure 9.16.



Simulation ID: RTCExample
Scenario ID: Base

Simulation mode
- ● Normal
- ○ LTS

LTS job list

Runoff input
- ☑ Runoff
- File: RTCExample.CRF

Dynamic simulation
- Model type: Dynamic Wave
- Time step: Min. 10, Max. 60, Factor 1.30
- Save every: Hour 0, Min. 1, Sec. 0

Hot start
- ☐ Hot start
- File:
- Hot start date/time: 1/ 1/2005 12:00:00 AM
- ☐ AD Hot start

Processes
- ☑ RTC
- ☐ Additional parameters

Pollutants
- ☐ Transport (AD)
- ☐ Water quality

Sediments
- ☐ Transport (ST)
- ☐ Graded sediments

Results
- ☑ Standard
- ☐ User-specifield
- ☑ LTS statistics
- Contents:
- ☑ Overwrite

Simulation I	Include to b	Description	Computatio	Computatio	Computatio	Hot start file	Overwrite
RTCExample	False	<Null>	Network	5/16/1953	5/18/1953	<Null>	Fals

Figure 9.16 The control simulation is activated on the network parameters tab page

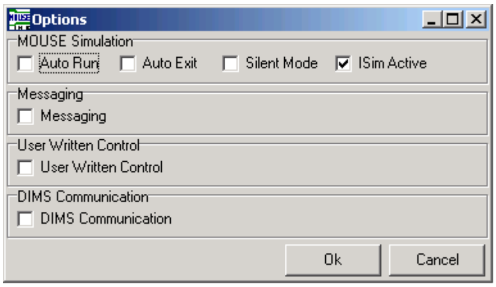
9.10 User Written Control

MOUSE supports User Written Control (UWC) for advanced RTC control of the system. This gives the user the possibility of controlling almost all aspects of MOUSE each time step and thereby implementing advanced control strategies in the system. UWC is only recommended for the advanced computer user since it involves programming experience. The currently supported language is Delphi the procedures and functions can however be called from any programming language including Visual Basic, C++ and others that support generating DLLs.

The user can execute code at 3 different steps in the code. The Initialization step is for executing code opening, preparing or reading files needed in the simulation. The Time step is the step in which the control strategies are run. The End step is the step in which files opened in the Initialization step can be closed. The three steps are defined under the mouse604.exe Simulation | Options dialog, where a filename and a procedure name are specified for each step under UWC. The compiled DLL with the UWC must be located in the same folder as the mouse_hd.dll file.

There are currently over 140 different procedures and functions exported from MOUSE ranging from setting PID constants runtime to retrieving the flow in a specific grid point in a link. Using the methods require the user to include the MOUSEDLL.pas file into the DLL Delphi project. An example of the use of this is listed below.

```
library Test;

uses
  SysUtils,
  Classes,
  MOUSEDLL; // <- Important

{$R *.res}

var
  OutFile : TextFile;

procedure InitStep;
begin
  AssignFile(OutFile, 'C:\MOUSEOutput.txt');
  Rewrite(OutFile);
end;

procedure TimeStep;
var
  NodeLevelString : String;
  MOUSETimeString : String;
begin
  NodeLevelString := Floattostr(GetNodeLevel('Node_11'));
  MOUSETimeString := FormatDateTime('YYYY-MM-DD hh:mm:ss', 
MOUSETime);
  Writeln(OutFile, NodeLevelString+' '+MOUSETimeString);
end;

procedure EndStep;
begin
  CloseFile(OutFile);
end;

exports
  InitStep,
  TimeStep,
  EndStep;

end.
```



MOUSE Simulation

☐ Auto Run ☐ Auto Exit ☐ Silent Mode ☑ ISim Active

Messaging

☐ Messaging

User Written Control

☐ User Written Control

DIMS Communication

☐ DIMS Communication

Figure 9.17 Options dialog of the simulation launcher

The example above simply writes the water level in a node and the corresponding time to a file, but the possibilities are endless.

The next example is the RTCExample provided in the installation where the actual control is handled from UWC. The RTC control in the UND file is not changed for this, which means the UWC RTC overrides the RTC in the UND. This provides a way of overriding standard RTC in e.g. emergency situations in the network. Alternatively the UWC RTC can be used alone. In this case the Control Functions and Logical Conditions are not needed for the Controlled Devices, since the actual control is handled externally. In addition to this, the control type number for the 3 PID controlled devices is changed to 5. This change is done using a text editor.

```
library RTCExample;

uses
  SysUtils,
  Classes,
  MOUSEDLL;

{$R *.res}

procedure TimeStep;
var LevelA, Level02, Level12, Level22,
    SetPoint1, Measured1, SetPoint2, Measured2, SetPoint3, Measured3,
    StartLevel, StopLevel, GatePosition, WeirPosition : Double;
begin
  LevelA    := GetSensorValue('Level_A');
  Level02   := GetSensorValue('Level_02');
  Level12   := GetSensorValue('Level_12');
  Level22   := GetSensorValue('Level_22');
```

```
//Pump_2_3
SetPoint1 := -0.6*LevelA+67;
SetPumpPIDSetpoint('Pump_2_3', SetPoint1);
Measured1 := Level02;
SetPumpPIDMeasurement('Pump_2_3', Measured1);

//Weir_12_3
SetPoint2 := -0.6*LevelA+67;
SetWeirPIDSetpoint('Weir_12_3', SetPoint2);
Measured2 := Level12;
SetWeirPIDMeasurement('Weir_12_3', Measured2);

//Orifice_22_3
SetPoint3 := -0.6*LevelA+67;
SetGatePIDSetpoint('Orifice_22_3', SetPoint3);
Measured3 := Level22;
SetGatePIDMeasurement('Orifice_22_3', Measured3);

//Pump_2a_3
StartLevel := LevelA+0.2;
StopLevel  := LevelA;
SetPumpStartLevel('Pump_2a_3', StartLevel);
SetPumpStopLevel('Pump_2a_3', StopLevel);

//Weir_12a_3
WeirPosition := LevelA;
SetWeirPosition('Weir_12a_3', WeirPosition);
```

```
  //Orifice_22a_3

  GatePosition := LevelA;

  SetGatePosition('Orifice_22a_3', GatePosition);

end;



exports

  TimeStep;



begin

end.
```

A documentation of the different procedures and functions can be found in the MOUSEDLL.pas file. DHI does not provide support in the Delphi Programming language. We are however open to suggestions on improving and adding functionality to this

10 Long term statistics

The Long Term Statistics (LTS) allows that a MOUSE network model with intermittent hydrological inputs can be set up for a time-efficient long-term simulation, covering a long, continuous historical period and the computation of relevant statistics for the operational variables of interest. LTS features an easy way to convert a usual MOUSE simulation covering a long historical period into a discontinuous series of relevant events feasible for the simulation, and to define various event-based and annual statistics for the selected parameters.

Based on the specified criteria, the system automatically selects the periods for dynamic pipe flow simulations during wet weather. This reduces the simulation time significantly, while preserving the accuracy in the simulation of relevant dynamic effects in the system - CSOs, surcharges, etc. For example, by running successive LTS simulations with the existing drainage system configuration and the planned upgrade, impacts of the planned investments (e.g. new sewers, retention tanks, control schemes) on the system performance can be reliably tested and evaluated based on the computed operational statistics. This allows the user to develop the optimal rehabilitation / upgrade strategy, e.g. for satisfying the requirements of environmental regulations.

It is possible to combine the MIKE 1D LTS hydrodynamic simulations with MIKE 1D pollution transport (AD) module. In this way, it is possible to estimate loads of pollutants annually or on an event based level.

Statistics on maximum water levels, discharges from weirs, etc., generated by simulating long continuous periods subject to impact of. recorded historical rainfalls, is a specific form of results from numerical simulation models particularly useful for the evaluation of the simulated system functionality. Statistics can be used to compare the effects of various mitigation measures in the planned rehabilitation process, the performance of various control strategies, the cost-benefit efficiency of the planned investments, as well as to determine whether the functionality of the sewer system is compliant with legislative requirements.

Functionality of urban drainage systems is often subject to requirements like:

- Flooding on terrain must not happen more than once every T years.
- Flooding of basements must not happen more than once every T years.
- Discharge from CSO weirs to receiving waters must not happen more than n times per year.
- Total volume from CSOs must not exceed a certain threshold amount per year.
- Total pollutant emission from the system must not exceed the specified threshold amount per year.

- Etc.

LTS is able to answer these and similar questions and thereby appears as an essential simulation tool for a reliable assessment of system performance.

Statistics on some operational parameters are only reliable if the simulated number of more or less independent events is sufficiently large, i.e. if the simulated historical period is sufficiently long. This, in turn, may imply very long simulation times (if 'normal' computing hardware is applied) and thereby spoil the feasibility of the analysis.

However, use of dynamic simulation for the hydraulic and pollution transport analysis of drainage networks is only beneficial in periods of high-flow dynamics. Otherwise, simpler and much faster simulation tools can be used with a similar level of accuracy. Since high-flow dynamics in urban drainage networks are normally associated with rainfall events, the dynamically simulated periods can be reduced to the wet periods, i.e. to only a few percent of the certain historical period.

The fundamental principle behind the LTS concept is therefore the ability to reduce the simulation time without losing significant information. LTS is able, based on the user-specified criteria, to eliminate the irrelevant periods from the simulated series, and to limit the dynamic simulations to a series of disconnected events, instead of the entire simulation period

10.1 Data Input

10.1.1 Job list

For the purpose of isolating the intervals relevant for dynamic simulation, LTS generates a job list on the basis of system information, the knowledge of time series for dry weather flow (i.e. wastewater), rainfall and other loads and the set of specified 'Job-List Criteria'. The simulation jobs from the list, limited by the simulation start time end the simulation stop time, may optionally be subject to further evaluation by a set of 'Runtime Criteria' during the simulation. Runtime Criteria are founded on the evaluation of the operational variables within the network itself which can potentially extend the simulations beyond the time intervals defined in the job list. Since the 'runtime criteria' are evaluated during the simulation, it is not possible to determine the exact duration of the dynamic simulation in advance.

Upon its creation, the job list can be edited, so that the actual start time and date and the estimated stop time for the individual jobs may be modified according to specific requirements

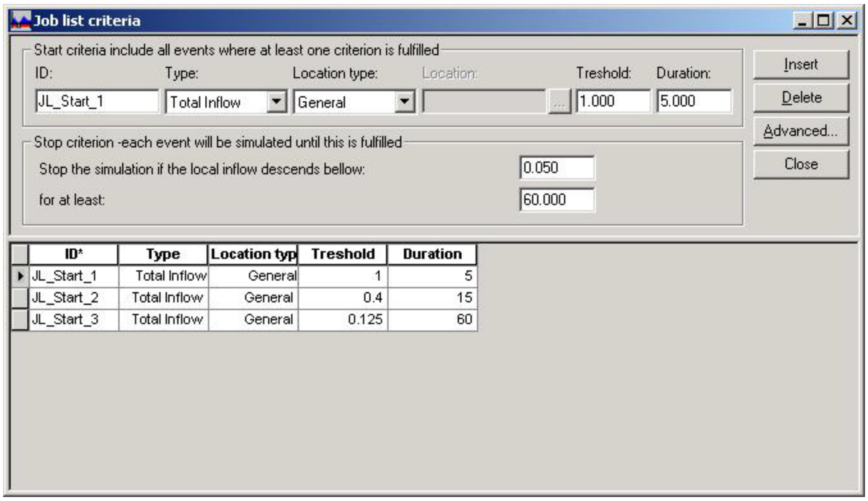
10.1.2 Job list criteria

Job list criteria are used to select the intervals which (at least) should be calculated with the hydrodynamic model during a computation with LTS.

The job list criteria represent actually various thresholds for the system loading, which are used to evaluate if the state of the system is within or outside the "dynamic" conditions. There are separate START criteria, which determine the start dates and times for the dynamic simulations, and a single STOP criterion, which is used for the estimation of the ending dates and times for dynamic simulations.

The START criteria are evaluated against the loading time series - dry weather flow, rainfall events, etc., for the system as a whole, for a part of the system, or for an individual sub-catchment. A number of START criteria can be defined, and if at least one of them is fulfilled, a certain "event" will be included into the job list.

The STOP criterion includes a threshold for the total load for the entire system, and an additional time condition which extends the simulation for the specified time after the loading has fallen below the threshold. The objective of the STOP criterion is to delimit the dynamic simulation, but without a potential loss of important information due to the time offset between the loads and effects in the sewer network.



Start criteria include all events where at least one criterion is fulfilled

ID:	Type:	Location type:	Location:	Treshold:	Duration:
JL_Start_1	Total Inflow	General		1.000	5.000

Stop criterion -each event will be simulated until this is fulfilled

Stop the simulation if the local inflow descends bellow: 0.050

for at least: 60.000

ID*	Type	Location typ	Treshold	Duration
JL_Start_1	Total Inflow	General	1	5
JL_Start_2	Total Inflow	General	0.4	15
JL_Start_3	Total Inflow	General	0.125	60

Figure 10.1 Job list criteria dialog

Table 10.1

Edit field	Description	Used or required by simulation	Field name in data structure
StartCriteriaID	Identifier	Yes	MUID
StartCriteriaType	Controls the type of criterion to be specified - in this version only “Total Inflow” can be selected	Yes	ConditionNo
StartCriteriaLocationType	For the criteria of type ‘Total Inflow’, it must be specified for which part of the system the inflow must be evaluated. ‘Location type’ can be either ‘General’ (the whole system) or ‘Individual’ (a single node).	Yes	LocationNo
StartCriteriaLocation	For the types ‘List’ and ‘Individual’, additional information must be specified in the ‘Location’ field. For the ‘List’, a node selection file (*NSE) must be specified. For the ‘Individual’ a single node name must be types or selected from the node list.	Yes, except LocationNo=1 (General)	LocationID
StartCriteriaThreshold	Defines the threshold that must be exceeded in order to evaluate the job start criterion as true.	Yes	StartValue
StartCriterionDuration	Criteria of type ‘Total Inflow’ can optionally be extended by specifying the duration of a continous period in which the threshold must be exceeded in order to evaluate the criterion as true. The default duration is zero.	Yes	StartTime
StopCriterionThreshold	Defines the lower threshold that must be achieved in order to evaluate the job stop criterion as true	Yes	StopValue
StopCriterionDuration	Specifies the duration of a continous period in which the stopping threshold must be fulfilled in order to evaluate the stop criterion as true. The default duration is zero	Yes	StopTime

10.1.3 Initial conditions for simulated events

Appropriate initial conditions for the individual jobs are essential for achieving realistic statistics in hydrodynamic LTS simulations.

LTS provides two different methods for initializing the system. The system can either be initialized in an empty state before each job (default behaviour), or a set of hotstart files can be provided. If the latter option is used, then the specific hotstart file used for a job is selected based on the inflow to the system at the start of the job. A detailed description of the method is presented below.

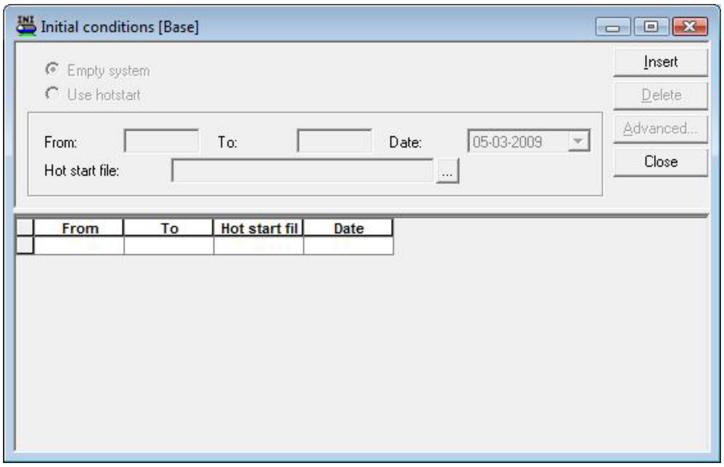
If the first option is selected (empty initial system), then the pipe system is emptied of water before each job in the LTS job list is started. Water levels in outlets will still be extended backward into the system similar to a 'normal' simulation without hotstart files (see MOUSE Pipe Flow reference manual). This option is valid only for storm drainage systems where there is no water in the system during dry periods. It is not recommended to use this option in any other situation.

If a set of hotstart files is provided, then each hotstart file must be provided with an inflow interval. This interval is the "validity interval", meaning that the specified hotstart file is valid (the file will be used) for all jobs in the job list where the inflow to the network at the beginning of the job is in that interval. Note that the inflow to the system when selecting a hotstart file is calculated from inflows to the system that are marked "runoff" or "other". Inflows marked as "dry weather flow" (DWF) are excluded from the total inflow when selecting a hotstart file. Inflow from the rainfall runoff simulation is considered "runoff" and therefore included in the total inflow calculation.

If the total inflow to the system at the beginning of job is outside all the validity intervals provided for the hotstart files, then the system will be initialized with empty pipes before the job is started.

In order to take daily variations in inflow into account, the time-of-day at which the values from the hotstart file should be used is set equal to the time-of-day when the job starts. The date however, is provided by the user. This means that care must be taken to eliminate the filling phase in the beginning of the hotstart files. This is achieved by running simulation that produce the hotstart files over two or more days before the date provided as hotstart date. It also means that the hotstart file must cover a period of at least 24 hours. If all 24 hours of the hotstart date are not covered by the hotstart file, then the simulation might fail.

This method of using the starting time of the job to account for daily variations has the drawback that weekly and seasonal variations cannot be taken into account.


Initial conditions [Base]
Insert
Delete
Advanced...
Close
From:
To:
Hot start file:
...
From
To
Hot start fil
Date

● Empty system
○ Use hotstart

Date: 05-03-2009

Figure 10.2 Initial conditions dialog

Table 10.2 The input on the initial conditions dialog

Edit field	Description	Used or required by simulation	Field name in data structure
Empty system / Use Hotstart	Radio buttons for the selection of the initial conditions mode	Yes	
Total inflow interval - From	Defines a lower threshold for intervals for total inflow (excluding DWF) which is used at the start of each job in the job list to evaluate what hotstart parameters to use.	Yes, if Hotstart	InitFrom
Total inflow interval - To	Defines an upper threshold for intervals for total inflow (excluding DWF) which is used at the start of each job in the job list to evaluate what the hotstart parameters to use	Yes, if Hotstart	InitTo

Table 10.2 The input on the initial conditions dialog

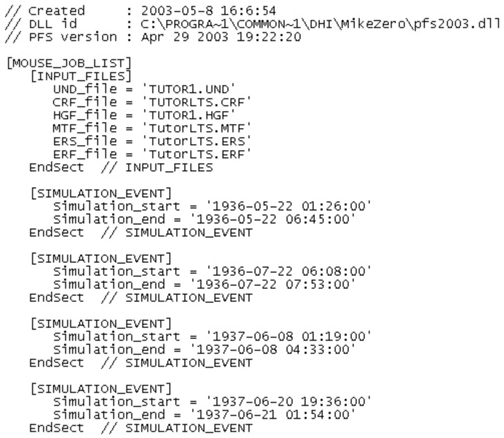
Edit field	Description	Used or required by simulation	Field name in data structure
Hotstart file	Specifies a hotstart file to be used for the system initialization in association with the specified range of total inflows (excluding DWF) detected at the beginning of certain job.	Yes, if Hotstart	HotStartFileName
Date	Specifies the date in the hotstart file to be used in the search for the hotstart conditions associated with the specified range of total inflows (excluding DWF) detected at the beginning of certain job.	Yes, If Hotstart	InitDate

10.1.4 Generating job list

On the basis of user-specified Job List criteria, the simulation input files, the simulation period (specified in the Start Computation dialog) and optionally, the initiation hot start files, a job list is automatically created.

The simulation periods for individual events represent the minimum simulation time, which may be extended in run-time according to the fulfilment of run-time criteria.

The job list is written into an ASCII file (*.MJL, written in the 'PFS' format), which contains all relevant information about input files for the simulation, definition of the time periods to be simulated (for the individual jobs), and optionally information about the hot start files and hotstart date/time. An example of a job list file is presented in Figure 10.3



```
// Created     : 2003-05-8 16:6:54
// DLL id      : C:\PROGRA~1\COMMON~1\DHI\MikeZero\pfs2003.dll
// PFS version : Apr 29 2003 19:22:20

[MOUSE_JOB_LIST]
   [INPUT_FILES]
      UND_file = 'TUTOR1.UND'
      CRF_file = 'TUTORLTS.CRF'
      HGF_file = 'TUTOR1.HGF'
      MTF_file = 'TutorLTS.MTF'
      ERS_file = 'TutorLTS.ERS'
      ERF_file = 'TutorLTS.ERF'
   EndSect  // INPUT_FILES

   [SIMULATION_EVENT]
      Simulation_start = '1936-05-22 01:26:00'
      Simulation_end = '1936-05-22 06:45:00'
   EndSect  // SIMULATION_EVENT

   [SIMULATION_EVENT]
      Simulation_start = '1936-07-22 06:08:00'
      Simulation_end = '1936-07-22 07:53:00'
   EndSect  // SIMULATION_EVENT

   [SIMULATION_EVENT]
      Simulation_start = '1937-06-08 01:19:00'
      Simulation_end = '1937-06-08 04:33:00'
   EndSect  // SIMULATION_EVENT

   [SIMULATION_EVENT]
      Simulation_start = '1937-06-20 19:36:00'
      Simulation_end = '1937-06-21 01:54:00'
   EndSect  // SIMULATION_EVENT
```

Figure 10.3 Example of a job list file

10.1.5 Edit job list

Optionally, the job list file can be edited before starting the LTS calculation. There is no dedicated dialog available for editing the job list.. Instead, Windows Notepad is used. Alternatively, any ASCII editor can be used. When editing the job list before starting the computation it is also possible to change the minimum and maximum default time step for each job. Per default, each job has assigned the same minimum and maximum time step, according to the global definition. This ca be changed by editing the values for Dt_MaxSec and Dt_MinSec (see below).

[SIMULATION_EVENT]

Simulation_start = '1937-06-08 01:19:00'

Simulation_end = '1937-06-08 04:33:00'

Dt_MaxSec = 60

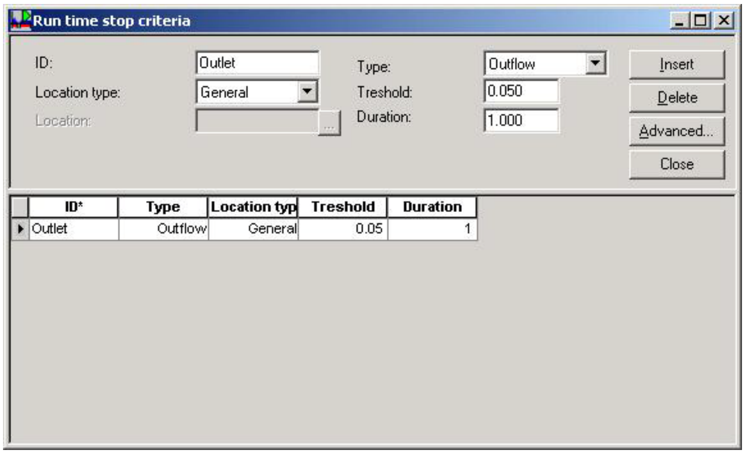
Dt_MinSec = 10

EndSect // SIMULATION_EVENT

10.1.6 Runtime stop criteria

Run-time Stop Criteria are evaluated during computations in order to determine the earliest time at which the simulation can be stopped without losing important information, i.e. in the closing phase of the simulated event. The evaluation of run-time stop criteria for the currently simulated event starts immediately after the preliminary event end (determined according to the job list) has been reached. The simulation continues as long as the runtime stop conditions are not fulfilled.

A number of independent Stop Criteria can be specified. They are evaluated through an Evaluation matrix.

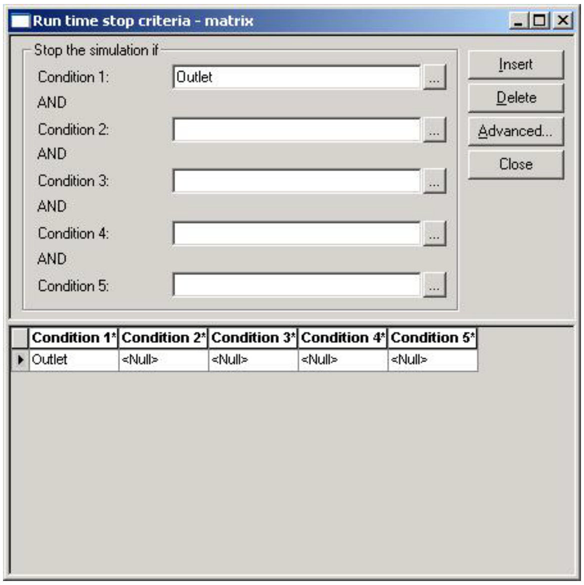

Run time stop criteria
Insert
Delete
Advanced...
Close

ID: Outlet
Location type: General
Location:
Type: Outflow
Treshold: 0.050
Duration: 1.000

ID*	Type	Location typ	Treshold	Duration
Outlet	Outflow	General	0.05	1

Figure 10.4 The runtime stop criteria dialog

Run-Time Stop Criteria Evaluation Matrix


Run time stop criteria - matrix
Insert
Delete
Advanced...
Close

Stop the simulation if
Condition 1: Outlet
AND
Condition 2:
AND
Condition 3:
AND
Condition 4:
AND
Condition 5:

Condition 1*	Condition 2*	Condition 3*	Condition 4*	Condition 5*
Outlet	<Null>	<Null>	<Null>	<Null>

Figure 10.5 The run time stop criteria maxtrix dialog

At every time step during computation (but only after the end time in the job list), each criterion is subject to logical evaluation as an element of an evaluation matrix. The evaluation matrix consists of an arbitrary number of lines, each containing up to five individual criteria. Each line of the evaluation matrix represents a combination of criteria which have to be fulfilled simultaneously (i.e. criteria are linked with the logical operator AND) if the combination is to be evaluated as TRUE and the simulation stopped.

If one line has been evaluated as FALSE, the evaluation algorithm jumps to the next line. This means that the successive lines are connected by the logical operator OR. The process continues until one line has been evaluated as TRUE (the simulation stops).

If none of the lines has been evaluated as TRUE, the simulation continues for one time step. Then, a new evaluation cycle is initiated and the entire evaluation repeats.

If no evaluation matrix is specified, periods specified in the job list will be simulated exactly, i.e. no runtime criteria will be evaluated.

Table 10.3

Edit field	Description	Used or required by simulation	Field name in data structure
ID	Each criterion is identified with a unique indentifier, which is used as a reference in the evaluation matrix.	Yes	MUID
Type	Controls the type of criterion to be specified - either total outflow, difference in inflow and outflow, total volume stored in the system, or filling degree in the system.	Yes	ConditionNo
Location	For the ‘List’, a node selection file (*.NSE) must be specified. For the ‘Individual’, a single node name must be types or selected fro the node list.	Yes	LocationNo

Table 10.3

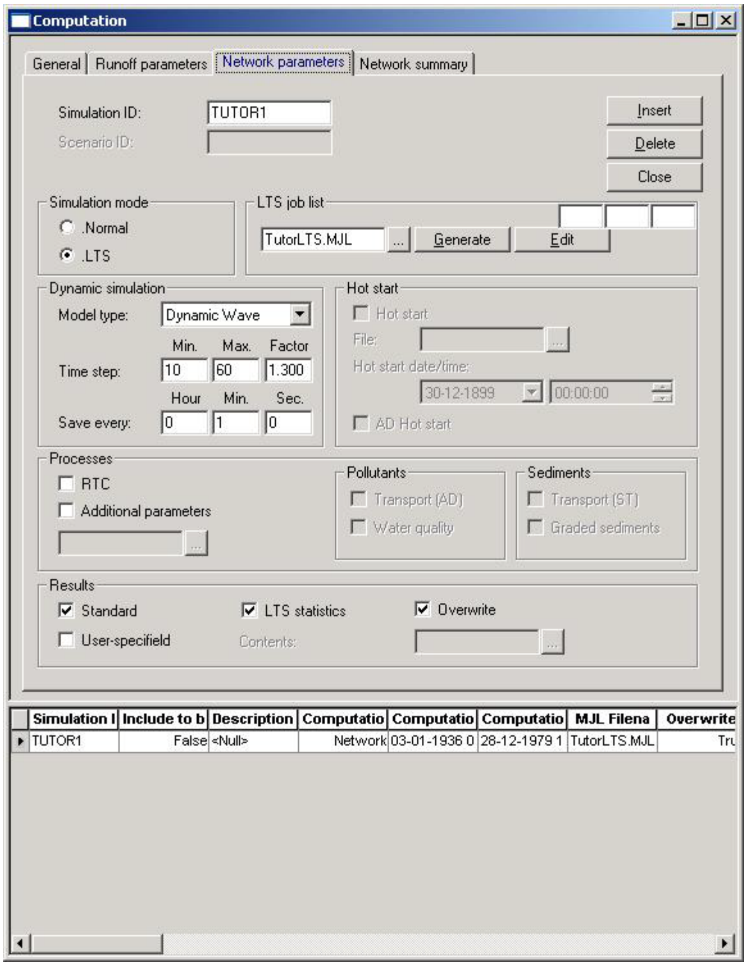
Edit field	Description	Used or required by simulation	Field name in data structure
Threshold	The residual value of the variable defined by the 'Type' that must be achieved in order to evaluate the criterion as true	Yes	StopValue
Duration	A period in which the parameter value must be below the threshold in order to evaluate the criterion as true.	Yes	StopTime

The specified runtime criteria are saved in the MOUSE LTS (*.MTF) file.

10.2 LTS Computations

Running an LTS computation (based on a job list) is usually beneficial if the total simulation period is long and the model is heavy. This has a potential of speeding up the simulation significantly, without the loss of vital information.

A LTS discontinuous computation is started from the usual Computation|Run MOUSE dialog and will be performed if the simulation mode is set to 'LTS'.



Simulation ID: TUTOR1

Scenario ID:

Simulation mode

○ .Normal

● .LTS

LTS job list: TutorLTS.MJL

Dynamic simulation

Model type: Dynamic Wave

	Min.	Max.	Factor
Time step:	10	60	1.300

	Hour	Min.	Sec.
Save every:	0	1	0

Hot start

☐ Hot start

File:

Hot start date/time: 30-12-1899 00:00:00

☐ AD Hot start

Processes

☐ RTC

☐ Additional parameters

Pollutants

☐ Transport (AD)

☐ Water quality

Sediments

☐ Transport (ST)

☐ Graded sediments

Results

☑ Standard

☐ User-specifield

☑ LTS statistics

☑ Overwrite

Contents:

	Simulation I	Include to b	Description	Computatio	Computatio	Computatio	MJL Filena	Overwrite
▸	TUTOR1	False	<Null>	Network	03-01-1936 0	28-12-1979 1	TutorLTS.MJL	Tru

Figure 10.6

Switching the simulation mode to 'LTS' opens a new part of the dialog, including fields for selecting, generating and editing a job list file.

Prior to the computation, a job list must be generated. Several different job lists can be generated and the appropriate one selected from the list.

While generating a job list, MOUSE scans the boundary inputs, and evaluates the job list start criteria in order to identify the events to be simulated. Each event is preliminarily delimited by the evaluation of the stop criterion.

Simultaneously, total loads (excluding DWF) at the simulation start for each event are determined, and the appropriate initialization file is selected. Please note that MOUSE LTS engine only handles a unique AD component. If several AD components are specified then MIKE 1D engine has to be used.

When using a job list in a LTS simulation, the specified simulation period may include the entire job list or only a small fraction - ultimately only a single event. This feature opens the possibility to use the same job list through different phases of the project - initially using only individual events for test simulations (or in the calibration process), and later extending the simulation to the entire job list. The advantage of using the job list instead of an "ordinary" continuous mode simulation is access to useful statistics available only with MOUSE LTS.

10.3 Result Files

A MOUSE LTS simulation generates the same types of result files as a normal MOUSE simulation ('standard' and 'user-specified'). Due to usually very long simulation periods, attention must be paid to keeping the size of the result files within reasonable limits. The standard MOUSE result file (*.PRF), where most of the computed time series are saved, becomes practically too large in long-term simulations, if the saving frequency is high. On the other hand, too coarse time resolution in a result file might make the saved results useless. Consequently, for simulations covering long periods, a user-specified result file will usually be of major interest.

10.3.1 User-Specified result files

The user-specified result file contains only selected time series of particular interest. This makes it especially practical for keeping the long-term results sufficiently resolved, and the standard result file may often be completely abandoned. Detailed information on the specification of the contents of user-specified result files is provided in the Chapter entitled ‘MOUSE Simulations’.

The specification of the contents of the user-specified result file for hydraulic variables (i.e. *.PRF file) applies also for the results of pollution transport model (AD), i.e. defines the user-specified *.TRF file. This practically means that at the specified locations, all computed water quality variables will be saved. Selective saving of AD variables can be further controlled through the DHIAPP.INI file, located in MIKE URBAN/Bin directory (section MouseTRAP, see below).

```
[MouseTRAP]
```

```
PRINT_WEIR_AD=1

PRINT_PUMP_AD=1

PRINT_NODE_AD=1

PRINT_GRID_AD=1

PRINT_GRID_MASS_AD=1

PRINT_MASS_BALANCE_AD=1
```

If the parameter value is changed to 0 (zero), then the item will not be written in the result file.

10.3.2 Statistics result file

When running any simulation over a longer period, focus of interest is usually turned to the operational statistics rather than to the raw time series. Therefore, MOUSE LTS provides an option to save an additional result file (*.ERF) containing various statistics (event-based and annual) over the individual computed time series and the system performance as the whole.

The statistics are computed according to the specification

The technical background for statistical computations and the process of specifying which statistical results to save are further described below

10.4 Specification of Statistic and Result Files

MIKE 1D LTS can generate a series of useful statistical results – events statistics, annual/monthly statistics and system statistics - related to the simulated period.

The events statistics - instantaneous extremes or accumulated values - are ranked according to the size in a falling order.

The annual/monthly statistics for specified locations and for the whole system, remain in chronological order.

Statistical results are stored in a statistical result file (*.ERF) (ASCII format) as well as in binary (*.res1D) files. *.ERF file can be loaded in MIKE View for presentation and generation of MU LTS statistic reports. MU LTS statistics results in *.res1D format will support future developments in MU result presentation. Currently they can be loaded into MIKE URBAN end presented as time simple series graphs, without special LTS features.

Contents of the statistical result file must be specified prior to a MIKE 1D LTS simulation. Contest statistics is defined through the 'Statistic specifications' dialog.

10.4.1 LTS Global Statistics Parameters - Event definitions

In terms of events statistics, MU LTS distinguishes events associated with extremes (maxima) of instantaneous variables (e.g. water level, discharge, mass transport), and intermittent (i.e. discontinuous) events where accumulated values (e.g. volume, pollution mass) and durations are calculated.

In order to perform the statistical analysis of the extreme events (maxima), MU LTS analyses the respective time series and identifies existence of independent extreme events. For instantaneous variables (e.g. water level, discharge, concentration, etc.), the event identification must take into account time between consecutive extreme events and the “depth” of the local minima, in order to eliminate dependent maxima and “noise” caused by numerical instabilities, a nearby pump action, etc. In other words, MU LTS applies two interevent criteria to identify independent extreme events:

1. Intervent time criterion Dt_c : two successive events (i.e. peak occurences) are considered independent if the time between the two events is larger than Dt_c.

2. Interevent level criterion p_c ($0 < p_c < 1$): two successive events are independent if the level between the events becomes smaller than p_c times the lower of the two events.

MU LTS considers two successive events as independent only if both criteria (1) and (2) are fulfilled.

Both for peak statistics and for accumulated flow statistics, two events are always considered as independent if the computation has been stopped in between by the specified stop criteria. Thus, specification of a very long dT - longer than any individual event to be simulated, would result in the number of peak values and the number of accumulated discharges and duration corresponding exactly to the number of simulated events.

Each MU LTS simulation job contains at least one statistical events. In some cases, several events may be identified within one MU LTS job.

Time criterion Dt_c is user-specified through MU LTS editor, separately for water levels (Max[H])), for discharges and velocities (Max[Q,V]) and for accumulated values and duration (Discharge [Q,T,n]).

Level criterion p_c is only accessible through initiation file DHIAPP.INI. For general information on DHIAAP.INI application, consult a separate document "DHI APP.INI and ADP Files Reference Manual.

Per default, the relevant initiation line looks as follows:

```
[MouseHD]

;  Long Term Simulation

;  Interevent level criterion.

; LTS_INTEREVENT_LEVEL_CRITERION=0.999
```

If a different p_c value is wanted, user must remove the leading semi-colon and specify the wanted p_c value $0 < p_c < 1$. Note that the default value 0.999 only removes "noise" of very small amplitude as well as prevents false extremes in case of a flat time series. Application of smaller values would reduce the number of identified independent events.

Note that for some variables (e.g. discharge) the specified p_c applies for the actual value of the variable. In other cases, e.g. water levels, p_c is applied on local water depth, i.e. water level subtracted local invert level. In principle, all variables are offset so that the minimum possible local value is zero.

Definition of independent extreme events for instantaneous variables is illustrated in several examples below.

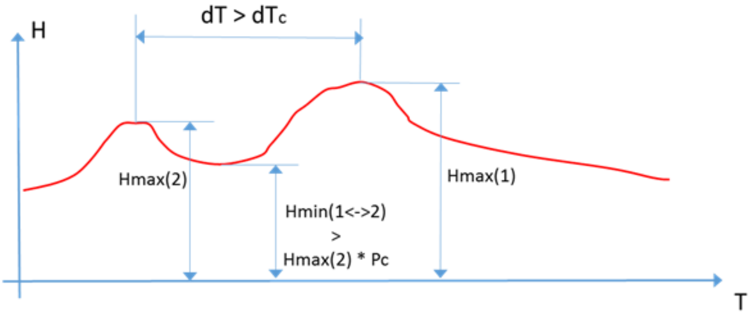


Figure 10.7 There are two water level (i.e. depth) peaks in the time series. The higher of the two depth peaks is clearly one extreme event in the current simulation. The other peak (Hmax(2)) is also a candidate for extreme depth event, because it occurs before the highest peak, longer that the specified dt_c criterion. But evaluation of level criterion is negative – the “bottom” between the two peaks is higher that the specified criterion p_c. Therefore, Hmax(2) is excluded as a separate, independent event and is considered as belonging to the same event leading to the true peak Hmax(1)

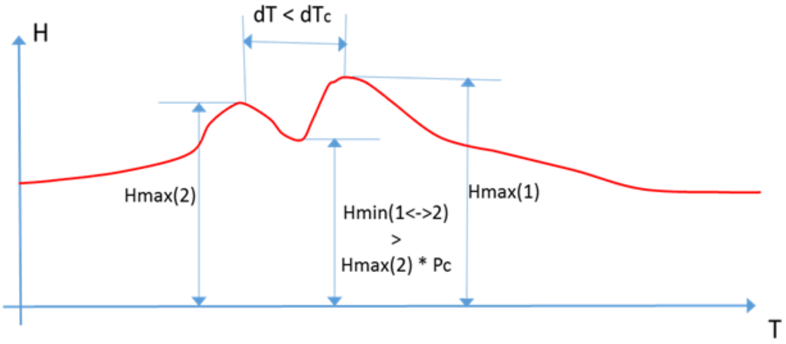


Figure 10.8 As in the previous example, there are two water level (i.e. depth) peaks in the time series. The higher of the two peaks is clearly one extreme event in the current simulation. The other peak (Hmax(2)) is also a candidate for extreme depth event, but evaluation of both criteria is negative – this peak occurs too close to the highest peak and the “bottom” between the two peaks is higher that the specified criterion p_c. Therefore, Hmax(2) is excluded as a separate, independent event. Rather, it is considered as belonging to the same event leading to the true peak Hmax(1)

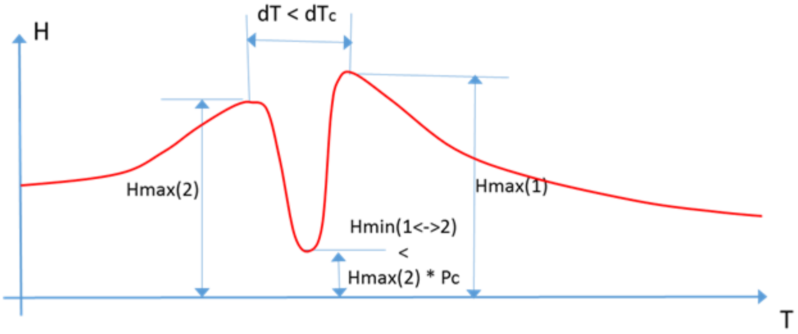


Figure 10.9 As in the prevoious examples, there are two water level (i.e. depth) peaks in the time series. The higher of the two depths (Hmax(1)) is clearly an extreme event in the current simulation. The other peak (Hmax(2)) is also a candidate for extreme depth event, but evaluation of ***dt_c*** criterion is negative – this peak occurs too close to the highest peak. Therefore, Hmax(2) is excluded as a separate, independent event and is considered as belonging to the same event leading to the true peak Hmax(1)

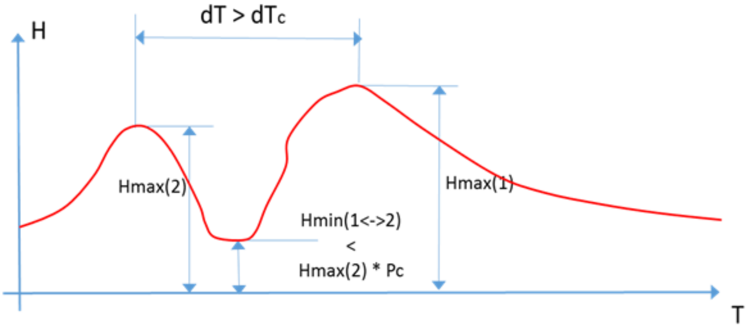


Figure 10.10 As in the previous examples, there are two water level (i.e. depth) peaks in the time series. The higher of the two peaks (Hmax(1)) is clearly an extreme event in the current simulation. The other peak (Hmax(2)) is also a candidate for extreme depth event. Both criteria are evaluated positively – this peak occurs long time before the highest peak (dT > dTc) and the “bottom” between the two peaks is lower that the specified criterion ***p_c***. Therefore, Hmax(2) is also identified as a separate, independent event.

For identification of intermittent events and statistics on accumulated variables (e.g. volume, mass) and duration, only time criterion ***Dt_c*** applies. Consecutive independent events are separated by time interval longer than ***dt_c***, where the underlying instantaneous variable (e.g. discharge for volume, mass transport for mass) is below the specified threshold (default value for the threshold is zero). E.g. two overflow events (i.e. overflow spills) are considered as independent overflow events if there is zero flow between them for a time longer than ***dt**$_c$*.

The threshold value other than zero is relevant for inherently continuous variable, e.g. discharge in pipes: even if the pipe is empty, due to the requirements of the algorithm, a small “numeric” discharge in the pipe will be reported. In order to consider such periods as “dry”, user may want to specify a small discharge threshold, other than zero. This feature is accessible through configuration file DHIAPP.INI. Per default, the relevant initiation line looks as follows:

```
[MouseHD]

;  Long Term Simulation

;  Discharge threshold for accumulated event values.

; LTS_DISCHARGE_THRESHOLD=0.0001
```

If wanted activated, user must remove the leading semi-colon and specify the wanted threshold (m3/s)

Examples of identifying independent events are shown in below.

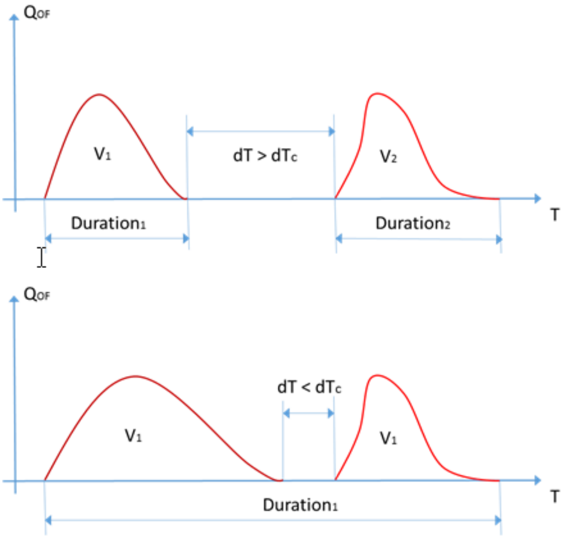
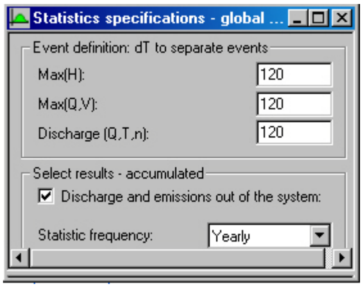


Figure 10.11 This example illustrates simulated sewer overflows, occurring as two spills separated by a period of time with zero flow. In the upper graph, the two spills are separated by ***dT*** which is longer than the specified **dT_c**. In this case, MU LTS considers the two spills as two independent overflow events and calculates their volumes and durations separately. In the lower graph, the seconds spill starts shortly after the first one has stopped, i.e. within time interval shorter than ***dTc***. In this case, MU LTS considers both spills as one overflow event (probably caused by the same meteorological event), and calculates the total volume and duration.

The main parameters which control the event definition for various types of statistics are specified in the editor 'Statistics Specification - Global'.

Also, this editor allows for toggling between the annual and monthly statistics and for requesting the statistics on system emissions (flows and pollutants out of the system).



Event definition: dT to separate events

Max(H): 120

Max(Q,V): 120

Discharge (Q,T,n): 120

Select results - accumulated

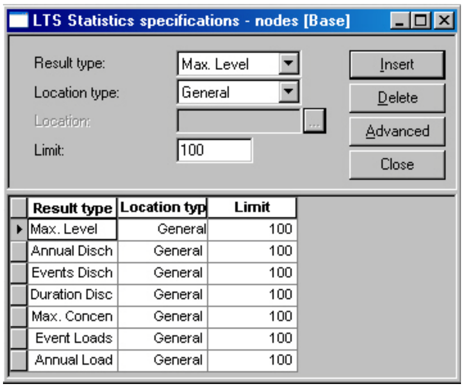
☑ Discharge and emissions out of the system:

Statistic frequency: Yearly

Figure 10.12 Global Statistics Specification

Result Type in nodes

The statistics for the results in model nodes is specified in editor 'LTS Statistics Specification - Nodes'



Result type: Max. Level

Location type: General

Location:

Limit: 100

Result type	Location typ	Limit
Max. Level	General	100
Annual Disch	General	100
Events Disch	General	100
Duration Disc	General	100
Max. Concen	General	100
Event Loads	General	100
Annual Load	General	100

Figure 10.13 LTS Statistics Specification - Nodes

The following types of statistics can be computed:Max levels:

Max levels:

Saves ranked peak water levels in nodes - instantaneous values.

Total discharge:

Saves ranked total event-accumulated discharges for weirs, pumps and outlets.

Total discharge annual/monthly:

Saves total accumulated yearly/monthly discharge for weirs, pumps and outlets in chronological order.

Duration discharge:

Saves ranked total accumulated event discharge duration for weirs, pumps and outlets.

Maximum Concentration:

Saves ranked peak concentrations in nodes (instantaneous values) for all simulated pollutants.

Event Loads:

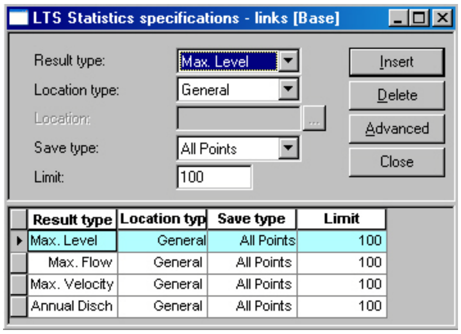
Saves ranked total event-accumulated mass flux for weirs, pumps and outlets for all simulated pollutants.

Annual Loads:

Saves total accumulated yearly/monthly mass flux for weirs, pumps and outlets for all simulated pollutants, in chronological order.

Result Type in links

The statistics for the results in model links is specified in editor 'LTS Statistics Specification - Links'



Result type: Max. Level
Location type: General
Location:
Save type: All Points
Limit: 100

Result type	Location typ	Save type	Limit
Max. Level	General	All Points	100
Max. Flow	General	All Points	100
Max. Velocity	General	All Points	100
Annual Disch	General	All Points	100

Figure 10.14 LTS Statistics Specification - Links

Max levels:

Saves ranked peak water levels in links - instantaneous values.

Max flow:

Saves ranked peak flows in links - instantaneous values.

Max velocity:

Saves ranked peak flow velocity in links - instantaneous values.

Total discharge annual/monthly:

Saves total accumulated yearly/monthly discharge for links in chronological order.

Location

For all result types it must be specified for which location(s) the desired results are to be saved. Location type can be either 'General' (the whole system), 'List' (a number of nodes/links specified on NSE/LSE/MUS-file), or 'Individual' (one node/link name). If 'List' or 'Individual' is selected, the supplementary information of selection file name or node name should be specified.

Additionally, some pre-defined groups can be specified for nodes by selecting type 'All Weirs', 'All Pumps', or 'All Outlets'.

Save Type

For links, it must be specified where in the link data should be saved. There are four saving possibilities:

Upstream grid point,

Downstream grid point,

Both upstream and downstream grid points,

All grid points.

Max number of events to save

For some statistics parameters, the number of events during a long MOUSE LTS computation can be very high and the statistics result file size can grow rapidly. Therefore, the statistical computation can be limited to a specified number of highest events, i.e. only the limited number of most significant events is included into the calculation. This is possible if the purpose of statistical calculation is to focus on extreme events only. In such a case, all insignificant events may be eliminated from the statistics as "irrelevant".

The maximum number of events must be specified within the rand 1-5000.

10.5 LTS Statistics Presentation

LTS Statistics results cannot be presented in MIKE URBAN. This functionality will be added in near future.

In the meantime, presentation facilities in MIKE View result viewing programme should be used.

11 Automatic pipe design with MOUSE

The 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.

However it should be stressed that pipe design is an iterative process, and it is therefor recommended to use the feature on models of moderate size in order to be able to make the necessary number of iterations.

11.1 Design Principles

The design is based on an iterative process, where pipe diameters are changed in order to fulfil the specified design criteria. During each iteration the diameter is kept constant, but the Manning number is changed (increased) during simulation if the criteria is not fulfilled.

In the present version the pipes are always designed from the principle of increasing the diameter until the maximum level upstream in the system is below the design criteria. Thus the module cannot be used to design pipes from downstream criteria.

After end of the simulation the maximum Manning number is used to calculate the new diameter which will then be used in next iteration.

This means that after a number of iteration - depending on how good the initial choice of diameters was - the module will stabilise results on the diameters needed to fulfil the design criteria.

Each pipe (group) is designed independently of the others, which means that the process can be long if there are many pipes, since changing of one diameter can often influence the design of both upstream and downstream pipes.

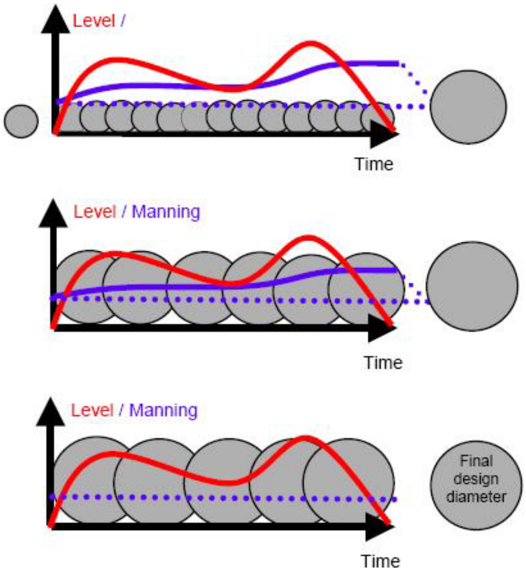


Figure 11.1 Design principle

11.2 Design Input

In the present version input data for the Pipe Design Module are specified in the Additional Parameters File (ADP). In order to activate the design module the simulation is started in the usual way after specification of the ADP file name on the Start Simulation Dialog.

The ADP file can either be written directly in a text editor, e.g. Notepad, or a wizard like tool can be utilized for writing the ADP-file for the design module (see further in 11.2.7). The format of the DESIGN section in the ADP file is as follows:

Table 11.1 The Design section of the ADP file

Keyword	No	Parameter	Description
UNIT_TYPE	1	UNITTYPE	Specifies the unit type. 1 - SI, 2 - US
DESIGNRESULT-FILE	1	“RESULTFILE“	File containing the history of the individual diameters. If present at simulation start a continuance of the previous iterations assumed.
DESIGNCON-STANTS	1-6	DC1-DC6	Parameters for internal use. Not mandatory.

Table 11.1 The Design section of the ADP file

Keyword	No	Parameter	Description
MANNINGITEM	1	DESIGNID	DesignID is used to identify the actual design group, but has no influence on the simulations.
	2	DESIGNTYPE	1 - Fixed invert levels for pipes 2 - Fixed top levels for pipes
	3	DESIGNCRITERIATYPE	1- Critical levels (specified on node dialog) 2 - Filling degree (value specified as parameter 10 FILLINGDEGREE)
	4	DESIGNGRPOUPTYPE	1- Group. All pipes are resized in parallel based on criteria in one node (value specified as parameter 6 UPSTREAMNODE) 2- Individual pipes. Each pipe is designed based on criteria in the immediate upstream node.
	5	ELEMENTS	LSE-file specifying the group of links to be designed
	6	UPSTREAMNODE	Upstream node where design criteria is evaluated. Only used for DESIGNGROUPTYPE = 1
	7	MIN	Minimum value of Manning number. Note that extreme low values can cause numerial problems during simulation. It is recommended to use values relatively close to the original value and run a few more iterations instead.
	8	MAX	Maximum value of Manning number. Note - see above
	9	LOWERLIMIT	Limit for evaluating the design criteria. Specified relative to water level of the design criteria.
	10	FILLINGDEGREE	Filling degree. Only used for DESIGNCRITERIA = 2.
Diameter Subtarget [Commercial_Pipes]	1	DIAMETER	List of commercial diameters. The diameters are not mandatory and should be given in ascending order. For UNIT_TYPE = 1 the unit is mm and for UNIT_TYPE = 2 the unit is inches.

11.2.1 Example of an ADP-file

```
[DESIGN]
  DESIGNRESULTFILE = ‘MyOwnResults.txt’
MANNINGITEM = ‘Manning’,1,1,2,’SelectedPipes.lse’,’’, 50, 200, -0.05, 1.0
[Commercial_Pipes]
Diameter = 50
Diameter = 100
Diameter = 200
EndSect // Commercial_Pipes
EndSect // DESIGN
```

11.2.2 Design Type

There are two design types available for designing pipes – fixed invert levels of pipes and fixed top levels of pipes.

When **fixed invert levels** are selected the network geometry is updated only with pipe diameters hence all invert levels are kept as in the original setup.

When **fixed top levels** are selected the top of each pipe is fixed, which means that all invert levels need to be updated according to the new diameters. All adjacent pipes are unchanged, hence when a nodes invert level is changed any attached pipe which was previously connected at the invert level of the manhole is now connected above invert of the manhole - at the original invert level of the manhole. A node invert level is never changed to a higher level than for the original setup.

Note: In the present version fixed top levels can only be used for pipes connected to manholes in both ends. If either upstream of downstream node is specified as a basin, the pipe will be designed according to the principle of fixed invert levels, regardless of design type. This is necessary in order to accommodate for the complexity of changed basin geometry, weir levels, pump start-/stop levels etc. which will often be present in basin structures.

11.2.3 Design Criteria

The design tool accommodates two types of design criteria – critical levels and filling degree. When the **Critical Levels** are applied, the pipes will be designed according to the critical level specified in the MOUSE nodes dialog.

The **Filling Degree** is in this case interpreted as the maximum water level in upstream node compared to the upstream pipe levels. E.g. if the filling degree is set to 0.8 the design criteria will be set as a water level corresponding to the pipe being filled 80% at the design node.

If more than one pipe is connected to the design node the lowest value for all connected pipes is applied in order to ensure that the criteria is fulfilled for all pipes.

11.2.4 Design Group Type

The pipes can be designed either as a group or on individual basis.

If the pipes are designed as a **group** their sizes will decrease or increase with the same factor in order to obtain the design criteria for one specified upstream node. This means that the pipe sizes will change with same percentage for all pipes.

When the design acts on **individual** pipes, each pipe will initially be automatically assigned to a design node corresponding to their immediate upstream node. Here the upstream node is defined as the node connected to the highest end of the pipe from geometric point of view.

If more than one pipe is connected to the same upstream node they will automatically be grouped together and thereby maintain their relative sizes.

11.2.5 Lower Limit

During the designing simulation there will be long periods where the water levels are below the design level. In order to avoid that these periods influence too much on the new pipe diameters a lower limit for designing is specified. Hence when the water level is below this limit the design module will not try to resize the current pipe size until the level rises above the limit again.

The limit is defined relative to water level of the design criteria. Thus a negative value indicate a lower limit below the design level and vice versa for a positive value.

11.2.6 Commercial Diameters

Through the iterations the pipe design will suggest diameters without taking into account that only some diameters are actually manufactured. Specifying the available pipe diameters can be used to let the module select the most appropriate standard pipe after end of simulation.

The standard pipe diameters should be given in increasing order and in millimetres (or inches for US-units).

The commercial diameters are an optional input.

11.2.7 Creating the ADP file for the design simulation

The ADP file can either be written directly in a text editor, e.g. Notepad, or a wizardlike tool can be utilized for writting the ADP-file for the design module. This tool can be activated under Tools|User Tools as seen in Figure 11.2.

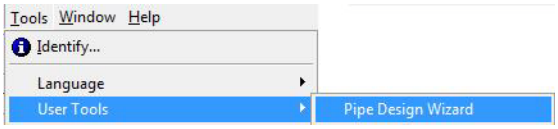
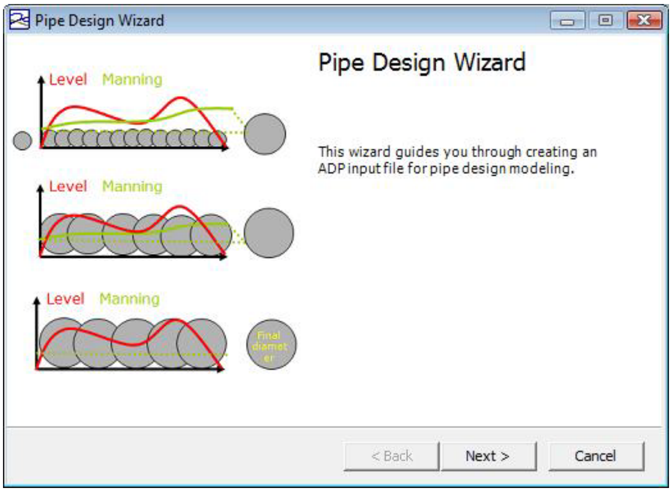

Tools
Window
Help
Identify...
Language
User Tools
Pipe Design Wizard

Figure 11.2 The tool for creating the ADP-file is now located under User Tools

When the tool is launched you are guided step-by-step setting up the ADP-file.


Pipe Design Wizard
Level
Manning
Level
Manning
Level
Manning
Final diameter
< Back
Next >
Cancel

Pipe Design Wizard

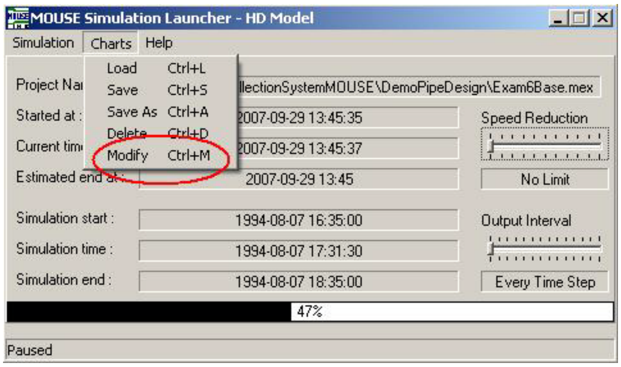
This wizard guides you through creating an ADP input file for pipe design modeling.

Figure 11.3 Start page of the wizard

11.3 Design Simulation and Output

Input data for the Pipe Design Module is specified in the Additional Parameters File (ADP). When the design module is in function the simulation is started the usual way (with specification of the ADP file name on the Start Simulation Dialog), but an additional dialog for viewing and editing the design data will be available in parallel to the MOUSE Simulation Launcher.

In the simulation launcher two new item types can be selected from the dialog <Charts>+<Modify>: Designs and Profiles.


MOUSE Simulation Launcher - HD Model
Simulation
Charts
Help
Load Ctrl+L
Save Ctrl+S
Save As Ctrl+A
Delete Ctrl+D
Modify Ctrl+M
Speed Reduction
No Limit
Output Interval
Every Time Step

Project Na llectionSystemMOUSE\DemoPipeDesign\Exam6Base.mex
Started at : 2007-09-29 13:45:35
Current tim 2007-09-29 13:45:37
Estimated end at : 2007-09-29 13:45
Simulation start : 1994-08-07 16:35:00
Simulation time : 1994-08-07 17:31:30
Simulation end : 1994-08-07 18:35:00
47%
Paused

Figure 11.4 For changing the charts displayed during simulation go to Charts|Modify

- “Designs” are time series plots for the nodes where design criteria has been specified. The default selection will include actual level, critical level and calculated factor of the manning number which is changed during the simulation
- “Profiles” are longitudinal profiles. All profiles which are saved as LPF files can be selected

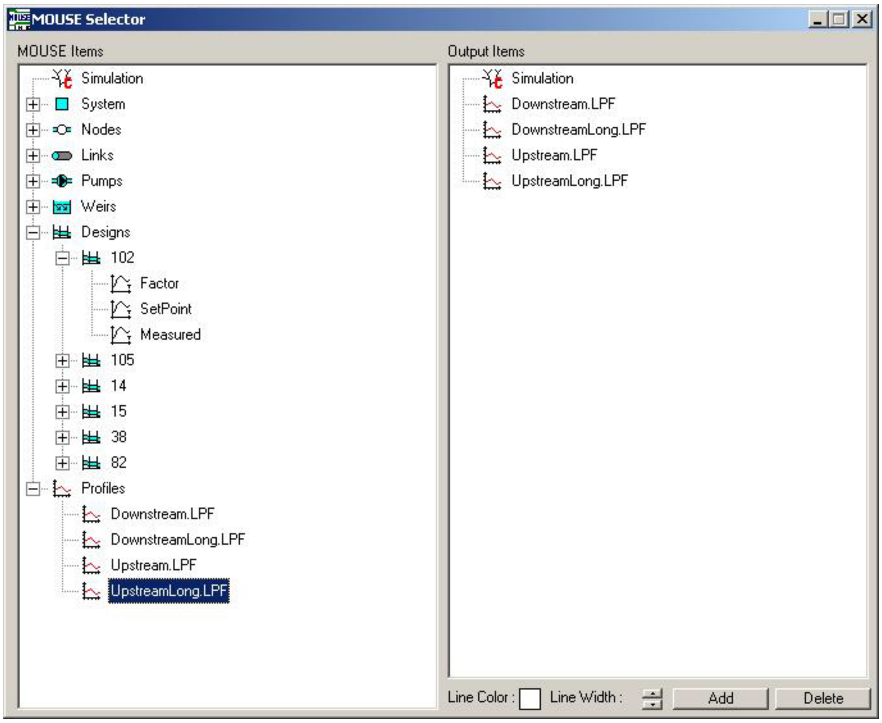
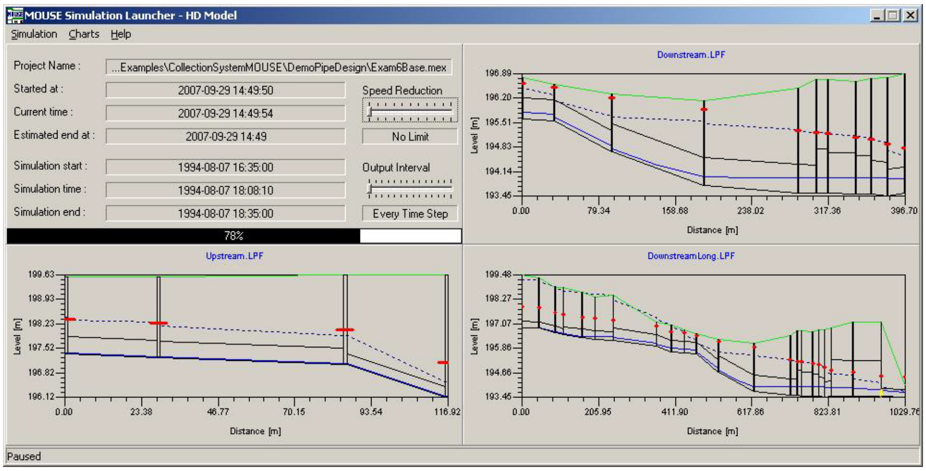

MOUSE Selector
MOUSE Items
Simulation
System
Nodes
Links
Pumps
Weirs
Designs
102
Factor
SetPoint
Measured
105
14
15
38
82
Profiles
Downstream.LPF
DownstreamLong.LPF
Upstream.LPF
UpstreamLong.LPF
Output Items
Simulation
Downstream.LPF
DownstreamLong.LPF
Upstream.LPF
UpstreamLong.LPF
Line Color :
Line Width :
Add
Delete

Figure 11.5 After choosing Charts|Modify above dialog displays

During the simulation it will hence be possible to see the results while simulated, see Figure 11.6.


MOUSE Simulation Launcher - HD Model
Simulation
Charts
Help
Speed Reduction
Output Interval
78%
Downstream.LPF
196.89
196.20
195.51
194.83
194.14
193.45
Level [m]
0.00
79.34
158.68
238.02
317.36
396.70
Distance [m]
Upstream.LPF
199.63
198.93
198.23
197.52
196.82
196.12
Level [m]
0.00
23.38
46.77
70.15
93.54
116.92
Distance [m]
DownstreamLong.LPF
199.48
198.27
197.07
195.86
194.66
193.45
Level [m]
0.00
205.95
411.90
617.86
823.81
1029.76
Distance [m]
Paused

Project Name : ...Examples\CollectionSystemMOUSE\DemoPipeDesign\Exam6Base.mex
Started at : 2007-09-29 14:49:50
Current time : 2007-09-29 14:49:54
Estimated end at : 2007-09-29 14:49
Simulation start : 1994-08-07 16:35:00
Simulation time : 1994-08-07 18:08:10
Simulation end : 1994-08-07 18:35:00
Speed Reduction: No Limit
Output Interval: Every Time Step

Figure 11.6 The results are displayed during the simulation. The critical levels used for the design are clearly marked on the profiles.

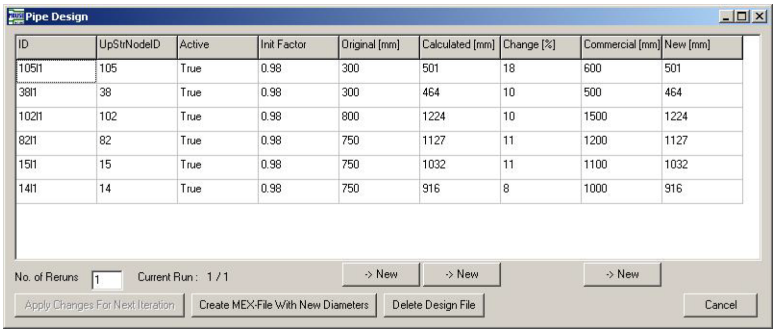
The additional dialog for pipe design is dominated by a grid with input parameters and results from the design, and can be used to adjust parameters for the design between the iterations.

The following information is available in the grid:

Table 11.2 Information found in the grid

ID	ID of the pipe
UpStrNodeID	ID of the node where design criteria are evaluated
Active	“True“ if the pipes should actually be designed - can be used to exclude one or more pipe iterations
Init Factor	Initial factor of Manning number for next iteration - values smaller than 1.0 will allow the pipe to decrease in size if necessary, however too small values can cause numerical problems. Values bigger than 1.0 are usually not relevant since it will cause the pipe to increase in size even if it's not necessary. Default is 0.98.
Original	Original diameter in mm or inches.
Calculated	Calculated diameter in mm or inches - result from last iteration.
%Change	Percentage change of diameter in last iteration
Commercial	Smallest available commercial diameter which correspond to the calculated diameter
New	Diameter in mm or inches which will be used as initial value for next iteration. Default is that results from the previous iteration will be used

Below the columns Original, Calculated and Commercial it is possible to press "->New", which will simply copy the column of diameter to the column "New", thus cause them to be used as initial values for next iteration.



ID	UpStrNodeID	Active	Init Factor	Original [mm]	Calculated [mm]	Change [%]	Commercial [mm]	New [mm]
105l1	105	True	0.98	300	501	18	600	501
38l1	38	True	0.98	300	464	10	500	464
102l1	102	True	0.98	800	1224	10	1500	1224
82l1	82	True	0.98	750	1127	11	1200	1127
15l1	15	True	0.98	750	1032	11	1100	1032
14l1	14	True	0.98	750	916	8	1000	916

No. of Reruns: 1
Current Run : 1 / 1

-> New
-> New
-> New
Apply Changes For Next Iteration
Create MEX-File With New Diameters
Delete Design File
Cancel

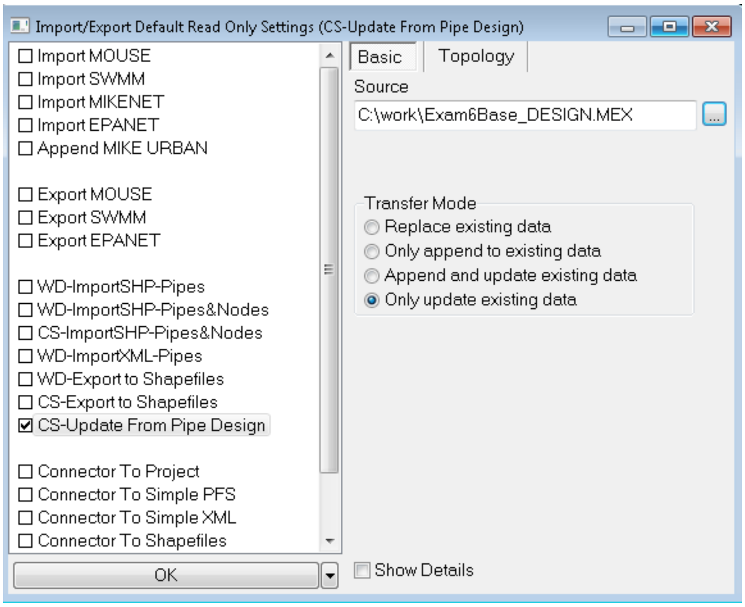
Figure 11.7 The pipe design dialog

Number of reruns can be used to make a number of iterations automatically - default value is 1.

If changes are made in the grid these must be saved in order to be used for the next iteration. Saving can be done by pressing "Apply changes for next iteration", but if data has been changed and a new iteration started, a message will occur asking if the changes should be used or lost.

Finally there is the possibility to "Create MEX-file With New Diameters" which will change the parameters for the specified elements and create a MEX file. This MEX file can be used to import and update the elements in MIKE URBAN once the optimal set of values has been identified. In order to import and update the elements in the network based on this MEX file you need to go to File|Import/Export and switch to the ‘Use Advanced’ mode. Under the list of pre-defined import jobs you will find the job ‘CS-Update From Pipe Design’. Choosing this option and the MEX file in question (see example in Figure 11.8) will update the elements in your MIKE URBAN model after pressing ‘OK’.

The option "Delete Design File" which will delete results of all the performed iterations and cause the design to start from scratch.


Import/Export Default Read Only Settings (CS-Update From Pipe Design)

☐ Import MOUSE
☐ Import SWMM
☐ Import MIKENET
☐ Import EPANET
☐ Append MIKE URBAN

☐ Export MOUSE
☐ Export SWMM
☐ Export EPANET

☐ WD-ImportSHP-Pipes
☐ WD-ImportSHP-Pipes&Nodes
☐ CS-ImportSHP-Pipes&Nodes
☐ WD-ImportXML-Pipes
☐ WD-Export to Shapefiles
☐ CS-Export to Shapefiles
☑ CS-Update From Pipe Design

☐ Connector To Project
☐ Connector To Simple PFS
☐ Connector To Simple XML
☐ Connector To Shapefiles

Source: C:\work\Exam6Base_DESIGN.MEX

Transfer Mode
○ Replace existing data
○ Only append to existing data
○ Append and update existing data
● Only update existing data

☐ Show Details

Figure 11.8 Choosing to import and update the elements based on the result of the pipe design

12 Modelling Water Quality with MOUSE

12.1 Key Features and Application Domain

MOUSE provides several modules for the simulation of and water quality for both urban catchments surfaces and sewer systems. Since pollutants are carried by sediment, sediment transport process 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 Surface Runoff Quality (SRQ), Pipe Sediment Transport (ST), Pipe Advection-Dispersion (AD), and Biological Processes (BP). Output from this, 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.

12.1.1 Surface Runoff Quality (SRQ)

The primary role of the Surface Runoff Quality (SRQ) 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 wash-out of dissolved pollutants in potholes and stilling basins.

12.1.2 Pipe Sediment Transport (ST)

Sediment deposits can greatly reduce the hydraulic capacity of sewer pipes by restricting their flow area and increasing the bed friction resistance. Pipe Sediment Transport (ST) can be used to 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 process 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.

12.1.3 Pipe Advection-Dispersion (AD)

Pipe Advection-Dispersion (AD) 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 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 can be linked to the MOUSE Long Term Statistics modelling capacities to provide long-term simulations of pollutant transport.

The Pipe-Advection module also includes the possibility of simulating water age and blend in percentages between two sources.

12.1.4 Biological Processes (BP)

Biological Processes (BP) works in conjunction with the Advection-Dispersion Module, 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. BP includes diurnal variation of foul flow discharges and user-specified concentrations of foul flow components. The sediment types included in the interaction with BP are foul flow organic sediments, and fine and course mineral in-pipe sediments originating from catchment runoff, potholes, and stilling basins. BP can account for:

- 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.

12.1.5 Interaction between water quality modules

The various water quality processes can be used independently or in conjunction, which depends on the level of required sophistication. However, using BP requires a simultaneous run of AD. Interaction between water quality modules shows the individual modules and their connections.

SRQ can be used by itself to study water quality and sediment transport processes on catchments. The result can be used as input to a river model (MIKE11) or to other water quality modules which deal with processes in the pipe model.

If only dissolved pollutants are of concern, it is sufficient to use the AD-module, possibly with BP in conjunction.

Finally, if the sediment transport is the only modelling purpose the ST module can be used separately.

Problem types and processes shows which water quality modules should be used for a list of problem types. Figure 12.1 gives an overview of the required modules for different pollutant and sediment types.

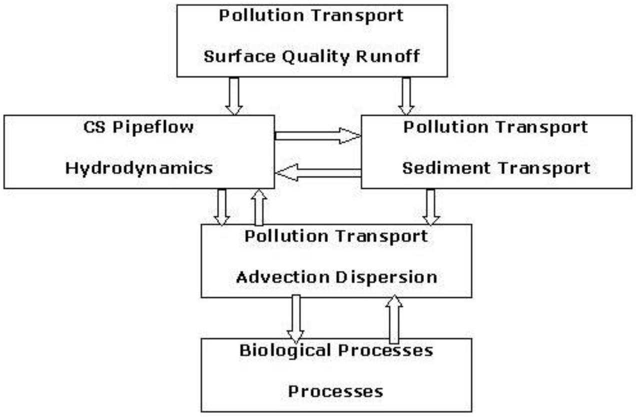

Pollution Transport
Surface Quality Runoff
CS Pipeflow
Hydrodynamics
Pollution Transport
Sediment Transport
Pollution Transport
Advection Dispersion
Biological Processes
Processes

Figure 12.1 Interaction between water quality modules

Table 12.1 Problem types and processes

Type of problem	Processes to be used
Single event pollution spill at weirs and pollution loads to treatment plants • Dissolved pollutants • Sediment attached pollutants • Sediments	• AD, AD/BP • AD/ST, AD/BP/ST • ST
Sedimentation in sewers • Uniform sediments • Graded sediments	ST
Pollution of receiving waters from surface catchments & pollution input to pipe system • Sediments • Sediments attached pollutants • Dissolved pollutants	SRQ

12.2 Surface Runoff Quality (SRQ)

Surface Runoff Quality is used to generate input for the sediment transport, advection-dispersion or water quality computation in the underlying pipe model. The Surface Runoff Quality module consists of three models:

- a model for the description of the accumulation (build-up) and wash-off of particles on the catchment surface;
- a model for the description of the surface transport of pollutants attached to the sediments;
- a model for the description of the build-up and wash-out of dissolved pollutants in gully pots.

The Build-up/Wash-off model consists of two sub-models: a model for the description of the accumulation of particles on the catchment and a model for the description of the detachment of particles by rainfall and subsequent routing of the wash-off by the overland flow.

The model works with two sediment fractions, a fine and a coarse, each fraction characterised by its mean diameter. The fine fraction typically has a particle size less than 0.1 mm and is limited in supply. On the contrary, the coarse fraction is typically unlimited in supply. Thus, for the coarse fraction, transport capacity is always satisfied and, therefore, the build-up description is only applicable to the fine sediment fraction to determine the mass available for transport.

The surface sediment transport model is based on the hydrological models available in MOUSE, i.e. the Surface Runoff Model A and the Surface Runoff Model B. In model sediment is routed according to the hydrological description in the surface runoff model. Output from the model consists of time series of sediment transport or concentration mass and accumulated mass for each sediment fraction. In addition, a time series of mass of the fine sediment fraction on the surface is available. The Sediment and Pollutants model is an optional model, which describes the attachment of pollutants to the sediment. Output from the model consists of time series of mass or concentration and ac-cumulated mass of pollutants attached to the two sediment fractions.

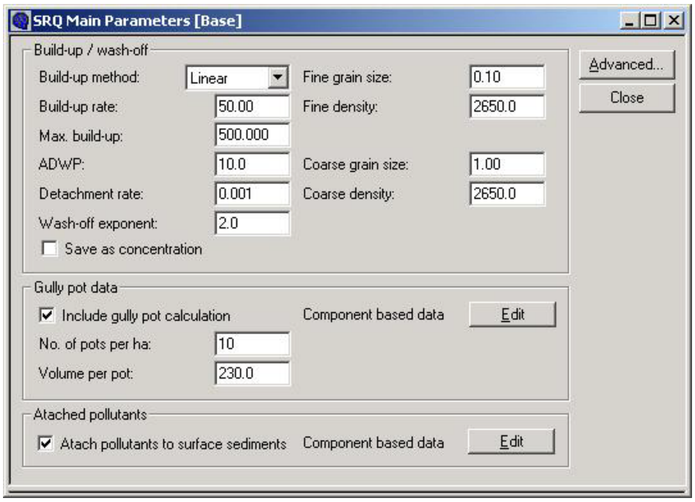
12.2.1 Surface Sediment Data Dialogs

The data for the surface sediment transport calculation is entered and edited through dedicated editors accessible under the 'MOUSE|Water Quality'-menu. The data for the surface sediment model consists of data for the Build-up/Wash-off model, data for the Sediment and Pollutant model and data for the Gully Pot model.

MOUSE| Water Quality| SRQ Main Parameters

The sediment parameters used in the build-up/wash-off computations are specified under 'MOUSE|Water Quality| SRQ Main Parameters.

The parameters for the build-up/wash-off model can be specified in the dialog displayed in the SRQ Main Parameters dialog Figure 12.2.The 'Build Up Method' defines build up method for the fine sediment on the surface. A linear build-up function, or an exponential function, can be chosen.



Build-up method: Linear
Build-up rate: 50.00
Max. build-up: 500.000
ADWP: 10.0
Detachment rate: 0.001
Wash-off exponent: 2.0
☐ Save as concentration
Fine grain size: 0.10
Fine density: 2650.0
Coarse grain size: 1.00
Coarse density: 2650.0
☑ Include gully pot calculation
Component based data
No. of pots per ha: 10
Volume per pot: 230.0
☑ Atach pollutants to surface sediments
Component based data

Figure 12.2 The SRQ Main Parameters dialog

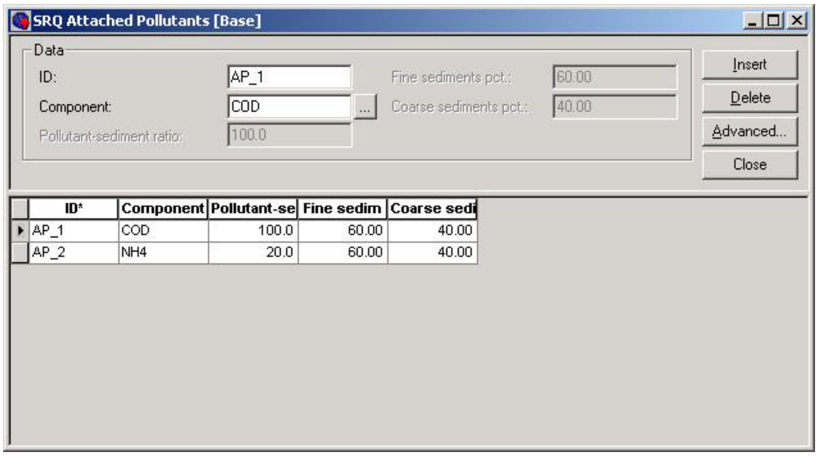
Most of the parameters are self-explanatory. ADWP stands for "Antecedent Dry Weather Period", and is responsible for the determination of initial amount of surface sediments.

Notice that if the load generated from the washoff simulation subsequently is supposed to be used as input in a pipe model, then it is imperative that the "Save as concentration" is left un-checked.

MOUSE| Water Quality| SRQ Attached Pollutants

Information related to the pollutants attached to the sediments on the catchment surface is edited under 'MOUSE|Water Quality| SRQ Attached Pollutants’.

The parameters of relevance include pollutant-sediment ratio and the distribution of the pollutant among the fine and coarse sediment fractions.



ID: AP_1
Component: COD
Pollutant-sediment ratio: 100.0
Fine sediments pct.: 60.00
Coarse sediments pct.: 40.00

ID*	Component	Pollutant-se	Fine sedim	Coarse sedi
AP_1	COD	100.0	60.00	40.00
AP_2	NH4	20.0	60.00	40.00

Figure 12.3 The SRQ Attached Pollutants dialog

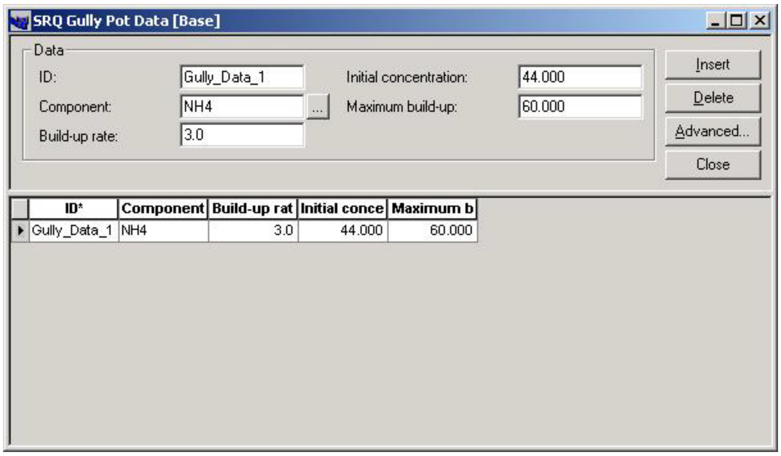
MOUSE| Water Quality| SRQ Gully Pot Data

The data supplied to the Gully pot model facilitate description of the build-up of dissolved pollutants in the gully pots during dry weather and the wash-out of dissolved pollutants from the gully pots during storm events.

Data for description of the build-up of pollutants for each component must be specified. During dry weather flow, the concentration of dissolved pollutants will build-up according to a user-specified linear build-up function, with a maximum threshold value. During storms, all sediment is routed straight through the gully pots and a simple mixing model is used to mix the incoming water with the gully pot liquor.

The data for the gully pots are specified and edited under 'MOUSE|Water Quality| SRQ Gully Pot Data - see Figure 12.4.

The parameters of relevance include initial concentration, build-up rate and maximum concentration for each of the pollutants



ID: Gully_Data_1
Component: NH4
Build-up rate: 3.0
Initial concentration: 44.000
Maximum build-up: 60.000

ID*	Component	Build-up rat	Initial conce	Maximum b
Gully_Data_1	NH4	3.0	44.000	60.000

Figure 12.4 The SRQ Gully Pot Data dialog

12.3 Advection-Dispersion (AD)

The Advection-Dispersion model can be used for calculation of the transport of dissolved or suspended substances, age of water, blend in percent between two sources, and for modelling of water temperature variation within the sewer network. The model is based on the one-dimensional transport equations for dissolved material. The equations reflect two transport mechanisms: the advective (or convective) transport with the mean flow velocity and the dispersive transport due to concentration gradients in the water. The transport equations are solved by use of an implicit finite difference scheme, which is fully time and space centred, in order to minimize the numerical dispersion. The main assumptions of the model are:

- The considered substance is completely mixed over the cross-sections. This implies that a source/sink term is considered to mix instantaneously over the cross-section;
- The substance is conservative or subject to a first order reaction (linear decay);
- Fick's diffusion law can be applied, i.e. the dispersive transport is proportional to the concentration gradient.

Special considerations have been given to the transport at manholes and other structures. More information on the technical background of the model is given in the ”MOUSE Pollution Transport Reference” manual.

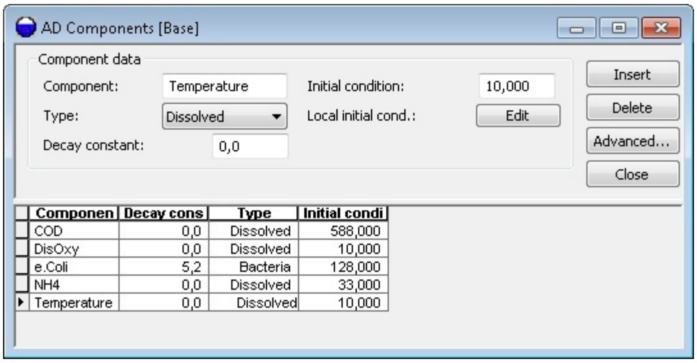
The Advection-Dispersion model requires two types of data: time series of concentrations at the model boundaries and data for full definition of the components to be modeled, e.g. initial concentrations, dispersion coefficients and decay rates.

12.3.1 Advection-Dispersion Data Dialogs

MOUSE| Water Quality| AD Components

Each of the components (substances) to be included in the Advection-Dispersion computations must be specified in the is Data dialog, shown in the AD Components dialog, shown in Figure 12.5. The naming of component is absolutely flexible, and no "reserved" or "standard" component names are prescribed.

Please note that when using MOUSE LTS engine only a single AD component should be specified. If several AD components have to be included in the simulation then use MIKE 1D engine.



Component: Temperature
Type: Dissolved
Decay constant: 0,0
Initial condition: 10,000
Local initial cond.: Edit

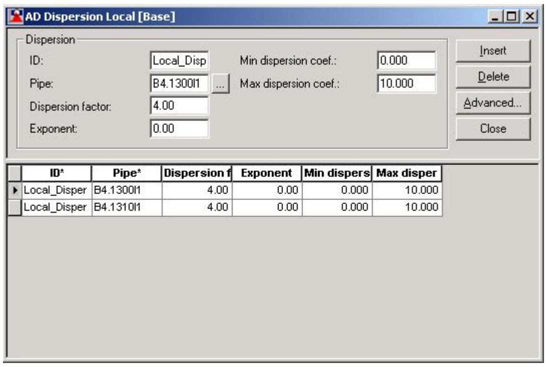
Componen	Decay cons	Type	Initial condi
COD	0,0	Dissolved	588,000
DisOxy	0,0	Dissolved	10,000
e.Coli	5,2	Bacteria	128,000
NH4	0,0	Dissolved	33,000
Temperature	0,0	Dissolved	10,000

Figure 12.5 The AD Components dialog

The specified pollution components can be declared as 'Dissolved', 'Suspended', 'Total', 'Bacteria', 'Temperature' ‘Water Age’, ‘Water Blend’ and 'Other'. This categorization is needed for correct handling of units: mass per volume (dissolved, suspended, total, other), counts per volume (bacteria), water age in hours, water blend in %, and degrees (temperature). When working with water-quality model, this categorization is further supplemented with other parameters, in order to apply the components in the WQ model properly. Practically, each of the specified components can be connected to a 'standard' component in the WQ module. By these means the WQ module 'knows' how to treat each component.

For each component, specification of an initial condition and decay coefficient can be specified. The decay coefficient cannot be given for water age and water blend type. If the specification of the initial concentration for a certain component has been omitted, a zero concentration is automatically applied. Water blend concentration must always be given as a number between 0 and 100 percent, and the sum of the two blend components must add up to 100 percent.

Local initial conditions for dissolved or suspended pollutants, water blend, water age and temperature can be specified under 'MOUSE|Water Quality|Conditions Local'., This editor can that be also be accessed directly from the 'AD Components' editor, by pressing the "Edit" button.



ID: Local_Disp
Pipe: B4.1300l1
Dispersion factor: 4.00
Exponent: 0.00
Min dispersion coef.: 0.000
Max dispersion coef.: 10.000

ID*	Pipe*	Dispersion f	Exponent	Min dispers	Max disper
Local_Disper	B4.1300l1	4.00	0.00	0.000	10.000
Local_Disper	B4.1310l1	4.00	0.00	0.000	10.000

Figure 12.6 The AD Initial Conditions Local dialog

The initial conditions are specified for individual nodes. The initial conditions in the connected conduits is calculated by linear interpolation of the concentrations specified in the upstream and downstream nodes.

By defining decay constants, non-conservative components can be specified. For such non-conservative component the concentration is assumed to decay according to the first order expression:

$$\frac{dC}{dt} = KC \tag{12.1}$$

where:

K = the decay coefficient (h^{-1})

C = the concentration

The decay constant is defined as an uniform decay over the entire model.

The AD model can be run with the components specifications only. In this case all model specific parameters (decay constant, dispersion coefficient, initial concentration) as well as boundary conditions are set to zero.

Please not that the sum of the two blend components must always add up to 100.

MOUSE | Water Quality | AD Dispersion

The dispersion coefficient is specified as a function of the flow velocity. The function is given as:

$$D = au^b \tag{12.2}$$

where:

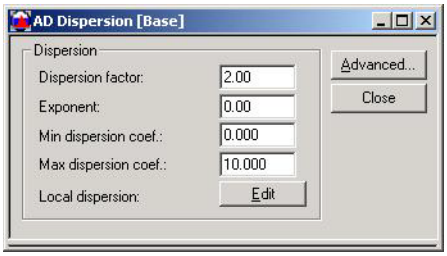
D = the dispersion coefficient (m2/s),

a = the dispersion factor,

u = the flow velocity (m/s),

b = a dimensionless exponent.

If the exponent is set equal to zero, then the dispersion coefficient is constant and independent of the flow velocity. The unit for the dispersion factor will then be m2/s. If the exponent is 1, i.e. the dispersion coefficient is a linear function of the flow velocity, then the unit of the dispersion factor will be meter, and the dispersion factor will in this case be equal to what is generally termed the dispersivity. It is possible to specify values of the minimum and the maximum dispersion coefficients, in order to limit the range of the dispersion coefficient calculated during the simulation.

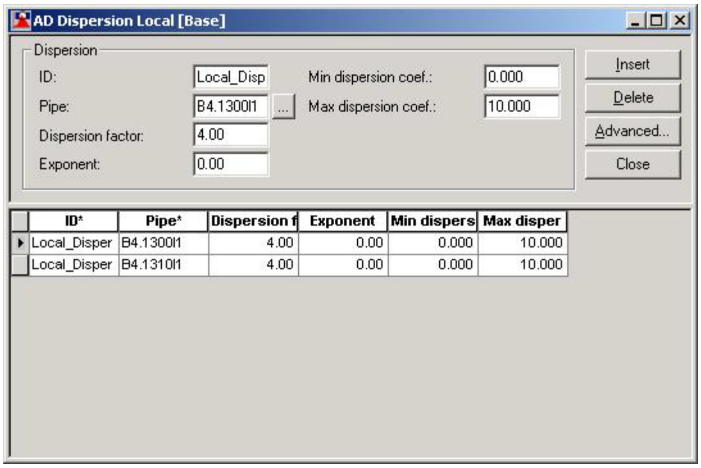


Dispersion factor: 2.00
Exponent: 0.00
Min dispersion coef.: 0.000
Max dispersion coef.: 10.000
Local dispersion:

Figure 12.7 The AD Dispersion dialog

The dispersion coefficient can be given either globally or locally. The global description will be used at all locations except for those pipes where local conditions have been specified in the menu. In the example above the global statement indicates that a constant dispersion coefficient of 2.00 m2/s is applicable, and independent of the flow velocity (the exponent is zero).

For two conduits (see the example below), the dispersion coefficients specified locally, 'overrule' the global specification and prescribe a constant dispersion coefficient of 4 m2/s.



ID: Local_Disp

Pipe: B4.1300l1

Dispersion factor: 4.00

Exponent: 0.00

Min dispersion coef.: 0.000

Max dispersion coef.: 10.000

ID*	Pipe*	Dispersion f	Exponent	Min dispers	Max disper
Local_Disper	B4.1300l1	4.00	0.00	0.000	10.000
Local_Disper	B4.1310l1	4.00	0.00	0.000	10.000

Figure 12.8 The AD Dispersion Local dialog

Advection-Dispersion and Open Boundary Conditions

The main purpose of specifying an open boundary condition is to enable exchange of mass between the MOUSE network and the surroundings (recipient) during the computations. If only inflow takes place at a boundary, then it is not necessary to specify a boundary as an open boundary. Open boundaries are normally located at outlets.

The presence of the open boundaries in the model is declared in the 'MOUSE|Boundary Conditions|External water levels’ (see description of boundary conditions).

A special consideration must be given to the open boundaries in the Advection-Dispersion model. At an open boundary a boundary time series, containing either transport or concentration in the receiving waters must be specified.

If the flow is directed out of the model, the concentrations or transport at the boundary are computed from the concentrations or transport within the model area. In this case the concentration or transport outside the model area is assumed not to influence the conditions in the model area.

If a flow reversal takes place so that flow is directed from the receiving waters into the model area, the specified time series of boundary concentrations or transport are used and assumed to be unaffected by the previous outflow from the model area. A parameter K-mix is used to ensure a smooth transition between the calculated concentration or transport at the boundary before the flow reversal and the specified time series of boundary concentrations. The description of K-mix is given in the "MOUSE Pollution Transport Reference" manual.

12.4 Biological Processes (BP)

This part of MOUSE deals with the transport and transformation of a number of chemical compounds and biological constituents found in sewer systems. These constituents are first of all organic material measured as biological or chemical oxygen demand (BOD and COD respectively), which can affect the oxygen content of the water. Other compounds are the nutrients (ammonia and phosphate), the heavy metals and various other organic or inorganic pollutants including bacteria from human and animal wastes.

The determinants that can be analyzed using BP are:

- BOD/COD (organic matter) and oxygen
- Ammonia and phosphate
- Heavy metals
- Bacteria : Total and Faecal Coliform and Streptococci
- Pollutants attached to sediments

The BP-module is coupled to the AD-module. This means that while the AD-module simulates the transport process, the BP-module deals with the simultaneous transforming processes of the com-pounds in the sewer. The AD-module is running in parallel to the BP-module when the WQ-module is activated.

The BP module has been constructed to be as flexible as possible. This has resulted in a structure where the determinants mentioned above can be studied individually or rather in the groups indicated above.

12.4.1 Biological Processes Dialog (MOUSE|Water Quality|WQ Process Model)

The determinants included in BP are distributed in 6 groups as shown in the menu.

Group 1 includes the aspects of the classical BOD/DO problem, e.g. the degradation of organic material and subsequent consumption of oxygen. The oxygen demand can be specified as BOD or as COD. The processes included in the simulation of these parameters are the degradation of organic material and the resulting oxygen consumption and the reaeration of oxygen.

Group 2 includes nutrients relevant for sewer systems, e.g. the reduced form of nitrogen - ammonia/ammonium and phosphate. The nutrients are modeled as conservative substances.

Group 3 is the heavy metals. The model does not a priori determines which metals to be included. This is defined by the user. The Heavy metals are modeled similarly to the nutrients, e.g. as conservative substances.

Group 4 is bacteria arising from human and animal wastes. Three groups of bacteria can be modeled: faecal coliform, total coliform and streptococci. The coliform bacteria are assumed to be subject to a decay (death) that depends on the temperature.

Group 5 and 6 are facilities for specifying the parameters needed when pollutants are modeled as sediment attached. This can be any pollutant (BOD, heavy metals etc).

The processes are described theoretically in the "MOUSE Biological Processes Reference" manual. Here, a general explanation of the processes and inventory of the required input parameters are given in connection to the discussion about relevant data forms.

The determinants included in this group is the BOD/COD and dissolved oxygen. The BOD is modeled as two fractions: the dissolved and readily degradable BOD and the suspended BOD that is degraded slower and of which a part is the heterotrophic bacteria responsible for the degradation of a part of the dissolved BOD. The dissolved BOD can also in some cases besides the degradation by the bacteria, be degraded in a biofilm at the inside of the sewer pipe and at the water surface.

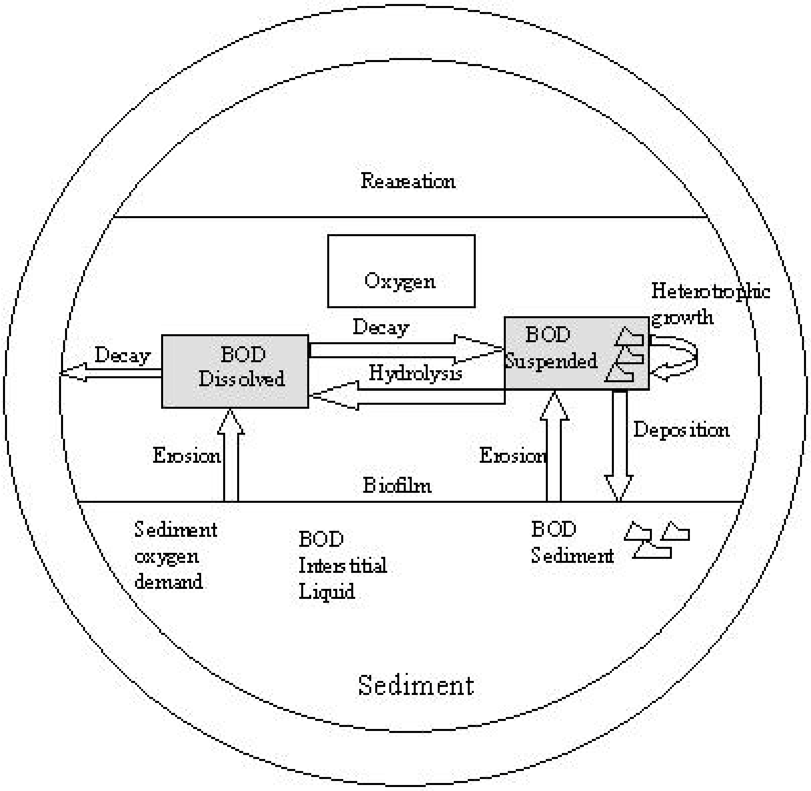
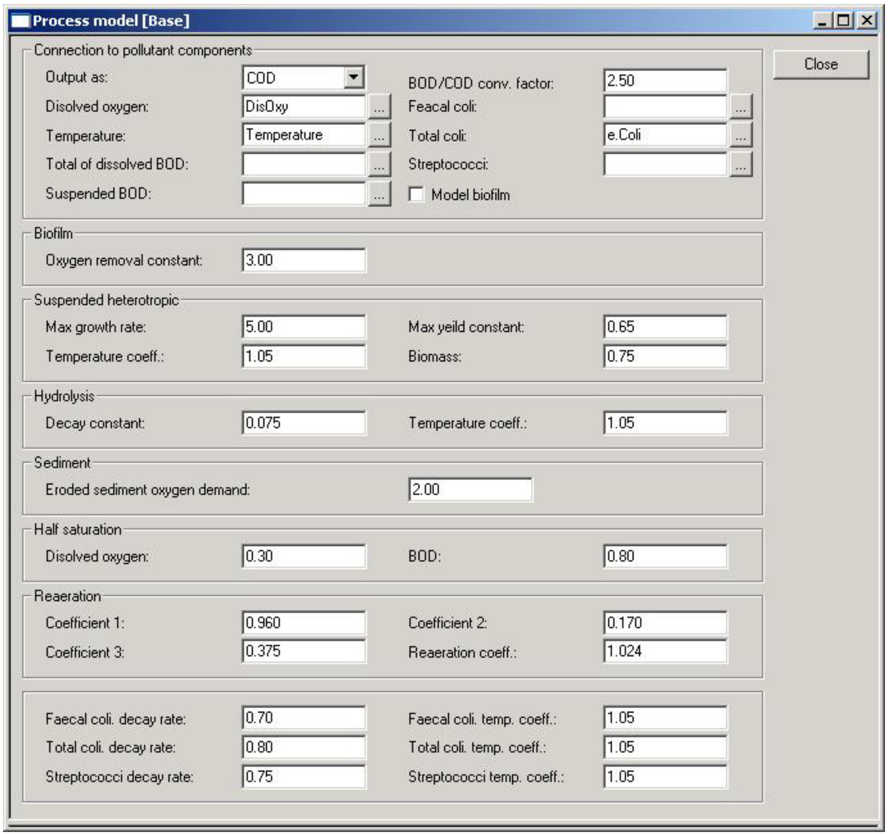

Reareation
Oxygen
Heterotrophic growth
Decay
BOD Suspended
Decay
BOD Dissolved
Hydrolysis
Deposition
Erosion
Erosion
Biofilm
Sediment oxygen demand
BOD Interstitial Liquid
BOD Sediment
Sediment

Figure 12.9 The processes included in the modelling of BOD/COD - dissolved oxygen problems in sewer pipes modeled with MOUSE

The general parameters for the BOD/COD - DO processes are inserted through the data dialog shown dialog in the WQ Process model dialog.



Connection to pollutant components

Output as: COD
Disolved oxygen: DisOxy
Temperature: Temperature
Total of dissolved BOD:
Suspended BOD:
BOD/COD conv. factor: 2.50
Feacal coli:
Total coli: e.Coli
Streptococci:
☐ Model biofilm

Biofilm

Oxygen removal constant: 3.00

Suspended heterotropic

Max growth rate: 5.00
Temperature coeff.: 1.05
Max yeild constant: 0.65
Biomass: 0.75

Hydrolysis

Decay constant: 0.075
Temperature coeff.: 1.05

Sediment

Eroded sediment oxygen demand: 2.00

Half saturation

Disolved oxygen: 0.30
BOD: 0.80

Reaeration

Coefficient 1: 0.960
Coefficient 3: 0.375
Coefficient 2: 0.170
Reaeration coeff.: 1.024

Faecal coli. decay rate: 0.70
Total coli. decay rate: 0.80
Streptococci decay rate: 0.75
Faecal coli. temp. coeff.: 1.05
Total coli. temp. coeff.: 1.05
Streptococci temp. coeff.: 1.05

Figure 12.10 The WQ Process model dialog

The reaeration is a relevant process in cases of non-full flow in the pipes. An empirical expression based on measurements in sewer pipes is provided to simulate the reaeration process. The reaeration rate is modeled as a function of the flow velocity, the mean hydraulic water depth and the pipe slope. The reaeration rate is also dependent on the temperature expresses by an Arrhenius temperature coefficient. The reaeration constant is calculated at 20°C and adjusted to the actual temperature by means of the temperature coefficient.

The constants in this expression can be changed by the user (totally three constants). Besides these constants a temperature coefficient can be specified.

The Group 4 of determinants is the bacteria that can be faecal and total coliform bacteria and streptococci. Most pathogenic microorganisms are usually unable to multiply or survive for extensive periods in the aquatic environment. Sedimentation, starvation, pH changes, temperature plus competition with and predation by other naturally occurring microorganisms are factors involved in the decay of pathogenic bacteria in the marine environment.

Escherichia coli (E. coli) is one of the dominant species in faeces from human and warm-blooded animals. The organism itself is normally considered non-pathogenic, but is very often used as indicator organism for faecal pollution and hence a potential of real pathogenic organisms (e.g. other bacteria and vira). Infections obtained from bathing waters are most often caused by vira, protozoas, Shigella and Campylobacter.

Enteric bacteria die-off can be modeled very well by a first order decay reaction. However, the die-off constant or decay rate is highly variable due to interaction by environmental factors on bacterial die-off. The main factors are suspected to be: light, temperature and salinity variations. In sewers, the only relevant factor among these to include is the temperature. Other less important factors are, e.g. predation,sedimentation and the amount of available nutrients.

For each bacteria group, a first order decay rate at 20ºC has to be specified together with a temperature coefficient.

12.5 Water quality (MIKE ECO Lab)

Hydrodynamic and advection-dispersion (transport) simulations can be coupled with MIKE ECO Lab for efficient and versatile simulation of water quality processes in sewer systems. This option opens for practically unlimited range of processes to be simulated, as long as these can be described by a consistent model.

MIKE ECO Lab is a highly flexible and open framework for formulation of water quality models. MIKE ECO Lab utilises a concept of templates where water quality models are defined transparently: Complete contents of the pre-defined templates supplied by the MU installation is accessible for review and can be modified without any need to involve the software vendor. Also, completely new templates can be developed by the user.

The MIKE URBAN installation includes two pre-defined water quality templates that can be applied “as is” - or they can be adjusted by the user to conform to the specific project requirements.

A MIKE ECO Lab template is as ASCII file which can be accessed and edited by the MIKE ECO Lab editor. Note that MIKE ECO Lab is part of MIKE Zero and it is necessary install MIKE Zero and acquire a license for MIKE ECO Lab in order to use MIKE ECO Lab with MIKE URBAN.

Please consult the MIKE Zero ECO Lab documentation on how to create or modify model templates, i.e. how to work with the MIKE ECO Lab editor.

Please also note that MIKE ECO Lab couplings are **only possible with the MIKE 1D engine**. The older MOUSE engine does not support MIKE ECO Lab couplings.

MIKE ECO Lab implementation in MIKE URBAN is limited to coupling the existing MIKE ECO Lab templates for MIKE URBAN with the hydrodynamic and transport model and to editing the MIKE ECO Lab model constants during calibration.

A simulation job with the coupled HD/AD and MIKE ECO Lab model is set up and launched from within MIKE URBAN in a usual way.

MIKE ECO Lab in MIKE URBAN is supported by a set of four MIKE ECO Lab editors in MIKE URBAN:

- Templates and Assignments
- State Variables
- Forcings
- Constants

These four editors and the database tables behind them are described below.

12.5.1 MIKE ECO Lab Templates and Assignments

This editor is used for loading (import) MIKE ECO Lab templates into MIKE URBAN and assigning them to the model elements – nodes and links. The data for this editor is stored in the msm_EcoLabTemplate MU database table.

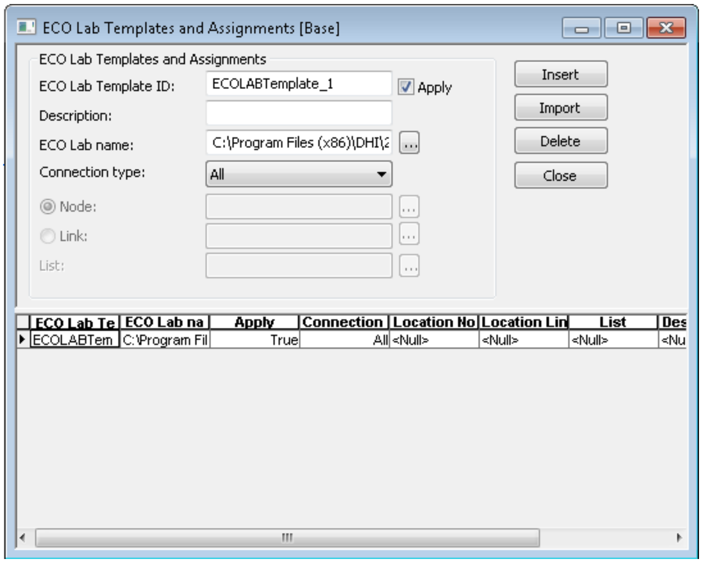
Before importing a template, a new table record must be created by <Insert> button. Then, the wanted template is browsed and selected. Actual loading of the selected template is done by clicking on <Import> button. The import scans the template and extracts information about state variables, forcings, and constants, and loads it into the respective tables. Contents of these tables is described in the following three sub-sections.

A template can be assigned to the entire model (i.e. to all points where HD/AD calculation is performed or to an arbitrary sub-set. A sub-set is defined by a MIKE URBAN Selection file (*.mus) or as an individual element (a node or a link).

Note that there can only be one MIKE ECO Lab template on a node or link. If a template is used for *All*, then a template on a single or multiple nodes/links will override the template on all elements.

If a specific template should be connected to both nodes and links, it is necessary to create a selection file (*.mus) file to link the desired features to the msm_EcoLabTemplate table to the same template but connected to nodes and links respectively.

The "Apply" check box can be used to disable a template.


ECO Lab Templates and Assignments [Base]
ECO Lab Templates and Assignments
Insert
Import
Delete
Close

ECO Lab Template ID: ECOLABTemplate_1 ☑ Apply

Description:

ECO Lab name: C:\Program Files (x86)\DHI\2

Connection type: All

● Node:

○ Link:

List:

	ECO Lab Te	ECO Lab na	Apply	Connection	Location No	Location Lin	List	Des
▸	ECOLABTem	C:\Program Fil	True	All	<Null>	<Null>	<Null>	<Nu

Figure 12.11 MIKE ECO Lab Templates and Assignments

Table 12.2 Field description for MIKE ECO Lab Templates editor

Edit field label	Database field	Description
ECO Lab Template ID	msm_EcoLabTemplate.MUID	User-specified template ID
Description	msm_EcoLabTemplate.Description	Free text description of the template
ECO Lab name	msm_EcoLabTemplate.TemplatePath	Path/filename of the MIKE ECO Lab template
Connection type	msm_EcoLabTemplate.ConnectionType	1 = All 2 = Indiviidual 3 = List
Node	msm_EcoLabTemplate.NodeID	Individual node assignment
Link	msm_EcoLabTemplate.LinkID	Individual link assignment
List	msm_EcoLabTemplate.ListName	Path/filename of the *.MUS file for the template assignment

Pressing <Import> button after the template has been already loaded (imported) into MIKE URBAN will re-load the template and any associations of the MIKE ECO Lab state variables with MIKE URBAN AD components, as well as any modifications of the MIKE ECO Lab constants will be lost.

12.5.2 MIKE ECO Lab State Variables

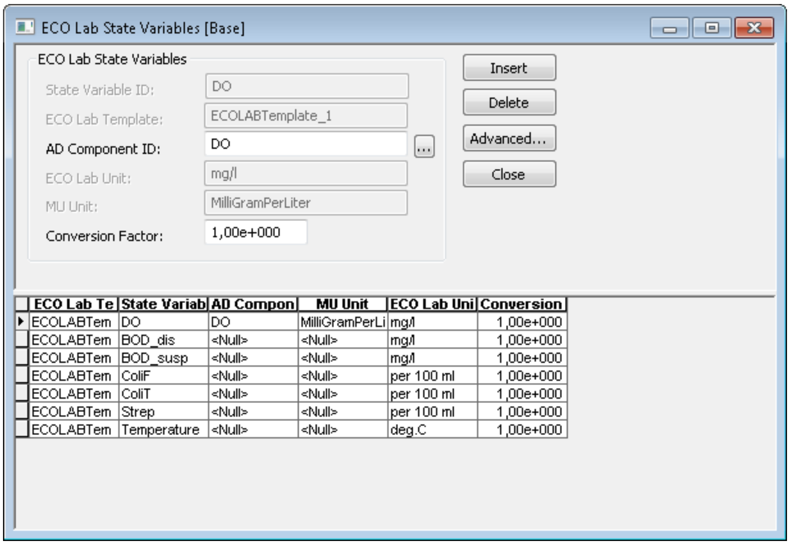
This editor is used for coupling MIKE URBAN AD components to MIKE ECO Lab state variables and for possible reconciliation of differences in units used in MIKE URBAN and in MIKE ECO Lab, respectively. The data for this editor is stored in the msm_EcoLab.Component MU database table.

In MIKE URBAN database table msm_ADComponent there should be an entry for every MIKE ECO Lab state variable defined in the MIKE ECO Lab template. This means that all MIKE ECO Lab state variables are subject to both AD transport mechanisms and to MIKE ECO Lab transformations.

AD components must be specified by the user. These AD components must represent exactly the same constituents as those in the MIKE ECO Lab template, but their names can be freely defined. The specified initial values for the AD components will be applied in the computation. Initial (default) values for the MIKE ECO Lab state variables defined in the template will be ignored.

The import operation described in the MIKE ECO Lab Templates and Assignments section will automatically create the required entries in this table. The user must couple each of these MIKE ECO Lab state variables (components) to the relevant AD component.

The "Conversion Factor" entry (default value 1, i.e. no conversion) can be set different from 1.0 if the units for the AD components and MIKE ECO Lab state variables are different. Please note that the AD components require a specific unit while MIKE ECO Lab does not require a specific unit for a specific component type.



ECO Lab State Variables

State Variable ID: DO
ECO Lab Template: ECOLABTemplate_1
AD Component ID: DO
ECO Lab Unit: mg/l
MU Unit: MilliGramPerLiter
Conversion Factor: 1,00e+000

	ECO Lab Te	State Variab	AD Compon	MU Unit	ECO Lab Uni	Conversion
▶	ECOLABTem	DO	DO	MilliGramPerLi	mg/l	1,00e+000
	ECOLABTem	BOD_dis	<Null>	<Null>	mg/l	1,00e+000
	ECOLABTem	BOD_susp	<Null>	<Null>	mg/l	1,00e+000
	ECOLABTem	ColiF	<Null>	<Null>	per 100 ml	1,00e+000
	ECOLABTem	ColiT	<Null>	<Null>	per 100 ml	1,00e+000
	ECOLABTem	Strep	<Null>	<Null>	per 100 ml	1,00e+000
	ECOLABTem	Temperature	<Null>	<Null>	deg.C	1,00e+000

Figure 12.12 MIKE ECO Lab State Variables

Table 12.3 Field description for MIKE ECO Lab State Variables editor

Edit field label	Database field	Description
Component ID	msm_EcoLabComponent.ComponentID	MIKE ECO Lab state variable identifier
MIKE ECO Lab Template	msm_EcoLabComponent.TempID	MIKE URBAN name of the template the component belongs to
AD Component ID	msm_EcoLabComponent.ADComponentID	MU AD component which is coupled to the MIKE ECO Lab state variable
ECO Lab Unit	msm_EcoLabComponent.EcoLabUnit	Unit for the MIKE ECO Lab state variable, as defined in the template
MU Unit	msm_EcoLabComponent.MUUnit	Unit for the MU AD Component, as defined in the AD model setup
Conversion factor	msm_EcoLabComponent.ConvFactor	Conversion factor for possible units conversion between MU and MIKE ECO Lab

Please note that the only active fields in this editor are those for coupling the MU AD components with MIKE ECO Lab state variables, and for specification of the conversion factor. The remaining fields are read-only.

<Insert> and <Delete> buttons have a functionality limited to very special applications. Inserting a new state variable in Mike Urban will have no effect whatsoever on the functionality of the MIKE ECO Lab template, as it will not be linked to any processes. On the other hand, deleting a state variable would destroy the MIKE ECO Lab template functionality, with unpredictable consequences.

12.5.3 MIKE ECO Lab Forcings

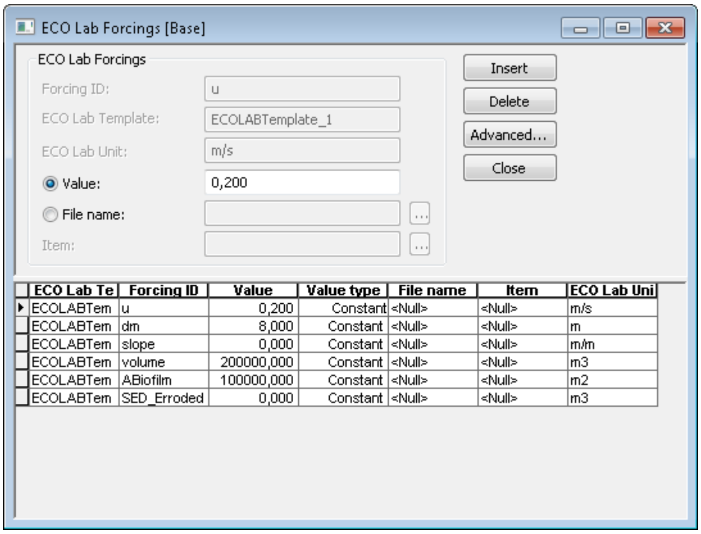
Forcings are external variables associated with computational points (i.e. are spatially fixed), constant or variable in time, which affect some of the processes on the state variables.

Forcings can be “Built-In” or “User Defined”. “Built-In” means that forcings are provided by the hydraulic model through coupling with MIKE ECO Lab. “User De-fined” forcings must be provided by the user, either as constant values or as time series (dfs0 file name and item in the file).

This editor is used for setting the User Defined forcings associated with the loaded MIKE ECO Lab templates. Built-In forcings are also imported into

MIKE URBAN, but specification of their values as constants or time series would not have any effect on the MIKE ECO Lab computations.

The data table in MIKE URBAN database containing MIKE ECO Lab forcings is msm_EcoLabForcing. The database records are created automatically during the template import. The table contains as many forcings as there are in the MIKE ECO Lab template.



Forcing ID: u

ECO Lab Template: ECOLABTemplate_1

ECO Lab Unit: m/s

● Value: 0,200

○ File name:

Item:

ECO Lab Te	Forcing ID	Value	Value type	File name	Item	ECO Lab Uni
ECOLABTem	u	0,200	Constant	<Null>	<Null>	m/s
ECOLABTem	dm	8,000	Constant	<Null>	<Null>	m
ECOLABTem	slope	0,000	Constant	<Null>	<Null>	m/m
ECOLABTem	volume	200000,000	Constant	<Null>	<Null>	m3
ECOLABTem	ABiofilm	100000,000	Constant	<Null>	<Null>	m2
ECOLABTem	SED_Erroded	0,000	Constant	<Null>	<Null>	m3

Figure 12.13 MIKE ECO Lab Forcings

Table 12.4 Field description for MIKE ECO Lab Forcings editor

Edit field label	Database field	Description
Forcing ID	msm_EcoLabForcing.ForcingID	MIKE ECO Lab forcing identifier
MIKE ECO Lab Template	msm_EcoLabForcing.TempID	MIKE URBAN name of the template the forcing belongs to
ECO Lab Unit	msm_EcoLabForcing.EcoLabUnit	Unit for the MIKE ECO Lab forcing, as defined in the template
Value	msm_EcoLabForcing.DefaultValue	Constant value for the forcing
File	msm_EcoLabForcing.FileName	Path/filename for dfs0 time series file containing forcing TS data
Item	msm_EcoLabForcing.ItemLabel	dfs0 item reference

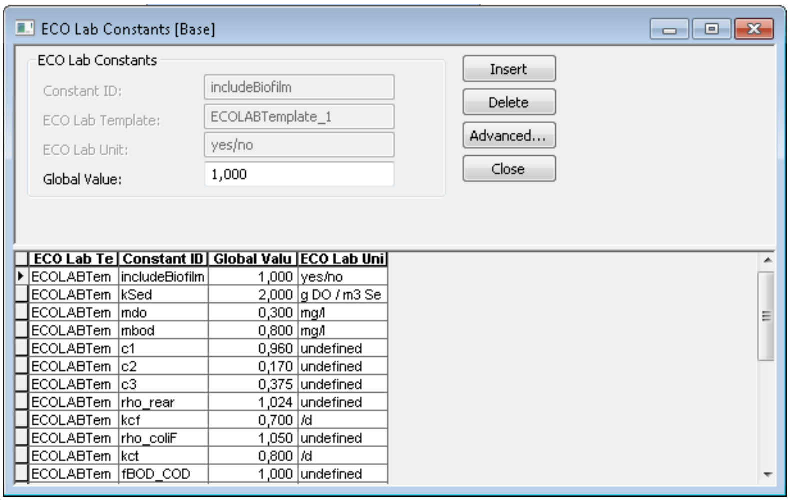
Please note that the only active fields are those for the specification of the forcing's constant value or the reference to dfs0 TS file. The remaining fields are read-only.

<Insert> and <Delete> buttons have a functionality limited to very special applications. Inserting a new forcing in Mike Urban will have no effect whatsoever on the functionality of the MIKE ECO Lab template, as it will not be linked to any processes. On the other hand, deleting a forcing would destroy the MIKE ECO Lab template functionality, with unpredictable consequences.

12.5.4 MIKE ECO Lab Constants

This editor is used for setting the values of constants used in the MIKE ECO Lab template, typically in relation to the model calibration.

The data table in MIKE URBAN database containing MIKE ECO Lab constants is msm_EcoLabCoeff. The database records are created automatically during the template import. The table contains as many constants as there are in the MIKE ECO Lab template.



Constant ID: includeBiofilm
ECO Lab Template: ECOLABTemplate_1
ECO Lab Unit: yes/no
Global Value: 1,000

ECO Lab Te	Constant ID	Global Valu	ECO Lab Uni
ECOLABTem	includeBiofilm	1,000	yes/no
ECOLABTem	kSed	2,000	g DO / m3 Se
ECOLABTem	mdo	0,300	mg/l
ECOLABTem	mbod	0,800	mg/l
ECOLABTem	c1	0,960	undefined
ECOLABTem	c2	0,170	undefined
ECOLABTem	c3	0,375	undefined
ECOLABTem	rho_rear	1,024	undefined
ECOLABTem	kcf	0,700	/d
ECOLABTem	rho_coliF	1,050	undefined
ECOLABTem	kct	0,800	/d
ECOLABTem	fBOD_COD	1,000	undefined

Figure 12.14 MIKE ECO Lab Constants

Table 12.5 Field description for MIKE ECO Lab Constants editor

Edit field label	Database field	Description
Constant ID	msm_EcoLabCoeff.CoeffID	MIKE ECO Lab constant identifier
MIKE ECO Lab Template	msm_EcoLabCoeff.TempID	MIKE URBAN name of the template the constant belongs to
Unit	msm_EcoLabCoeff.Unit	Unit for the MIKE ECO Lab constant, as defined in the template
Global Value	msm_EcoLabCoeff.CoeffValue	Value of the constant

Please note that the only active field in the one containing the constant value. The remaining fields are read-only.

<Insert> and <Delete> buttons have a functionality limited to very special applications. Inserting a new constant in Mike Urban will have no effect whatsoever on the functionality of the MIKE ECO Lab template, as it will not be linked to any processes. On the other hand, deleting a constant would destroy the MIKE ECO Lab template functionality, with unpredictable consequences.

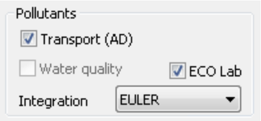
12.5.5 Running MIKE ECO Lab simulation

To run a MIKE ECO Lab simulation, the network model must be prepared in consistency with the MIKE ECO Lab template which is to be simulated. This means that it should contain at least those AD components which are to be coupled with MIKE ECO Lab state variables.

Further, the wanted MIKE ECO Lab template should be imported to MIKE URBAN. Possible unit inconsistencies must be considered and resolved by providing proper conversion factors. Values of the constant should be reviewed and modified, as appropriate. User-defined forcings should be specified, either as constants or as time series.

The network simulation with MIKE ECO Lab is activated by checking “Transport(AD)” and “ECO Lab” checkboxes, as shown in Figure 12.15. Default integration method (EULER) can be applied in most cases.

The simulation will result in three files containing all the hydrodynamic information (*.res1d), advection and dispersion results for the state variables (*AD.res1D) and the auxiliary variables, processes and derived outputs included in the MIKE ECO Lab results file (ADEL.res1D).



Pollutants

☑ Transport (AD)

☐ Water quality ☑ ECO Lab

Integration EULER

Figure 12.15 MIKE ECO Lab Check

12.6 Sediment Transport (ST)

MOUSE supports the computation of non-cohesive sediment transport by means of four different sediment transport formulae. Each of the sediment transport formulae can be used in any of the sediment transport models available in MOUSE ST. The sediment transport models are of different levels of complexity. The most simple model calculates only the hydraulic resistance from sediment deposits in the pipes and the most complex model is a full morphological model for graded sediment. The sediment transport models and the sediment transport formulae are briefly described below. For more detailed technical outlines of the sediment transport models and the sediment transport formula, refer to the “MOUSE Pollution Transport Reference” manual.

12.6.1 The Sediment Transport Models in MOUSE ST

The sediment transport models in MOUSE ST can be divided into two groups:

- explicit models,
- morphological models.

The main difference between the two model types is that in the explicit models the bed levels are fixed during the simulation. In the morphological models the bed levels / sediment deposits are updated dynamically during the simulation.

The Explicit Sediment Transport Models

In the explicit sediment transport models the bed level is fixed throughout the simulation and the only feed-back from the sediment transport computations to the hydrodynamics is established via the Manning number. The sediment transport capacity is calculated in time and space as an explicit function of the corresponding hydrodynamic flow parameters, i.e. the model calculates a potential sediment transport and no sediment is moved around in the system. The explicit models are in general used to:

- investigate the hydraulic capacity in pipes with sediment deposits;
- calculation of the sediment transport at locations where the bed levels changes are insignificant during a simulation.

In order to select an explicit sediment transport model, leave the checkbox "Apply Morphological Modeling” on the dialog 'MOUSE | Water Quality | ST Main Parameters' unchecked.

The Morphological Models

The morphological models in MOUSE TRAP ST are:

- morphological model for uniform sediment,
- morphological model for non-uniform sediment.

In the morphological models the sediment transport continuity equation is solved, based on the corresponding values of the hydrodynamic parameters (i.e. discharge, water levels, etc). The feedback to the hydrodynamic module is established through dynamically changed flow area and flow resistance number.

The morphological model for uniform sediment uses one grain diameter at each point in the sewer system to calculate the sediment transport. In the morphological model for non-uniform sediment it is possible to calculate the sediment transport from a range of grain diameters. For the transport of fine organic particles some of the non-uniform sediment fractions can be chosen to be transported by use of the Advection-Dispersion transport formulation instead of the non-cohesive sediment transport formula, as the transport of particles transported in suspension is better described by used of the advection-dispersion equation. The fractions transported by the advection-dispersion module are still included in the morphological calculation, but the description of erosion/deposition is changed.

Note, the morphological models require sediment or bed level boundary conditions at all inflow boundaries. In order to chose the morphological sediment transport model type, check the checkbox "Apply Morphological Modeling" on the dialog 'MOUSE | Water Quality | ST Main Parameters'.

12.6.2 The Transport Formulae - Short Description

The following four non-cohesive sediment transport formulae are implemented in MOUSE TRAP ST:

- Engelund - Hansen;
- Ackers - White;
- Engelund - Fredsøe - Deigaard;
- van Rijn.

Short descriptions of the sediment transport formulae are given below. No general guidelines can be given for the preference of one formulation over another, as the applicability of each depends on a number of factors.

All these formulae demonstrate that the sediment transport is a highly non-linear function of the flow velocity: depending on the formulation, the sediment transport is proportional to the velocity raised to the power from 3 to 5. Hence, the user should be very careful in the description of the hydrodynamics in the model setup.

The Ackers-White formulae

The formulae determine directly the total sediment transport. The formulae are semi-empirical, based partly on dimensional analysis and partly on physical arguments.

The Engelund-Hansen formula

The formula determines the total sediment transport directly. It has been derived from consideration of the work done by the flow on the sediment being transported. Originally, the formula was derived for a dune covered bed, but it was found applicable to the upper regimes (plane bed and anti-dunes) as well.

The Engelund-Fredsøe-Deigaard formulae

The formulae calculate the total transport as the sum of the bed load transport and the suspended transport. The sediment transport is calculated from the skin friction, i.e. the shear stress acting on the surface of the bed. In this formulae it is possible to describe the development of sand dunes in pipes and hence include the resulting friction into the computations. The total bed resistance is then calculated as the sum of a contribution from the skin friction acting on the dune and an expansion loss behind the dune.

The van Rijn formulae

In the van Rijn sediment transport formulae the sediment transport is divided into bed load and suspended load. The bed load is calculated from the saltation height, the particle velocity and the bed load concentration. The bed load computations follow the approach of Bagnold (1973), which assumes that the motion of the bed load particles is dominated by the gravity forces. When the bed shear velocity exceeds the fall velocity sediment is transported in suspension.

The suspended load is calculated as the depth integration of the local concentration and flow velocity. The method uses the reference concentration computed from the bed load transport. The formula has been verified for particles in the range 200 - 2000 mm. The verification based on 600 data sets, showed that 77% of the predicted bed load rates were within 0.5 and 2 times the observed values, van Rijn (1984a). The verification for the suspended load, using 800 data sets showed that 76% of the predicted values were within 0.5 and 2 times the observed values, van Rijn (1984b).

12.6.3 The Flow Resistance in Sewer Systems with Sediment Deposits

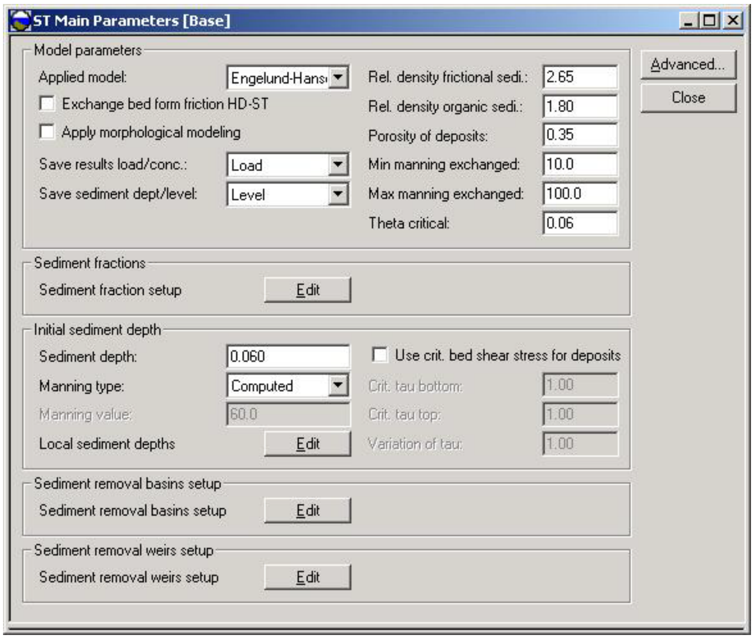
The hydraulic resistance in the sewer originates from the pipe wall and from the sediment deposits on the bottom of the sewer. The resistance from the sediment deposits consists of two contributions, one part originates from the grain friction and the other part originates from the expansion loss behind the bed forms. The dimensions of the bed forms are determined by the sediment transport and the flow, the resistance from the bed forms can be described through sediment transport formulae.

The average shear stress in a pipe with sediment deposits is calculated from the Einstein side-wall elimination procedure. The calculation is based on the pipe roughness and the bed shear stress calculated from the sediment deposits.

12.6.4 Sediment Transport Data Dialogs

The data dialogs under the Menu option "MOUSE|Water Quality" contain input parameters to the sediment transport model - MOUSE ST. The data are e.g. sediment grain diameter, relative density of the sediment, critical bed shear stress, type of sediment transport model, etc.

MOUSE | Water Quality | ST Main Parameters



Model parameters
Applied model: Engelund-Hans
☐ Exchange bed form friction HD-ST
☐ Apply morphological modeling
Save results load/conc.: Load
Save sediment dept/level: Level
Rel. density frictional sedi.: 2.65
Rel. density organic sedi.: 1.80
Porosity of deposits: 0.35
Min manning exchanged: 10.0
Max manning exchanged: 100.0
Theta critical: 0.06

Sediment fractions
Sediment fraction setup

Initial sediment depth
Sediment depth: 0.060
Manning type: Computed
Manning value: 60.0
Local sediment depths
☐ Use crit. bed shear stress for deposits
Crit. tau bottom: 1.00
Crit. tau top: 1.00
Variation of tau: 1.00

Sediment removal basins setup
Sediment removal basins setup

Sediment removal weirs setup
Sediment removal weirs setup

Figure 12.16 ST Main Parameters dialog

Selection of the sediment transport formulae and type of model takes place in the Data Dialog "MOUSE| Water Quality| ST Main Parameters".

In a case when it has been chosen to calculate the bed shear stress during the simulation, the updated values will be used in the hydraulic computations. This implies that the Manning number specified for the bed may vary during the simulation. A detailed description of how the shear stress is calculated is given in the “MOUSE Pollution Transport Reference” manual. If it has been chosen to update the bed level then a morphological model is run during the simulation. Hence, boundary conditions in terms of time series of sediment transport, bed levels and/or available fraction percentages, must be specified at every point where inflow occurs in the sewer network. If it has been chosen not to update the bed level, then an explicit sediment transport model is run during the simulation and boundary conditions for sediment transport are not required. The results of an explicit sediment transport calculation represent the potential sediment transport.

Additional parameters to the sediment transport modeling are also specified in this dialog. Default values are supplied automatically for all parameters, but as these parameters are very important for the calculation of the sediment transport, this menu must be edited before a sediment transport calculation can be carried out.

The various model parameters represent the following:

- Relative Density of the sediment, (density of the mineral sediment, relative to water)
- Porosity, (sediment porosity)
- Theta critical, (the critical Shields' parameter)
- Relative Density of fine sediment (AD), (density of the fine sediment containing organic material, relative to water)

The default values of computation parameters are typical values, but the actual values of viscosity, density and porosity should be substituted as appropriate. The parameter Theta critical should not be altered unless the user is familiar with the consequences. For a more detailed description of these parameters refer to the related equations in the “MOUSE Pollution Transport Reference” manual.

The initial depth of the sediment deposits, the bed shear stress caused by the sediment deposits and variation of the critical bed shear stress as a function of the initial sediment depth.

The depth of the initial sediment deposits should be specified relative to the bottom level of the pipe.

The bed shear stress from the sediment deposits can be specified in three ways:

- the bed shear stress is calculated from the local mean grain diameter according to:

$$M = \frac{25{,}4}{(2{,}5d)^{1/6}} \tag{12.3}$$

where d is the mean grain diameter,

- the bed shear stress is the Manning's M taken from the dialog.

If “Computed” is selected then the software computes an equivalent roughness from the sediment grain size.

The variation of the critical bed shear stress can be given as a function of the initial sediment depth as long as only erosion occurs. If deposition occurs, the critical bed shear stress is set equal to the Shields critical bed shear stress. This value of the critical bed shear stress will be used as long as the depth of the sediment deposit is larger than the maximum depth to which the initial sediment deposit has been eroded.

This can be summarized in the following expression:

If the actual sediment depth > maximum eroded sediment depth then

$$\tau_c = \tau_{c,\,Shields} \tag{12.4}$$

else

$$\tau_C = \tau_{bot} + (\tau_{top} - \tau_{bot}) \times \left(\frac{y_{sediment}}{y_{initialdepth}}\right)^{EXP} \tag{12.5}$$

where:

τ_{bot} = the critical bed shear stress at the bottom of the pipe,

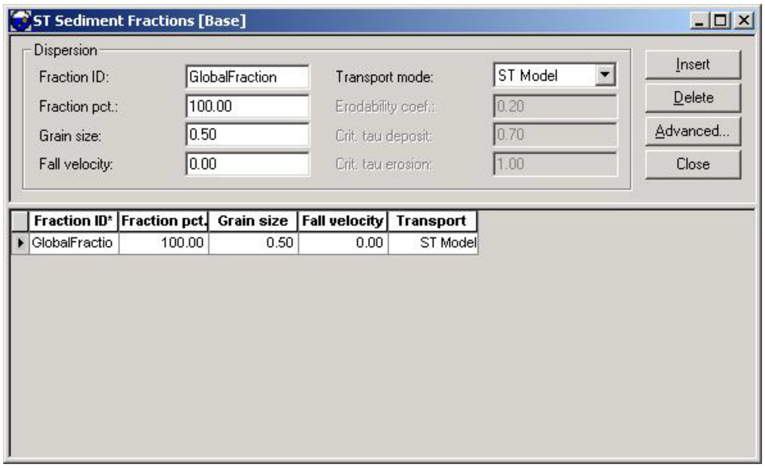
τ_{bot} = the critical bed shear stress at top of the initial sediment deposit,

$y_{sediment}$ = the actual depth of the sediment deposit,

$y_{initial\ depth}$ = the initial depth of the sediment deposit.

MOUSE| Water Quality | ST Sediment Fractions

The grain diameter may be specified for nodes in the network. In the links connecting two nodes with different specified grain diameters, the grain sizes are linearly interpolated.



Fraction ID: GlobalFraction
Fraction pct.: 100.00
Grain size: 0.50
Fall velocity: 0.00
Transport mode: ST Model
Erodability coef.: 0.20
Crit. tau deposit: 0.70
Crit. tau erosion: 1.00

Fraction ID*	Fraction pct.	Grain size	Fall velocity	Transport
GlobalFractio	100.00	0.50	0.00	ST Model

Figure 12.17 ST Sediment Fractions dialog

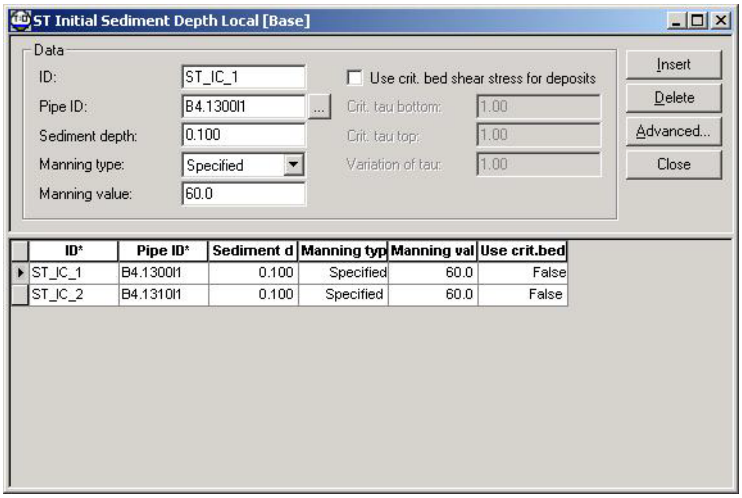
This dialog allows for the specification of the input data required for the simulation of graded sediment transport and sediment sorting.

The bed material is represented by two layers, an active layer overlying an inactive, passive layer. Each layer is divided into an equal number of fractions. A mean grain size for each fraction and the percentage distribution for both layers must be specified. The sum of the initial percentages for all the layers must equal 100 %. The model assumes that the initial division between active and passive layer is 50-50 %. If the fall velocity is specified to zero, then it will be calculated by the program. The equations used for the fall velocity calculation are given in the “MOUSE Pollution Transport Reference” manual.

The 'Transport Mode' combo-box allows for the choice whether the sediment fraction is transported with ordinary non-cohesive sediment transport formulations, or the fraction is transported by the Advection-Dispersion formulations. The transport of fine organic sediment fractions is better described by use of an Advection-Dispersion transport formulation instead of the non-cohesive sediment transport formulae. If it has been selected to transport the fraction with the Advection-Dispersion model then the critical shear stress for deposition τ_{cd}, the critical shear stress for erosion τ_{ce} and the erodibility coefficient must be specified. The morphological changes resulting from the sediments transported by the Advection-Dispersion formulations are still included in the calculation of the bed level, but the changes in bed levels are now calculated from source/sink terms. For further information, refer to the “MOUSE Pollution Transport Reference” manual.

MOUSE | Water Quality | ST Initial Sediment Depth Local

The initial sediment depth can be specified either locally or globally, globally it is done on “MOUSE| Water Quality| Main Parameters”.



ID: ST_IC_1

Pipe ID: B4.1300l1

Sediment depth: 0.100

Manning type: Specified

Manning value: 60.0

☐ Use crit. bed shear stress for deposits

Crit. tau bottom: 1.00

Crit. tau top: 1.00

Variation of tau: 1.00

ID*	Pipe ID*	Sediment d	Manning typ	Manning val	Use crit.bed
ST_IC_1	B4.1300l1	0.100	Specified	60.0	False
ST_IC_2	B4.1310l1	0.100	Specified	60.0	False

Figure 12.18 ST Initial Sediment Depth Local

MOUSE| Water Quality| ST Sediment Removal Basins

Sediments can be removed from nodes (tanks, basins) according to the formula:

$$E = 1 - \left(1 + \frac{W_s}{nx(Q/A)}\right)^{-n} \tag{12.6}$$

where:

E = efficiency

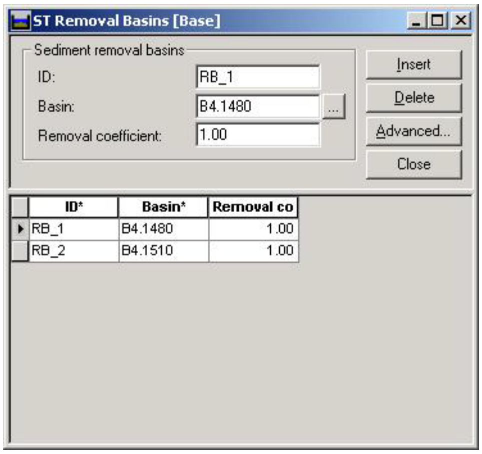
W_s = setting velocity

n = removal coefficient

Q = flow into basin

A = surface area of the basin

The calculated efficiency multiplied with the inflowing sediment transport gives the amount of the sediment retained in the structure. There is no updating of the bed level in the structure.


ST Removal Basins [Base]
Sediment removal basins
...
Insert
Delete
Advanced...
Close

ID: RB_1
Basin: B4.1480
Removal coefficient: 1.00

ID*	Basin*	Removal co
RB_1	B4.1480	1.00
RB_2	B4.1510	1.00

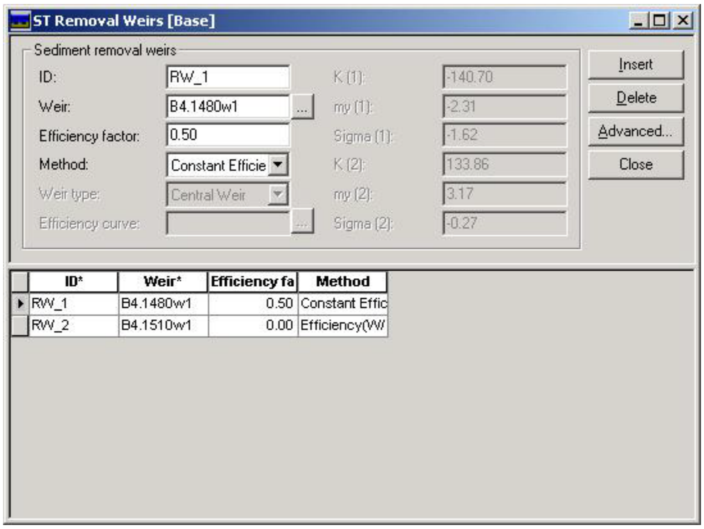
Figure 12.19 ST Removal Basins dialog

MOUSE | Water Quality| ST Sediment Removal Weirs

There are three methods available for removal of sediments in weirs.

- Constant efficiency factor.
- User-specified relation between efficiency and discharge towards the weir.
- Efficiency as a function of fall velocity and velocity of the approaching flow.

Depending on which type of weirs is selected (central weir or vortex separator), different default values of constants which are used in a reversed normal distribution curve will be given.



Sediment removal weirs

ID: RW_1
Weir: B4.1480w1
Efficiency factor: 0.50
Method: Constant Efficie
Weir type: Central Weir
Efficiency curve:
K (1): -140.70
my (1): -2.31
Sigma (1): -1.62
K (2): 133.86
my (2): 3.17
Sigma (2): -0.27

ID*	Weir*	Efficiency fa	Method
RW_1	B4.1480w1	0.50	Constant Effic
RW_2	B4.1510w1	0.00	Efficiency(W/

Figure 12.20 ST Removal Weirs dialog

12.6.5 Boundary Conditions for the Sediment Transport Model

Boundary conditions for the ST module can be specified via sediment transport results from a surface calculation (.SMR-file) and/or via the boundary connection system. It is possible to connect sediment loads or concentrations to nodes in this system either as time series, constant values or cyclic patterns. Boundary conditions must be specified at all inflow points in the sediment transport model. Hence, if a water level at an outlet results in inflow to the model, a boundary condition must be specified at this location for the morphological model. An appropriate boundary condition at such a location will often be to specify the sediment transport equal to zero.

Computation with Sediment Transport model

A ST simulation is started from the MOUSE Pipe Flow Computation Dialog. Both the explicit and the morphological sediment transport model run in parallel with the hydrodynamic model.

The main limitations to the time step in the sediment transport calculation are the boundary conditions. Both the hydrodynamic and the morphological boundary conditions should be well resolved. Finally the calculated changes in the bed level should be sufficiently small in order not to generate instabilities in the hydrodynamic calculations.

Notice that additional functionality for the ST module may be available through input data in the DHIAPP.ini or additional parameter file (*.adp).

12.7 Storm Water Quality

The MOUSE Stormwater Quality modelling features provide possibilities in conjunction with MOUSE AD for modelling transport of pollutants from surface water and through the pipe network. The module adds functionality to MOUSE focusing on wash off and transport of suspended sediments, heavy metals and dissolved pollutants from surface water and infiltration. Additionally, the module allows modeling of "Treatment nodes". These are typically stormwater ponds, small wetlands, infiltration and retention basins.

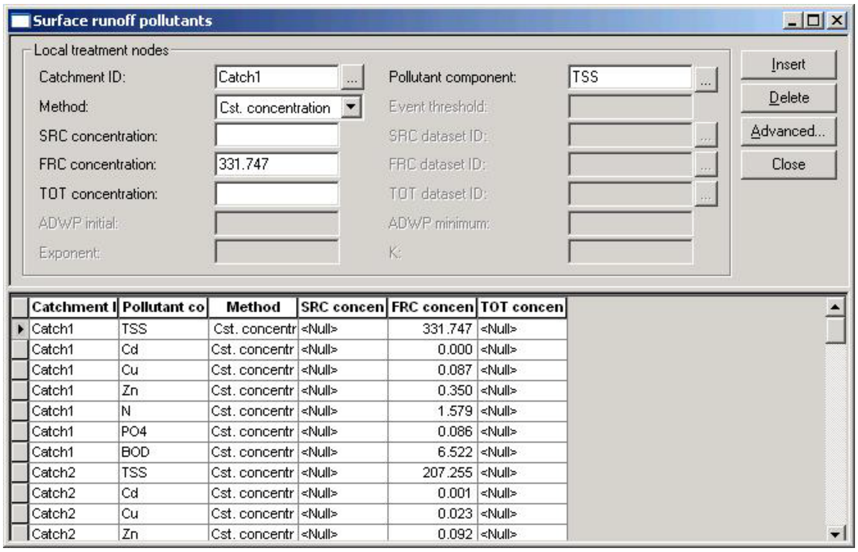
Although the module is aimed at stormwater applications it can easily be applied for combined sewer systems. For combined systems dry weather concentrations can be described. This enables modelling of spatially and time varying distribution of pollutants concentrations in domestic sewage.

12.7.1 Assigning concentrations of pollutants to runoff and infiltrations

It is possible to assign variable concentrations to the runoff and infiltration for each pollutant component specified in the MU CS Pollution Transport module.

The functionality is aimed at modelling the pollutant loads from runoff and infiltration water and is implemented in all the runoff models of the MOUSE engine.

The concentrations of a given pollutant can be given for each specific catchment. For one catchment it is possible to give a concentration on the slow runoff component (SRC) and the fast runoff component (FRC) or override these settings stating a concentration for the total runoff (TOT). Instead of fixed concentrations it is possible to state a name of a tabular data set. The tabular data set should then hold a table with concentrations versus normalised flow, i.e. the runoff divided by the total catchment area.

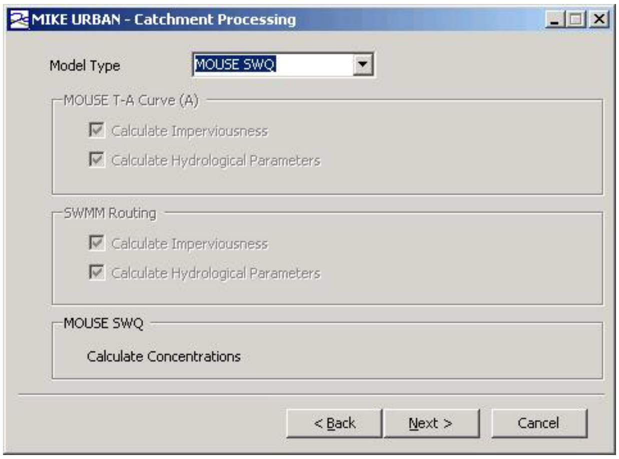

Surface runoff pollutants
Local treatment nodes
Insert
Delete
Advanced...
Close

Catchment ID: Catch1
Method: Cst. concentration
SRC concentration:
FRC concentration: 331.747
TOT concentration:
ADWP initial:
Exponent:
Pollutant component: TSS
Event threshold:
SRC dataset ID:
FRC dataset ID:
TOT dataset ID:
ADWP minimum:
K:

Catchment I	Pollutant co	Method	SRC concen	FRC concen	TOT concen
Catch1	TSS	Cst. concentr	<Null>	331.747	<Null>
Catch1	Cd	Cst. concentr	<Null>	0.000	<Null>
Catch1	Cu	Cst. concentr	<Null>	0.087	<Null>
Catch1	Zn	Cst. concentr	<Null>	0.350	<Null>
Catch1	N	Cst. concentr	<Null>	1.579	<Null>
Catch1	PO4	Cst. concentr	<Null>	0.086	<Null>
Catch1	BOD	Cst. concentr	<Null>	6.522	<Null>
Catch2	TSS	Cst. concentr	<Null>	207.255	<Null>
Catch2	Cd	Cst. concentr	<Null>	0.001	<Null>
Catch2	Cu	Cst. concentr	<Null>	0.023	<Null>
Catch2	Zn	Cst. concentr	<Null>	0.092	<Null>

Figure 12.21 Assigning surface runoff pollutants

The concentrations given as input data described above come from a consideration of pollutant loads and land use types for the various catchments. These can be difficult to assess. The requirements for a GIS analysis, using MIKE URBAN, are thematic maps of different land-use types such as buildings, traffic area, industrial area and areas with dense traffic.The catchment processing tool, MOUSE|Catchment Tools|Catchment Processing, can be used to assign concentrations based on these thematic maps.


MIKE URBAN - Catchment Processing
MOUSE T-A Curve (A)
SWMM Routing
MOUSE SWQ
< Back
Next >
Cancel

Model Type: MOUSE SWQ

MOUSE T-A Curve (A)
☑ Calculate Imperviousness
☑ Calculate Hydrological Parameters

SWMM Routing
☑ Calculate Imperviousness
☑ Calculate Hydrological Parameters

MOUSE SWQ
Calculate Concentrations

Figure 12.22 Using the catchment processing tool for assigning concentrations

Concentrations for different pollutants are assigned to the various land-use types and are combined with the sub-catchments. Typical pollutant concentrations are provided in Table 12.6.

Table 12.6 Concentration for various pollutants

Pollutant	Normal conc.	Low conc.	High conc.	Baseflow conc.	Unit
TSS	100	30	400	5	mg/l
Cd	0.5	0.15	1.5	0.005	µg/l
Cu	30	15	100	5	µg/l
Zn	130	50	400	10	µg/l
Tot-N	1.8	0.6	3.5	0.2	mg/l
PO4	100	25	300	0.01	µg/l
BOD	0.075*TSS	0.075*TSS	0.075*TSS	0.075*TSS	mg/l

Table 12.7 The edit fields for the surface runoff pollutants dialog

Edit field	Description
Catchment ID	The ID of the catchment. It is recommeended to add pollutants only to catchments with a surface runoff areas
Pollutant component	The ID name of the pollutant component in the AD model
Method	Method = Cst concentration (method 1), Table concentration (method 2), EMC formula (method 3) For all methods values in the fields for total concentrations (TOT) overrides settings specified for SRC and FRC.
SRC concentration	Constant concentration value for the slow runoff component. Only used in RDI models.
FRC concentration	Constant concentration value for the fast runoff component. This is also the component used in the model A, B, C and UHM.
TOT concentration	Constant concentration value for the total runoff component. Can be used in all models.
SRC dataset ID	Only applied with method 2. ID identifier for the tabular dataset applied to the slow runoff component.
FRC dataset ID	Only applied with method 2. ID identifier for the tabular data set applied to the fast runoff component

Table 12.7 The edit fields for the surface runoff pollutants dialog

Edit field	Description
TOT dataset ID	Only applied with method 2. ID identifier for the tabular data set applied to the total runoff component
K	Only applied with method 3. K factor used in the formula.
Event threshold	Threshold that defines when a runoff event starts and stops. Events with flows less than the treshold are seen as dry weather when calculating the antcendent dry weather period (ADWP)
ADWP initial	The initial ADWP used before the first runoff event is detected in the data.
ADWP minimum	If the duration between two events is smaller than the ADWP minimum then the two events are perceived as only one event.
Exponent	Only applied with method 3. Exponent used in the formula.

Cst. concentration (method 1)

The specified constant values of concentrations are assigned to the runoff and infiltration water.

Table concentration (method 2)

The concentration is based on a lookup in a set of tabular data given in the MOUSE menu system for tabular data, MOUSE|Curves & Relations. The runoff discharge divided with the total catchment area is the value from which the concentration is determined, i.e. c(t) = f(Q(t)/Atot). Values are interpolated linearly between datasets. The tabular data editor in MOUSE is missing the Conc. vs. Q/A datatype. Data should be entered as an “Undefined” table type. The first column should hold flow/total catchment area values and the second column, concentrations.

Please verify the columns for Q/A and concentration.

EMC formula (method 3)

The event mean concentration (EMC) is computed based on the following formula.

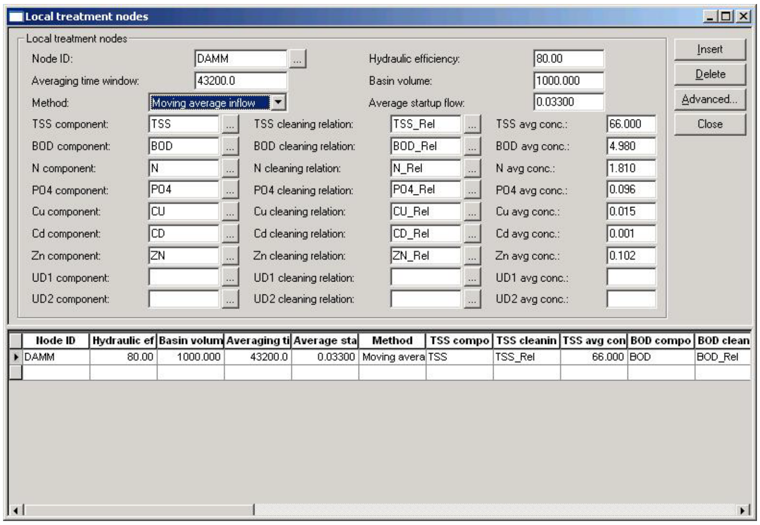
$$EMC = SMC \cdot K \cdot ADWP^{exponent} \tag{12.7}$$

As site mean concentration (SMC) is taken the specified constant concentration value (FRC Concentration). The K-factor and the "exponent" is given as input data and the antecedent dry weather period (ADWP) is determined from the runoff from the given catchment. The method only applies for the FRC runoff component.

12.7.2 Local treatment nodes

Through specification of local treatment node it is possible to apply a removal efficiency to nodes in the model. This enables modelling of stormwater ponds, small wetlands, infiltration and retention basins.

Based on the layout of the basin or pond, the hydraulic load and the type of pollutant the removal efficiency is computed and the corresponding amount of pollutant is removed during the simulation with the MIKE URBAN Collection System Pollutant Transport module.



Node ID: DAMM
Averaging time window: 43200.0
Method: Moving average inflow
Hydraulic efficiency: 80.00
Basin volume: 1000.000
Average startup flow: 0.03300
TSS component: TSS
TSS cleaning relation: TSS_Rel
TSS avg conc.: 66.000
BOD component: BOD
BOD cleaning relation: BOD_Rel
BOD avg conc.: 4.980
N component: N
N cleaning relation: N_Rel
N avg conc.: 1.810
PO4 component: PO4
PO4 cleaning relation: PO4_Rel
PO4 avg conc.: 0.096
Cu component: CU
Cu cleaning relation: CU_Rel
Cu avg conc.: 0.015
Cd component: CD
Cd cleaning relation: CD_Rel
Cd avg conc.: 0.001
Zn component: ZN
Zn cleaning relation: ZN_Rel
Zn avg conc.: 0.102
UD1 component:
UD1 cleaning relation:
UD1 avg conc.:
UD2 component:
UD2 cleaning relation:
UD2 avg conc.:

Node ID	Hydraulic ef	Basin volum	Averaging ti	Average sta	Method	TSS compo	TSS cleanin	TSS avg con	BOD compo	BOD clean
DAMM	80.00	1000.000	43200.0	0.03300	Moving avera	TSS	TSS_Rel	66.000	BOD	BOD_Rel

Figure 12.23 Local treatment nodes

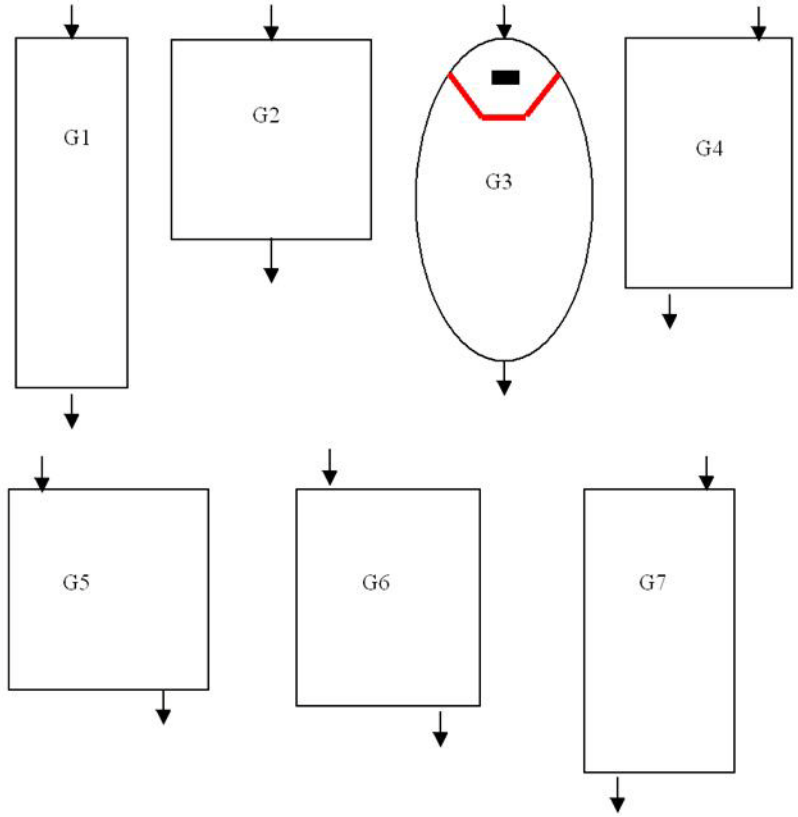


Table 12.8 The edit fields for local treatment nodes

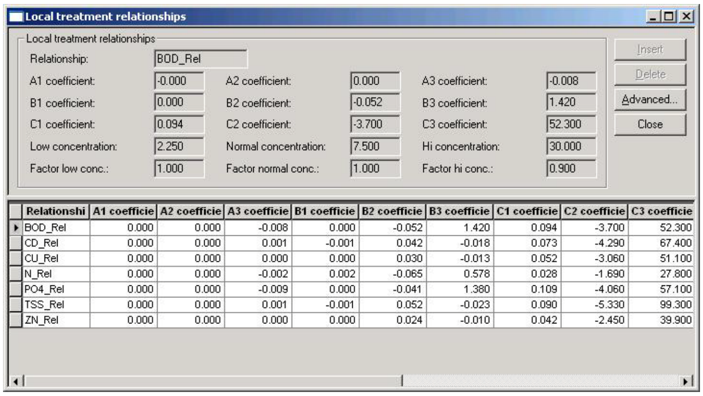
Edit field	Description
Node ID	The node ID that acts as a local treatment node. Typically basins, but not required.
Hydraulic efficiency	The hydraulic efficiency is determined on basis of the shape and layout of the pond. The following figures and table shows a number of examples that can help to determine the hydraulic coefficient. When decided on the basin type that resembles the one that needs modelling the hydraulic efficiency can be interpreted from the table below. The values have been obtained from numerical simulations with the 2D model MIKE 21, which is recommended if high accuracy is of importance.
Basin volume	Basin volume used for the computation. If a negative value, e.g. -1 is stated, then the volume is computed from the geometry given.

Table 12.8 The edit fields for local treatment nodes

Edit field	Description
Average time window	In connection with the computation of the removal efficiency the average inflow to the basin is used. The average is computed as a moving average with a time window equal to the average time window. As default this is set to 43200 minutes (one month).
Method	The volume load is defined as the average flow divided with the basin volume. If Average Method is 1 then the volume load is computed as the initial AvgFlow parameter divided by BasinVol, i.e. no moving average is computed. If AvgMethod is 2 then it is the moving average that is divided by BasinVol
Average start up flow	Until the average window is filled up with real model data the average start up is used as an initial value.
xxx component	ID identifier of the attaining pollutant component in MOUSE AD.
xxx cleaning relation	The final removal efficiency is computed based on a polynomium. The nine coefficients for this are given in the local treatment coefficients.
xxx average concentration	The computed removal efficiency is adjusted based on the level of the incoming average concentration to the node. The average concentration is computed the same way as the average flow and is using the same average time and average method.

Figure 12.24 Various ponds and basins

The treatment node relations are used from the treatment relationships, see Figure 12.25.



Local treatment relationships

Relationship: BOD_Rel

A1 coefficient: -0.000
A2 coefficient: 0.000
A3 coefficient: -0.008
B1 coefficient: 0.000
B2 coefficient: -0.052
B3 coefficient: 1.420
C1 coefficient: 0.094
C2 coefficient: -3.700
C3 coefficient: 52.300
Low concentration: 2.250
Normal concentration: 7.500
Hi concentration: 30.000
Factor low conc.: 1.000
Factor normal conc.: 1.000
Factor hi conc.: 0.900

Relationshi	A1 coefficie	A2 coefficie	A3 coefficie	B1 coefficie	B2 coefficie	B3 coefficie	C1 coefficie	C2 coefficie	C3 coefficie
BOD_Rel	0.000	0.000	-0.008	0.000	-0.052	1.420	0.094	-3.700	52.300
CD_Rel	0.000	0.000	0.001	-0.001	0.042	-0.018	0.073	-4.290	67.400
CU_Rel	0.000	0.000	0.000	0.000	0.030	-0.013	0.052	-3.060	51.100
N_Rel	0.000	0.000	-0.002	0.002	-0.065	0.578	0.028	-1.690	27.800
PO4_Rel	0.000	0.000	-0.009	0.000	-0.041	1.380	0.109	-4.060	57.100
TSS_Rel	0.000	0.000	0.001	-0.001	0.052	-0.023	0.090	-5.330	99.300
ZN_Rel	0.000	0.000	0.000	0.000	0.024	-0.010	0.042	-2.450	39.900

Figure 12.25 Local treatment relationship

Table 12.9 Proportions of the basins or ponds

Type	Length [m]	Width [m]	Depth [m]
G1	300	60	1
G2	132	132	0.5, 1 and 2
G3	200	114	1
G4	188	96	0.5, 1 and 2
G5	132	132	1
G6	144	124	1
G7	236	76	1

Table 12.10 Edit fields for local treatment relationships

Edit field	
Relationship	ID for the relation
A1-3, B1-3, C1-3	Coefficients used in the polynomium
Low concentration	Typical low concentration of incoming pollutants to the treatment node
High concentration	Typical high concentration of incoming pollutants to the treatment node
Normal concentration	Typical normal concentration of incoming pollutants to the treatment node
Factor low conc	Correction factor for changing the computed removal efficiency over the time period of average time is equal to the low concentration

Table 12.10 Edit fields for local treatment relationships

Edit field	
Facor normal conc	Correction factor for changing the computed removal efficiency over the time period of average time is equal to the normal concentration
Factor hi conc	Correction factor for changing the computed removal efficiency over the time period of average time is equal to the high concentration

For average concentrations which are somewhere in between the sets of data given for low, normal and high, the correction factor is linearly interpolated. Typical correction factors are given in below table.

Table 12.11 Typical correction factors

Pollutant	TSS	BOD	Ntot	PO4	Cu	Cd	Zn
Low conc	0.7	1.0	2.2	1.1	0.6	0.5	0.6
Normal conc	1.0	1.0	1.0	1.0	1.0	1.0	1.0
High conc	1.1	0.9	0.4	0.6	1.6	1.4	1.7

The polynomium for calculating the removal efficiency in % is stated as

$$R = (A_1 h^2 + B_1 H + C_1) \cdot q^2 + (A_2 h^2 + B_2 H + C_2) \cdot q^2 + (A_3 h^2 + B_3 H + C_3) \cdot q^2 \tag{12.8}$$

where

h: hydraulic efficiency (%)

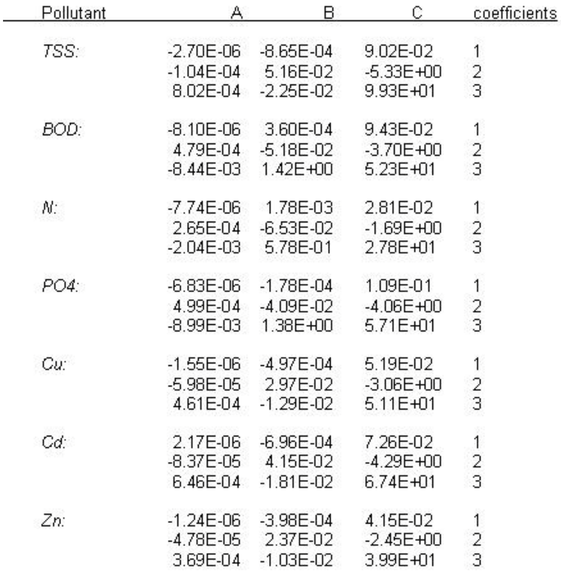
q: volume load 0.001 $(m^3/h)m^3$

Table 12.12 Hydraulic efficiency corresponding to basin G1-G7

Type	Hydraulic efficiency
G1	0.93
G2	0.3
G3	0.96
G4	0.58
G5	0.52
G6	0.39
G7	0.69

Table 12.12 Hydraulic efficiency corresponding to basin G1-G7

Type	Hydraulic efficiency
G2, 2 m depth	0.17
G2, 0.5 m depth	0.37
G4, 2 m depth	0.34
G4, 0.5 m depth	0.7



Pollutant	A	B	C	coefficients
TSS:	-2.70E-06	-8.65E-04	9.02E-02	1
	-1.04E-04	5.16E-02	-5.33E+00	2
	8.02E-04	-2.25E-02	9.93E+01	3
BOD:	-8.10E-06	3.60E-04	9.43E-02	1
	4.79E-04	-5.18E-02	-3.70E+00	2
	-8.44E-03	1.42E+00	5.23E+01	3
N:	-7.74E-06	1.78E-03	2.81E-02	1
	2.65E-04	-6.53E-02	-1.69E+00	2
	-2.04E-03	5.78E-01	2.78E+01	3
PO4:	-6.83E-06	-1.78E-04	1.09E-01	1
	4.99E-04	-4.09E-02	-4.06E+00	2
	-8.99E-03	1.38E+00	5.71E+01	3
Cu:	-1.55E-06	-4.97E-04	5.19E-02	1
	-5.98E-05	2.97E-02	-3.06E+00	2
	4.61E-04	-1.29E-02	5.11E+01	3
Cd:	2.17E-06	-6.96E-04	7.26E-02	1
	-8.37E-05	4.15E-02	-4.29E+00	2
	6.46E-04	-1.81E-02	6.74E+01	3
Zn:	-1.24E-06	-3.98E-04	4.15E-02	1
	-4.78E-05	2.37E-02	-2.45E+00	2
	3.69E-04	-1.03E-02	3.99E+01	3

Figure 12.26 The coefficients for a number of pollutants have been determined through MIKE 21 simulations

In order to avoid unrealistic results a number of bounds have been set:

- Removal efficiency must be between 1 and 95%
- Volume load must be between 2.5 e-3 and 30 e-3 ([m3/ h]/m3)
- Hydraulic efficiency must be between 30% and 90%

To sum up the methodology, for each treatment node and for each pollutant:

1. The hydraulic efficiency is determined based on the size and shape of the basin.
2. The volume load is determined as average flow divide with the basin volume.
3. The removal efficiency is calculated based on the volume load, hydraulic efficiency and the pollutant relation given by the nine coefficients.

4. The average pollutant concentration is determined.
5. The removal efficiency is computed and adjusted with the correction factor.
6. The removed load for the current time step is computed and subtracted in the MU CS Pollution Transport module.

MIKE URBAN CS - SWMM

User Guide

13 Hydraulic Network Modelling with SWMM

13.1 Terms and Concept

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.

This Chapter provides a comprehensive guideline for the preparation of the basic SWMM hydrodynamic simulation models. Modelling of network hydrodynamics in SWMM requires understanding of the information requirements. On the other hand, detailed knowledge of the computational theory is not essential.

The modelling process consists of the following distinct steps:

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

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

A SWMM network within MIKE URBAN can be defined in one of the following ways. Please note that MIKE URBAN supports the EPA SWMM5 format (if you have a SWMM44 model you can locate conversion tools on the US EPA website).

- Import of external data (e.g. GIS) into MIKE URBAN CS SWMM network
- Copying network data from MIKE URBAN CS Asset network into MIKE URBAN CS SWMM network
- Copying network data from MIKE URBAN CS MOUSE network into MIKE URBAN CS SWMM network
- Graphical data digitalisation and manual data typing within MIKE URBAN

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

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

A SWMM model consists of the following elements:

- Nodes
- Conduits
- Orifices
- Pumps
- Weirs
- Outlets

13.2 Nodes

Nodes are hydraulic elements such as **Junctions**, **Outfalls**, **Storage Units** and **Flow Dividers** used in modelling a CS SWMM storm and sewer collection system.

Junctions are defined as conveyance system nodes where links join together. Physically they can represent the confluence of natural surface channels, manholes in a sewer system, or pipe connection fittings. External inflows can enter the system at junctions. Excess water at a junction can become partially pressurized during surcharge events and can either be lost from the system or be allowed to pond atop the junction and subsequently drain back through it.

Outfalls are defined as terminal nodes of the conveyance system and used to define final downstream boundaries under Dynamic Wave flow routing. For other types of flow routing they behave as a junction. Only one link can be incident on an outfall node.

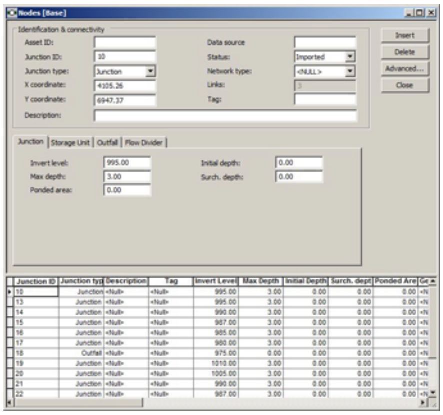
Storage Units are defined as conveyance system nodes that provide storage volume. Physically they could represent something as small as a catchbasin or as large as a lake. They are allowed to have any shape as described by a function or table of surface area versus height.

Flow Dividers are defined as conveyance system nodes used to split the total flow between exactly to two outflow conduits in a prescribed manner. Flow dividers are only active under Uniform or Kinematic Wave routing and are treated as simple junctions under Dynamic Wave routing.

The CS SWMM Nodes editor organizes the related input data for the hydraulic elements into the following groups:

Identification and Connectivity - General identification and location information

- Geometrical and Hydraulic Properties - Model related data
- Junction - Geometry information
- Storage Unit - Geometry information
- Outfall - Outfall information
- Flow Divider - Flow Divider information



Asset ID:
Junction ID: 10
Junction type: Junction
X coordinate: 4105.26
Y coordinate: 6947.37
Description:
Data source
Status: Imported
Network type: <NULL>
Links: 3
Tag:

Invert level: 995.00
Max depth: 3.00
Ponded area: 0.00
Initial depth: 0.00
Surch. depth: 0.00

Junction ID	Junction typ	Description	Tag	Invert Level	Max Depth	Initial Depth	Surch. dept	Ponded Are	Ge
10	Junction	<Null>	<Null>	995.00	3.00	0.00	0.00	0.00	<N
13	Junction	<Null>	<Null>	995.00	3.00	0.00	0.00	0.00	<N
14	Junction	<Null>	<Null>	990.00	3.00	0.00	0.00	0.00	<N
15	Junction	<Null>	<Null>	987.00	3.00	0.00	0.00	0.00	<N
16	Junction	<Null>	<Null>	985.00	3.00	0.00	0.00	0.00	<N
17	Junction	<Null>	<Null>	980.00	3.00	0.00	0.00	0.00	<N
18	Outfall	<Null>	<Null>	975.00	0.00	0.00	0.00	0.00	<N
19	Junction	<Null>	<Null>	1010.00	3.00	0.00	0.00	0.00	<N
20	Junction	<Null>	<Null>	1005.00	3.00	0.00	0.00	0.00	<N
21	Junction	<Null>	<Null>	990.00	3.00	0.00	0.00	0.00	<N
22	Junction	<Null>	<Null>	987.00	3.00	0.00	0.00	0.00	<N

Figure 13.1 The CS SWMM Nodes Editor

Identification and Connectivity

Table 13.1 The Nodes Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Asset ID	Id in the asset network	No	AssetName
Junction ID	Id of the node	Yes	MUID
Tag	Optional label used to identify or catagorize the junction	No	Tag
X Coordinate	X coordinate	Yes	
Y Coordinate	Y coordinate	Yes	
Data source	Reference to an external data source from which the record has been imported from	No	DataSource
Status	Status according to user specified list	No	Element_S

Table 13.1 The Nodes Identification and Connectivity Group

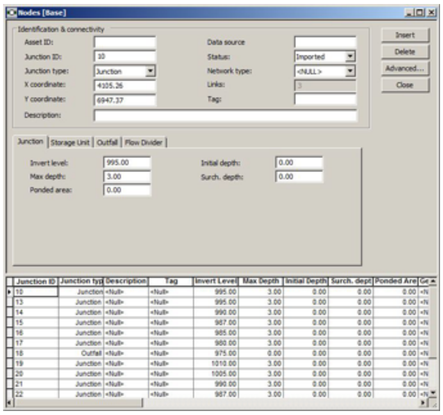
Edit field	Description	Used or required by simulations	Field name in datastructure
Network type	Network type describes the type of network i.e Stormwater, Combined or separate. The list of network types can be extended by user. Network type can be specified for each hydraulic element.	No	NetTypeNo
Links		No	
Description	User's descriptive information related to the node	No	Description

Geometrical and Hydraulic Properties

Table 13.2 The Nodes Geometrical and Hydraulic Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Junction Type	Definition of node as either Junction, Storage Unit, Outfall or Flow Divider	Yes	TypeNo

Junction



Asset ID:
Junction ID: 10
Junction type: Junction
X coordinate: 4105.26
Y coordinate: 6947.37
Description:
Data source
Status: Imported
Network type: <NULL>
Links: 3
Tag:

Invert level: 995.00
Max depth: 3.00
Ponded area: 0.00
Initial depth: 0.00
Surch. depth: 0.00

Junction ID	Junction typ	Description	Tag	Invert Level	Max Depth	Initial Depth	Surch. dept	Ponded Are	Ge
10	Junction	<Null>	<Null>	995.00	3.00	0.00	0.00	0.00	<N
13	Junction	<Null>	<Null>	995.00	3.00	0.00	0.00	0.00	<N
14	Junction	<Null>	<Null>	990.00	3.00	0.00	0.00	0.00	<N
15	Junction	<Null>	<Null>	987.00	3.00	0.00	0.00	0.00	<N
16	Junction	<Null>	<Null>	985.00	3.00	0.00	0.00	0.00	<N
17	Junction	<Null>	<Null>	980.00	3.00	0.00	0.00	0.00	<N
18	Outfall	<Null>	<Null>	975.00	0.00	0.00	0.00	0.00	<N
19	Junction	<Null>	<Null>	1010.00	3.00	0.00	0.00	0.00	<N
20	Junction	<Null>	<Null>	1005.00	3.00	0.00	0.00	0.00	<N
21	Junction	<Null>	<Null>	990.00	3.00	0.00	0.00	0.00	<N
22	Junction	<Null>	<Null>	987.00	3.00	0.00	0.00	0.00	<N

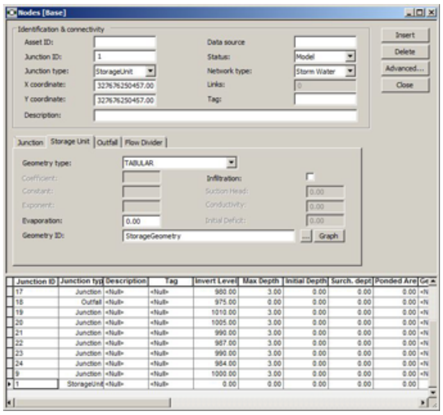
Figure 13.2 The CS SWMM Nodes - Junction Editor

Table 13.3 The Nodes Junction

Edit field	Description	Used or required by simulations	Field name in datastructure
Ponded Area	Area occupied by ponded water atop the junction after flooding occurs. If the Allow Ponding analysis option is turned on, a non-zero value of this parameter will allow ponded water to be stored and subsequently returned to the conveyance system when capacity exists. Default is, allow ponding is not turned on.	No	Apond
Invert Level	Invert elevation of the node	Yes	Einv
Max Depth	Maximum depth of node (i.e. from invert to ground surface)	Yes	Dmax
Inital Depth	Initial depth of water at the node at the start of the simulation	No	D0
Surch. Depth	Additional depth of water beyond the maximum depth that is allowed before the junction floods. This parameter can be used to model Bolted / Sealed manholes	No	Dsur

Storage Unit

The geometry of a Storage Unit can be defined by a Functional geometry or a Tabular geometry description. As an example a Storage Unit whose cross-sectional area remains constant with depth (e.g. a cylinder or cube) can be described by using a FUNCTIONAL geometry type whose coefficient equals the cross-sectional area and whose exponent is 0.



Asset ID:
Junction ID: 1
Junction type: StorageUnit
X coordinate: 327676250457.00
Y coordinate: 327676250457.00
Description:
Data source
Status: Model
Network type: Storm Water
Links: 0
Tag:

Geometry type: TABULAR
Coefficient:
Constant:
Exponent:
Evaporation: 0.00
Geometry ID: StorageGeometry
Infiltration: ☐
Suction Head: 0.00
Conductivity: 0.00
Initial Deficit: 0.00

Junction ID	Junction typ	Description	Tag	Invert Level	Max Depth	Initial Depth	Surch. dept	Ponded Are	Ge
17	Junction	<Null>	<Null>	980.00	3.00	0.00	0.00	0.00	<N
18	Outfall	<Null>	<Null>	975.00	0.00	0.00	0.00	0.00	<N
19	Junction	<Null>	<Null>	1010.00	3.00	0.00	0.00	0.00	<N
20	Junction	<Null>	<Null>	1005.00	3.00	0.00	0.00	0.00	<N
21	Junction	<Null>	<Null>	990.00	3.00	0.00	0.00	0.00	<N
22	Junction	<Null>	<Null>	987.00	3.00	0.00	0.00	0.00	<N
23	Junction	<Null>	<Null>	990.00	3.00	0.00	0.00	0.00	<N
24	Junction	<Null>	<Null>	984.00	3.00	0.00	0.00	0.00	<N
9	Junction	<Null>	<Null>	1000.00	3.00	0.00	0.00	0.00	<N
1	StorageUn	<Null>	<Null>	0.00	0.00	0.00	0.00	0.00	

Figure 13.3 The CS SWMM Nodes - Storage Unit Editor

Table 13.4 The Nodes Storage Unit

Edit field	Description	Used or required by simulations	Field name in datastructure
Geometry Type	Definition of the type of geometry description of the Storage Unit (**Functional** or **Tabular**)	Yes	GeomTypeNo
Coefficient	Coefficient for the **Functional** relationship between storage depth and surface area	Yes, if Functional is chosen	GeomCoeff
Exponent	Exponent for the **Functional** relationship between storage depth and surface area	Yes, if Functional is chosen	GeomExponent
Constant	Constant value for the **Functional** relationship between storage depth and surface area	Yes, if Functional is chosen	GeomConst
Evaporation	Fraction of potential evaporation realized	No	Fevap
Geometry ID	ID of the **Tabular** geometrical description. The geometry is specified relative to the invert level. This ensures reusability of the geometry for different Storage Units with different inverts	Yes if Tabular is chosen	GeomID
Infiltration	Option to apply optional Green-Ampt infiltration parameters so that the storage unit can serve as an infiltration basin	Yes	StorageInfiltrationNo

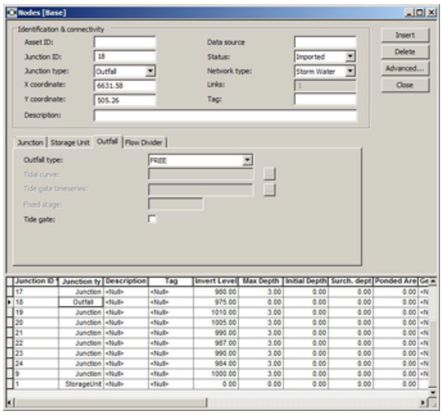
Table 13.4 The Nodes Storage Unit

Edit field	Description	Used or required by simulations	Field name in datastructure
Suction Head	Average value of soil capillary suction along wetting front	Yes, if Infiltration is chosen	StorageSuctionHead
Conductivity	Soil saturated hydraulic conductivity	Yes, if Infiltration is chosen	StorageConductivity
Initial Deficit	Fraction of soil volume that is initially dry	Yes, if Infiltration is chosen	StorageInitialDeficit

Outfall

Definition of an outfall in an urban drainage network is optional for uniform flow (UF) and Kinematic Wave (KW) routing. However when applying Dynamic Wave routing at least one outfall in the urban drainage system must be defined. It is possible to select between five types of outfalls, **FREE**, **NORMAL**, **FIXED**, **TIDAL** and **TIMESERIES**.

- FREE outfall bases the outfall stage on the smaller of the critical and normal depths of flow in the connecting conduit.
- NORMAL outfall uses just the normal depth of flow.
- FIXED outfall uses a constant water elevation at all times.
- TIDAL outfall has outfall stage varying in a repeating fashion over a tidal period.
- TIMESERIES outfall allows the outfall stage varying in any specified manner over time.



Asset ID:
Junction ID: 18
Junction type: Outfall
X coordinate: 6631.58
Y coordinate: 505.26
Description:
Data source
Status: Imported
Network type: Storm Water
Links: 1
Tag:

Outfall type: FREE
Tidal curve:
Tide gate timeseries:
Fixed stage:
Tide gate: ☐

Junction ID	Junction ty	Description	Tag	Invert Level	Max Depth	Initial Depth	Surch. dept	Ponded Are	Ge
17	Junction	<Null>	<Null>	980.00	3.00	0.00	0.00	0.00	<N
18	Outfall	<Null>	<Null>	975.00	0.00	0.00	0.00	0.00	<N
19	Junction	<Null>	<Null>	1010.00	3.00	0.00	0.00	0.00	<N
20	Junction	<Null>	<Null>	1005.00	3.00	0.00	0.00	0.00	<N
21	Junction	<Null>	<Null>	990.00	3.00	0.00	0.00	0.00	<N
22	Junction	<Null>	<Null>	987.00	3.00	0.00	0.00	0.00	<N
23	Junction	<Null>	<Null>	990.00	3.00	0.00	0.00	0.00	<N
24	Junction	<Null>	<Null>	984.00	3.00	0.00	0.00	0.00	<N
9	Junction	<Null>	<Null>	1000.00	3.00	0.00	0.00	0.00	<N
1	StorageUnit	<Null>	<Null>	0.00	0.00	0.00	0.00	0.00	

Figure 13.4 The CS SWMM Nodes - Outfall Editor

Table 13.5 The Nodes Outfall

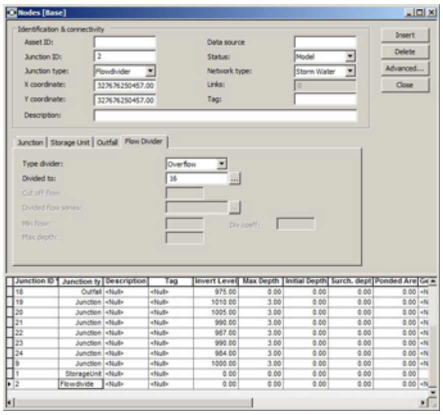
Edit field	Description	Used or required by simulations	Field name in datastructure
Outfall Type	If Node Type is selected to Outfall then type of Tide Gate must be selected	Yes	FlapGate-TypeNo
Tidal Curve	Outfall stage as a function of hour of day over a complete tidal cycle. Curve is specified in Tabular Data	Yes if Tide Gate is of type **TIDAL**	TideGateID
Outfall Time Series	Outfall stage variation over time. Time series specified in Time Series	Yes if Tide Gate is of type **TIME-SERIES**	TideGateTSID
Fixed Stage	Fixed position of Tide Gate	Yes if Tide Gate is of type **FIXED**	FixedStage
Tide Gate	If TideGate (FlapGate) present then backwater is prevented.	No	FlapGateNo

Flow Divider

Three different types of dividers can be specified for each divider node

- OVERFLOW divider, all flow above the non-diverted conduits full flow depth is diverted
- CUTOFF divider, all flow above the cutoff level is diverted.
- TABULAR divider, a curve is supplied that specifies the amount of diverted flow to total flow.

- WEIR divider the diverted flow is linearly proportional to the total flow in excess of some minimum flow



Asset ID:
Junction ID: 2
Junction type: Flowdivider
X coordinate: 327676250457.00
Y coordinate: 327676250457.00
Description:
Data source
Status: Model
Network type: Storm Water
Links: 0
Tag:

Type divider: Overflow
Divided to: 16

	Junction ID	Junction ty	Description	Tag	Invert Level	Max Depth	Initial Depth	Surch. dept	Ponded Are	Ge
	18	Outfall	<Null>	<Null>	975.00	0.00	0.00	0.00	0.00	<N
	19	Junction	<Null>	<Null>	1010.00	3.00	0.00	0.00	0.00	<N
	20	Junction	<Null>	<Null>	1005.00	3.00	0.00	0.00	0.00	<N
	21	Junction	<Null>	<Null>	990.00	3.00	0.00	0.00	0.00	<N
	22	Junction	<Null>	<Null>	987.00	3.00	0.00	0.00	0.00	<N
	23	Junction	<Null>	<Null>	990.00	3.00	0.00	0.00	0.00	<N
	24	Junction	<Null>	<Null>	984.00	3.00	0.00	0.00	0.00	<N
	9	Junction	<Null>	<Null>	1000.00	3.00	0.00	0.00	0.00	<N
	1	StorageUnit	<Null>	<Null>	0.00	0.00	0.00	0.00	0.00	
▸	2	Flowdivide	<Null>	<Null>	0.00	0.00	0.00	0.00	0.00	<N

Figure 13.5 The CS SWMM Nodes - Flow DIvider Editor

Table 13.6 The Flow Divider

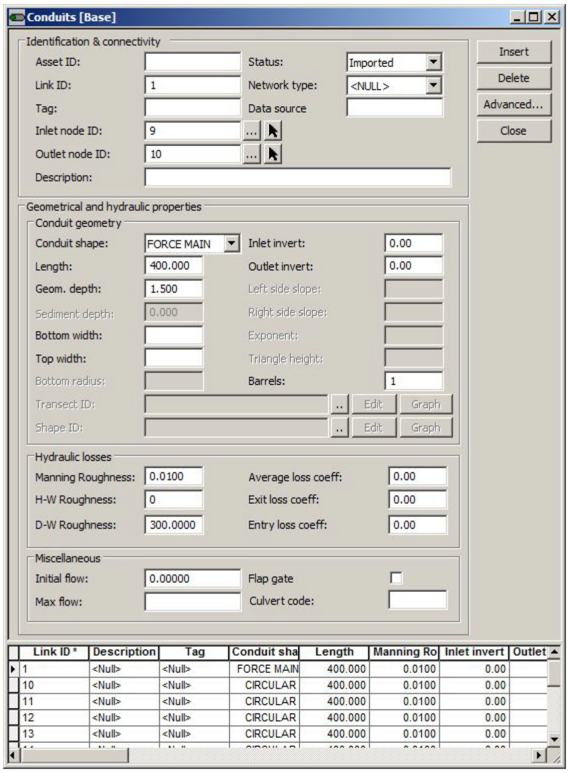
Edit field	Description	Used or required by simulations	Field name in datastructure
Type Divider	Type of Divider is selected to either **Overflow**, **Cutoff**, **Tabular**, or **Weir**	Yes, if Junction Type is a Flow Divider	DividerTypeNo
Divided To	The ID of the link that receives the diverted flow	Yes	LinkID
Cut Off Flow	Cutoff flow value used for a **Cutoff** divider	Yes, if Type Divider is **Cutoff**	CutoffFlow
Divided Flow Series	ID of diversion flow series for a **Tabular** divider	Yes, if Type Divider is **Tabular**	DivertedFlowID
Min Flow	Minimum flow at which diversion continues for a **Weir** divider	Yes, if Type Divider is **Weir**	DivertedMinFlow
Max Depth	Maximum depth above invert at which proportional diversion continues for a **Weir** divider	Yes, if Type Divider is **Weir**	DivertedMaxDepth
Div Coeff	Coefficient which determines what fraction between the min and max flow the diverted flow will be given the fraction that current water depth is of the max. depth for a **Weir** divider	Yes, if Type Divider is **Weir**	DivertionCoeff

13.3 Conduits

Conduits are pipes or channels that move water from one node (Junctions, Storage Units, Flow Dividers and Outfalls) to another node in the drainage system. Their cross-sectional shapes can be selected from a variety of standard open and closed geometries. Irregular natural cross-section shapes are also supported.

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

- Identification and Connectivity - General identification and location information
- Geometrical and Hydraulic Properties - Model related data
- Conduit Geometry - Geometry information
- Hydraulic Losses - Model Headlosses
- Miscellaneous



Identification & connectivity

Asset ID:
Status: Imported
Link ID: 1
Network type: <NULL>
Tag:
Data source
Inlet node ID: 9
Outlet node ID: 10
Description:

Geometrical and hydraulic properties

Conduit geometry

Conduit shape: FORCE MAIN
Inlet invert: 0.00
Length: 400.000
Outlet invert: 0.00
Geom. depth: 1.500
Left side slope:
Sediment depth: 0.000
Right side slope:
Bottom width:
Exponent:
Top width:
Triangle height:
Bottom radius:
Barrels: 1
Transect ID:
Shape ID:

Hydraulic losses

Manning Roughness: 0.0100
Average loss coeff: 0.00
H-W Roughness: 0
Exit loss coeff: 0.00
D-W Roughness: 300.0000
Entry loss coeff: 0.00

Miscellaneous

Initial flow: 0.00000
Flap gate ☐
Max flow:
Culvert code:

Link ID *	Description	Tag	Conduit sha	Length	Manning Ro	Inlet invert	Outlet
1	<Null>	<Null>	FORCE MAIN	400.000	0.0100	0.00	
10	<Null>	<Null>	CIRCULAR	400.000	0.0100	0.00	
11	<Null>	<Null>	CIRCULAR	400.000	0.0100	0.00	
12	<Null>	<Null>	CIRCULAR	400.000	0.0100	0.00	
13	<Null>	<Null>	CIRCULAR	400.000	0.0100	0.00	

Figure 13.6 The CS SWMM Conduits Editor

Identification and Connectivity

Table 13.7 The Conduits Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Asset ID	Id in the asset network	No	AssetName
Link ID	Id of the Link	Yes	MUID
Tag	Optional label used to identify or catagorize the junction	No	Tag
Inlet Node ID	Upstream Node	Yes	
Outlet Node ID	Downstream Node	Yes	
Data source	Reference to an external data source from which the record has been imported from	No	DataSource
Status	Status accoording to user specified list	No	Element_S
Network type	Network type describes the type of network i.e Stormwater, Combined or separate. The list of network types can be extended by user. Network type can be specified for each hydraulic element.	No	NetTypeNo
Description	User's descriptive information related to the node	No	Description

Geometrical and Hydraulic Properties

Table 13.8 The Conduits Geometrical and Hydraulic Properties

Edit field	Description	Used or required by simulations	Field name in data-structure
Conduit Geometry			
Conduit Shape	Cross section shape	Yes	ShapeTypeNo
Length	Length of conduit	Yes	Length
Geom. Depth	Depth or Diameter of conduit	Yes all shapes except IRREGULAR shape	Depth
Sediment Depth	Sediment depth in conduit	Yes, if CIRCULAR shape	SedimentDepth

Table 13.8 The Conduits Geometrical and Hydraulic Properties

Edit field	Description	Used or required by simulations	Field name in data-structure
Bottom Width	Width of bottom	Yes, if MODBASKETHANDLE shape	BottomWidth
Top Width	Width of top	Yes, if RECT_CLOSED, RECT_OPEN, TRAPEZOIDAL, TRIANGULAR, ARCH, PARABOLIC, POWER, RECT_TRIANGULAR, and RECT_ROUND shapes	TopWidth
Bottom Radius	Radius of bottom	Yes, if RECT_ROUND shape	BottomRadius
Transect ID	ID of Transect ID which is defined in Transects	Yes, if IRREGULAR	TransectID
Inlet Invert	Offset heigth of upstream end of conduit above the invert elevation of its upstream node		InletInvert
Outlet Invert	Offset heigth of downstream end of conduit above the invert elevation of its downstream node		OutletInvert
Left Side Slope	Left side slope of trapez	Yes, if TRAPEZOIDAL shape	LeftSideSlope
Right Side Slope	Right side slope of trapez	Yes if TRAPEZOIDAL shape	RightSideSlope
Exponent	Exponent in power function for describing the shape	Yes, if POWER	Exponent
Triangle Height	Height of triangular shape	Yes, if RECT_TRIANGULAR shape	TriangleHeight

Table 13.8 The Conduits Geometrical and Hydraulic Properties

Edit field	Description	Used or required by simulations	Field name in data-structure
Barrels	A conduit can have a multiple number of barrels meaning that it consists of a multiple number of parallel pipes of equal size, slope and roughness. Default value is 1.	Yes	Barrels
Hydraulic Losses			
Roughness	Manning, n-value for the conduit	Yes	Roughness
Entry Loss Coeff	Entry Loss coefficient (using entrance velocity)	No	EntryLossCoeff
Average Loss Coeff	Average loss coefficient (using average velocity in conduit)	No	AvgLossCoeff
Exit Loss Coeff	Exit loss coefficient (using exit velocity)	No	ExitLossCoeff
Miscellaneous			
Initial Flow	Flow in conduit at start of simulation	No	InitialFlow
Flap Gate	Flapgate prevents backwater flow. Default is No (not present)	No	FlapGateNo
Max Flow	Maximum flow allowed	Yes	MaxFlow
Culvert Code	Culvert type code	Yes	

Minor losses are only computed for the DW, Dynamic wave routing option. The minor losses are computed as $K^*v^2/2g$.

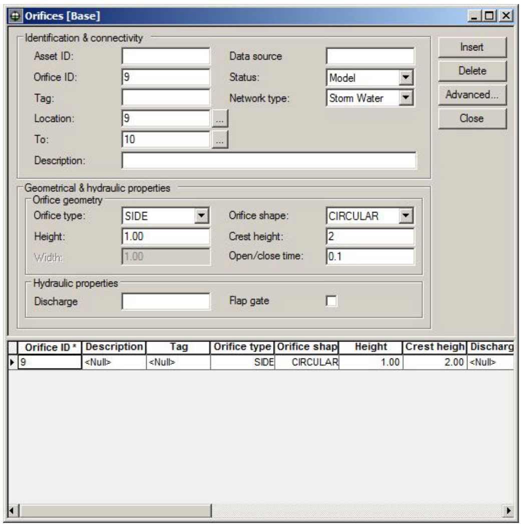
13.4 Orifices

Orifices are openings in the wall of a manhole junction or storage unit used to model outlet and diversion structures in drainage systems. Orifices belong to the group of Link components connecting two nodes. An orifice can have either a circular or rectangular shape, be located either at the bottom or along the side of the upstream node, and have a flap gate to prevent backflow.

Orifices attached to storage units are active under all types of flow routing. Those attached to other types of nodes are active only under Dynamic Wave flow routing, otherwise the flow through them is zero. The flow through an orifice is computed based on the area of its opening, its discharge coefficient, and the head difference across the orifice. The area of an orifice's opening can be controlled dynamically through user-defined control rules.

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

- Identification and Connectivity - General identification and location information
- Geometrical and Hydraulic Properties - Model related data
- Orifices Geometry - Geometry information
- Hydraulic Properties - Model Headlosses



Asset ID:
Orifice ID: 9
Tag:
Location: 9
To: 10
Description:
Data source
Status: Model
Network type: Storm Water

Orifice type: SIDE
Height: 1.00
Width: 1.00
Orifice shape: CIRCULAR
Crest height: 2
Open/close time: 0.1

Discharge
Flap gate ☐

Orifice ID *	Description	Tag	Orifice type	Orifice shap	Height	Crest heigh	Discharg
9	<Null>	<Null>	SIDE	CIRCULAR	1.00	2.00	<Null>

Figure 13.7 The CS SWMM Orifices Editor

Identification and Connectivity

Table 13.9 The Orifices Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Asset ID	Id in the asset network	No	AssetName
Orifice ID	Id of the Orifice	Yes	MUID
Tag	Optional label used to identify or catagorize the junction	No	Tag
Location	Node location of orifice	Yes	

Table 13.9 The Orifices Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
To	Discharge to Node	Yes	
Data source	Reference to an external data source from which the record has been imported from	No	DataSource
Status	Status according to user specified list	No	Element_S
Network type	Network type describes the type of network i.e Stormwater, Combined or separate. The list of network types can be extended by user. Network type can be specified for each hydraulic element.	No	NetTypeNo
Description	User's descriptive information related to the node	No	Description

Geometrical and Hydraulic Properties

Table 13.10 The Orifices Geometrical and Hydraulic Properties

Edit field	Description	Used or required by simulations	Field name in datastructure
Orifices Geometry			
Orifice Type	Type of orifice, SIDE or BOTTOM	Yes	TypeNo
Orifice Shape	The geometry of the orifice opening, CIRCULAR or RECT_CLOSED	Yes	ShapeTypeNo
Height	Height/Diameter of shape	Yes	Height
Width	Width of RECT_CLOSED shape	Yes, if Orifice shape is RECT_CLOSED	Width
Crest Height	Offset height of orifice's bottom from invert of upstream node	Yes (if Orifice Type is BOTTOM then Crest Height is set zero in SWMM engine)	CrestHeight
Hydraulic Properties			

Table 13.10 The Orifices Geometrical and Hydraulic Properties

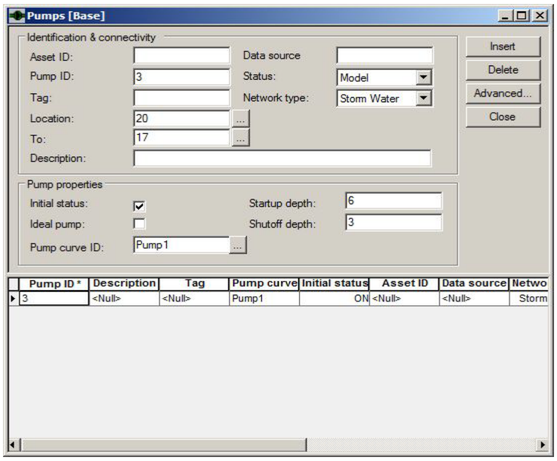
Edit field	Description	Used or required by simulations	Field name in data-structure
Discharge Coeff	Discharge coefficient	Yes	DischargeCoeff
Flap gate	Flapgate prevents reverse flow through the orifice, default is No	No	FlapGateNo

13.5 Pumps

Pumps are drainage system links used to lift water to higher elevations. The relation between a pump's flow rate and conditions at its end nodes is described by a pump curve. MIKE URBAN CS SWMM represents a pump as a link connecting two nodes, where the pump itself is placed at the upstream node.

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

- Identification and Connectivity - General identification and location information
- Pump Properties - Model related data



Identification & connectivity

Asset ID:
Pump ID: 3
Tag:
Location: 20
To: 17
Description:
Data source
Status: Model
Network type: Storm Water

Pump properties

Initial status: ☑
Ideal pump: ☐
Pump curve ID: Pump1
Startup depth: 6
Shutoff depth: 3

Pump ID *	Description	Tag	Pump curve	Initial status	Asset ID	Data source	Netwo
3	<Null>	<Null>	Pump1	ON	<Null>	<Null>	Storm

Figure 13.8 The CS SWMM Pumps Editor

Identification and Connectivity

Table 13.11 The Pumps Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Asset ID	Id in the asset network	No	AssetName
Pump ID	Id of the Pump	Yes	MUID
Tag	Optional label used to identify or catagorize the junction	No	Tag
Location	Location of Pump	Yes	
To	Receiving Node	Yes	
Data source	Reference to an external data source from which the record has been imported from	No	DataSource
Status	Status accoording to user specified list	No	Element_S
Network type	Network type describes the type of network i.e Stormwater, Combined or separate. The list of network types can be extended by user. Network type can be specified for each hydraulic element.	No	NetTypeNo
Description	User's descriptive information related to the node	No	Description

Pump Properties

Table 13.12 The Pumps Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Pump Curve ID	Reference to a Pump curve defined as one of the four possible pump curve types in Tabular Data (Curves).	Yes	PumpCurveID
Initial Status	The initial status is On	No	InitialStatusNo

For each individual pump a pump curve must be defined. Pump curves are defined in Tabular Data (Curves). Following four different types of pump curves are supported:

- Type1 - an off-line pump with a wet well where flow increases incrementally with wet well volume
- Type2 - an in-line pump where flow increases incrementally with node depth
- Type3 - an in-line pump where flow varies continuously with head difference between the inlet and outlet nodes
- Type4 - a variable speed in-line pump where flow varies continuously with node depth

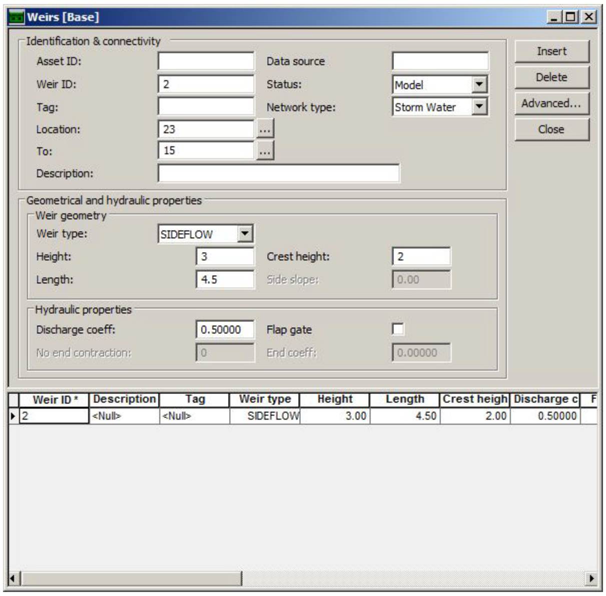
The on/off status of pumps can be controlled dynamically through user-defined Control Rules.

13.6 Weirs

Weirs, like orifices, are used to model outlet and diversion structures in a drainage system. Weirs are typically located in a chamber of a manhole, along the side of a channel, or along the top of a storage unit. MIKE URBAN CS SWMM represents a weir as a link connecting two nodes, where the weir itself is placed at the upstream node. A flap gate can be included to prevent backflow.

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

- Identification and Connectivity - General identification and location information
- Geometrical and Hydraulic Properties - Model related data
- Weir Geometry - Geometry information
- Hydraulic Properties


Weirs [Base]
Identification & connectivity
Geometrical and hydraulic properties
Weir geometry
Hydraulic properties
Insert
Delete
Advanced...
Close

Asset ID:
Weir ID: 2
Tag:
Location: 23
To: 15
Description:
Data source
Status: Model
Network type: Storm Water

Weir type: SIDEFLOW
Height: 3
Length: 4.5
Crest height: 2
Side slope: 0.00

Discharge coeff: 0.50000
No end contraction: 0
Flap gate ☐
End coeff: 0.00000

Weir ID *	Description	Tag	Weir type	Height	Length	Crest heigh	Discharge c	F
2	<Null>	<Null>	SIDEFLOW	3.00	4.50	2.00	0.50000	

Figure 13.9 The CS SWMM Weir Editor

Identification and Connectivity

Table 13.13 The Weir Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Asset ID	Id in the asset network	No	AssetName
Weir ID	Id of the Weir	Yes	MUID
Tag	Optional label used to identify or catagorize the junction	No	Tag
Location	Node location of Weir	Yes	
To	Receiving Node	Yes	
Data source	Reference to an external data source from which the record has been imported from	No	DataSource
Status	Status accoording to user specified list	No	Element_S

Table 13.13 The Weir Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Network type	Network type describes the type of network i.e Stormwater, Combined or separate. The list of network types can be extended by user. Network type can be specified for each hydraulic element.	No	NetTypeNo
Description	User's descriptive information related to the node	No	Description

Geometrical and Hydraulic Properties

Table 13.14 The Weir Geometrical and Hydraulic Properties

Edit field	Description	Used or required by simulations	Field name in datastructure
Weir Geometry			
Weir Type	Type of weir can chosen from a prefined list of four types of weirs	Yes	TypeNo
Height	Height of weir	Yes	Height
Crest Height	Offset of weir crest above invert elevation of node	Yes	CrestHeight
Length	Length of weir	Yes	Length
Side Slope	Side slope of weir of type V-NOTCH and TRAPEZOIDAL		SideSlope
Hydraulic Properties			
Discharge Coeff	Weir discharge coefficient	Yes	DischargeCoeff
Flap gate	Flap gate to prevent backflow	No	FlapGateNo
No End Contraction	Number of end contractions for Transverse and Trapezoidal weir	Yes, if weir type is transverse or trapezoidal	NoEndContractions
End Coeff	Discharge coefficient for triangular ends of trapezoidal weir	Yes, if weir type is trapezoidal	Cd2

Table 13.14 The Weir Geometrical and Hydraulic Properties

Edit field	Description	Used or required by simulations	Field name in data-structure
Coefficient Curve	Weir curve that relates discharge coefficient to head	No	CoeffCurveID
Road Width	Width of the road lanes and shoulders	Yes, if weir coefficient curve is specified	RoadWidth
Road Surface	Type of road surface, PAVED or GRAVEL	Yes, if weir coefficient curve is specified	RoadSurf

Four varieties of weirs (Weir Type) are available, each with a slightly different formula for computing flow as a function of area, discharge coefficient and head difference across the weir:

- Transverse (rectangular shape)
- Sideflow (rectangular shape)
- V-notch (triangular shape)
- Trapezoidal (trapezoidal shape)
- Roadway (Roadway shape)

Weirs attached to storage units are active under all types of flow routing. Those attached to other types of nodes are active only under Dynamic Wave flow routing, otherwise the flow through them is zero.

The height of the weir crest above the inlet node invert can be controlled dynamically through user-defined Control Rules. This feature can be used to model RTC controlled Weirs and inflatable dams.

The type of weir ROADWAY, models roadway overtopping using the FHWA HDS-5(Hydraulic Design Series 5) method and would typically be used in parallel with a culvert conduit

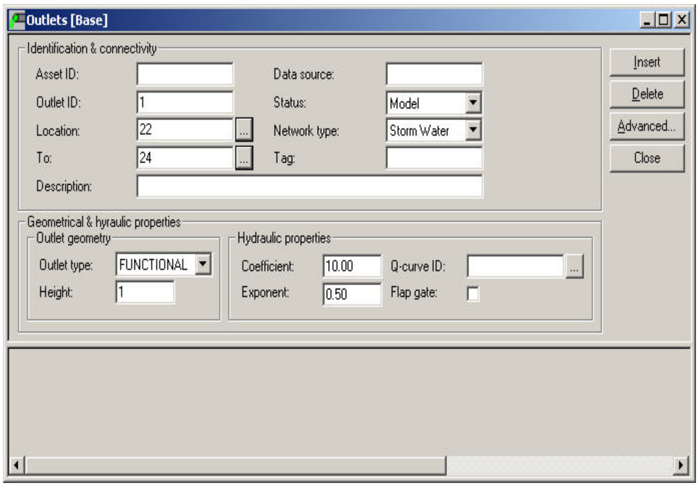
13.7 Outlets

Outlets are flow control devices which are typically used to control outflows from storage units. They are used to model special flow-head relations that cannot be characterized by either orifices or weirs. MIKE URBAN CS SWMM represents an Outlet as a link connecting two nodes, where the outlet itself is placed at the upstream node.

Outlets attached to storage units are active under all types of flow routing. Those attached to other types of nodes are active only under Dynamic Wave flow routing, otherwise the flow through them is zero.

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

- Identification and Connectivity - General identification and location information
- Geometrical and Hydraulic Properties - Model related data
- Outlet Geometry - Geometry information
- Hydraulic Properties


Outlets [Base]
Identification & connectivity
Geometrical & hyraulic properties
Outlet geometry
Hydraulic properties
Insert
Delete
Advanced...
Close

Asset ID:
Outlet ID: 1
Location: 22
To: 24
Description:
Data source:
Status: Model
Network type: Storm Water
Tag:
Outlet type: FUNCTIONAL
Height: 1
Coefficient: 10.00
Exponent: 0.50
Q-curve ID:
Flap gate: ☐

Figure 13.10 The CS SWMM Outlets Editor

Identification and Connectivity

Table 13.15 The Outlets Identification and Connectivity

Edit field	Description	Used or required by simulations	Field name in datastructure
Asset ID	Id in the asset network	No	AssetName
Outlet ID	Id of the Outlet	Yes	MUID
Tag	Optional label used to identify or catagorize the junction	No	Tag
Location	Node location of Outlet	Yes	
To	Recieving Node	Yes	
Data source	Reference to an external data source from which the record has been imported from	No	DataSource
Status	Status accoording to user specified list	No	Element_S

Table 13.15 The Outlets Identification and Connectivity

Edit field	Description	Used or required by simulations	Field name in datastructure
Network type	Network type describes the type of network i.e Stormwater, Combined or separate. The list of network types can be extended by user. Network type can be specified for each hydraulic element.	No	NetTypeNo
Description	User's descriptive information related to the node	No	Description

Geometrical and Hydraulic Properties

Table 13.16 The Outlets Geometrical and Hydraulic Properties

Edit field	Description	Used or required by simulations	Field name in datastructure
Outlet Geometry			
Outlet Type	Outlet Type, 1=TABUAR/DEPTH, 2 = TABULAR/HEAD 3 = FUNCIONAL /DEPTH 4 = FUNCTIONAL / HEAD	Yes	TypeNo
Height	Minimum water depth at upstream node of outflow to occur	Yes	Height
Hydraulic Properties			
Coefficient	Coefficient of power function that relates outflow to head across the outlet fir a Functional outlet	Yes, if Outlet Type is Functional	Qcoeff
Q-curve ID	ID of rating curve with outflow rate as a function of head across the outlet for a tabular outlet	Yes if Outlet Type is Tabular	QcurveID
Exponent	Exponent of power function that relates outflow to head across the outlet fir a Functional outlet	Yes, if Outlet Type is Functional	Qexpon
Flap Gate	Flap gate	Yes	FlapGateNo

An outlet can have a flap gate which restricts flow to only one direction and prevents backwater flow.

The flow through an outlet is determined by a user-supplied function or table of flow versus head difference across the outlet (Q-curve ID). This flow can be controlled dynamically through user-defined Control Rules.

13.8 Transects

Transects refer to the geometric data that describe how bottom elevation varies with horizontal distance over the cross section of a natural channel or irregular-shaped conduit. The figure below displays an example of a transect for a natural channel.

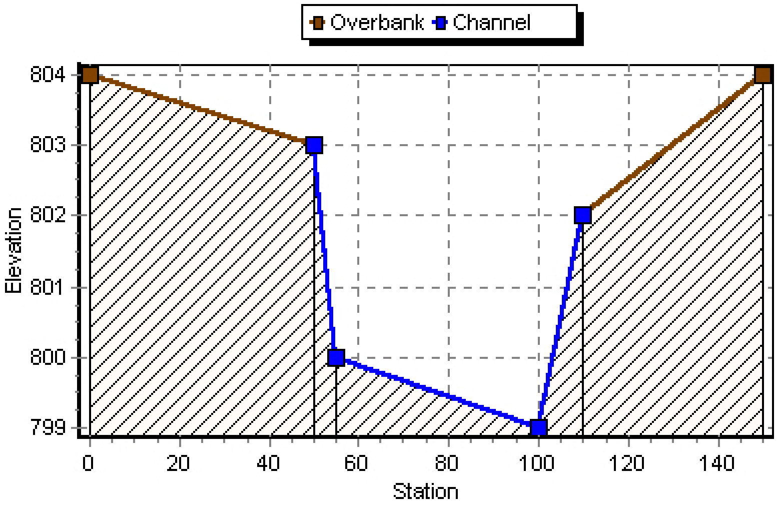


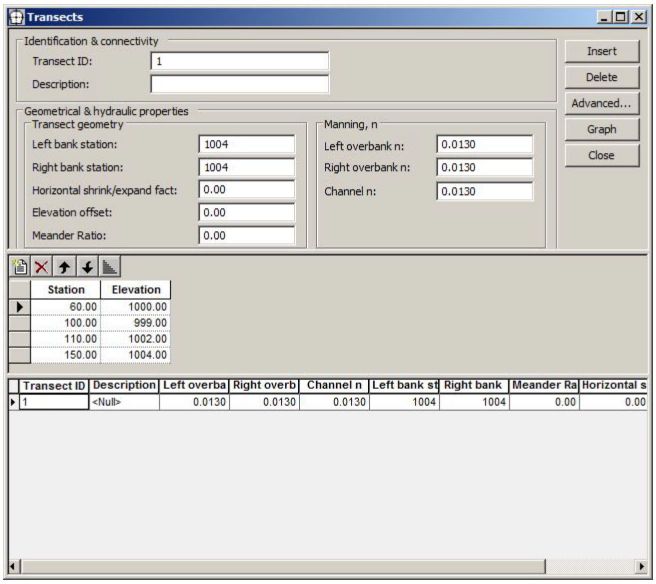
Figure 13.11 Definition of the Transect cross-sectional data

Each transect are given an unique ID. When a IRREGULAR shape is chosen as the shape type for a conduit then a reference to one of the user defined Transects must be made for representing the crossectional shape.

The SWMM engine converts these data into tables of area, top width, and hydraulic radius versus channel depth. In addition, as shown in the figure above, each Transect can have a left and right overbank section whose Manning's roughness can be different from that of the main channel. This feature can provide more realistic estimates of channel conveyance under high flow conditions.

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

- Identification and Connectivity - General identification and location information
- Geometrical and Hydraulic Properties - Model related data
- Transect Geometry - Geometry information
- Manning, n - Roughness for Channel and Overbanks



Identification & connectivity

Transect ID: 1

Description:

Geometrical & hydraulic properties

Transect geometry

Left bank station: 1004

Right bank station: 1004

Horizontal shrink/expand fact: 0.00

Elevation offset: 0.00

Meander Ratio: 0.00

Manning, n

Left overbank n: 0.0130

Right overbank n: 0.0130

Channel n: 0.0130

Station	Elevation
60.00	1000.00
100.00	999.00
110.00	1002.00
150.00	1004.00

Transect ID	Description	Left overba	Right overb	Channel n	Left bank st	Right bank	Meander Ra	Horizontal s
1	<Null>	0.0130	0.0130	0.0130	1004	1004	0.00	0.00

Figure 13.12 The SWMM Transects Editor Dialog

The above Transect can be viewed graphically by clicking on the Graph button

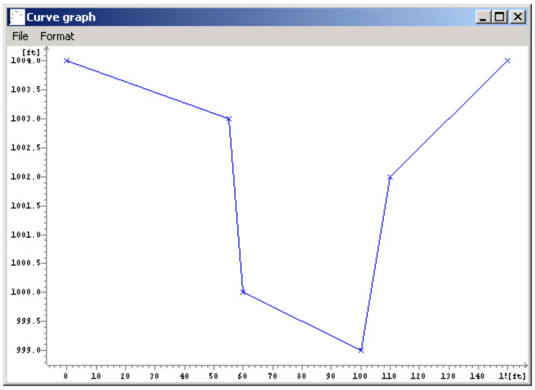


Figure 13.13 Graphically display of the transect

Identification and Connectivity

Table 13.17 The Transect Identification and Connectivity

Edit field	Description	Used or required by simulations	Field name in datastructure
Transect ID	ID of the transect	Yes	MUID
Description	User's descriptive information related to the node	No	Description

Geometrical and Hydraulic Properties

Table 13.18 The Transect Geometrical and Hydraulic Properties

Edit field	Description	Used or required by simulations	Field name in datastructure
Transect geometry			
Left bank station	Left overbank station	Yes	Xleft
Right bank station	Right overbank station	Yes	Xright
Horizontal shrink/expand factor	Horizontal shrink/expand factor	Yes	Wfactor
Elevation offset	Elevation offset	Yes	Eoffset
Meander Ratio	Meander ratio	Yes	LFactor
Manning, n			
Left overbank n	Manning for left overbank	Yes	Nright
Right overbank, n	Manning for right overbank	Yes	Nleft
Channel, n	Manning for channel	Yes	Nchannel

Cross-sectional data, (Station, Elevation)

Table 13.19 The Transect Cross-sectional data

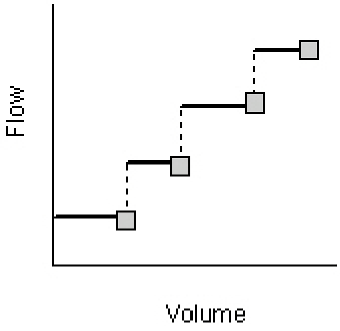
Edit field	Description	Used or required by simulations	Field name in data-structure
Station	Station, distance of the crosssection station from some fixed reference	Yes	Station
Elevation	Elevation of the channel bottom at a cross-section station relative to some fixed relative reference	Yes	Elevation

13.9 Tabular Data (Curves)

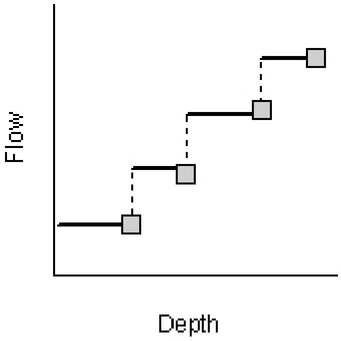
Tabular Data (Curves) are used to describe a functional relationship between two quantities. Following types of curves are available:

- **Storage** - defines the geometry of a Storage Unit node by describing the variation of surface area as a function of water depth
- **Diversion** - defines how the diverted outflow from a Flow Divider node is related to the total inflow.
- **Tidal** - defines how the stage at an Outfall node varies by the hour of the day.
- **Rating** - relates flow through an Outlet link to the head difference across the outlet
- **Pump** - relates flow through a Pump link to the depth or volume at the upstream node or to the head delivered by the pump. Following four types of Pump relations are available

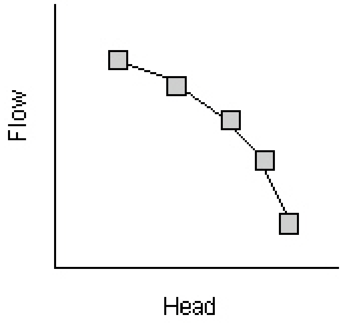
Type 1 - an off-line pump with a wet well where flow increases incrementally with the available wet well volume.



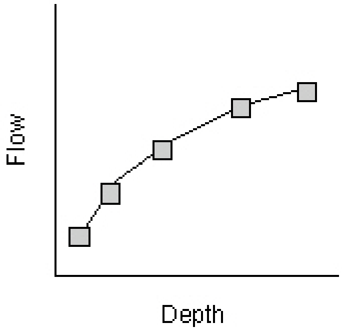
Type 2 - an in-line pump where flow increases incrementally with inlet node depth.



Type 3 - an in-line pump where flow varies continuously with head difference between the inlet and outlet nodes.

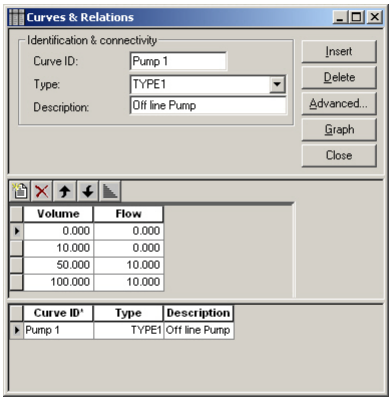


Type 4 - a variable speed in-line pump where flow varies continuously with inlet node depth.



The Tabular Data (Curves) editor organizes the related input data into the following groups:

- Identification and Connectivity - General identification and location information



Curve ID: Pump 1
Type: TYPE1
Description: Off line Pump

Volume	Flow
0.000	0.000
10.000	0.000
50.000	10.000
100.000	10.000

Curve ID'	Type	Description
Pump 1	TYPE1	Off line Pump

Figure 13.14 The SWMM Tabular Data (Curves) dialog

Identification and Connectivity

Table 13.20 The Tabular Data (Curves) Identification and Connectivity

Edit field	Description	Used or required by simulations	Field name in datastructure
Curve ID	ID of the Curve	Yes	MUID
Type	Type of Curve	Yes	TypeNo
Description	User's descriptive information related to the node	No	Description

Tabular data in Grid

Table 13.21 The Tabular data values

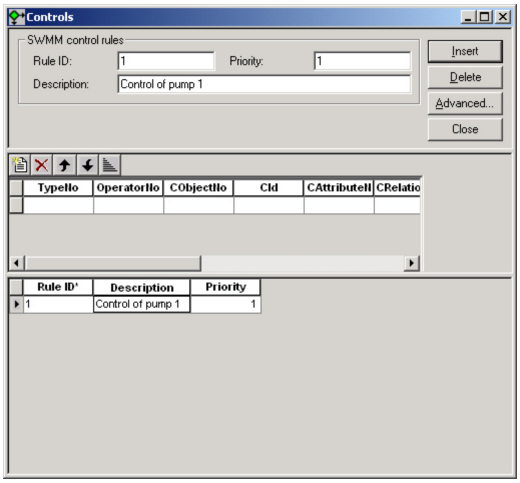
Edit field	Description	Used or required by simulations	Field name in data-structure
‘Value1’	Value1, depended on Type of curve (Depth, Inflow, Hour, Volume, Head)	Yes	Value1
‘Value2’	Value2, depended on Type of curve (Area, Outflow, Stage, Flow)	Yes	Value2

13.10 Controls

Controls is used to determine pumps and regulators will be controlled based on simulation time or conditions at specific nodes and links.

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

- SWMM control rules - Identification and Connectivity
- The control rules are specified in a upper grid


Controls
Insert
Delete
Advanced...
Close
TypeNo
OperatorNo
CObjectNo
CId
CAttributeN
CRelatio

SWMM control rules

Rule ID: 1

Priority: 1

Description: Control of pump 1

Rule ID*	Description	Priority
1	Control of pump 1	1

Figure 13.15 The SWMM Controls Editor Dialog

SWMM control rules

Table 13.22 The SWMM control rules Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Rule ID	ID of the specific rule	Yes	MUID
Priority	Priority of the rule	Yes	Priority
Description	User's descriptive information related to the node	No	Description

Table 13.23 The upper grid Group

Edit field	Description	Used or required by simulations	Field name in datastructure
TypeNo	Condition or Action	Yes	TypeNo
OperatorNo	IF, AND, OR, THEN, ELSE	Yes	Operators
CObjectNo	Node, Link, Pump, Orifice, Weir or Simulation	Yes	CObjectNo
CId	ID of the object	Yes	CId
CAttributeNo	Depth, Head, Inflow, Flow, Setting, Time, Date, Clocktime	Yes	CAttributeNo
CRelationNo	=, <, >, <>, <=,>=	Yes	CRelationNo
AObjectNo	Node, Link, Pump, Orifice, Weir or Simulation	Yes	AObjectNo
AId	ID of the object	Yes	AId
AAttributeNo	Node, Link, Pump, Orifice, Weir or Simulation	Yes	AAttributeNo
RValue	Value	Yes	RValue

Please note there can only be either a Condition or an Action in one line.

14 Hydrological Modelling with SWMM

14.1 Terms and Concepts

Various hydrological processes that produce runoff from urban areas can be modelled by SWMM. These include:

- time-varying rainfall
- evaporation of standing surface water
- snow accumulation and melting
- rainfall interception in depression storage
- infiltration of rainfall into upper soil layers
- percolation of infiltrated water into groundwater layers
- interflow between groundwater and the conveyance system
- nonlinear reservoir routing of overland flow

Spatial variability in all of these processes is achieved by dividing a study area into a smaller collection of homogeneous areas. In the literature different terms are used for describing the delineation of large study areas into smaller sub-areas each containing its own fraction of pervious and impervious areas.

Following terminology for the delineation of areas is widely used in North America

- Basin
- Sub-basin
- Watershed/Sewershed
- Catchment

Throughout this documentation the term Catchments will be used as term for the smallest hydrological unit of land.

Overland flow can be routed between catchments or to load points of a drainage conveyance system.

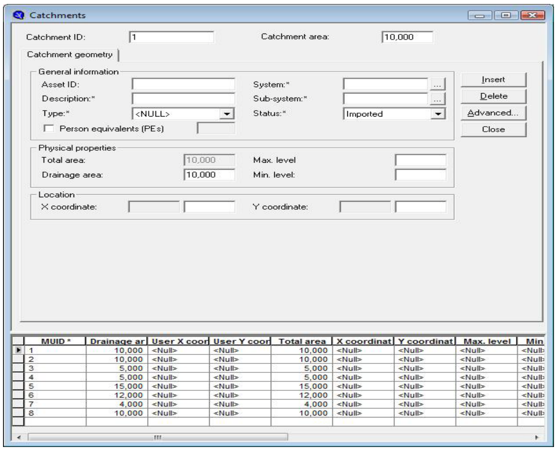
14.2 Catchments

Catchments are hydrological units of land whose topography cause surface runoff to drain to a single point. The user is responsible for the delineation of a watershed/sewershed into a compatible number of catchments, and for identifying the outlet point of each catchment. Outlet points can be either nodes of the conveyance system or other catchments.

Catchments can be divided into pervious and impervious sub-areas. Surface runoff can infiltrate into the upper soil zone of the pervious sub-area, but not through the impervious sub-area. Impervious areas are themselves divided into two sub-areas - one that contains depression storage and another that does not. A portion of the runoff flow from one sub-area in a catchment can be routed to the other sub-area, or both sub-areas can drain to the catchment outlet.

The CS SWMM Catchments Editor organizes the related input data for the catchment into the following groups. The catchment data in the Catchment Editor is shared by CS models:

- General information
- Physical Properties
- Location


Catchments
Catchment geometry
General information
Asset ID:
Description:*
System:*
Sub-system:*
☐ Person equivalents (PEs)
Physical properties
Max. level
Min. level:
Location
X coordinate:
Y coordinate:
Insert
Delete
Advanced...
Close

Catchment ID: 1
Catchment area: 10,000
Type:* <NULL>
Status:* Imported
Total area: 10,000
Drainage area: 10,000

	MUID *	Drainage ar	User X coor	User Y coor	Total area	X coordinat	Y coordinat	Max. level	Min
▶	1	10,000	<Null>	<Null>	10,000	<Null>	<Null>	<Null>	<Null>
	2	10,000	<Null>	<Null>	10,000	<Null>	<Null>	<Null>	<Null>
	3	5,000	<Null>	<Null>	5,000	<Null>	<Null>	<Null>	<Null>
	4	5,000	<Null>	<Null>	5,000	<Null>	<Null>	<Null>	<Null>
	5	15,000	<Null>	<Null>	15,000	<Null>	<Null>	<Null>	<Null>
	6	12,000	<Null>	<Null>	12,000	<Null>	<Null>	<Null>	<Null>
	7	4,000	<Null>	<Null>	4,000	<Null>	<Null>	<Null>	<Null>
	8	10,000	<Null>	<Null>	10,000	<Null>	<Null>	<Null>	<Null>

Figure 14.1 The SWMM Catchments Editor

Description of the input fields for catchments

Table 14.1 The Catchment table

Edit field	Description	Used or required by simulations	Field name in datastructure
Catchment ID	Id in the catchment	Yes	MUID
Status	Status according to user specified list	No	Element_S
Type	Type which describes the type of catchment. The list of catchment types is user sepcified.	No	NetTypeNo

Location

Table 14.2 The Catchment Location Group

Edit field	Description	Used or required by simulations	Field name in datastructure
System	User defined list	No	SystemID
Sub-system	User defined list	No	SubSystemID
X-coordinate	X - coordinate of center of catchment	No	X
Y-coordinate	Y - coordinate of center of catchment	No	Y

Physical Properties

Table 14.3 The Catchment Physical Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Total area	Total area of polygon	Yes	Area_C
Drainage area	User defined catchment area - overwrites Total area	No	Area
Max. level	Catchment max. level, can be calculated from DEM by GIS function	No	MaxLevel
Min. level	Catchment min. level, can be calculated from DEM by GIS function	No	MinLevel

General Information

Table 14.4 The Catchment General Information Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Description	Descriptive user field	No	Description

14.3 Surface Routing

The concept of the surface runoff used by CS SWMM is illustrated in the figure below. Each catchment surface is treated as a nonlinear reservoir. Inflow comes from precipitation and upstream catchments. There are several outflows, including infiltration, evaporation, and surface runoff.

The capacity of this "reservoir" is the maximum depression storage, which is the maximum surface storage provided by ponding, surface wetting, and interception.

Surface runoff per unit area, Q, occurs only when the depth of water in the "reservoir" exceeds the maximum depression storage, d_p, in which case the outflow is given by Manning's equation (Kinematic Wave).

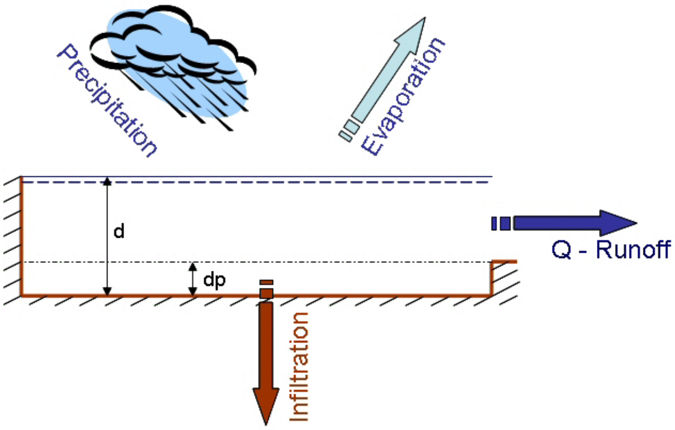


Figure 14.2 Nonlinear Reservoir Concept

Depth of water over the catchment is continuously updated with time by solving numerically a water balance equation over the catchment.

The CS SWMM Routing Editor organizes the related input data for the catchment concerning the surface runoff into following groups:

- Identification and Connectivity

- Hydrology and Hydraulic Properties

Identification and Connectivity

Table 14.5 The Routing Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Catchment ID	ID of the Catchment	Yes	CatchID
Load Point ID	ID of Node that receives the surface runoff	Yes, If RouteToNo is OUTLET	OutletNodeID
Outlet Catchment ID	ID of Catchment that receives the surface runoff	Yes, If RoutToNo is IMPERV or PERV	OutletCatchmentID
Tag	Optional label used to identify or categorize the junction	No	Tag
Description	User's descriptive information related to the node	No	Description
Raingage ID	ID of Rain Gauge containing the precipitation used for this catchment	Yes	RaingageID

Hydrology and Hydraulic Properties

Table 14.6 The Routing Hydrology and Hydraulic Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Width	Width of catchment	Yes	Width
Ground Slope	Ground slope of catchment in percentage	Yes	Slope
Imperviousness	Impervious part of the catchment area in percentage	Yes	Impervious
Imperv. Manning	Manning Number for the Impervious part of the area	Yes	NImperv
Pervious Manning	Manning Number for the Pervious part of the area	Yes	NPerv
Imperv. d. Storage	Size of Impervious Depression Storage	Yes	DImperv
Perv. d. Storage	Size of Pervious Depression Storage	Yes	DPerv
%DCIA w/o d. storage	Percent of the impervious area with no depression storage	Yes	ImpervPctZero

Table 14.6 The Routing Hydrology and Hydraulic Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Subarea routing	Where does the surface routing go to	Yes	RouteToNo
Curb length	Total length of Curbs in the catchment. Only used when pollutant buildup is normalized to curb length	No	CurbLength
N-Perv Pattern	Operational monthly pattern that adjust pervious Mannings N	Optional	NPervPatternID
DStore Pattern	Optional monthly pattern that adjust depresion storage	Optional	DstorePatternID
Infil Pattern	Optional Monthly pattern that adjust infiltration rate	Optional	InfilPatternID

14.4 Infiltration

Infiltration is the process of rainfall penetrating the ground surface into the soil over the pervious areas of catchments. CS SWMM supports three choices for modelling infiltration:

Horton's Equation

This method is based on empirical observations showing that infiltration decreases exponentially from an initial maximum rate to some minimum rate over the course of a long rainfall event. Input parameters required by this method include the maximum and minimum infiltration rates, a decay coefficient that describes how fast the rate decreases over time, and a regeneration constant that describes the restoration of infiltration rate during dry periods.

Green-Ampt Method

This method for modelling infiltration assumes that a sharp wetting front exists in the soil column, separating soil with some initial moisture content below from saturated soil above. The input parameters required are the initial moisture deficit of the soil, the soil's hydraulic conductivity, and the suction head at the wetting front.

Curve Number Method

This approach is adopted from the NRCS (SCS) Curve Number method for estimating runoff. It assumes that the total infiltration capacity of a soil can be found from the soil's tabulated Curve Number. During a rain event this capacity is depleted as a function of cumulative rainfall and remaining capacity. The input parameters for this method are the curve number, the soil's hydraulic conductivity (used to estimate a minimum separation time for distinct rain events), and a regeneration constant that describes the restoration of infiltration capacity during dry periods.

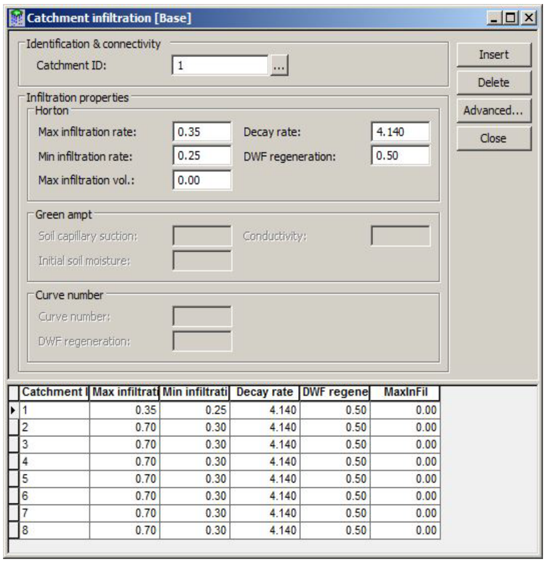
Modified Green-Ampt

This is a modified version of the original Green-Ampt procedure by not depleting moisture deficit in the top surface layer of soil during the initial periods of low rainfall as was done in the original method. This change can produce more realistic infiltration behavior for storms with long initial periods where the rainfall intensity is below the soil's saturated hydraulic conductivity.

Only one of the above infiltration methods can be used in one simulation - the option for which method is used is set in Run SWMM5, tabsheet RUNOFF.

The CS SWMM Infiltration Editor organizes the related input data for these three infiltration approaches into following groups:

- Identification and Connectivity
- Infiltration Properties
  - Horton
  - Green-Ampt
  - Curve Number
  - Modified Green-Ampt


Catchment infiltration [Base]
Identification & connectivity
Infiltration properties
Horton
Green ampt
Soil capillary suction:
Conductivity:
Initial soil moisture:
Curve number
Curve number:
DWF regeneration:
Insert
Delete
Advanced...
Close

Catchment ID: 1

Max infiltration rate: 0.35

Decay rate: 4.140

Min infiltration rate: 0.25

DWF regeneration: 0.50

Max infiltration vol.: 0.00

Catchment I	Max infiltrati	Min infiltrati	Decay rate	DWF regene	MaxInFil
1	0.35	0.25	4.140	0.50	0.00
2	0.70	0.30	4.140	0.50	0.00
3	0.70	0.30	4.140	0.50	0.00
4	0.70	0.30	4.140	0.50	0.00
5	0.70	0.30	4.140	0.50	0.00
6	0.70	0.30	4.140	0.50	0.00
7	0.70	0.30	4.140	0.50	0.00
8	0.70	0.30	4.140	0.50	0.00

Figure 14.3 The SWMM Infiltration Editor

Identification and Connectivity

Table 14.7 The Infiltration Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Catchment ID	ID of the Catchment	Yes	CatchID

Infiltration Properties

Table 14.8 The Infiltration Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Horton			
Max infiltration rate	Max Infiltration Rate	Yes	MaxRate
Min infiltration rate	Min Infiltration Rate	Yes	MinRate
Max infiltration vol.	Max Infiltration Volume	Yes	MaxInFil
Decay rate	Decay Rate	Yes	DecayRate
DWF regeneration	DWF regeneration factor	Yes	HRegen
Green-Ampt			
Soil capillary suction	Soil Capillary Suction	Yes	Suction
Initial soil moisture	Inital saturated soil moisture	Yes	InitDef
Conductivity	Soil saturated hydraulic conductivity	Yes	Conduct
Curve Number			
Curve Number	Curve No	Yes	RunoffCN
DWF regeneration	DWF regeneration factor	Yes	CRegen

14.5 RDII

Rainfall Dependent Inflow/Infiltration (RDII) - These are stormwater flows that enter sanitary or combined sewers due to "inflow" from direct connections of downspouts, sump pumps, foundation drains, etc. as well as "infiltration" of subsurface water through cracked pipes, leaky joints, poor manhole connections, etc.

RDII can be computed for a given rainfall record based on set of triangular unit hydrographs (UH) that determine a short-term, intermediate-term, and long-term inflow response for each interval of rainfall. Each unit hydrograph is defined by three parameters:

- R: the fraction of rainfall volume that enters the sewer system
- T: the time from the onset of rainfall to the peak of the UH in hour
- K: the ratio of time to recession of the UH to the time to peak

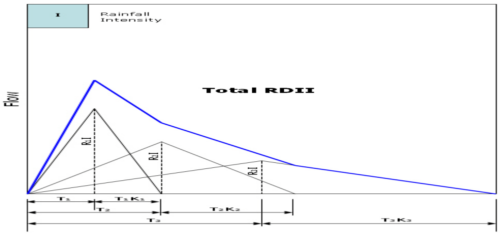
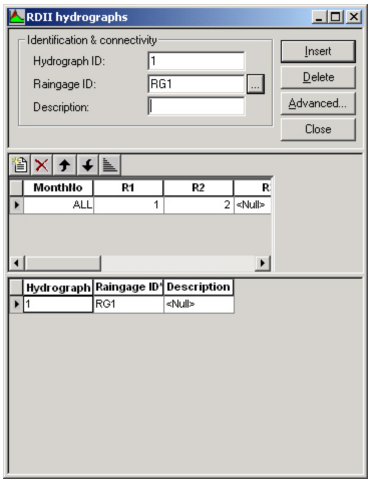


Figure 14.4 RTK - concept

A CS SWMM RDII Hydrograph can contain up to 12 sets of unit hydrographs (one for each month of the year), and each set can consist of up to 3 individual hydrographs (for short-term, intermediate-term, and long-term responses, respectively).

The CS SWMM RDII Hydrographs Editor organizes the related input data for the RDII Unit Hydrograph approach into following groups:

- Identification and Connectivity
- Tabular Data


RDII hydrographs
Identification & connectivity
Insert
Delete
Advanced...
Close

Hydrograph ID: 1

Raingage ID: RG1

Description:

MonthNo	R1	R2	R
ALL	1	2	<Null>

Hydrograph	Raingage ID	Description
1	RG1	<Null>

Figure 14.5 The SWMM RDII Hydrographs Editor

Identification and Connectivity

Table 14.9 The Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Hydrograph ID	ID of the RDII Hydrograph (UH Group)	Yes	MUID
Raingage ID	ID of the Raingage	Yes	RaingageID
Description	User descriptive text	No	Description

Tabular Data

Table 14.10 The Tabular Data Group

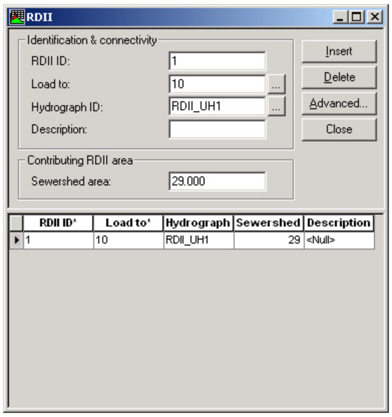
Edit field	Description	Used or required by simulations	Field name in datastructure
MonthNo	Individual Months or all months	Yes	MonthNo
R1	The fraction of rainfall volume that enters the sewer system	Yes	R1
T1	The time from the onset of rainfall to the peak of the UH in hour	Yes	T1
K1	The ratio of time to recession of the UH to the time to peak	Yes	K1
R2	The fraction of rainfall volume that enters the sewer system	Yes	R2
T2	The time from the onset of rainfall to the peak of the UH in hour	Yes	T2
K2	The ratio of time to recession of the UH to the time to peak	Yes	K2
R3	The fraction of rainfall volume that enters the sewer system	Yes	R3
T3	The time from the onset of rainfall to the peak of the UH in hour	Yes	T3
K3	The ratio of time to recession of the UH to the time to peak	Yes	K3
IA_Max1	Short term maximum depth	Yes	IA_Max1
IA_Rec1	Short term recovery rate	Yes	IA_Rec1
IA_Init1	Short term initial depth	Yes	IA_Init1
IA_Max2	Medium term maximum depth	Yes	IA_Max2
IA_Rec2	Medium term recovery rate	Yes	IA_Rec2

Table 14.10 The Tabular Data Group

Edit field	Description	Used or required by simulations	Field name in datastructure
IA_Init2	Medium term initial depth	Yes	IA_Init2
IA_Max3	Long term maximum depth	Yes	IA_Max3
IA_Rec3	Long term recovery rate	Yes	IA_Rec3
IA_Init3	Long term initial depth	Yes	IA_Init3

After defining the RDII Hydrographs it is necessary to connect them to a node of the conveyance system and the area of the surrounding sewershed that contributes RDII flow. This done by the CS SWMM RDII Editor which organizes the related data into following groups:

- Identification and Connectivity
- Contributing RDII area



Identification & connectivity

RDII ID: 1

Load to: 10

Hydrograph ID: RDII_UH1

Description:

Contributing RDII area

Sewershed area: 29.000

RDII ID*	Load to*	Hydrograph	Sewershed	Description
1	10	RDII_UH1	29	<Null>

Figure 14.6 The SWMM RDII Editor

Identification and Connectivity

Table 14.11 The Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
RDII ID	ID of the RDII	Yes	MUID
Load To	ID of Node which recieves RDII	Yes	NodeID

Table 14.11 The Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Hydrograph ID	ID of Hydrograph defined in the CS RDII Hydrographs Editor	Yes	HydrographID
Description	User descriptive text	No	Description

Contributing RDII Area

Table 14.12 The Contributing RDII Area Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Sewershed area	Area that contributes with RDII inflow	Yes	SewerArea

14.6 Aquifers

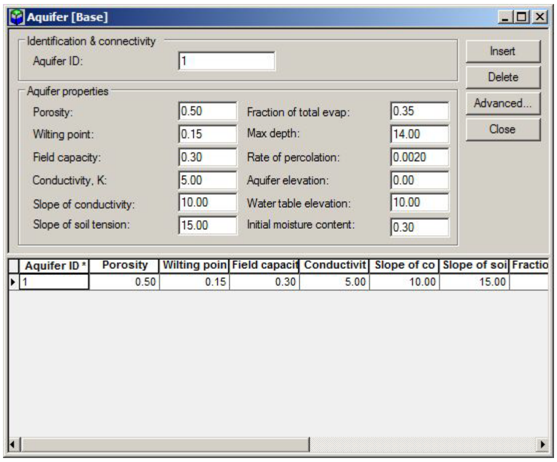
Aquifers are sub-surface groundwater areas used to model the vertical movement of water infiltrating from the catchments which lie above them. They also permit the infiltration of groundwater into the conveyance system, or exfiltration of surface water from the conveyance system, depending on the hydraulic gradient that exists. The same aquifer object can be shared by several catchments.

Aquifers are represented using two zones - an unsaturated zone and a saturated zone. Their behaviour is characterized using such parameters as soil porosity, hydraulic conductivity, evapotranspiration depth, bottom elevation, and loss rate to deep groundwater. In addition, the initial water table elevation and initial moisture content of the unsaturated zone must be supplied.

Aquifers are connected to catchments and to nodes of the conveyance system through the CS SWMM Groundwater Editor.

The CS SWMM Aquifer Editor organizes the related input data for the Aquifer into following groups:

- Identification and Connectivity
- Aquifer Properties



Aquifer ID: 1

Porosity: 0.50
Wilting point: 0.15
Field capacity: 0.30
Conductivity, K: 5.00
Slope of conductivity: 10.00
Slope of soil tension: 15.00
Fraction of total evap: 0.35
Max depth: 14.00
Rate of percolation: 0.0020
Aquifer elevation: 0.00
Water table elevation: 10.00
Initial moisture content: 0.30

Aquifer ID *	Porosity	Wilting poin	Field capacit	Conductivit	Slope of co	Slope of soi	Fractio
1	0.50	0.15	0.30	5.00	10.00	15.00	

Figure 14.7 The SWMM Aquifer Editor

Identification and Connectivity

Table 14.13 The Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Aquifer ID	ID of the Aquifer	Yes	MUID

Aquifer Properties

Table 14.14 The Aquifer Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Porosity	Soil porosity (fraction)	Yes	Por
Wilting Point	Soil wilting point (fraction)	Yes	WP
Field Capacity	Soil field capacity (fraction)	Yes	FC
Conductivity, K	Saturated hydraulic conductivity	Yes	K
Slope of Conductivity	Slope of hydraulic conductivity versus moisture content curve	Yes	Kslope
Slope of soil tension	Slope of soil tension versus moisture content curve	Yes	Yslope
Fraction of total evap.	Fraction of total evaporation available for evapotranspiration	Yes	UEF

Table 14.14 The Aquifer Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Max Depth	Maximum depth into lower saturated zone over which evapotranspiration can occur	Yes	LED
Rate of percolation	Rate of percolation from saturated zone to deep groundwater when water table is at ground surface	Yes	GWR
Aquifer Elevation	Elevation of the bottom of the aquifer	Yes	BE
Water table elevation	Water table elevation at the start of simulation	Yes	WTE
UMC	Unsaturated zone moisture content at the start of simulation	Yes	UMC

14.7 Groundwater

A sketch of the two-zone groundwater model that is used in EPA SWMM is shown in Figure 14.8. The upper zone is unsaturated at a varying moisture content of q. The lower zone is saturated and therefore its moisture content is fixed at the soil porosity h. The fluxes shown in the figure, expressed as volume per unit area per unit time, consist of the following:

- f_I infiltration from the surface
- f_{EU} evapotranspiration from the upper zone which is a fixed fraction of the un-used surface evaporation
- f_U percolation from the upper to lower zone which depends on the upper zone moisture content q and depth d_U
- f_{EL} evapotranspiration from the lower zone, which is a function of the depth of the upper zone d_U
- f_L percolation from the lower zone to deep groundwater which depends on the lower zone depth d_L
- f_G lateral groundwater interflow to the conveyance network which depends on the lower zone depth d_L as well as depths in the receiving channel or node.

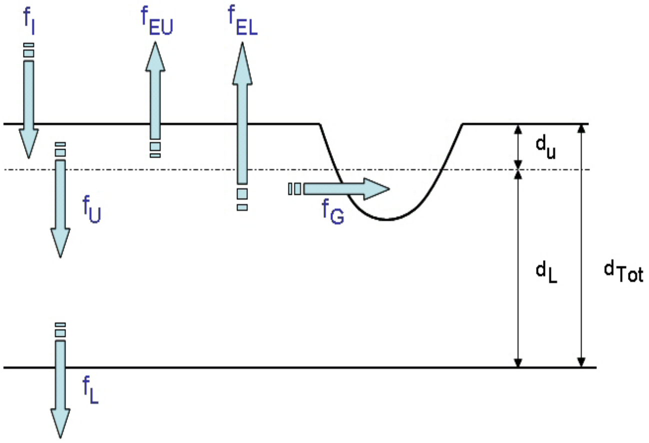
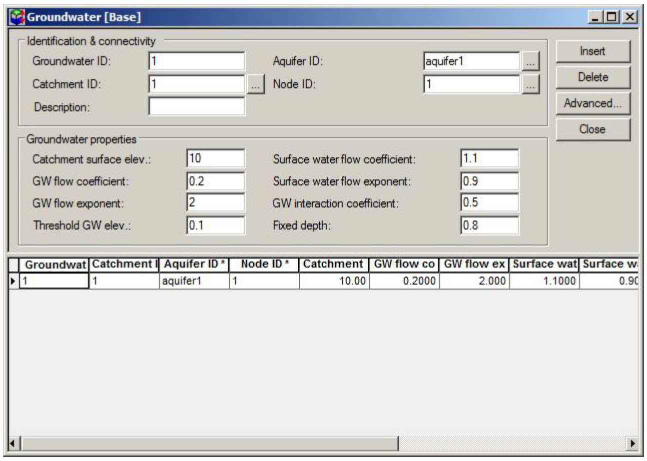


Figure 14.8 Goundwater Modelling Concept

After computing the water fluxes that exist at a given point in time, a mass balance is written for the change in water volume stored in each zone so that a new water table depth and unsaturated zone moisture content can be computed for the next time step.

The CS SWMM Groundwater Editor organizes the related input data for the Groundwater into following groups:

- Identification and Connectivity
- Groundwater Properties



Identification & connectivity

Groundwater ID: 1
Catchment ID: 1
Description:
Aquifer ID: aquifer1
Node ID: 1

Groundwater properties

Catchment surface elev.: 10
GW flow coefficient: 0.2
GW flow exponent: 2
Threshold GW elev.: 0.1
Surface water flow coefficient: 1.1
Surface water flow exponent: 0.9
GW interaction coefficient: 0.5
Fixed depth: 0.8

Groundwat	Catchment I	Aquifer ID *	Node ID *	Catchment	GW flow co	GW flow ex	Surface wat	Surface w
1	1	aquifer1	1	10.00	0.2000	2.000	1.1000	0.9

Figure 14.9 The SWMM Groundwater Editor

Identification and Connectivity

Table 14.15 The Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Groundwater ID	ID of the Groundwater	Yes	MUID
Catchment ID	ID of the Catchment	Yes	SubCatchID
Description	User's descriptive information	No	Description
Aquifer ID	ID of the Aquifer	Yes	AquiferID
Node ID	ID of Node	Yes	NodeID

Groundwater Properties

Table 14.16 The Groundwater Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Catchment Surface Elevation	Surface elevation of Catchment	Yes	SurfElev
GW flow coefficient	Groundwater flow coefficient	Yes	A1
GW flow exponent	Groundwater flow exponent	Yes	B1
Surface water flow coefficient	Surface water flow coefficient	Yes	A2
Surface water flow exponent	Surface water flow exponent	Yes	B2
GW interaction coefficient	Surface water - Groundwater interaction coefficient	Yes	A3
Fixed Depth	Fixed depth of surface water at receiving node	Yes	TW

14.8 Snowpack

The snow melt routine is part of the runoff modelling process. It updates the state of the snow packs associated with each catchment by accounting for snow accumulation, snow redistribution by areal depletion and removal operations, and snow melt via heat budget accounting. Any snow melt coming off the pack is treated as an additional rainfall input into the runoff process.

At each runoff time step the following computations are made:

1. Air temperature and melt coefficients are updated according to the calendar date.

2. Any precipitation that falls as snow is added to the snow pack.

3. Any excess snow depth on the plowable area of the pack is redistributed according to the removal parameters established for the pack.

4. Areal coverages of snow on the impervious and pervious areas of the pack are reduced according to the Areal Depletion Curves defined for the study area.

5. The amount of snow in the pack that melts to liquid water is found using:

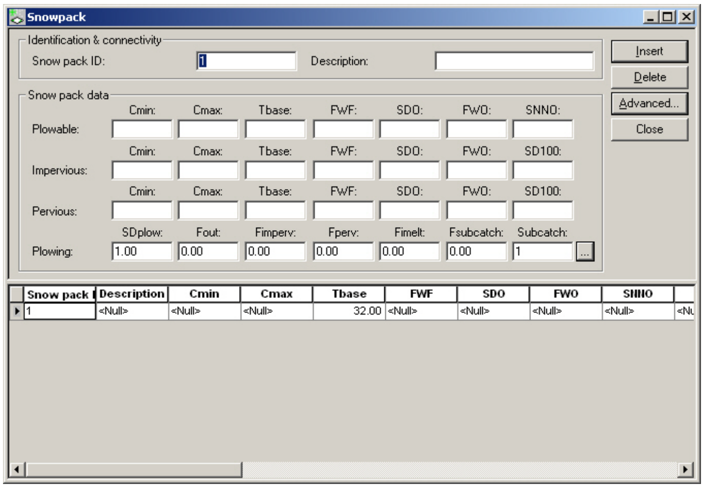
- a heat budget equation for periods with rainfall, where melt rate increases with increasing air temperature, wind speed, and rainfall intensity
- a degree-day equation for periods with no rainfall, where melt rate equals the product of a melt coefficient and the difference between the air temperature and the pack's base melt temperature.

6. If no melting occurs, the pack temperature is adjusted up or down based on the product of the difference between current and past air temperatures and an adjusted melt coefficient. If melting occurs, the temperature of the pack is increased by the equivalent heat content of the melted snow, up to the base melt temperature. Any remaining melt liquid beyond this is available to runoff from the pack.

7. The available snow melt is then reduced by the amount of free water holding capacity remaining in the pack. The remaining melt is treated the same as a direct rainfall input onto the catchment.

The CS SWMM Snowpack Editor and the CS SWMM Climatology Editor (described in next chapter) contains the above information required for characterizing the modelling of the snowfall and snow melt processes. The CS SWMM Snowpack Editor organizes the data into following groups

- Identification and Connectivity
- SnowPack Properties


Snowpack
Identification & connectivity
Description:
Snow pack data
Plowable:
Impervious:
Pervious:
Cmin:
Cmax:
Tbase:
FWF:
SD0:
FW0:
SNN0:
SD100:
Insert
Delete
Advanced...
Close

Snow pack ID: 1

	SDplow:	Fout:	Fimperv:	Fperv:	Fimelt:	Fsubcatch:	Subcatch:
Plowing:	1.00	0.00	0.00	0.00	0.00	0.00	1

Snow pack	Description	Cmin	Cmax	Tbase	FWF	SDO	FWO	SNNO	
1	<Null>	<Null>	<Null>	32.00	<Null>	<Null>	<Null>	<Null>	<Nu

Figure 14.10 The SWMM Snowpack Editor

Identification and Connectivity

Table 14.17 The Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
SnowPackID	ID of the Snowpack	Yes	MUID
Description	User’s descriptive information	No	Description

Snowpack Properties

Table 14.18 The Snowpack Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Plowable			
Cmin	Minimum melt coefficient	Yes	Cmin1
Cmax	Maximum melt coefficient	Yes	Cmax1
Tbase	Snow melt base temperature	Yes	Tbase1
FWF	Ratio of free water holding the capacity of snow depth	Yes	Fwf1
SD0	Initial snow depth	Yes	Sd01
FW0	Initial free water in snow pack	Yes	Fw01
SNN0	Fraction of impervious area with depression storage that can be plowed	Yes	Snn0

Table 14.18 The Snowpack Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Impervious			
Cmin	Minimum melt coefficient	Yes	Cmin1
Cmax	Maximum melt coefficient	Yes	Cmax1
Tbase	Snow melt base temperature	Yes	Tbase1
FWF	Ratio of free water holding the capacity of snow depth	Yes	Fwf1
SD0	Initial snow depth	Yes	Sd01
FW0	Initial free water in snow pack	Yes	Fw01
SD100	Snow depth above which there is 100% cover	Yes	SD1002
Pervious			
Cmin	Minimum melt coefficient	Yes	Cmin1
Cmax	Maximum melt coefficient	Yes	Cmax1
Tbase	Snow melt base temperature	Yes	Tbase1
FWF	Ratio of free water holding the capacity of snow depth	Yes	Fwf1
SD0	Initial snow depth	Yes	Sd01
FW0	Initial free water in snow pack	Yes	Fw01
SD100	Snow depth above which there is 100% cover	Yes	SD1003
Plowing			
SDplow	Depth of snow on plowable area at which redistribution through plowing occurs	Yes	SDplow
Fout	Fraction of excess snow on plowable area transferred out of watershed	Yes	Fout
Fimperv	Fraction of excess snow on plowable area transferred to impervious area by plowing	Yes	Fimperv
Fperv	Fraction of excess snow on plowable area transferred to pervious area by plowing	Yes	Fperv
Fimelt	Fraction of excess snow on plowable area converted into imeediate melt	Yes	Fimelt

Table 14.18 The Snowpack Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Fsubcatch	Fraction of excess snow on plowable area transferred to pervious area in another catchment	Yes	Fsubcatch
Subcatch	ID of catchment recieving the Fsubcatch fraction of transferred snow	Yes	SubcatchID

14.9 Climatology

Climate-related variables used for computing runoff and snow melt are grouped in the Climatology object:

- Temperature
- Evaporation
- Wind Speed
- Snow Melt
- Areal Depletion

Temperature data are used when simulating snowfall and snow melt processes during runoff calculations. If these processes are not being simulated then no temperature data are needed.

Temperature data can be supplied from one of the following sources:

- a user-supplied time series of point values (values at intermediate times are found by interpolation)
- a NCDC 3200 or 3210 climatological file containing daily minimum and maximum values (SWMM fits a sinusoidal curve through these values depending on the day of the year).

For user-supplied time series, temperatures are in degrees F for US units and degrees C for metric units. Note that the NCDC climatological file can also be used to supply evaporation and wind speed as well.

Evaporation can occur for standing water on catchment surfaces, for subsurface water in groundwater aquifers, and from water held in storage units. Evaporation rates can be stated as:

- a single constant value
- a set of monthly average values
- a user-supplied time series of daily values
- daily values read from a NCDC climatological file

If a NCDC file is used then a set of monthly pan coefficients should also be supplied to convert the pan evaporation data to free-water-surface values.

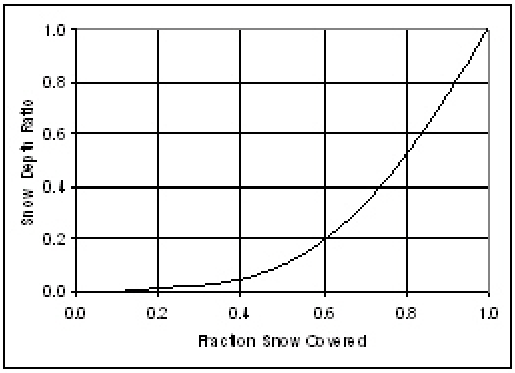
Wind speed is an optional climatological variable that is only used for snow melt calculations. There can be use either a set of monthly average speeds or wind speed data contained in the same NCDC climatological file used for daily min/max temperatures.

Snow melt parameters are climatological variables that apply across the entire model area when simulating snowfall and snow melt. They include:

- the temperature at which precipitation falls as snow
- heat exchange properties of the snow surface
- study area elevation, latitude, and longitude correction

For additional information of the Snow fall and Snow melt processes please consult the chapter Snowpack.

Areal Depletion refers to the tendency of snow to melt non-uniformly over the surface of a catchment. As the melting process proceeds, the area covered by snow is reduced. This behaviour can be described by an *Areal Depletion Curve*. It plots the fraction of total area that remains snow covered against the ratio of the actual snow depth to the depth at which there is 100% snow cover. A typical ADC for a natural area is shown below.



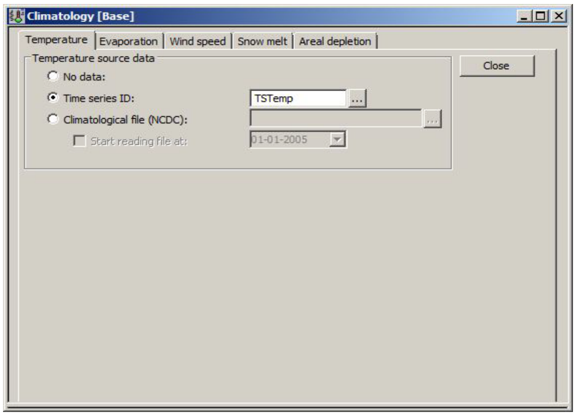
Two such curves can be supplied, one for impervious areas and another for pervious areas.

The CS SWMM Climatology Editor organizes these data into following tabsheets and groups:

- Temperature Tabsheet
- Evaporation Tabsheet
- Wind Speed Tabsheet

- Snow melt Tabsheet
- Areal Depletion Tabsheet

Temperature data



Climatology [Base]
Temperature
Evaporation
Wind speed
Snow melt
Areal depletion
Close

Temperature source data

○ No data:

● Time series ID: TSTemp

○ Climatological file (NCDC):

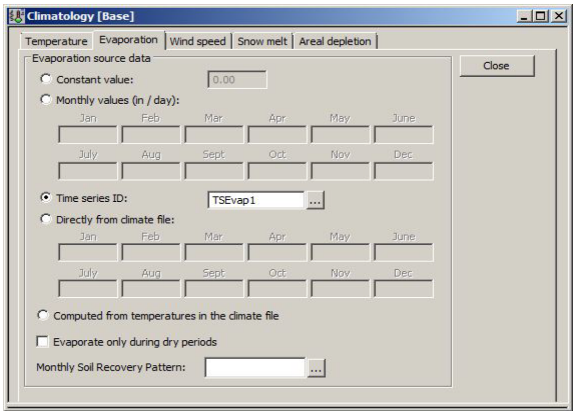
☐ Start reading file at: 01-01-2005

Figure 14.11 The SWMM Climatology | Temperature Editor

Table 14.19 The Climatology | Temperature Tabsheet

Edit field	Description	Used or required by simulations	Field name in datastructure
	Radio Button Types	No	TypeNo
No data	TypeNo=1, No temperature data	Yes if TypeNo = 1	
Time series ID	TypeNo=2, Temperature data read from Time Series	Yes if TypeNo = 2	TimeSeriesID
Climatological file (NCDC)	TypeNo=3, Temperature data read from NCDC file	Yes if TypeNo = 3	Filename
Start date	TypeNo=3, Start date for reading from NCDC data file	Yes if TypeNo = 3	Start

Evaporation Data



Evaporation source data

○ Constant value: 0.00

○ Monthly values (in / day):

● Time series ID: TSEvap1

○ Directly from climate file:

○ Computed from temperatures in the climate file

☐ Evaporate only during dry periods

Monthly Soil Recovery Pattern:

Figure 14.12 The SWMM Climatology | Evaporation Editor

Table 14.20 The Climatology | Evaporation Tabsheet

Edit field	Description	Used or required by simulations	Field name in datastructure
	Radio button types.	Yes	TypeNo
Climatological file (NCDC) selection at temperature file	Evaporation data are read from the NCDC data file which is specified on the Temperature Tabsheet	Yes, if TypeNo = 4	Filename
Constant Value	Evaporation is set to constant value	Yes if TypeNo = 1	ConstVale
Jan	Monthly evaporation data for January entered by user	Yes if TypeNo = 2	Value1
Feb	Monthly evaporation data for February entered by user	Yes if TypeNo = 2	Value2
Mar	Monthly evaporation data for March entered by user	Yes if TypeNo = 2	Value3
Apr	Monthly evaporation data for April entered by user	Yes if TypeNo = 2	Value4
May	Monthly evaporation data for May entered by user	Yes if TypeNo = 2	Value5
June	Monthly evaporation data for June entered by user	Yes if TypeNo = 2	Value6
July	Monthly evaporation data for July entered by user	Yes if TypeNo = 2	Value7
Aug	Monthly evaporation data for August entered by user	Yes if TypeNo = 2	Value8

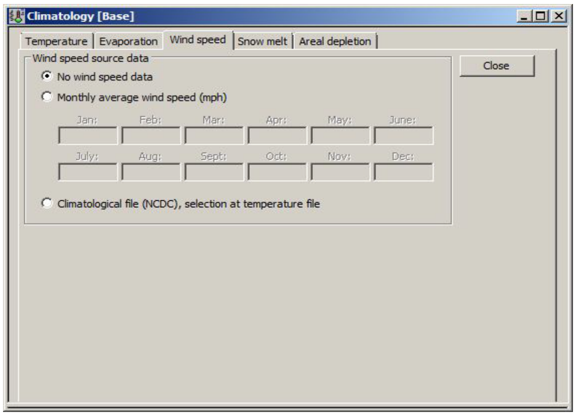
Table 14.20 The Climatology | Evaporation Tabsheet

Edit field	Description	Used or required by simulations	Field name in datastructure
Sept	Monthly evaporation data for September entered by user	Yes if TypeNo = 2	Value9
Oct	Monthly evaporation data for October entered by user	Yes if TypeNo = 2	Value10
Nov	Monthly evaporation data for November entered by user	Yes if TypeNo = 2	Value11
Dec	Monthly evaporation data for December entered by user	Yes if TypeNo = 2	Value12
Time series ID	Evaporation data read from Time Series	Yes if TypeNo = 3	TimeSeriesID
Jan	Evaporation data for January read from NCDC file	Yes if TypeNo = 4	Pan1
Feb	Evaporation data for February read from NCDC file	Yes if TypeNo = 4	Pan2
Mar	Evaporation data for March read from NCDC file	Yes if TypeNo = 4	Pan3
Apr	Evaporation data for April read from NCDC file	Yes if TypeNo = 4	Pan4
May	Evaporation data for May read from NCDC file	Yes if TypeNo = 4	Pan5
June	Evaporation data for June read from NCDC file	Yes if TypeNo = 4	Pan6
July	Evaporation data for July read from NCDC file	Yes if TypeNo = 4	Pan7
Aug	Evaporation data for August read from NCDC file	Yes if TypeNo = 4	Pan8
Sept	Evaporation data for September read from NCDC file	Yes if TypeNo = 4	Pan9
Oct	Evaporation data for October read from NCDC file	Yes if TypeNo = 4	Pan10
Nov	Evaporation data for November read from NCDC file	Yes if TypeNo = 4	Pan11
Dec	Evaporation data for December read from NCDC file	Yes if TypeNo = 4	Pan12
Computed from temperatures in the climate files	Evaporation calculated based on temperature in the temperature file	Yes if TypeNo = 5	Defined at temperature tab in climatology dialog

Table 14.20 The Climatology | Evaporation Tabsheet

Edit field	Description	Used or required by simulations	Field name in datastructure
Monthly Soil Recovery Pattern	Monthly infiltration recovery rate	All	SoilRecoveryPatternID
Evaporate only during dry periods	Evaporation occur only during dry periods	All	DryOnlyNo

Windspeed data


Climatology [Base]
Temperature
Evaporation
Wind speed
Snow melt
Areal depletion
Close
Jan:
Feb:
Mar:
Apr:
May:
June:
July:
Aug:
Sept:
Oct:
Nov:
Dec:

Wind speed source data

● No wind speed data

○ Monthly average wind speed (mph)

○ Climatological file (NCDC), selection at temperature file

Figure 14.13 The SWMM Climatology | Wind Speed Editor

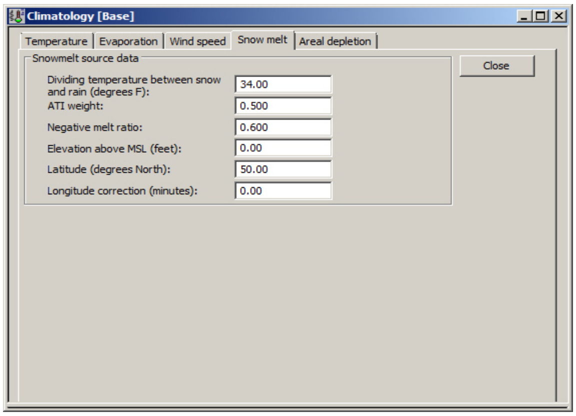
Table 14.21 The Climatology | WindSpeed Tabsheet

Edit field	Description	Used or required by simulations	Field name in datastructure
	Radio button types.	Yes	WindSpeedTypeNo
No WindSpeed data	No Windspeed data should be used	No, if WindSpeedTypeNo = 1 (NONE)	

Table 14.21 The Climatology | WindSpeed Tabsheet

Edit field	Description	Used or required by simulations	Field name in datastructure
Climatological file (NCDC) selection at temperature file	WindSpeed data are read from the NCDC data file which is specified on the Temperature Tabsheet	Yes, if WindSpeedTypeNo = 3 (FILE)	FileName
Jan	Monthly evaporation data for January entered by user	Yes, if WindSpeedTypeNo 2	Value1
Feb	Monthly evaporation data for February entered by user	Yes, if WindSpeedTypeNo 2	Value2
Mar	Monthly evaporation data for March entered by user	Yes, if WindSpeedTypeNo 2	Value3
Apr	Monthly evaporation data for April entered by user	Yes, if WindSpeedTypeNo 2	Value4
May	Monthly evaporation data for May entered by user	Yes, if WindSpeedTypeNo 2	Value5
June	Monthly evaporation data for June entered by user	Yes, if WindSpeedTypeNo 2	Value6
July	Monthly evaporation data for July entered by user	Yes, if WindSpeedTypeNo 2	Value7
Aug	Monthly evaporation data for August entered by user	Yes, if WindSpeedTypeNo 2	Value8
Sept	Monthly evaporation data for September entered by user	Yes, if WindSpeedTypeNo 2	Value9
Oct	Monthly evaporation data for October entered by user	Yes, if WindSpeedTypeNo 2	Value10
Nov	Monthly evaporation data for November entered by user	Yes, if WindSpeedTypeNo 2	Value11
Dec	Monthly evaporation data for December entered by user	Yes, if WindSpeedTypeNo 2	Value12

Snowmelt data



Snowmelt source data

Dividing temperature between snow and rain (degrees F): 34.00
ATI weight: 0.500
Negative melt ratio: 0.600
Elevation above MSL (feet): 0.00
Latitude (degrees North): 50.00
Longitude correction (minutes): 0.00

Figure 14.14 The SWMM Climatology | Snow Melt Editor

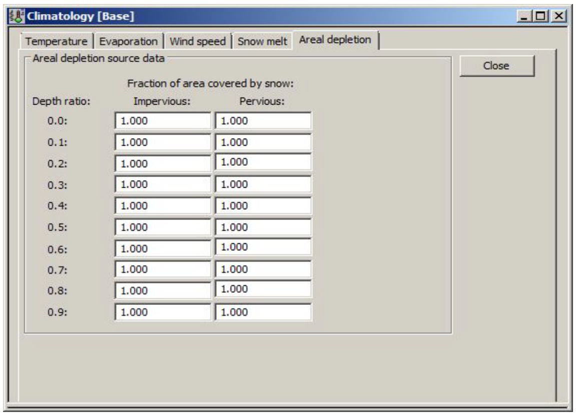
Table 14.22 The Climatology | Snowmelt Tabsheet

Edit field	Description	Used or required by simulations	Field name in datastructure
Dividing temperature between snow and rain (degrees F)	Temperature below which precipitation falls as snow	Yes	SnowTemp
ATI Weight	This parameter reflects to what degree heat transfer within a snow pack during non-melt periods is affected by prior air temperatures. Smaller values reflect a thicker surface layer of snow which result in reduced rates of heat transfer. Values must be between 0 and 1.The default value is 0.5.	Yes	Atiwt
Negative melt ratio	This is the ratio of the heat transfer coefficient of a snow pack during non-melt conditions to the coefficient during melt conditions. It must be a number between 0 and 1. The default value is 0.6.	Yes	Rnm

Table 14.22 The Climatology | Snowmelt Tabsheet

Edit field	Description	Used or required by simulations	Field name in datastructure
Elevation above MSL (feet)	Enter the average elevation above mean sea level for the study area, in feet or meters. This value is used to provide a more accurate estimate of atmospheric pressure. The default is 0.0, which results in a pressure of 29.9 inches Hg. The effect of wind on snow melt rates during rainfall periods is greater at higher pressures, which occur at lower elevations.	Yes	Elev
Latitude (degrees North)	Enter the latitude, in degrees North, of the study area. This number is used when computing the hours of sunrise and sunset, which in turn are used to extend min/max daily temperatures into continuous values. The default is 50 degrees North.	Yes	Lat
Longitude correction (minutes)	This is a correction, in minutes of time, between true solar time and the time on clocks. It depends on a location's longitude (q) and the standard meridian of its time zone (SM) through the expression 4 (q-SM). This correction is used to adjust the hours of sunrise and sunset when extending daily min/max temperatures into continuous values. The default value is 0	Yes	Dtlong

Areal depletion data



Areal depletion source data

Fraction of area covered by snow:

Depth ratio:	Impervious:	Pervious:
0.0:	1.000	1.000
0.1:	1.000	1.000
0.2:	1.000	1.000
0.3:	1.000	1.000
0.4:	1.000	1.000
0.5:	1.000	1.000
0.6:	1.000	1.000
0.7:	1.000	1.000
0.8:	1.000	1.000
0.9:	1.000	1.000

Figure 14.15 The SWMM Climatology | Areal Depletion Editor

Table 14.23 The Climatology | Areal depletion Tabsheet

Edit field	Description	Used or required by simulations	Field name in datastructure
Imervious			
0.0	Fraction of impervious area covered by snow at interval [0.0-0.1]	Yes	AdcImPerv0
0.1	Fraction of impervious area covered by snow at interval]0.1-0.2]	Yes	AdcImPerv1
0.2	Fraction of impervious area covered by snow at interval]0.2-0.3]	Yes	AdcImPerv2
0.3	Fraction of impervious area covered by snow at interval]0.3-0.4]	Yes	AdcImPerv3
0.4	Fraction of impervious area covered by snow at interval]0.4-0.5]	Yes	AdcImPerv4
0.5	Fraction of impervious area covered by snow at interval]0.5-0.6]	Yes	AdcImPerv5
0.6	Fraction of impervious area covered by snow at interval]0.6-0.7]	Yes	AdcImPerv6
0.7	Fraction of impervious area covered by snow at interval]0.7-0.8]	Yes	AdcImPerv7

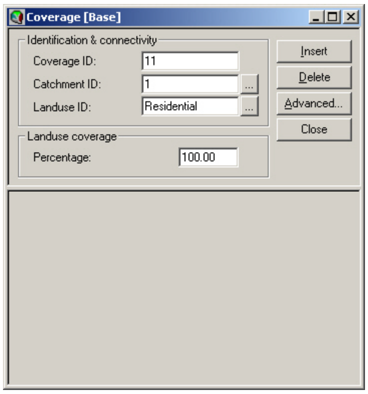
Table 14.23 The Climatology | Areal depletion Tabsheet

Edit field	Description	Used or required by simulations	Field name in datastructure
0.8	Fraction of impervious area covered by snow at interval]0.8-0.9]	Yes	AdcImPerv8
0.9	Fraction of impervious area covered by snow at interval]0.9-1.0]	Yes	AdcImPerv9
	Pervious		
0.0	Fraction of pervious area covered by snow at interval [0.0-0.1]	Yes	AdcPerv0
0.1	Fraction of pervious area covered by snow at interval]0.1-0.2]	Yes	AdcPerv1
0.2	Fraction of pervious area covered by snow at interval]0.2-0.3]	Yes	AdcPerv2
0.3	Fraction of pervious area covered by snow at interval]0.3-0.4]	Yes	AdcPerv3
0.4	Fraction of pervious area covered by snow at interval]0.4-0.5]	Yes	AdcPerv4
0.5	Fraction of pervious area covered by snow at interval]0.5-0.6]	Yes	AdcPerv5
0.6	Fraction of pervious area covered by snow at interval]0.6-0.7]	Yes	AdcPerv6
0.7	Fraction of pervious area covered by snow at interval]0.7-0.8]	Yes	AdcPerv7
0.8	Fraction of pervious area covered by snow at interval]0.8-0.9]	Yes	AdcPerv8
0.9	Fraction of pervious area covered by snow at interval]0.9-1.0]	Yes	AdcPerv9

14.10 Coverage

The CS SWMM Coverage Editor organizes the related input data for the coverage into following groups:

- Identification and Connectivity
- Landuse Coverage



Coverage [Base]

Identification & connectivity

Coverage ID: 11

Catchment ID: 1

Landuse ID: Residential

Landuse coverage

Percentage: 100.00

Figure 14.16 The SWMM Coverage Editor Dialog

Identification and Connectivity

Table 14.24 The Coverage Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
CoverageID	ID of Coverage	Yes	MUID
Catchments	ID of Catchment	Yes	SubCatchID
LanduseID	ID of Landuse	Yes	LandUseID

Landuse coverage

Table 14.25 The Landuse Coverage Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Percentage	Percentage of the catchment area covered by this type of landuse	Yes	Percentage

14.11 LID Controls

LID Controls are low impact development practices designed to capture surface runoff and provide some combination of detention, infiltration, and evapotranspiration to it.

They are considered as properties of a given sub catchment, similar to how Aquifers and Snow Packs are treated. SWMM can explicitly model five different generic types of LID controls:

Bio-retention Cells

Bio-retention Cells are depressions that contain vegetation grown in an engineered soil mixture placed above a gravel drainage bed. They provide storage, infiltration and evaporation of both direct rainfall and runoff captured from surrounding areas. Rain gardens, street planters, and green roofs are all variations of bio-retention cells.

Infiltration Trenches

Infiltration Trenches are narrow ditches filled with gravel that intercept runoff from upslope impervious areas. They provide storage volume and additional time for captured runoff to infiltrate the native soil below.

Continuous Porous Pavement

Continuous Porous Pavement systems are excavated areas filled with gravel and paved over with a porous concrete or asphalt mix. Normally all rainfall will immediately pass through the pavement into the gravel storage layer below it where it can infiltrate at natural rates into the site's native soil. Block Paver systems consist of impervious paver blocks placed on a sand or pea gravel bed with a gravel storage layer below. Rainfall is captured in the open spaces between the blocks and conveyed to the storage zone and native soil below.

Rain Barrels

Rain Barrels (or Cisterns) are containers that collect roof runoff during storm events and can either release or re-use the rainwater during dry periods.

Vegetative Swales

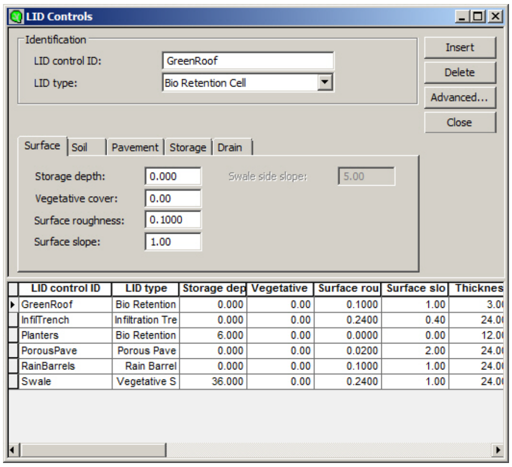
Vegetative Swales are channels or depressed areas with sloping sides covered with grass and other vegetation. They slow down the conveyance of collected runoff and allow it more time to infiltrate the native soil beneath it.

Bio-retention cells, infiltration trenches, and porous pavement systems can all contain optional underdrain systems in their gravel storage beds to convey captured runoff off of the site rather than letting it all infiltrate. They can also have an impermeable floor or liner that prevents any infiltration into the native soil from occurring. Infiltration trenches and porous pavement systems can also be subjected to a decrease in hydraulic conductivity over time due to clogging.

Although some LID practices can also provide significant pollutant reduction benefits, at this time SWMM only models their hydrologic performance

The CS SWMM LID Controls Editor organizes the data input for the different types of LID practices. The input data is organized into following group and tabs:

- Identification
- Tabs: Surface, Soil, Pavement, Storage, Drain and Pollutant Removals

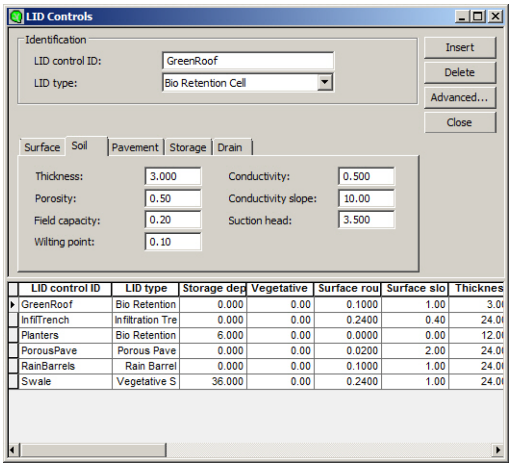

LID Controls
Identification
Insert
Delete
Advanced...
Close
Surface
Soil
Pavement
Storage
Drain

LID control ID: GreenRoof
LID type: Bio Retention Cell

Storage depth: 0.000
Vegetative cover: 0.00
Surface roughness: 0.1000
Surface slope: 1.00
Swale side slope: 5.00

LID control ID	LID type	Storage dep	Vegetative	Surface rou	Surface slo	Thicknes
GreenRoof	Bio Retention	0.000	0.00	0.1000	1.00	3.0
InfilTrench	Infiltration Tre	0.000	0.00	0.2400	0.40	24.0
Planters	Bio Retention	6.000	0.00	0.0000	0.00	12.0
PorousPave	Porous Pave	0.000	0.00	0.0200	2.00	24.0
RainBarrels	Rain Barrel	0.000	0.00	0.1000	1.00	24.0
Swale	Vegetative S	36.000	0.00	0.2400	1.00	24.0

Figure 14.17 The CS SWMM LID Controls Editor - Surface

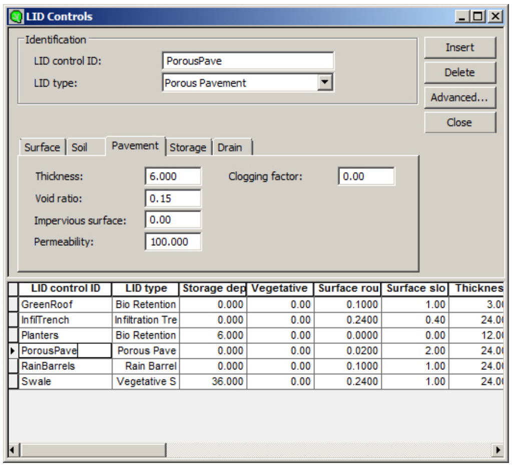

LID Controls
Identification
Insert
Delete
Advanced...
Close
Surface
Soil
Pavement
Storage
Drain

LID control ID: GreenRoof
LID type: Bio Retention Cell

Thickness: 3.000
Porosity: 0.50
Field capacity: 0.20
Wilting point: 0.10
Conductivity: 0.500
Conductivity slope: 10.00
Suction head: 3.500

LID control ID	LID type	Storage dep	Vegetative	Surface rou	Surface slo	Thicknes
GreenRoof	Bio Retention	0.000	0.00	0.1000	1.00	3.0
InfilTrench	Infiltration Tre	0.000	0.00	0.2400	0.40	24.0
Planters	Bio Retention	6.000	0.00	0.0000	0.00	12.0
PorousPave	Porous Pave	0.000	0.00	0.0200	2.00	24.0
RainBarrels	Rain Barrel	0.000	0.00	0.1000	1.00	24.0
Swale	Vegetative S	36.000	0.00	0.2400	1.00	24.0

Figure 14.18 The CS SWMM LID Controls Editor - Soil

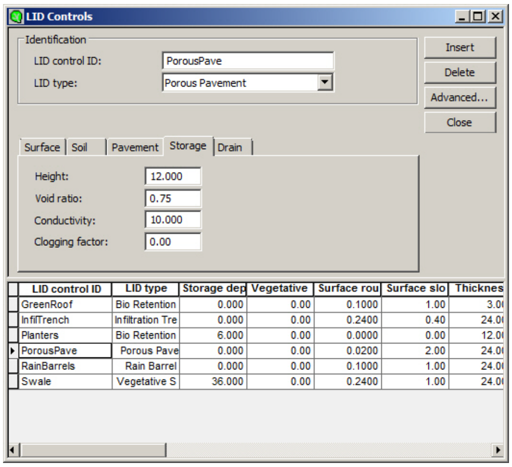

LID Controls
Identification
Insert
Delete
Advanced...
Close
Surface
Soil
Pavement
Storage
Drain

LID control ID: PorousPave
LID type: Porous Pavement

Thickness: 6.000
Clogging factor: 0.00
Void ratio: 0.15
Impervious surface: 0.00
Permeability: 100.000

LID control ID	LID type	Storage dep	Vegetative	Surface rou	Surface slo	Thicknes
GreenRoof	Bio Retention	0.000	0.00	0.1000	1.00	3.0
InfilTrench	Infiltration Tre	0.000	0.00	0.2400	0.40	24.0
Planters	Bio Retention	6.000	0.00	0.0000	0.00	12.0
PorousPave	Porous Pave	0.000	0.00	0.0200	2.00	24.0
RainBarrels	Rain Barrel	0.000	0.00	0.1000	1.00	24.0
Swale	Vegetative S	36.000	0.00	0.2400	1.00	24.0

Figure 14.19 The CS SWMM LID Controls Editor - Pavement

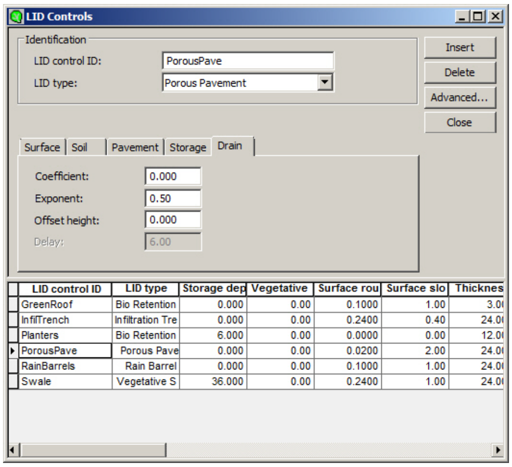

LID Controls
Identification
Insert
Delete
Advanced...
Close
Surface
Soil
Pavement
Storage
Drain

LID control ID: PorousPave
LID type: Porous Pavement

Height: 12.000
Void ratio: 0.75
Conductivity: 10.000
Clogging factor: 0.00

LID control ID	LID type	Storage dep	Vegetative	Surface rou	Surface slo	Thicknes
GreenRoof	Bio Retention	0.000	0.00	0.1000	1.00	3.0
InfilTrench	Infiltration Tre	0.000	0.00	0.2400	0.40	24.0
Planters	Bio Retention	6.000	0.00	0.0000	0.00	12.0
PorousPave	Porous Pave	0.000	0.00	0.0200	2.00	24.0
RainBarrels	Rain Barrel	0.000	0.00	0.1000	1.00	24.0
Swale	Vegetative S	36.000	0.00	0.2400	1.00	24.0

Figure 14.20 The CS SWMM LID Controls Editor - Storage



LID control ID: PorousPave
LID type: Porous Pavement

Coefficient: 0.000
Exponent: 0.50
Offset height: 0.000
Delay: 6.00

LID control ID	LID type	Storage dep	Vegetative	Surface rou	Surface slo	Thicknes
GreenRoof	Bio Retention	0.000	0.00	0.1000	1.00	3.0
InfilTrench	Infiltration Tre	0.000	0.00	0.2400	0.40	24.0
Planters	Bio Retention	6.000	0.00	0.0000	0.00	12.0
PorousPave	Porous Pave	0.000	0.00	0.0200	2.00	24.0
RainBarrels	Rain Barrel	0.000	0.00	0.1000	1.00	24.0
Swale	Vegetative S	36.000	0.00	0.2400	1.00	24.0

Figure 14.21 The CS SWMM LID Controls Editor - Drain

The Pollutant Removal page of the LID Control Editor allows one to specify the degree to which pollutants are removed by an LID control unit as seen by the flow leaving through its underdrain system. Hence it only applies to LID practices that contain an underdrain (bio-retention cells,permeable pavement, infiltration trenches, and rain barrels).

The page contains a data entry grid with the chosen pollutants names and the percent removal which is treated by the LID unit (second editable column). The default status is blank which is interpreted as the percent removal being 0.

The removals specified on this tab are applied to the unit's underdrain when it sends flow onto either a subcatchment or into a conveyance system node. They do not apply to any surface flow that leaves the LID unit.

As an example (taken from EPA SWMM5.1 Help manual), if the runoff treated by the LID unit had a TSS concentration of 100 mg/L and a removal percentage of 90, then if 5 cfs flowed from its drain into a conveyance system node the mass loading contribution to the node would be 100 x (100 - 90) x 5 x (28.3 L/ft3) = 1,415 mg/sec. If in addition the unit had a surface outflow of 1 cfs into the same node, the mass loading from this flow stream would be 100 x 1 x 28.3 = 2,830 mg/sec.

Identification

Table 14.26 The LID Controls Identification Group

Edit field	Description	Used or required by simulations	Field name in datastructure
LID Control ID	ID of Low Impact Design, LID	Yes	MUID
LID Type	Type of LID 1: Bio Retention Cell 2: Porous Pavement 3: Infiltration Trench 4: Rain Barrel 5: Vegetatvie Swale	Yes	LIDTypeNo

LID control data specification

Table 14.27 The LID data specification

Edit field	Description	Used or required by simulations	Field name in datastructure
Storage Depth	When confining walls or berms are present this is the maximum depth to which water can pond above the surface of the unit before overflow occurs (in inches or mm). For LIDs that experience overland flow it is the height of any surface depression storage. For swales, it is the height of its trapezoidal cross section.	Yes If LID-TypeNo = (1,2,3,5)	StorHt
Vegetative Cover Fraction	The fraction of the storage area above the surface that is filled with vegetation	Yes If LID-TypeNo = (1,2,3,5)	VegFrac
Surface Roughness	Manning's n for overland flow over the surface of porous pavement or a vegetative swale (see this table for suggested values). Use 0 for other types of LIDs.	Yes If LID-TypeNo = (1,2,3,5)	Rough
Surface Slope	Slope of porous pavement surface or vegetative swale (percent). Use 0 for other types of LIDs	Yes If LID-TypeNo = (1,2,3,5)	Slope

Table 14.27 The LID data specification

Edit field	Description	Used or required by simulations	Field name in datastructure
Swale Side Slope	Slope (run over rise) of the side walls of a vegetative swale's cross section. This value is ignored for other types of LIDs	Yes If LID-TypeNo = 5	Xslope
Thickness	The thickness of the soil layer (inches or mm). Typical values range from 18 to 36 inches (450 to 900 mm) for rain gardens, street planters and other types of land-based bio-retention units, but only 3 to 6 inches (75 to 150 mm) for green roofs.	Yes If LID-TypeNo = 1	SThick
Porosity	The volume of pore space relative to total volume of soil (as a fraction).	Yes If LID-TypeNo = 1	Por
Field Capacity	Volume of pore water relative to total volume after the soil has been allowed to drain fully (as a fraction). Below this level, vertical drainage of water through the soil layer does not occur.	Yes If LID-TypeNo = 1	FC
Wilting Point	Volume of pore water relative to total volume for a well dried soil where only bound water remains (as a fraction). The moisture content of the soil cannot fall below this limit.	Yes If LID-TypeNo = 1	WP
Conductivity	Hydraulic conductivity for the fully saturated soil (in/hr or mm/hr)	Yes If LID-TypeNo = 1	Ksat
Conductivity Slope	Slope of the curve of log(conductivity) versus soil moisture content (dimensionless). Typical values range from 5 for sands to 15 for silty clay	Yes If LID-TypeNo = 1	Kcoeff
Suction Head	The average value of soil capillary suction along the wetting front (inches or mm). This is the same parameter as used in the Green-Ampt infiltration model	Yes If LID-TypeNo = 1	Suct

Table 14.27 The LID data specification

Edit field	Description	Used or required by simulations	Field name in datastructure
Thickness	The thickness of the pavement layer (inches or mm). Typical values are 4 to 6 inches (100 to 150 mm)	Yes If LID-TypeNo = 2	PThick
Void Ratio	The volume of void space relative to the volume of solids in the pavement for continuous systems or for the fill material used in modular systems. Typical values for pavements are 0.12 to 0.21. Note that porosity = void ratio / (1 + void ratio).	Yes If LID-TypeNo = 2	PVratio
Impervious Surface	Ratio of impervious paver material to total area for modular systems; 0 for continuous porous pavement systems	Yes If LID-TypeNo = 2	FracImp
Permeability	Permeability of the concrete or asphalt used in continuous systems or hydraulic conductivity of the fill material (gravel or sand) used in modular systems (in/hr or mm/hr). The permeability of new porous concrete or asphalt is very high (e.g., hundreds of in/hr) but can drop off over time due to clogging by fine particulates in the runoff	Yes If LID-TypeNo = 2	Perm
Clogging Factor	Number of pavement layer void volumes of runoff treated it takes to completely clog the pavement. Use a value of 0 to ignore clogging. Clogging progressively reduces the pavement's permeability in direct proportion to the cumulative volume of runoff treated.	Yes If LID-TypeNo = 2	PVclog
Height	This is the height of a rain barrel or thickness of a gravel layer (inches or mm). Crushed stone and gravel layers are typically 6 to 18 inches (150 to 450 mm) thick while single family home rain barrels range in height from 24 to 36 inches (600 to 900 mm).	Yes If LID-TypeNo = (1,2,3,4)	Height

Table 14.27 The LID data specification

Edit field	Description	Used or required by simulations	Field name in datastructure
Void Ratio	The volume of void space relative to the volume of solids in the layer. Typical values range from 0.5 to 0.75 for gravel beds. Note that porosity = void ratio / (1 + void ratio).	Yes If LID-TypeNo = (1,2,3)	SVratio
Conductivity	The maximum rate at which water can flow out the bottom of the layer after it is first constructed (in/hr or mm/hr).	Yes If LID-TypeNo = (1,2,3)	Filt
Clogging Factor	Total volume of treated runoff it takes to completely clog the bottom of the layer divided by the void volume of the layer	Yes If LID-TypeNo = (1,2,3)	SVclog
Apply		Yes	DrainNo
Coefficient	Coefficient *C* and exponent *n* that determines the rate of flow through the underdrain as a function of height of stored water above the drain height.	Yes If LID-TypeNo = (1,2,3,4)	Coeff
Exponent	Coefficient *C* and exponent *n* that determines the rate of flow through the underdrain as a function of height of stored water above the drain height.	Yes If LID-TypeNo = (1,2,3,4)	Expon
Offset Height	Height of any underdrain piping above the bottom of a storage layer or rain barrel (inches or mm).	Yes If LID-TypeNo = (1,2,3,4)	Offset
Delay	The number of dry weather hours that must elapse before the drain line in a rain barrel is opened (the line is assumed to be closed once rainfall begins).	Yes If LID-TypeNo = 4	Delay
Open Level	The height in the drain´s storage Layer that causes the drain to automatically open when the water level rises above it. Default is 0 which indicates disabled	No	OpenLev

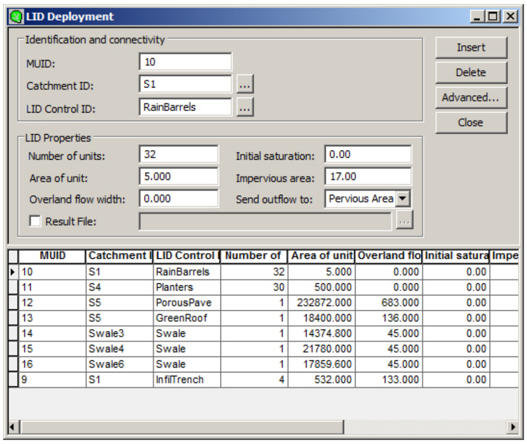
Table 14.27 The LID data specification

Edit field	Description	Used or required by simulations	Field name in datastructure
Closed Level	The height in the drain´s storage level that causes the drain to automatically close when the water level falls below it. Default is 0	No	ClosedLev
Control Curve	The name of an optional control curve that adjust the computed drain flow as function of the head of water above the drain. Leave blank if not applicable	No	ControlCurveID

14.12 LID Deployment

In the CS SWMM LID Deployment Editor the LID controls are deployed to individual catchments. The type of LID control and number are defined as well as the properties for each LID control. The input data is organized into following groups:

- Identification and Connectivity
- LID Properties



Identification and connectivity

MUID: 10
Catchment ID: S1
LID Control ID: RainBarrels

LID Properties

Number of units: 32
Area of unit: 5.000
Overland flow width: 0.000
Initial saturation: 0.00
Impervious area: 17.00
Send outflow to: Pervious Area
☐ Result File:

MUID	Catchment I	LID Control I	Number of	Area of unit	Overland flo	Initial satura	Impe
10	S1	RainBarrels	32	5.000	0.000	0.00	
11	S4	Planters	30	500.000	0.000	0.00	
12	S5	PorousPave	1	232872.000	683.000	0.00	
13	S5	GreenRoof	1	18400.000	136.000	0.00	
14	Swale3	Swale	1	14374.800	45.000	0.00	
15	Swale4	Swale	1	21780.000	45.000	0.00	
16	Swale6	Swale	1	17859.600	45.000	0.00	
9	S1	InfilTrench	4	532.000	133.000	0.00	

Figure 14.22 The CS SWMM LID Deployment Editor

Identification and Connectivity

Table 14.28 The LID Deployment Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
MU ID	MIKE URBAN ID	Yes	MUID
Catchment ID	Id of Catchment	Yes	CatchID
LID Control ID	ID of LID control to be deployed	Yes	LidID

LID Deployment specification

Table 14.29 The LID deployment specification

Edit field	Description	Used or required by simulations	Field name in datastructure
Number of units	Number of replicate LID units deployed within the catchment	Yes	ReplicateNumber
Area of unit	The surface area devoted to each replicate LID unit (sq. ft or sq. m).	Yes	Area
Overland flow width	The width of the outflow face of each identical LID unit (in ft or m). This parameter only applies to LID processes such as Porous Pavement and Vegetative Swales that use overland flow to convey surface runoff off of the unit. (The other LID processes, such as Bio-Retention Cells and Infiltration Trenches simply spill any excess captured runoff over their berms.)	Yes	Width

Table 14.29 The LID deployment specification

Edit field	Description	Used or required by simulations	Field name in datastructure
Initial saturation	For Bio-Retention Cells this is the degree to which the unit's soil is initially filled with water (0 % saturation corresponds to the wilting point moisture content, 100 % saturation has the moisture content equal to the porosity). The storage zone beneath the soil zone of the cell is assumed to be completely dry. For other types of LIDs it corresponds to the degree to which their storage zone is initially filled with water	Yes	InitSat
Impervious Area	The percent of the impervious portion of the subcatchment's non-LID area whose runoff is treated by the LID practice. (E.g., if rain barrels are used to capture roof runoff and roofs represent 60% of the impervious area, then the impervious area treated is 60%). If the LID unit treats only direct rainfall, such as with a green roof, then this value should be 0. If the LID takes up the entire subcatchment then this field is ignored.	Yes	FromImp
Send flow to	Select this option if the outflow from the LID is returned onto the subcatchment's pervious area rather than going to the subcatchment's outlet. An example of where this might apply is a rain barrel whose contents are used to irrigate a lawn area. This field is ignored if the LID takes up the entire subcatchment.	Yes	ToPervNo
Result File	Report file option	Yes	RptFileNo
Filename	The name of an optional file where detailed time series results for the LID will be written	Yes	RptFileName

15 Time Series and Boundary Conditions in SWMM

Time Series representing rainfall, evaporation, water stages etc. are important values to model simulations. The time series normally originate from field measurements. The term ‘Boundary Condition’ is used to described the loading of these different timeseries to the model.

Visualization of boundary conditions

The boundary conditions can be visualized on the map. Figure 15.1 shows how where you can toggle the visualization on/off in the TOC. 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. In order to display the changes, make sure to save your edits and to regenerate the relevant layers. This is done by right-clicking on the layer that needs to be updated and then choosing the ‘Regenerate’ option.

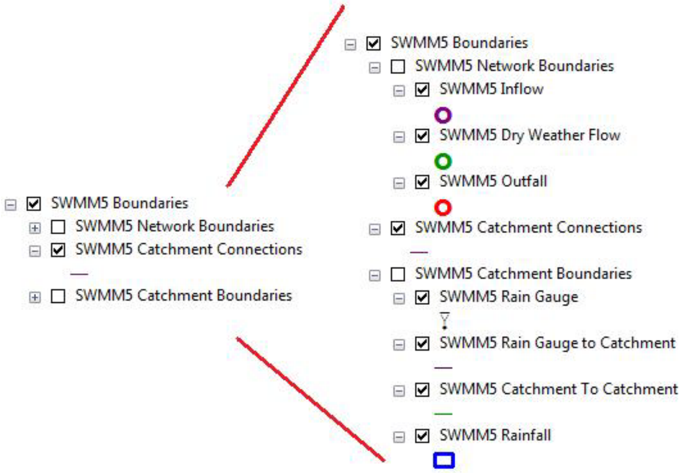

☑ SWMM5 Boundaries
☐ SWMM5 Network Boundaries
☑ SWMM5 Catchment Connections
☐ SWMM5 Catchment Boundaries
☑ SWMM5 Boundaries
☐ SWMM5 Network Boundaries
☑ SWMM5 Inflow
☑ SWMM5 Dry Weather Flow
☑ SWMM5 Outfall
☑ SWMM5 Catchment Connections
☐ SWMM5 Catchment Boundaries
☑ SWMM5 Rain Gauge
☑ SWMM5 Rain Gauge to Catchment
☑ SWMM5 Catchment To Catchment
☑ SWMM5 Rainfall

Figure 15.1 SWMM Boundaries

The boundaries that can be seen on the map are explained in below.

- **SWMM5 Inflow** shows the nodes with an inflow load
- **SWMM5 Dry Weather Flow** shows nodes where dry weather flow has been assigned

- **SWMM5 Outfall** shows outfalls where a water level has been assigned. The boundary condition will only be displayed if the Outfall type is set to one of the following type: Fixed (3), Tidal (4) or Timeseries (5).
- **SWMM5 Catchment Connection** shows the link between the center of the catchment and the node it is connected to
- **SWMM5 Raingauge to Catchment** displays the link between the raingauge and the center of the catchment when a raingauge is specified in the Routing dialog
- **SWMM5 Catchment to Catchment** displays the link between the centers of two connected catchments
- **SWMM5 Rainfall** shows the catchments where a raingauge has been assigned (this one is similar to the SWMM5 Raingauge to Catchment but will show all catchments where a raingauge has been assigned to)

An example on how this may look is shown in Figure 15.2.

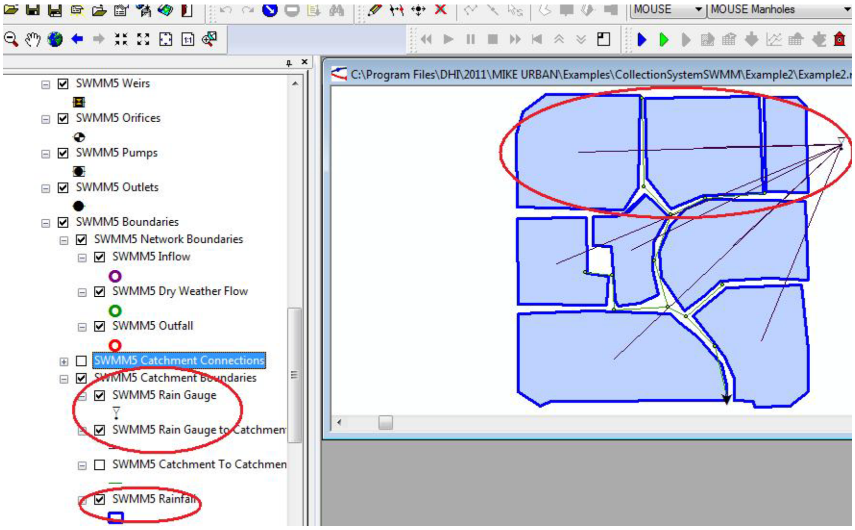

MOUSE
MOUSE Manholes
C:\Program Files\DHI\2011\MIKE URBAN\Examples\CollectionSystemSWMM\Example2\Example2.
SWMM5 Weirs
SWMM5 Orifices
SWMM5 Pumps
SWMM5 Outlets
SWMM5 Boundaries
SWMM5 Network Boundaries
SWMM5 Inflow
SWMM5 Dry Weather Flow
SWMM5 Outfall
SWMM5 Catchment Connections
SWMM5 Catchment Boundaries
SWMM5 Rain Gauge
SWMM5 Rain Gauge to Catchmen
SWMM5 Catchment To Catchmen
SWMM5 Rainfall

Figure 15.2 Example of the visualization of boundary conditions

15.1 Time Series

Time Series can be user specified or originate from external measured data of i.e. rainfall, evaporation, inflows to nodes of the drainage system, and water stage at outfall boundary nodes.

The SWMM engine supports two time series formats. Time series values can either be in **date / time / value format** or in **time / value format**, where each entry is separated by one or more spaces or tab characters.

For the date / time / value format which is the only format supported in MIKE URBAN interface, dates are entered as month/day/year (e.g., 7/21/2004) and times as 24-hour military time (e.g., 8:30 pm is 20:30).

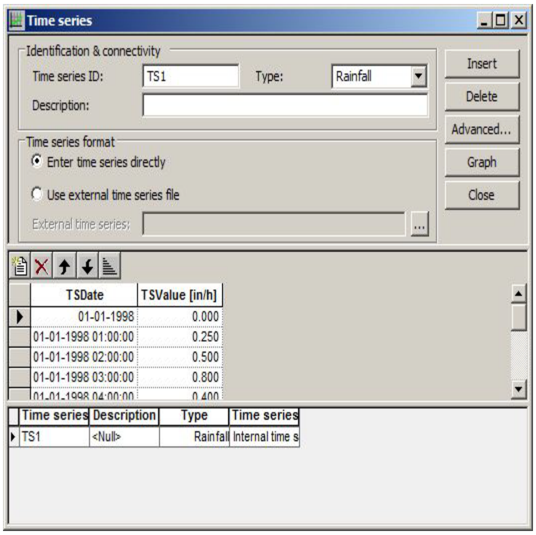
For the time / value format, times are entered as hours since simulation start. If the time / value format has been used outside MIKE URBAN interface, then it is converted during import to the date / time /value format given the start of simulation date as initial start date.

Following types of time series can be defined:

- Temperature data
- Evaporation data
- Rainfall data
- Water stage at outfall nodes
- External inflow hydrographs at drainage system nodes
- External inflow pollutographs at drainage system nodes.

The CS SWMM Time Series editor organizes the related input data for the different timeseries into the following groups:

- Identification and Connectivity - General identification and location information



Identification & connectivity
Time series ID: TS1
Type: Rainfall
Description:

Time series format
● Enter time series directly
○ Use external time series file
External time series:

TSDate	TSValue [in/h]
01-01-1998	0.000
01-01-1998 01:00:00	0.250
01-01-1998 02:00:00	0.500
01-01-1998 03:00:00	0.800
01-01-1998 04:00:00	0.400

Time series	Description	Type	Time series
TS1	<Null>	Rainfall	Internal time s

Figure 15.3 The SWMM Time Series Editor

Identification and Connectivity

Table 15.1 The Time Series Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Time Series ID	ID of Time Series	Yes	MUID
Type	Type of Time Series (Rainfall, Temperature, Inflow Hydrograph, Evaporation, Outfall Stage and Pollutographs)	Yes	TypeNo
Time Series options	Use either external time series file or enter time series values directly	Yes	TimeSeriesTypeNo
External-Time-SeriesID	External time series id	Yes if TimeSeriesTypeNo = 1	ExternalTimeSeriesID
Description	User's descriptive information	No	Description

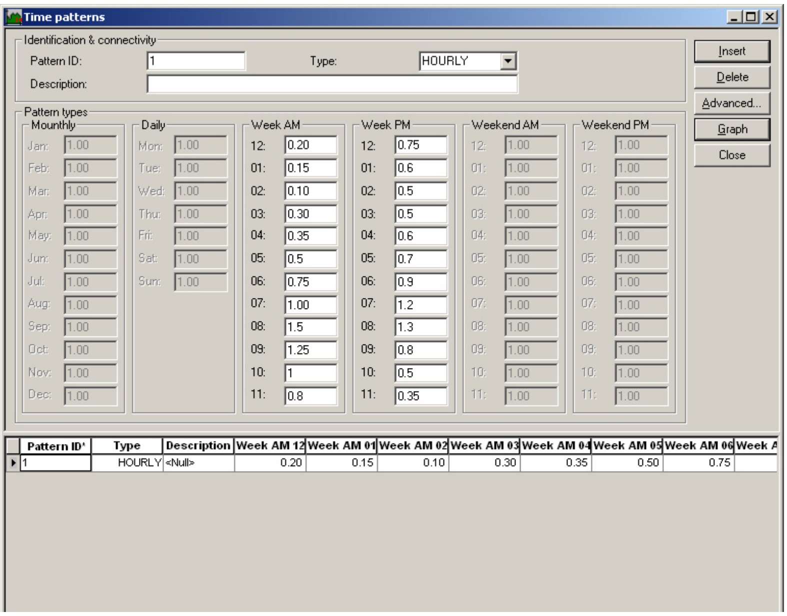
15.2 Time Patterns

Time Patterns are used to described cyclic behaviour of any timeseries. Following four different types of patterns can be supplied

- Monthly
- Daily
- Hourly
- Weekend

The CS SWMM Time Patterns editor organizes the related input data for the different time patterns into the following groups:

- Identification and Connectivity - General identification and location information
- Pattern types - definition of each pattern.



Identification & connectivity

Pattern ID: 1

Type: HOURLY

Description:

Pattern types

Mounthly		Daily		Week AM		Week PM		Weekend AM		Weekend PM	
Jan:	1.00	Mon:	1.00	12:	0.20	12:	0.75	12:	1.00	12:	1.00
Feb:	1.00	Tue:	1.00	01:	0.15	01:	0.6	01:	1.00	01:	1.00
Mar:	1.00	Wed:	1.00	02:	0.10	02:	0.5	02:	1.00	02:	1.00
Apr:	1.00	Thu:	1.00	03:	0.30	03:	0.5	03:	1.00	03:	1.00
May:	1.00	Fri:	1.00	04:	0.35	04:	0.6	04:	1.00	04:	1.00
Jun:	1.00	Sat:	1.00	05:	0.5	05:	0.7	05:	1.00	05:	1.00
Jul:	1.00	Sun:	1.00	06:	0.75	06:	0.9	06:	1.00	06:	1.00
Aug:	1.00			07:	1.00	07:	1.2	07:	1.00	07:	1.00
Sep:	1.00			08:	1.5	08:	1.3	08:	1.00	08:	1.00
Oct:	1.00			09:	1.25	09:	0.8	09:	1.00	09:	1.00
Nov:	1.00			10:	1	10:	0.5	10:	1.00	10:	1.00
Dec:	1.00			11:	0.8	11:	0.35	11:	1.00	11:	1.00

Pattern ID*	Type	Description	Week AM 12	Week AM 01	Week AM 02	Week AM 03	Week AM 04	Week AM 05	Week AM 06	Week A
1	HOURLY	<Null>	0.20	0.15	0.10	0.30	0.35	0.50	0.75	

Figure 15.4 The SWMM Pattern Editor

Identification and Connectivity

Table 15.2 The Time Patterns Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Pattern ID	ID of Pattern	Yes	MUID
Type	Type of pattern (Monthly, Daily, Hourly, Weekend)	Yes	TypeNo
Description	User's descriptive information	No	Description

Time Pattern Data

Table 15.3 The Time Pattern data

Edit field	Description	Used or required by simulations	Field name in datastructure
Jan	Pattern value for January	Yes, if Monthly	Mdr1
Feb	Pattern value for February	Yes, if Monthly	Mdr2
Mar	Pattern value for March	Yes, if Monthly	Mdr3

Table 15.3 The Time Pattern data

Edit field	Description	Used or required by simulations	Field name in datastructure
Apr	Pattern value for April	Yes, if Monthly	Mdr4
May	Pattern value for May	Yes, if Monthly	Mdr5
Jun	Pattern value for June	Yes, if Monthly	Mdr6
Jul	Pattern value for July	Yes, if Monthly	Mdr7
Aug	Pattern value for August	Yes, if Monthly	Mdr8
Sep	Pattern value for September	Yes, if Monthly	Mdr9
Oct	Pattern value for October	Yes, if Monthly	Mdr10
Nov	Pattern value for November	Yes, if Monthly	Mdr11
Dec	Pattern value for December	Yes, if Monthly	Mdr12
Mon	Pattern value for Monday	Yes, if Daily	Day1
Tue	Pattern value for Tuesday	Yes, if Daily	Day2
Wed	Pattern value for Wednesday	Yes, if Daily	Day3
Thu	Pattern value for Thursday	Yes, if Daily	Day4
Fri	Pattern value for Friday	Yes, if Daily	Day5
Sat	Pattern value for Saturday	Yes, if Daily	Day6
Sun	Pattern value for Sunday	Yes, if Daily	Day7
12	Pattern value for 12AM week	Yes, if Hourly	AM12
01	Pattern value for 01AM week	Yes, if Hourly	AM1
02	Pattern value for 02AM week	Yes, if Hourly	AM2
03	Pattern value for 03AM week	Yes, if Hourly	AM3
04	Pattern value for 04AM week	Yes, if Hourly	AM4
05	Pattern value for 05AM week	Yes, if Hourly	AM5
06	Pattern value for 06AM week	Yes, if Hourly	AM6
07	Pattern value for 07AM week	Yes, if Hourly	AM7
08	Pattern value for 08AM week	Yes, if Hourly	AM8
09	Pattern value for 09AM week	Yes, if Hourly	AM9
10	Pattern value for 10AM week	Yes, if Hourly	AM10
11	Pattern value for 11AM week	Yes, if Hourly	AM11
12	Pattern value for 12PM week	Yes, if Hourly	PM12
01	Pattern value for 01PM week	Yes, if Hourly	PM1
02	Pattern value for 02PM week	Yes, if Hourly	PM2
03	Pattern value for 03PM week	Yes, if Hourly	PM3
04	Pattern value for 04PM week	Yes, if Hourly	PM4
05	Pattern value for 05PM week	Yes, if Hourly	PM5
06	Pattern value for 06PM week	Yes, if Hourly	PM6
07	Pattern value for 07PM week	Yes, if Hourly	PM7

Table 15.3 The Time Pattern data

Edit field	Description	Used or required by simulations	Field name in datastructure
08	Pattern value for 08PM week	Yes, if Hourly	PM8
09	Pattern value for 09PM week	Yes, if Hourly	PM9
10	Pattern value for 10PM week	Yes, if Hourly	PM10
11	Pattern value for 11PM week	Yes, if Hourly	PM11
12	Pattern value for 12AM weekend	Yes, if Weekend	AMW12
01	Pattern value for 01AM weekend	Yes, if Weekend	AMW1
02	Pattern value for 02AM weekend	Yes, if Weekend	AMW2
03	Pattern value for 03AM weekend	Yes, if Weekend	AMW3
04	Pattern value for 04AM weekend	Yes, if Weekend	AMW4
05	Pattern value for 05AM weekend	Yes, if Weekend	AMW5
06	Pattern value for 06AM weekend	Yes, if Weekend	AMW6
07	Pattern value for 07AM weekend	Yes, if Weekend	AMW7
08	Pattern value for 08AM weekend	Yes, if Weekend	AMW8
09	Pattern value for 09AM weekend	Yes, if Weekend	AMW9
10	Pattern value for 10AM weekend	Yes, if Weekend	AMW10
11	Pattern value for 11AM weekend	Yes, if Weekend	AMW11
12	Pattern value for 12PM weekend	Yes, if Weekend	PMW12
01	Pattern value for 01PM weekend	Yes, if Weekend	PMW1
02	Pattern value for 02PM weekend	Yes, if Weekend	PMW2
03	Pattern value for 03PM weekend	Yes, if Weekend	PMW3
04	Pattern value for 04PM weekend	Yes, if Weekend	PMW4
05	Pattern value for 05PM weekend	Yes, if Weekend	PMW5

Table 15.3 The Time Pattern data

Edit field	Description	Used or required by simulations	Field name in datastructure
06	Pattern value for 06PM weekend	Yes, if Weekend	PMW6
07	Pattern value for 07PM weekend	Yes, if Weekend	PMW7
08	Pattern value for 08PM weekend	Yes, if Weekend	PMW8
09	Pattern value for 09PM weekend	Yes, if Weekend	PMW9
10	Pattern value for 10PM weekend	Yes, if Weekend	PMW10
11	Pattern value for 11PM weekend	Yes, if Weekend	PMW11

15.3 Raingauges

Rain Gauges supply precipitation data for one or more catchments in a study area. The rainfall data can be either a user-defined time series (specified in Time Series as Type Rainfall) or come from an external file.

Several different popular rainfall file formats currently in use are supported, as well as a standard user-defined format. These are:

- DSI-3240 and related formats which record hourly rainfall at U.S. National Weather Service (NWS) and Federal Aviation Agency stations, available online from the National Climatic Data Center (NCDC) at www.ncdc.noaa.gov/oa/ncdc.html.
- DSI-3260 and related formats which record fifteen minute rainfall at NWS stations, also available online from NCDC.
- HLY03 and HLY21 formats for hourly rainfall at Canadian stations, available online from Environment Canada at www.climate.weatheroffice.ec.gc.ca.
- FIF21 format for fifteen minute rainfall at Canadian stations, also available online from Environment Canada.
- a standard user-prepared format where each line of the file contains the station ID, year, month, day, hour, minute, and non-zero precipitation reading, all separated by one or more spaces. An excerpt from the user-prepared format might look as follows:

STA01 2004 6 12 00 00 0.12

STA01 2004 6 12 01 00 0.04

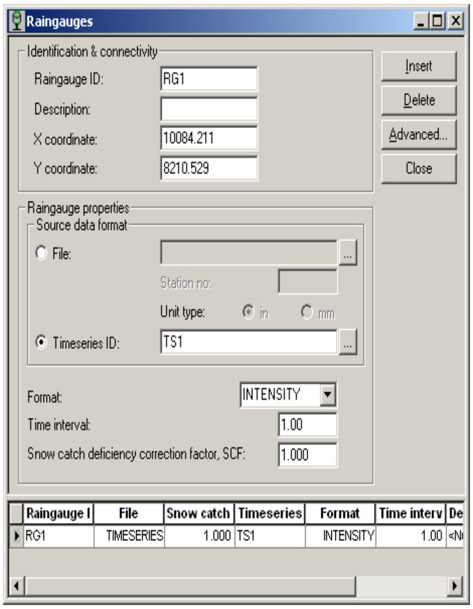
STA01 2004 6 22 16 00 0.07

When a rain gauge is designated as receiving its rainfall data from a file, the user must supply the name of the file and the name of the recording station referenced in the file.

For the standard user-prepared format, the rainfall type (e.g., intensity or volume), recording time interval, and depth units must also be supplied as rain gauge properties. For the other file types these properties are defined by their respective file format and are automatically recognized by SWMM.

The CS SWMM Raingauges editor organizes the related input data for the rainfall data into the following groups:

- Identification and Connectivity - General identification and location information
- Raingauge properties



Raingauge ID: RG1

Description:

X coordinate: 10084.211

Y coordinate: 8210.529

○ File:

Station no:

Unit type: ● in ○ mm

● Timeseries ID: TS1

Format: INTENSITY

Time interval: 1.00

Snow catch deficiency correction factor, SCF: 1.000

Raingauge I	File	Snow catch	Timeseries	Format	Time interv	De
RG1	TIMESERIES	1.000	TS1	INTENSITY	1.00	<N

Figure 15.5 The SWMM Raingauge Editor

Identification and Connectivity

Table 15.4 The Raingauge Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Raingauge ID	ID of Raingauge	Yes	MUID
X coordinate	X coordinate of raingauge	Yes	
Y coordinate	Y coordinate of raingauge	Yes	
Description	User's descriptive information	No	Description

Table 15.5 The Raingauge Properties Group

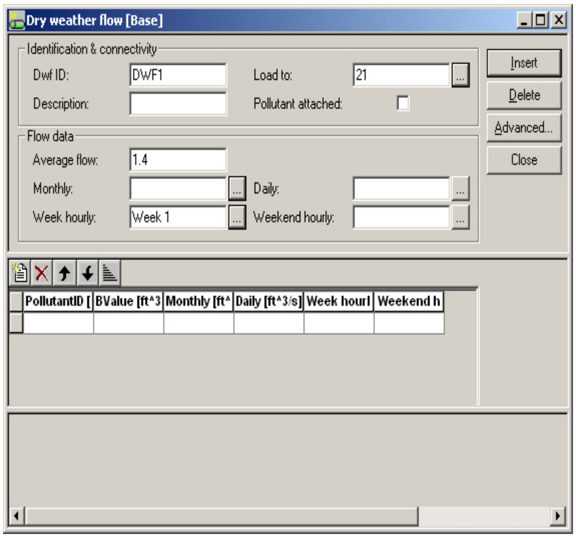
Edit field	Description	Used or required by simulations	Field name in datastructure
	Radio button, TypeNo = File or TimeSeries	Yes	TypeNo
File	Name of rainfall data file	Yes if TypeNo = File	FileNameSeries
Station no	NStation number	Yes if TypeNo = File	StationNumber
Unit type	Unit type (in or mm)	Yes if TypeNo = File	UnitNo
Timeseries ID	ID of Time Series	Yes if TypeNo = TimeSeries	TimeSeriesID
Format	Format of the rainfall data	Yes if TypeNo = TimeSeries	FormNo
Time interval	Time interval in timeseries	Yes if TypeNo = TimeSeries	TimeInterval
Snow catch defience correction factor, SCF	Snow catch defience correction factor	Yes	Scf

15.4 Dry Weather Flow

Dry Weather Inflows are continuous inflows that typically reflect the contribution from sanitary sewage in sewer systems or base flows in pipes and stream channels. They are represented by an average inflow rate that can be periodically adjusted on a monthly, daily, and hourly basis by applying Time Patterns multipliers to this average value.

The CS SWMM Dry Weather Flow editor organizes the related input data for each dry weather inflow into the following groups:

- Identification and Connectivity - General identification and location information
- Flow data - flow data and specification of pattern
- Pollutant attached to flow - upper grid in dialog



Dwf ID: DWF1
Load to: 21
Description:
Pollutant attached: ☐
Average flow: 1.4
Monthly:
Daily:
Week hourly: Week 1
Weekend hourly:

PollutantID [	BValue [ft^3	Monthly [ft^	Daily [ft^3/s]	Week hourl	Weekend h

Figure 15.6 The SWMM Dry Weather Flow Editor

Identification and Connectivity

Table 15.6 The Dry Weather Flow Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
DWF ID	ID Dry Weather Flow	Yes	MUID
Load To	ID of Node ID for DWF load	Yes	NodeID
Pollutant attached	Pollutant attached to dry weather flow	Yes	PollutNo
Description	User's descriptive information	No	Description

Table 15.7 The Dry Weather Flow Data Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Average Flow	Average flow value of DWF	Yes	FlowValue
Monthly	ID of Pattern of Monthly type	No	PatternMonthID
Daily	ID of Pattern of Daily type	No	PatternWeekID
Week hourly	ID of Pattern of Hourly type	No	PatternWeek-HourlyID
Weekend hourly	ID of Pattern of Weekend type	No	PatternWeek-endHourlyID

Table 15.8 The Pollutant Data in upper grid attached to dry weather flow

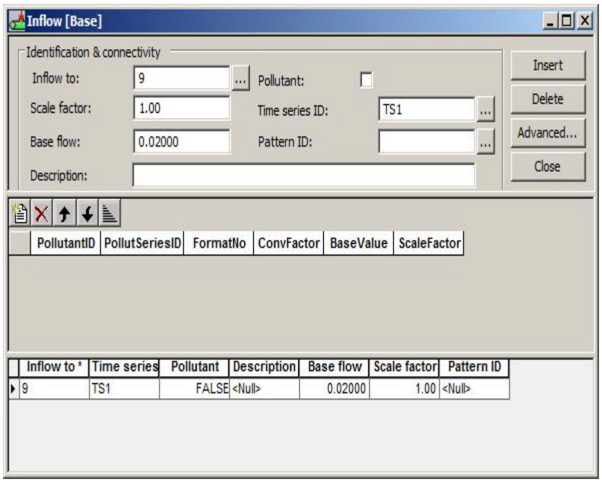
Edit field	Description	Used or required by simulations	Field name in datastructure
PollutantID	ID of Pollutant	Yes, if pollutant attached	PollutantID
BValue	Base value of pollutant	Yes, if pollutant attached	BValue
Monthly	ID of Pattern of Monthly type	Yes, if pollutant attached	PatternMonthID
Daily	ID of Pattern of Daily type	Yes, if pollutant attached	PatternWeekID
Week hourly	ID of Pattern of Hourly type	Yes, if pollutant attached	PatternWeek-HourlyID
Weekend hourly	ID of Pattern of Weekend type	Yes, if pollutant attached	PatternWeek-endHourlyID

15.5 Inflow

Direct Inflows are user-defined time series of inflows added directly into a node. They can be used to perform flow and water quality routing in the absence of any runoff computations (as in a study area where no catchments are defined). A pollutant component can only be defined as inflow to the system if it is attached to a flow component.

The CS SWMM Inflow editor organizes the related input data for each inflow into the following groups:

- Identification and Connectivity - General identification and location information
- Pollutant attached to flow - upper grid in dialog



Inflow to: 9

Pollutant: ☐

Scale factor: 1.00

Time series ID: TS1

Base flow: 0.02000

Pattern ID:

Description:

PollutantID	PollutSeriesID	FormatNo	ConvFactor	BaseValue	ScaleFactor

Inflow to *	Time series	Pollutant	Description	Base flow	Scale factor	Pattern ID
9	TS1	FALSE	<Null>	0.02000	1.00	<Null>

Figure 15.7 The SWMM Inflow Editor

Identification and Connectivity

Table 15.9 The Inflow Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Inflow to	ID of node where inflow should be loaded to	Yes	NodeID
Time series ID	ID of flow time series	Yes	FlowSeriesID
Pollutant	Pollutant attached to flow	Yes	PollutNo
Base Flow	Base flow	Yes	BaseFlowValue
Scale Factor	Scaling factor multiplied to the time series value	Yes	ScaleFlowFactor
Pattern ID	Baseline pattern multiplied to the baseline value	Yes	BaselinePatternID
Description	User’s descriptive information	No	Description

Table 15.10 The Dry Weather Flow Data Group

Edit field	Description	Used or required by simulations	Field name in datastructure
PollutantID	ID of pollutant	Yes, if pollutant attached to flow	PollutantID
Pollut-SeriesID	ID of pollutograph	Yes, if pollutant attached to flow	PollutSeriesID
FormatNo	Format (MASS, CONCEN)	Yes, if pollutant attached to flow	FormatNo
ConvFactor	Conversion factor	Yes, if pollutant attached to flow	ConvFactor

16 Project Options and Simulations in SWMM

The project and simulations options for SWMM are defined in the ‘Simulation | Run SWMM5’.

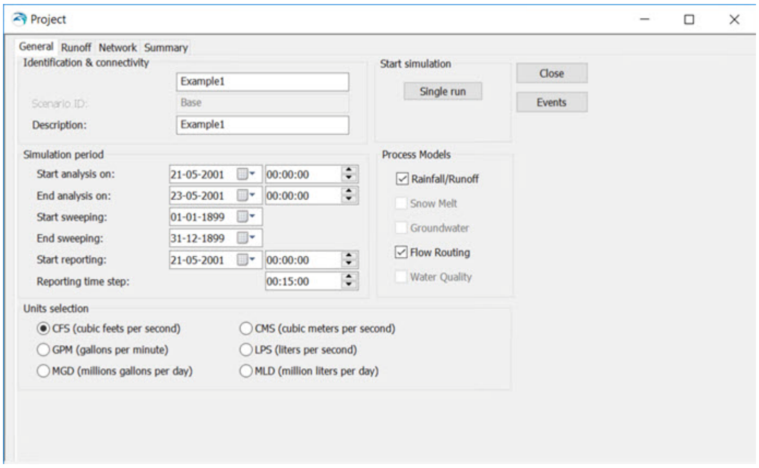
The CS-SWMM Run SWMM5 editor is split into following four tabsheets:

- General
- Runoff
- Network
- Summary

16.1 The General Simulation Settings

The General page is organized into following groups:

- Identification and Connectivity
- Simulation Period
- Start Simulation
- Units Selection (main unit group saved to file for engine simulation)



Identification & connectivity
Example1
Scenario ID: Base
Description: Example1

Simulation period
Start analysis on: 21-05-2001 00:00:00
End analysis on: 23-05-2001 00:00:00
Start sweeping: 01-01-1899
End sweeping: 31-12-1899
Start reporting: 21-05-2001 00:00:00
Reporting time step: 00:15:00

Process Models
☑ Rainfall/Runoff
☐ Snow Melt
☐ Groundwater
☑ Flow Routing
☐ Water Quality

Units selection
● CFS (cubic feets 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 16.1 The SWMM Simulation General Editor dialog

Identification and Connectivity

Table 16.1 The Simulation, General Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in data structure
Simulation ID	ID of simulation - user specified. ID will be reflected in the name of project and result files	Yes	MUID
Scenario ID	ID of selected Scenario - Activating scenario is done in the Scenario Manager dialog	Yes	ActiveJob
Description	User's descriptive information	No	Description

Simulation Period

Table 16.2 The Simulation, General Simulation Period Group

Edit field	Description	Used or required by simulations	Field name in data structure
Start Analysis on	Start analysis date / time	Yes	StartDate + StartTime
End Analysis on	End analysis date / time	Yes	EndDate + EndTime
Sweeping Start	Start sweeping date	Yes	SweepStartDate
Sweeping End	End sweeping date	Yes	SweepEndDate
Start Reporting	Start reporting date / time	Yes	ReportStartDate + ReportStartTime
Reporting time step	Saving time step to report	Yes	ReportStep

Start Simulation

Table 16.3 The Simulation, General Start Simulation Group

Edit field	Description	Used or required by simulations	Field name in data structure
Single Run	Initiate simulation		
Rainfall/Runoff	Include Rainfall / Runoff	Yes	IncludeRainfallRunoffNo
Snow Melt	Include Snow Melt	Yes	IncludeSnowMeltNo
Groundwater	Include Groundwater	Yes	InlcudeGroundwaterNo
Flow Routing	Include Flow Routing	Yes	IncludeFlowRoutingNo
Water Quality	Include Water Quality	Yes	IncludeWaterQualityNo

Units selection

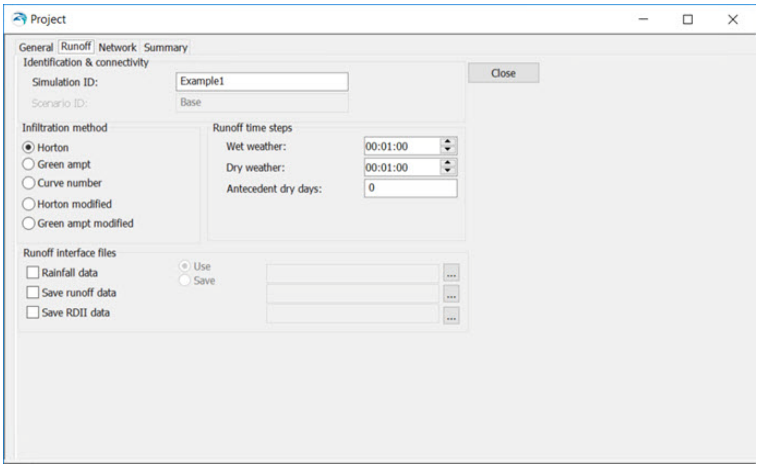
Table 16.4 The Simulation, General Units selection Group

Edit field	Description	Used or required by simulations	Field name in data structure
Units Selection	Selection of main Unit for saving to SWMM5 engine	Yes	UnitNo

16.2 The Runoff Simulation Settings

The Runoff page is organized into following groups:

- Identification and Connectivity
- Infiltration Method
- Runoff time steps
- Runoff interface files


Project
General
Runoff
Network
Summary
Identification & connectivity
Close
Infiltration method
Runoff time steps
Runoff interface files
...
...
...

Simulation ID: Example1

Scenario ID: Base

● Horton
○ Green ampt
○ Curve number
○ Horton modified
○ Green ampt modified

Wet weather: 00:01:00

Dry weather: 00:01:00

Antecedent dry days: 0

☐ Rainfall data
☐ Save runoff data
☐ Save RDII data

● Use
○ Save

Figure 16.2 The SWMM Simulation Runoff Editor Dialog

Identification and Connectivity

Table 16.5 The Simulation Runoff Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in data structure
Simulation ID	ID of simulation - user specified. ID will be reflected in the name of project and result files	Yes	MUID
Scenario ID	ID of selected Scenario - Activating scenario is done in the Scenario Manager dialog	Yes	ActiveJob

Infiltration Method

Table 16.6 The Simulation, Runoff Infiltration Method Group

Edit field	Description	Used or required by simulations	Field name in data structure
Horton	Horton Infiltration method	No	InfiltrationTypeNo
Green ampt	Green ampt Infiltration method	No	InfiltrationTypeNo
Curve Number	SCS Hydrology - or curve number method	No	InfiltrationTypeNo

Table 16.6 The Simulation, Runoff Infiltration Method Group

Edit field	Description	Used or required by simulations	Field name in data structure
Modified Horton	Modified Horton infiltration method	No	Infiltration-TypeNo
Modified Green Ampt	Modified Green-Ampt Infiltration method	No	Infiltration-TypeNo

Runoff Time Steps

Table 16.7 The Simulation, Runoff Time Steps Group

Edit field	Description	Used or required by simulations	Field name in data structure
Wet Weather	Time step length used to compute runoff from catchments	Yes	WetStep
Dry Weather	Time step length used for runoff computations (consisting essentially of pollutant buildup) during periods when there is no rainfall and no ponded water at junctions.	Yes	DryStep
Antecedent dry days	The number of days with no rainfall prior to start of the simulation.	Yes	DryDays

Runoff Interface Files

Table 16.8 The Simulation, Runoff Interface Files Group

Edit field	Description	Used or required by simulations	Field name in data structure
Rainfall data	Include Rainfall data file	No	RainfallFileNo
Use/Save	Optional to read information from already created Rainfall file or create a Rainfall file	Yes if Rainfall-FileNo is true	RainfallFileUs-eSaveNo
Rainfall-Name	Rainfall file name	No	RainfallFile-Name
Save Runoff data	Save to runoff file (RunoffUs-eSaveNo = Save)	No	RunoffFileNo
RunoffFile-Name	Runoff file name	Yes if Runoff-FileNo is true	RunoffFileName

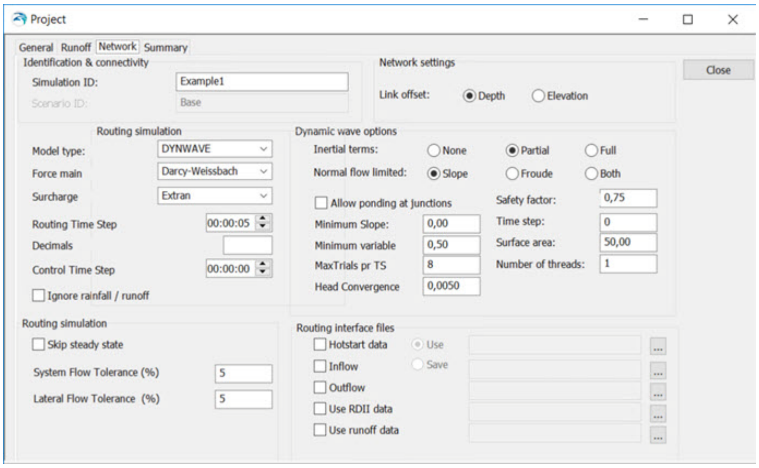
Table 16.8 The Simulation, Runoff Interface Files Group

Edit field	Description	Used or required by simulations	Field name in data structure
Save RDII	Save to RDII file (RDIIFileUseSaveNo = Save)	No	RDIIFileNo
RDIIFileName	Save to RDII file name	Yes if RDII FileNo is true	RDIIFileName

16.3 The Network Simulation Settings

The Network page is organized into following groups:

- Identification and Connectivity
- Routing simulation
- Dynamic wave options
- Routing interface files



Identification & connectivity

Simulation ID: Example1

Scenario ID: Base

Network settings

Link offset: ● Depth ○ Elevation

Routing simulation

Model type: DYNWAVE

Force main: Darcy-Weissbach

Surcharge: Extran

Routing Time Step: 00:00:05

Decimals:

Control Time Step: 00:00:00

☐ Ignore rainfall / runoff

Dynamic wave options

Inertial terms: ○ None ● Partial ○ Full

Normal flow limited: ● Slope ○ Froude ○ Both

☐ Allow ponding at junctions

Minimum Slope: 0,00

Minimum variable: 0,50

MaxTrials pr TS: 8

Head Convergence: 0,0050

Safety factor: 0,75

Time step: 0

Surface area: 50,00

Number of threads: 1

Routing simulation

☐ Skip steady state

System Flow Tolerance (%): 5

Lateral Flow Tolerance (%): 5

Routing interface files

☐ Hotstart data ○ Use ○ Save

☐ Inflow

☐ Outflow

☐ Use RDII data

☐ Use runoff data

Figure 16.3 The SWMM Simulation Network Editor Dialog

Identification and Connectivity

Table 16.9 The Simulation, Network Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in data structure
Simulation ID	ID of simulation - user specified. ID will be reflected in the name of project and result files	Yes	MUID
Scenario ID	ID of selected Scenario - Activating scenario is done in the Scenario Manager dialog	Yes	ActiveJob

Routing Simulation

Table 16.10 The Simulation, Network Routing Simulation Group

Edit field	Description	Used or required by simulations	Field name in data structure
Model Type	Steady State, Kinematic Wave or Dynamic Wave	Yes	FlowRoutingNo
Routing Time Step	Routing timestep for simulation	Yes	RoutingStep
Surcharge	Method that will be used to handle surcharge conditions. Extran Method option uses a variation of the surface algorithm to update nodal heads when all connecting links become full. The Slot uses a Preismann Slot to add a small amount of virtual top surface width top surface to full flowing pipes.	Yes	SurchargeMethodNo
Decimals	Decimals that can be added to the routing stepping	Yes	RoutingStep-Decimals
Control Time Step	Control rule timestep	Yes, if control is used	ControlStep

Dynamic Wave Options

Table 16.11 The Simulation, Network Dynamic Wave Options Group

Edit field	Description	Used or required by simulations	Field name in data structure
Inertial Terms	Indicates how initial terms in the St. Venants momentum equation will be handled	Yes	InitialDampingNo
Safety Factor	This is a safety factor, between 10 and 200%, which is applied to variable time step computed from the Courant criterion. If Safety Factor is set to 0.0 then the variable timestep option is not used	Yes	VariableStep
Allow Ponding	Allows flow to pond at the surface during node surcharge for re-entering the network when hydraulic capacity is regained. The size of the surface area for ponding is a input parameter supplied for the individual junction	Yes	AllowPondingNo
Time Step	This is a time step, in seconds, used to artificially lengthen conduits so that they meet the Courant stability criterion under full-flow conditions (i.e., the travel time of a wave will not be smaller than the specified conduit lengthening time step). As this value is decreased, fewer conduits will require lengthening. A value of 0 means that no conduits will be lengthened.	Yes	LengtheningStep
Minimum Slope	When this option is non-zero then the computed slope of a conduit can not be below this value	Yes	MinimumPipeSlope
Surface Area	Minimum area of a junction. If 0 is entered a default value of 4ft (12.566 square ft) is used	Yes	MinSurfArea
Compatibility	Indicates which version of SWMM's dynamic wave solution methods should be used, SWMM5, SWMM 4 or SWMM 3	Yes	CompatibilityNo

Table 16.11 The Simulation, Network Dynamic Wave Options Group

Edit field	Description	Used or required by simulations	Field name in data structure
Surface area	This is a minimum surface area used at nodes when computing changes in water depth. If 0 is entered, then the default value of 12.566 ft2 (i.e., the area of a 4-ft diameter manhole) is used	Yes	MinSurfArea
Max trials per TS	This is the maximum number of trials that SWMM uses at each time step to reach convergence when updating hydraulic heads at the conveyance system's nodes. The default value is 8	Yes	MaxTrials
Head Convergence	When the difference in computed head at each node between successive trials is below this value the flow solution for the current time step is assumed to have converged. The default tolerance is 0.005 ft (0.0015 m).	Yes	HeadTolerance

Routing Interface Files

Table 16.12 The Simulation Network Routing Interface Files Group

Edit field	Description	Used or required by simulations	Field name in data structure
HotStart data	Start date and time for HotStart	No	HotStartFileNo
Use/Save	Optional to read information from already created HotStart file or create a HotStart file	No	HotStartFileUseSaveNo
HotStartFileName	HotStart file name	No	HotStartFileName
Inflow	Include inflow	No	InflowFileNo
InflowFileName	Inflow filename	Yes if InflowFileNo = True	InflowFileName
Outflow	Save outflow	No	OutFlowFileNo
OutFlowFileName	Save outflow to filename	Yes if OutflowFileNo = True	OutFlowFileName

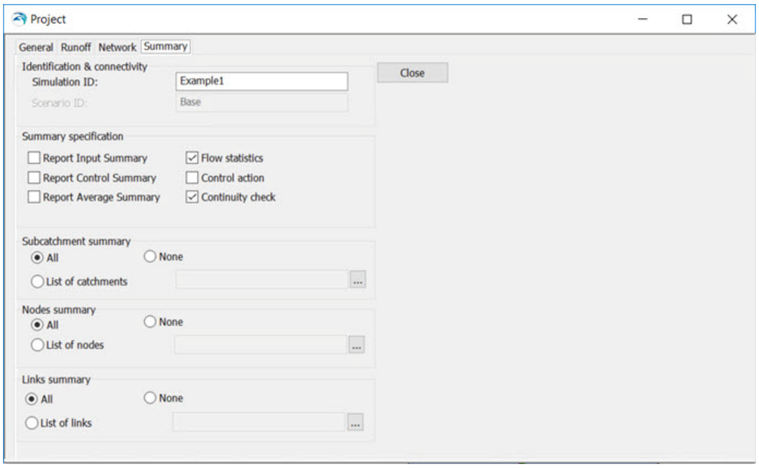
Table 16.12 The Simulation Network Routing Interface Files Group

Edit field	Description	Used or required by simulations	Field name in data structure
Use RDII data	Include RDII file	No	RDIIFileNo
RDIIFile-Name	RDII filename	Yes if RDIIFile-NameNo = True	RDIIFileName
Use Runoff data	Include Runoff file	No	RunoffFileNo
RunoffFile-Name	Runoff filename	No	RunoffFileName

16.4 The Summary Simulation Settings

The Summary page is organized into following groups:

- Identification and Connectivity
- Summary Specification
- Sub catchment Summary
- Nodes Summary
- Links Summary



Identification & connectivity
Simulation ID: Example1
Scenario ID: Base

Summary specification
☐ Report Input Summary
☐ Report Control Summary
☐ Report Average Summary
☑ Flow statistics
☐ Control action
☑ Continuity check

Subcatchment summary
● All ○ None
○ List of catchments

Nodes summary
● All ○ None
○ List of nodes

Links summary
● All ○ None
○ List of links

Figure 16.4 The SWMM Simulation Summary Editor Dialog

Identification and Connectivity

Table 16.13 The Simulation, Summary Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in data structure
Simulation ID	ID of simulation - user specified. ID will be reflected in the name of project and result files	Yes	MUID
Scenario ID	ID of selected Scenario - Activating scenario is done in the Scenario Manager dialog	Yes	ActiveJob

Summary Specifications

Table 16.14 The Simulation, Summary Specifications Group

Edit field	Description	Used or required by simulations	Field name in data structure
Report Input data	Specifies whether or not summary of the input data should be provided in the output report. The default is No.	No	ReportInputNo
Report Control Summary	Specifies whether or not summary of the control data should be provided in the output report.. Default is No	No	ReportControlNo
Report Average Summary	Specifies whether or not summary of the average data should be provided in the output report.. Default is No	No	ReportAverageNo
Flow Statistics	Specifies whether or not summary flowstatistics should be reported or not. The default is Yes.	No	FlowStatsNo
Continuity Check	Specifies whether or not summary of continuity checks should be reported or not. The default is Yes.	No	ContinuityNo
Control action	Specifies whether or not summary of control actions taken during simulation should be listed or not. The default is No.	No	ControlsNo

Subcatchment Summary

Table 16.15 The Simulation, Summary Subcatchment Summary Group

Edit field	Description	Used or required by simulations	Field name in data structure
All, None or List of catchment	All. None, List of catchments	Yes	CatchmentsNo
CatchmentsFileName	Filename. A selection file *.mus is expected	Yes, if CatchmentsNo = List of Catchments	CatchmentsFileName

Node Summary

Table 16.16 The Simulation, Summary Node Summary Group

Edit field	Description	Used or required by simulations	Field name in data structure
All, None or List of nodes	All. None, List of nodes	Yes	NodesNo
NodesFileName	Filename. A selection file *.mus is expected	Yes, if NodesNo = List of Nodes	NodesFileName

Link Summary

Table 16.17 The Simulation, Summary Link Summary Group

Edit field	Description	Used or required by simulations	Field name in data structure
All, None or List of Links	All. None, List of Links	Yes	LinksNo
LinksFileName	Filename. A selection file *.mus is expected	Yes, if LinksNo = List of Links	LinksFileName

17 Water Quality Modelling with SWMM

17.1 Terms and Concepts

MIKE URBAN CS SWMM models both the quantity and quality of runoff generated within each catchment, and the flow rate, flow depth, and quality of water in each pipe and channel during a simulation period comprised of multiple time steps.

Water quality routing within conduit links assumes that the conduit behaves as a continuously stirred tank reactor (CSTR). Although a plug flow reactor assumption might be more realistic, the differences will be small if the travel time through the conduit is on the same order as the routing time step. The concentration of a constituent exiting the conduit at the end of a time step is found by integrating the conservation of mass equation, using average values for quantities that might change over the time step such as flow rate and conduit volume.

Water quality modelling within storage unit nodes follows the same approach used for conduits. For other types of nodes that have no volume, the quality of water exiting the node is simply the mixture concentration of all water entering the node.

The production and fate of pollutant loads associated with this runoff is also modelled. The following processes can be modelled for any number of user-defined water quality constituents:

- dry-weather pollutant buildup over different land uses
- pollutant washoff from each land use during storm events
- direct contribution of rainfall deposition
- reduction in dry-weather buildup due to street cleaning
- reduction in washoff load due to BMPs
- entry of dry weather sanitary flows and user-specified external inflows at any point in the conveyance system
- routing of water quality constituents through the pipe/channel network
- reduction in constituent concentration through treatment at junction nodes or in storage units.

17.2 Land Uses

Land Uses are categories of activities or land surface characteristics that are assigned to catchment areas. Examples of land use activities are residential, commercial, industrial, and undeveloped. Land surface characteristics might include roof tops, lawns, paved roads, undisturbed soils, etc. Land uses are used solely to allow spatial variation in pollutant buildup and washoff rates.

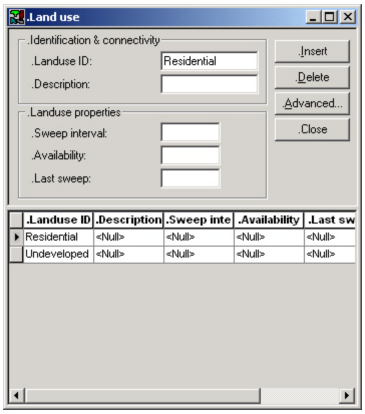
There is a complete freedom in defining land uses and assigning them to catchment areas. One approach is to assign a mix of land uses to each catchment, in which case all land uses in the catchment will have the same pervious/impervious characteristics. If this is not appropriate then it is possible to create catchments that have just a single land use classification along with a set of pervious/impervious characteristics that reflects the classification.

The following processes can be associated with each land use category:

- Pollutant Buildup
- Pollutant Washoff
- Street Cleaning

The CS SWMM Land Use editor organizes the related input data for the land uses into the following groups:

- Identification and Connectivity - General identification and location information
- Land Use Properties - Land Use related data



.Landuse ID: Residential
.Description:
.Sweep interval:
.Availability:
.Last sweep:

.Landuse ID	.Description	.Sweep inte	.Availability	.Last sw
Residential	<Null>	<Null>	<Null>	<Null>
Undeveloped	<Null>	<Null>	<Null>	<Null>

Figure 17.1 The SWMM Land Uses Editor

Identification and Connectivity

Table 17.1 The Land Use Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Landuse ID	ID of Land Use	Yes	MUID
Description	User's descriptive information	No	Description

Land Use Properties

Table 17.2 The Land Use Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Sweep Interval	Days between sweeping	Yes	SweepInterval
Availability	Fraction of pollutant buildup available for removal by street sweeping	Yes	Availability
LastSweep	Days since last sweeping at start of the simulation	Yes	LastSweep

17.3 Buildup

Pollutant Buildup that accumulates over a category of Land Use is described by either a mass per unit of catchment area or per unit of curb length. The amount of buildup as a function of days of dry weather can be computed using one of the following functions:

Power Function

Pollutant buildup (B) accumulates proportional to time (t) raised to some power, until a maximum limit is achieved,

$$B = Min(C_1, C_2 t^{C_3}) \quad (17.1)$$

where C1 = maximum buildup possible (mass per unit of area or curb length), C2 = buildup rate constant, and C3 = time exponent.

Exponential Function

Buildup follows an exponential growth curve that approaches a maximum limit asymptotically,

$$B = C_1(1 - e^{-C_2 t}) \tag{17.2}$$

where C1 = maximum buildup possible (mass per unit of area or curb length) and C2 = buildup rate constant (1/days).

Saturation Function

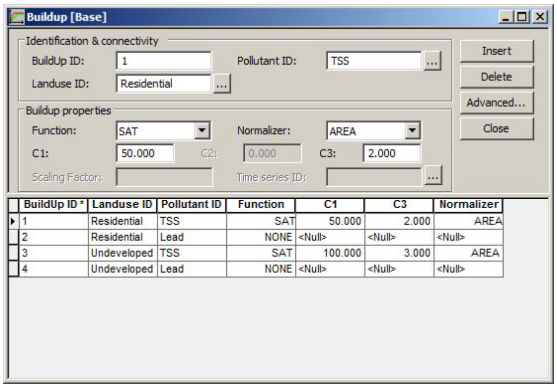
Buildup begins at a linear rate which proceeds to decline constantly over time until a saturation value is reached,

$$B = \frac{C_1 t}{C_2 + t} \tag{17.3}$$

where C1 = maximum buildup possible (mass per unit area or curb length) and C2 = half-saturation constant (days to reach half of the maximum buildup).

The CS SWMM Buildup Editor organizes the related input data for the pollutant buildup into the following groups:

- Identification and Connectivity - General identification and location information
- Buildup Properties - Buildup related data



BuildUp ID: 1

Pollutant ID: TSS

Landuse ID: Residential

Function: SAT

Normalizer: AREA

C1: 50.000

C2: 0.000

C3: 2.000

Scaling Factor:

Time series ID:

BuildUp ID *	Landuse ID	Pollutant ID	Function	C1	C3	Normalizer
1	Residential	TSS	SAT	50.000	2.000	AREA
2	Residential	Lead	NONE	<Null>	<Null>	<Null>
3	Undeveloped	TSS	SAT	100.000	3.000	AREA
4	Undeveloped	Lead	NONE	<Null>	<Null>	<Null>

Figure 17.2 The SWMM Buildup Editor

Identification and Connectivity

Table 17.3 The Buildup Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
BuildUp ID	ID of Buildup	Yes	MUID
Pollutant ID	ID of Pollutant	Yes	PollutantID
Landuse ID	ID of Landuse	Yes	LanduseID

Buildup Properties

Table 17.4 The Buildup Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Function	Function Type (Power, Exponential and Saturated)	Yes	FuncTypeNo
C1	Coefficient	Yes	C1
C2	Coefficient	Yes, if Function is Power or Exponential	C2
C3	Coefficient	Yes, if Function is Power or Saturation	C3
Normalizer	Normalizer to Area or to Curb Length	Yes	NormalizerNo
Scaling Factor	A scaling factor used to adjust buildup rates listed in the time series	Yes, if Function is external time series	C2_Scalingfactor
Time Series ID	Time series containing buildup rates	Yes, if Function is external time series	ExternalTimeSeriesID

17.4 Washoff

Pollutant Washoff from a given land use category during wet weather periods can be described in one of the following ways:

Exponential Washoff

The washoff load (W) in units of mass per hour is proportional to the product of runoff raised to some power and to the amount of buildup remaining, i.e.,

$$W = C_1 q^{C_2} B \tag{17.4}$$

where C1 = washoff coefficient, C2 = washoff exponent, q = runoff rate per unit area (inches/hour or mm/hour), and B = pollutant buildup in mass (lbs or kg) per unit area or curb length. Washoff mass units are the same as used to express the pollutant's concentration (milligrams, micrograms, or counts).

Rating Curve Washoff

The rate of washoff W in mass per second is proportional to the runoff rate raised to some power, i.e.,

$$W = C_1 Q^{C_2} \tag{17.5}$$

where C1 = washoff coefficient, C2 = washoff exponent, and Q = runoff rate in user-specified flow units.

Event Mean Concentration

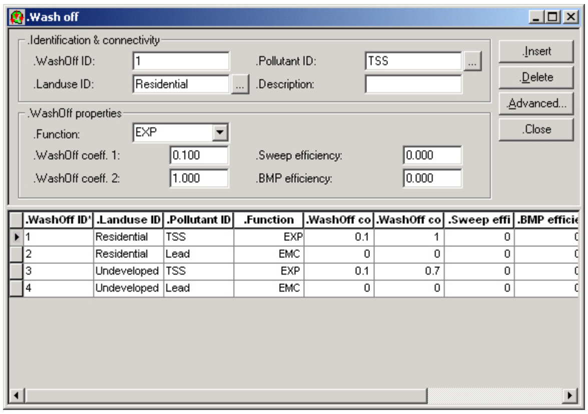
This is a special case of Rating Curve Washoff where the exponent is 1.0 and the coefficient C1 represents the concentration of any and all runoff in mass per liter (the conversion between user-specified flow units used for runoff and liters is handled internally by SWMM).

Note that in each case buildup is continuously deplenished as washoff proceeds, and washoff ceases when there is no more buildup available.

Washoff loads for a given pollutant and land use category can be reduced by a fixed percentage by specifying a BMP Removal Efficiency which reflects the effectiveness of any BMP controls associated with the land use.

The CS SWMM Buildup Editor organizes the related input data for the pollutant washoff into the following groups:

- Identification and Connectivity - General identification and location information
- Washoff Properties - Washoff related data



.WashOff ID: 1

.Pollutant ID: TSS

.Landuse ID: Residential

.Description:

.Function: EXP

.WashOff coeff. 1: 0.100

.Sweep efficiency: 0.000

.WashOff coeff. 2: 1.000

.BMP efficiency: 0.000

.WashOff ID'	.Landuse ID	.Pollutant ID	.Function	.WashOff co	.WashOff co	.Sweep effi	.BMP efficie
1	Residential	TSS	EXP	0.1	1	0	
2	Residential	Lead	EMC	0	0	0	
3	Undeveloped	TSS	EXP	0.1	0.7	0	
4	Undeveloped	Lead	EMC	0	0	0	

Figure 17.3 The SWMM Washoff Editor

Identification and Connectivity

Table 17.5 The Washoff Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Washoff ID	ID of Washoff	Yes	MUID
Pollutant ID	ID of Pollutant	Yes	PollutantID
LandUse ID	ID of Landuse	Yes	LanduseID
Description	User's descriptive information	No	Description

Washoff Properties

Table 17.6 The Washoff Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Function	Function Type (Exponential, RC and EMC)	Yes	FuncTypeNo
Washoff Coeff 1	Coefficient	Yes	C1
Washoff Coeff 2	Coefficient	Yes, if Function is Exponential or RC	C2
Sweep efficiency	Street sweeping removal efficiency in percentage	Yes	SweepEfficiency
BMP efficiency	Removal efficiency of BMP in percentage	Yes	BMPEfficiency

17.5 Loading

Input loadings of pollutants from external and dry weather inflows are supplied through time series data associated with particular nodes of the collection system.The definition of the boundary connection of external timeseries to a node is done in the CS SWMM Loading Editor.

The CS SWMM Loading Editor organizes the related data for setting up the boundary connection of the pollutant inflow timeseries to the model into the following groups:

- Identification and Connectivity - General identification and location information
- Loading Properties - Initial Buildup data

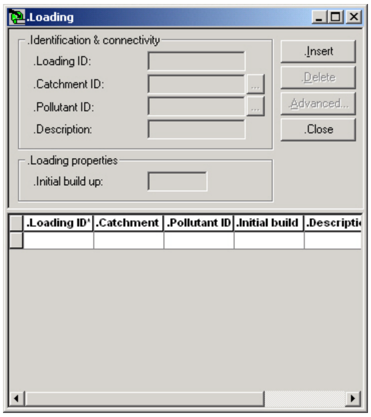

.Loading
.Identification & connectivity
.Loading ID:
.Catchment ID:
.Pollutant ID:
.Description:
.Insert
.Delete
.Advanced...
.Close
.Loading properties
.Initial build up:
.Loading ID'
.Catchment
.Pollutant ID
.Initial build
.Descripti

Figure 17.4 The SWMM Loading Editor dialog

Identification and Connectivity

Table 17.7 The Loading Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Loading ID	ID of Loadpoint (Node)	Yes	MUID
Catchment ID	ID of Catchment	Yes	CatchID
Pollutant ID	ID of Pollutant	Yes	PollutantID
Description	User’s descriptive information	No	Description

Loading Properties

Table 17.8 The Loading Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Initial buildup	Initial buildup of pollutant	Yes	InitBuildUp

17.6 Pollutant

The generation, inflow and fate of any number of user-specified pollutants can be modelled. Concentrations in rainfall, in groundwater, in inflow/infiltration is modelled.

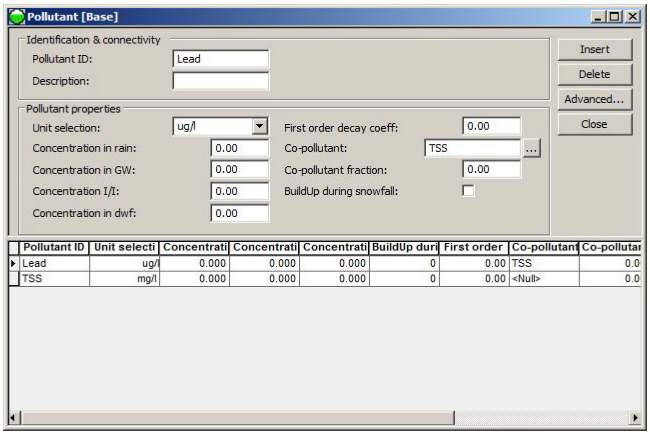
There can be used following different units for the individual user defined pollutant units (milligrams/liter, micrograms/liter, or counts/liter)

Co-pollutants can also be modelled. For example, pollutant X can have a co-pollutant Y, meaning that the runoff concentration of X will have some fixed fraction of the runoff concentration of Y added to it.

The pollutant buildup and washoff on catchment areas are determined by the Land Uses assigned to those areas.

The CS SWMM Pollutant Editor organizes the related input data for the pollutants into the following groups:

- Identification and Connectivity - General identification and location information
- Pollutant Properties - Pollutant related data



Identification & connectivity

Pollutant ID: Lead

Description:

Pollutant properties

Unit selection: ug/l

Concentration in rain: 0.00

Concentration in GW: 0.00

Concentration I/I: 0.00

Concentration in dwf: 0.00

First order decay coeff: 0.00

Co-pollutant: TSS

Co-pollutant fraction: 0.00

BuildUp during snowfall: ☐

Pollutant ID	Unit selecti	Concentrati	Concentrati	Concentrati	BuildUp duri	First order	Co-pollutant	Co-pollutar
Lead	ug/l	0.000	0.000	0.000	0	0.00	TSS	0.0
TSS	mg/l	0.000	0.000	0.000	0	0.00	<Null>	0.0

Figure 17.5 The SWMM Pollutant Editor

Identification and Connectivity

Table 17.9 The Pollutant Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Pollutant ID	ID of Pollutant	Yes	MUID
Description	User's descriptive information	No	Description

Pollutant Properties

Table 17.10 The Pollutant Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Unit Selection	Unit selection of pollutant	Yes	TypeNo
Concentration in rain	Concentration of pollutant in rain	Yes	Crain
Concentration in GW	Concentration of pollutant in groundwater	Yes	Cgw
Concentration I/I	Concentration of pollutant in I/I	Yes	Cii
First order decay coeff	First order decay coefficient	Yes	Kdecay
Co-pollutant	ID of the co-pollutant	Yes	CoPollut
Co-pollutant fraction	Fraction of co-pollutant	Yes	CoFract

Table 17.10 The Pollutant Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Buildup during snowfall	Buildup during snowfall. Yes if buildup only occurs during snowfall, no otherwise	Yes	SnowFlag
Concentration dwf	Concentration of pollutant in DWF	Yes	Cdwf

17.7 Local Treatments

Removal of pollutants contained in the flow into a node can be modelled by assigning a set of treatment functions to the node. Treatment functions can be any well-formed mathematical expression involving:

- pollutant concentrations at the node's inlet (use the pollutant name to represent a concentration)
- removals of other pollutants (use R_ prepended to the pollutant name to represent removal)
- process variables which include:
- FLOW for flow rate into node (user's flow units)
- DEPTH for water depth above node invert (ft or m)
- AREA for node surface area (ft2 or m2)
- DT for routing time step (sec)
- HRT for hydraulic residence time (hours)

The result of the treatment function can be either a concentration (denoted by the letter **C**) or a fractional removal (denoted by **R**).

For example, a first-order decay expression for BOD exiting from a storage node might be:

$$C = BOD \bullet e^{-0{,}05 \cdot HRT} \tag{17.6}$$

while having the removal of some trace pollutant be proportional to the removal of total suspended solids (TSS) would be written as:

$$R = 0{,}75 \bullet R\_TSS \tag{17.7}$$

The CS SWMM Treatment Editor organizes the related input data for the local treatment into the following groups:

- Identification and Connectivity - General identification and location information

- Local Treatment Properties - Treatment function

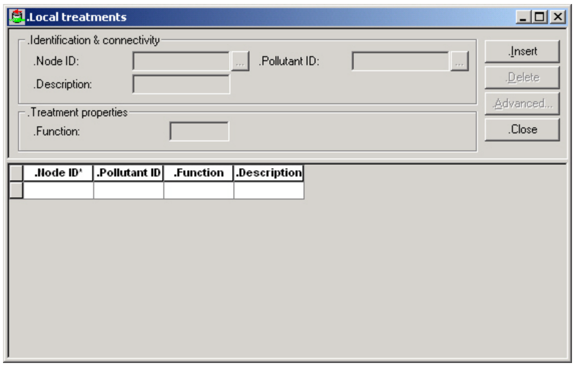

.Local treatments
.Identification & connectivity
.Node ID:
.Pollutant ID:
.Description:
.Treatment properties
.Function:
.Insert
.Delete
.Advanced...
.Close
.Node ID'
.Pollutant ID
.Function
.Description

Figure 17.6 The SWMM Treatment Editor dialog

Identification and Connectivity

Table 17.11 The Pollutant Identification and Connectivity Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Node ID	ID of Node where local treatment is performed	Yes	MUID
Pollutant ID	ID of pollutant	Yes	PollutantID
Description	User's descriptive information	No	Description

Treatment Properties

Table 17.12 The Treatment Properties Group

Edit field	Description	Used or required by simulations	Field name in datastructure
Function	Function expressing the treatment result in terms of pollutant concentrations, pollutant removals, and other standard variables.	Yes	Function

INDEX

B
Basin geometry 157
Biological Processes 302

C
Capacity Curve 157
Conduits 358
Control action 245
Controllable devices 251
Cross sections 66

F
Flooding 207
Flow Dividers 350
FST 208

H
Hydrological models 73

I
Inlet 49

J
Job list 267
Junctions 350

K
Kinematic Wave 80

L
Linear Reservoir 84
Long Term Statistics 265

N
Nodes and Structures 22

O
Orifice 44
Outfalls 350

P
PID 246
Pipe Design 289
Pipes and Canals 36
Pump 54

Q
QH Relation 157

R
RDI .92

S
Sediment transport 301
Sensor 242
Soakaway. 29
Storage Units 350

T
Time series 137
Time-Area Curve 157
Time-Area Method 75

U
Unit Hydrograph Method 89

W
Weirs 41