

MIKE URBAN Tutorials

Step-by-Step Training Guide



PLEASE NOTE

COPYRIGHT	This document refers to proprietary computer software which is pro- tected by copyright. All rights are reserved. Copying or other repro- duction 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 Agree- ment without these limitations and exclusions on its liability. These limitations and exclusions will apply notwithstanding any failure of essential purpose of any limited remedy.





MIK	E URB	AN tuto	rials Step-by-Step Training Guide
1	Tutor	ials Ove	Prview11Model Manager Tutorials11MOUSE Tutorial11SWMM Tutorial11Water Distribution Tutorials11
2	Mode	l Manag	jer Tutorials
	2.1	Import o	f data: Importing ArcView Shapefiles 13 GIS Water Distribution Network Data 14 GIS Storm- and Wastewater Network Data 14
		2.1.1 2.1.2	Step-by-Step: Using the import/export wizard14Importing Water Distribution Data - using the advanced mode25Prepared input and Output Files31Reviewing Import Results31
		2.1.3	Importing Wastewater Network Data - using the advanced mode
	2.2		f data: Exporting MIKE URBAN Data to ESRI Shapefiles - using the ad- mode 41 Prepared input and Output Files
	2.3	Toolbox 2.3.1 2.3.2 2.3.3 2.3.4 2.3.5 2.3.6	43Introduction43Lateral Snapping Tool43Overview43Lateral Snapping: Step-by-Step Procedure44Cross-section Generation Tool51Overview51Cross-section Generation: Step-by-Step Procedure51Auto Connection Tool57Overview57Auto Connection: Step-by-Step Procedure58Sequential Labelling Tool64Overview64Sequential Labelling: Step-by-Step Procedure65Catchment Slope and Length Tool67
			Catchment Slope and Length: Step-by-Step Procedure
3			rial
	3.1 3.2	3.1.1 Definitio 3.2.1 3.2.2	und
		3.2.3	Step 3: Graphical Digitalisation of the Network80Adding nodes81



	3.3	3.2.4 Definition 3.3.1 3.3.2 3.3.3 3.3.4 3.3.5	Adding links
	3.4	MOUSE	Simulations 95 The Runoff Simulation 96 The Network Simulation 98
	3.5	Result P 3.5.1 3.5.2 3.5.3 3.5.4 3.5.5	resentation99Step 1: Loading Results100Step 2: Viewing Result Time Series101Step 3: Profile Plots and Animations102Step 4: Adding Result Layers105Step 5: Result Statistics106
4	SWM 4.1 4.2 4.3	Backgrou Data . 4.2.1	al 109 und 109
	4.4	4.3.1 4.3.2	Step 1: Inserting the rain gauge parameters 112 Step 2: Editing the rain gauge parameters 113 atchments and editing the catchment parameters 114 Step 1: Adding the catchments 115 Step 2: Assigning catchment parameters 117 Step 3: Assigning routing parameters 119
	4.5 4.6		Step 4: Assigning catchment infiltration data121on122on Results125
5	2D O 5.1 5.2	Introduct	Flow Example 127 ion 127 the model 127
6	Water 6.1	r distrib	ution tutorials129I - Defining a Pipe Network System129Defining the Project130Dynamic Data Input131Defining Junction Node Data132Horizontal Plan Options134Displaying Multiple Dialog Boxes135Network Definition Methods136Defining Tank Data137

	6.1.4	Defining Pipe Data		
	6.1.5	Defining Pump Data		
		Flow Pumps		
		Head Pumps		
		Note		
		Graphically Defining Pumps		
	6.1.6	Opening a MIKE URBAN WD Data File		
	6.1.7	Saving Your Data		
	6.1.8	Performing the Analysis		143
	6.1.9	Prepared Input and Output Files		
	6.1.10	Viewing the Analysis Results		
		EPANET Analysis Results		
		Analysis Results Table		
		Component Browser		
		Horizontal Plan Graphical Plots		
~ ~	1	Prepared input and Output Files		
6.2		2 - Pressure reducing valve static analysis		
	6.2.1	Defining a Pressure Reducing Valve		
		Inserting a Pressure Reducing Valve		
		Note		
		Defining PRV Properties		
	6.2.2	Defining a Junction Node Demand Change		
	6.2.3	Defining a Global Demand Change		
	0.2.0	Field Calculator		
	6.2.4	Defining a Pipe Status Change		
	6.2.5	Prepared input and Output Files		
	6.2.6	Reviewing the Analysis Results		160
	6.2.7	Comparing the Analysis Results		160
		Viewing the Comparison Results		.161
		Compared Results of LESSON2B.RES and LESSON2C.RES .		
6.3	Lesson	3 - Extended Period Analysis		
	6.3.1	Defining an Extended Period Analysis		
	6.3.2	Defining and Applying a Demand Pattern		
		Field Calculator		
	6.3.3	Defining Storage Tank Data		167
	6.3.4	Defining Control Rules		168
	6.3.5	Performing an Extended Period Analysis		169
	6.3.6 6.3.7	Prepared Input and Output Files		170 170
	0.3.7	EPANET Analysis Results		
		Results Statistics		
		Results Browser		
		Time Series Plot		
		Map Window Thematic Plots		
	6.3.8	Reviewing Extended Period Analysis Results		



6.4	Lesson	4 - Fire Flow Analysis
	6.4.1	Specifying a Design Fire Flow Rate
		Tabular results
	6.4.2	Specifying a Design Fire Flow Pressure
		Prepared Input and Output Files
	6.4.3	Reviewing the Analysis Results
6.5	Lesson	5 - Water Quality–Source Tracing Analysis
	6.5.1	Defining a Source Tracing Analysis
	6.5.2	Defining the Source Node
	6.5.3	Prepared Input and Output Files
	6.5.4	Performing Source Node Water Results
	6.5.5	Percentage of Source Node Water Results
	6.5.6	Forward and Backward Tracing of Flow
	6.5.7	Reviewing the Analysis Results
6.6	Lesson	6 - Water Quality - Water Age Analysis
	6.6.1	Defining a Water Age Analysis
		Initial Water Quality
	6.6.2	Prepared Input and Output Files
	6.6.3	Water Age Results 197
		Results Statistics
	6.6.4	Reviewing the Analysis Results
6.7		7 - Water Quality - Constituent Chlorine Analysis
	6.7.1	Defining a Constituent Analysis
	6.7.2	Defining Constituent Data
	6.7.3	Prepared Input and Output Files
	6.7.4	Constituent Chlorine Decay Results
		Result Statistics
	6.7.5	Reviewing the Analysis Results
6.8		8 - Distributed Demands and Pressure Zones
	6.8.1	Distributed Demands
	6.8.2	Prepared Input and Output Files
	6.8.3	Reviewing the Demand Distribution Results
		Demand Statistics
6.9		9 - Demand Allocation
	6.9.1	Demand Points Import
	6.9.2	Demand Geocoding
	6.9.3	Demand Aggregation
0.40	6.9.4	Prepared Input and Output Files
6.10		10 - Water Hammer Analysis with Surge Protection
	6.10.1	Without surge protection 225
	6.10.2	With surge protection (air-chamber)

MIKE URBAN TUTORIALS

Step-by-Step Training Guide





1 Tutorials Overview

In this Step-By-Step Training Guide you will find tutorials organized in the following categories:

- Model Manager Tutorials
- MOUSE Tutorial
- SWMM Tutorial
- Water Distribution Tutorials

Below follows a brief overview on the tutorials and where to find the data for them.

Model Manager Tutorials

A number of step-by-steps on how to import data by using either the import/export wizard or the advanced import/export mode. An example on how to export data is also given. Data is provided in Examples\ModelManager. Also a tutorial on how to use the Collection System Toolbox is provided.

MOUSE Tutorial

A step-by-step on how to build a simple MOUSE model within MIKE URBAN from scratch and run a simulation and view the results. Data is provided in Examples\CollectionSystemMOUSE\Tutorial1.

SWMM Tutorial

A step-by-step on how to build a simple SWMM model within MIKE URBAN from scratch and run a simulation and view the results. Data is provided in Examples\CollectionSystemSWMM\Example3.

Water Distribution Tutorials

A number of step-by-steps covering various aspects of Water Distribution modelling with MIKE URBAN. Data is provided in Examples\WaterDistribution and the subfolders found here.





2 Model Manager Tutorials

2.1 Import of data: Importing ArcView Shapefiles

This tutorial takes you step-by-step, illustrating how to use MIKE URBAN to import and export ArcView water distribution and sewer network data.The first part will show how to the import data using the import wizard directly (a collection system is used in the example) - and the latter sections will show how this is done in the advanced mode (for both water distribution and collection system).

MIKE URBAN can import and export network data as ESRI ArcView shape files, allowing the network data to be directly imported and exported into and from ArcView. For example, by exporting the water distribution or sewer network model from MIKE URBAN as an ArcView shape file (.SHP), a GIS database can be quickly developed for a client. Once this data has been imported into the GIS application, join operations can be performed with other database tables. By using geocoding, SQL, and topological queries, the GIS system can provide a wide range of useful thematic maps of water pressure, water quality constituent concentration, etc.

The GIS data used in this tutorial is of an actual water distribution and sewer network from a major metropolitan city. This lesson demonstrates the procedure for exporting and importing GIS data from and back into MIKE URBAN. The project that is used in this lesson is shown in Figure 2.1.

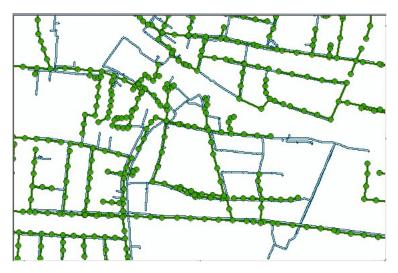


Figure 2.1 The water distribution and sewer network from a major metropolitan city, which is used in the lesson



GIS Water Distribution Network Data

Most municipal GIS systems that contain water distribution network data only store information concerning the pipe data (i.e., location, size, age, etc.) and do not contain information concerning starting and ending junction nodes, pumps, demand patterns, etc. This is because these GIS databases are mainly used for inventory asset management and not for water distribution network modeling. If they do contain data on storage tanks, valves, and pumps, the data stored has to do with inventory management and equipment maintenance—and not information that MIKE URBAN WATER can use for modeling.

Therefore, although MIKE URBAN exports all of the water distribution network link and node information (i.e., pipes, valves, pumps, junction nodes, storage tanks, and reservoirs) to the ArcView shape file, most municipal GIS users are typically only interested in the exported pipe information. Because of this GIS data shortage, MIKE URBAN has been developed so that it does not require any other information than pipe information when importing GIS data for defining a water distribution model. MIKE URBAN will automatically determine the water distribution topological references while it imports the GIS pipe data. However, this requires the modeler to add information defining the storage tanks, reservoirs, pumps, and valves that make up the water distribution network.

GIS Storm- and Wastewater Network Data

Contrary to water distribution GIS systems, GIS systems which contain storm- or wastewater collection network data usually contain information concerning both network pipes and nodes. This is because the nodes (manholes) are physically present in the collection system networks and are not fictive as in the water distribution networks.

However, it is seldom the case that explicit information about the starting and ending nodes for pipes (which is essential for the network topology) is available. Moreover, the nodes contained in the GIS typically do not physically coincide with the starting and ending points of the pipes: a pipe frequently extends over several nodes, as well as no nodes are present at the starting or at the ending points of a pipe. As it is the case with water distribution GIS system, the lack of exact network topology information is due to the fact that most GIS systems have been established with the view of inventory asset management and not for numerical modeling.

MIKE URBAN has been developed to support import of such deficient GIS data and automatic creation of the correct network topology.

2.1.1 Step-by-Step: Using the import/export wizard

To import ArcView GIS wastewater network data into MIKE URBAN, start by creating a new project or opening an existing project. We will create a new



project in this tutorial by selecting File | New and defining the project name and the unit system. Note that it is also possible to use existing model, such as water distribution model to add the wastewater data to it.

 The Create New Project dialog box will appear as shown in Figure 2.2. Select the subdirectory in which to save data, specify the filename IMPORT.MDB and select «OK».

ີ Create New Pi	roject			— ×-
Look in:	퉬 ImportShp	•	G 🤌 📂 🛄 🗸	
Ca.	Name	*	Date modified	Type
Recent Places		No items match your s	earch.	
Desktop				
Libraries				
Computer				
	•			+
Network	Database name:	import.mdb		Create
	Files of type:	MIKE URBAN Database (*.mdb)	-	Cancel
		Open as read-only		
-Working Mode	Coord	finate System		
Collection S		iew or change the settings for the coor em, press 'Edit Coordinate System'.	dinate	
MOUSE SWMM		NING! not all settings can be change	d later.	
Asset		Edit Coordinate System		
🔘 Water Distril		System		
Model	_	I-CUSTOM	•	
			_	

Figure 2.2 Create new project dialog box

2. Next, select File | Import and Export. This will display the Import dialog as shown in Figure 2.3. Press 'Next' to continue.





Figure 2.3 The start page of the import wizard

 Choose to import from ESRI Shape and select the directory where the shape files are located, see Figure 2.4. The default installation of MIKE URBAN will mean that the shape files for this tutorial are found in the example folder: \Examples\ModelManager\Tutorial1. Press 'Next' to continue.

S Import/Export Wizard			- • •
Select Import or Exp	port		
Mode			
Import-Export:			
Import from	ESRI Shape 🔻	 C:\Program Files (x86)\DH 	HI\2016\MIK
C Export to	MOUSE	r	
🔿 Load Import/Export config	file		
		< Back Next	> Cancel

Figure 2.4 Specifying format and files to import from



4. Now specify the name of the import job configuration. This will allow you to reuse the import at a later stage, see Figure 2.5. Press 'Next' to continue.

S Import/Export Wizar	ď		×
Specifying Job	o Configuration		
		Job and Table Configurations	
New Job	ImportFromShape		
💿 Edit Job			
💿 Copy Job			
💿 Delete Job			
🔘 Rename Job			
Topology			
Snap tolerance:			
		< Back Next > Cancel	

Figure 2.5 Specifying the job configuration

5. Choose the transfer settings for the import job (refer to chapter on import/export on these settings). As data is being imported into a new empty database the 'Import and Replace' option is being used, see Figure 2.6.



S Import/Export Wizard	
Select Job Settings	
Transfer Mode Replace existing data Only append to existing data Append and update existing data Only update existing data Delete existing data	
Settings Import from Export to	x86)\DHI\2016\MIKE URBAN\Examples\M
	<pre></pre>



6. You now need to create a table configuration for each table that you wish to import data into, see Figure 2.7. Start by creating one for the nodes table. Press 'Next' to continue. This step will take a couple of seconds.

A Import/Export Wizard			
Specifying Table C	Configuration		
		Table Configurations	
Current Job	ImportFromShape	····· ImportFromShape	•
New Table Config	NodesImport		
🔘 Edit Table Config	· · · · · · · · · · · · · · · · · · ·		
🔘 Copy Table Config			
🔘 Delete Table Config			
🔘 Rename Table Config	·		
No changes on section leve	I		
Info: When pressing [Next] dat	ta will be loaded this process may	take some time.	
		< Back Next >	Cancel



7. Select the source table, i.e. where data comes from, and select the target table, i.e. where data is imported into. The source table for the nodes



is the shapefile named cs_nodes.shp, the target table is the nodes table for MOUSE, msm_Node. See also Figure 2.8. Press 'Next' to continue.

Import/Export Wizard			×
Select Source, Target and	Transfer M	lode	
Select Source Table:	Select Target Tab	ole:	
Replicate Source Datastructure in Ta	arget		
Transfer Mode			
Import and Replace	Source ID	_X	~
C Update C Update and Append	Target ID	_X	~
C Append			

Figure 2.8 Selecting source and target tables

8. Now you need to map data. This is done by first pressing the 'Automap' button. Automap will map identical variable names as these are assumed to be mapped to each other. If this is not the case the assignments can be changed. For the other fields the assignment needs to be done as seen in Figure 2.9. Press 'Next' to continue.



 Target	Source	A	Automap
 SHAPE	Shape		
 MUID			Clear Assignments
 TypeNo			Advanced Edit
 InvertLevel			
GroundLevel	GROUNDLEVE	,	
Diameter	DIAMETER		
Source	Source Unit		Specify Units
			Clear Unit



9. You will now be prompted if you have more table configurations to continue with or if you are ready to proceed with import. As you wish to import the pipes as well, choose to continue with adding table configurations, Figure 2.10, and press 'Next'.

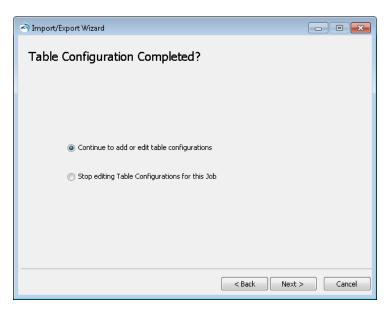


Figure 2.10 Continue to add table configurations

10. The table configuration for the pipes table needs to done now as shown in . Press 'Next' to continue.

齐 Import/Export Wizard			
Specifying Table C	Configuration		
		Table Configurations	
Current Job	ImportFromShape	ImportFromShape	
New Table Config	pipesImport		
🔘 Edit Table Config			
🔘 Copy Table Config			
🔘 Delete Table Config			
🔘 Rename Table Config			
🔘 No changes on section leve			
7-6 10h		Later error Pers	
Inro: when pressing [Next] dat	ta will be loaded this process may	take some time,	
		< Back Next >	Cancel

Figure 2.11 Adding a new table configuration for the pipes

11. Select the source table, i.e. where data comes from, and select the target table, i.e. where data is imported into. The source table for the pipes is the shapefile named cs_pipes.shp, the target table is the nodes table for MOUSE, msm_Link (for further on the tables please refer to MIKE URBAN Tables documentation). Press 'Next' to continue.

ব্দ Import/Export Wizard		- • •
Select Source, Target and	l Transfer M	ode
Select Source Table:	Select Target Tab	e:
cs-pipes 🔻	msm_Link	-
Replicate Source Datastructure in T	arget	
Transfer Mode		
 Replace existing data 	Source ID	_FromNodeID 👻
Only append to existing data Append and update existing data	Target ID	_FromNodeID 👻
 Append and opdate existing data Only update existing data 		
		< Back Next > Cancel

Figure 2.12 Selecting source and target tables

12. Now you need to map data for the pipes. This is done by first pressing the 'Automap' button. Automap will map identical variable names as these are assumed to be mapped to each other. If this is not the case the assignments can be changed. For the other fields the assignment needs to be done as seen in Figure 2.13. Press 'Next' to continue.

S Ir	mport/Export Wizard		
As	signment Spe	cification	
	Target	Source	Automap
	MUID		Clear Assignments
	TypeNo		
	UpLevel		Advanced Edit
	DwLevel		
•	Length	SHAPE_LENG	•
	UpLevel_C		.
	Source	Source Unit	Specify Units
*			Clear Unit
			< Back Next > Cancel

Figure 2.13 Assignment specification for the pipes table

13. You will now be prompted if you have more table configurations to continue with or if you are ready to proceed with import. As you are done choose to stop adding or editing tables, Figure 2.14, and press 'Next'.



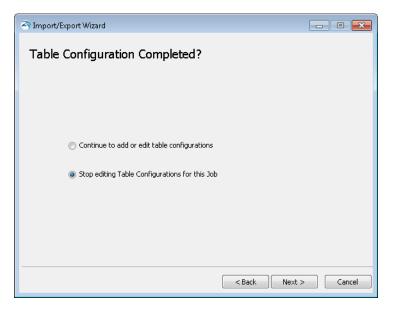


Figure 2.14 Stop adding table configurations

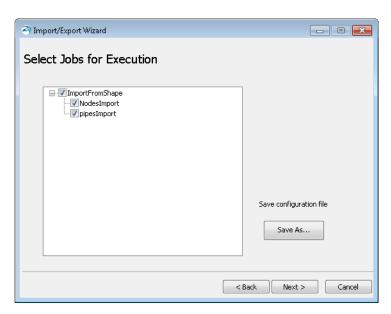
14. The prompt will ask if the import job is complete or if you wish to add more jobs. As you are done, choose as shown in Figure 2.15.

Job Configuration Completed?	
Continue to add or edit jobs	
Proceed with import/export of data	
	< Back Next > Cancel
	<back next=""> Cancel</back>

Figure 2.15 The job configuration is complete

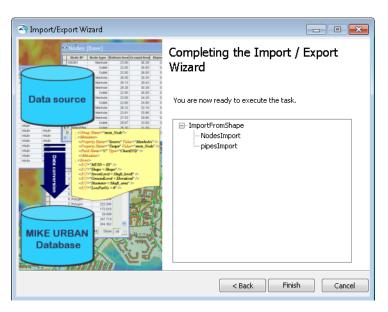
15. You can choose to save the configuration file for later use and to toggle jobs and table configurations on/off in the execution as shown in Figure 2.16. Press 'Next' to continue.







16. The final page - press 'Finish' to execute the import, Figure 2.17. This may take some time depending on the amount of data being imported.





The result of the import is seen in Figure 2.18.



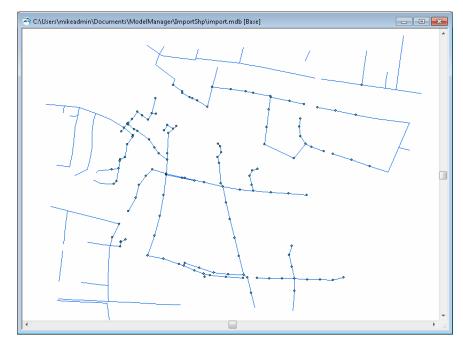


Figure 2.18 The result from the import.

2.1.2 Importing Water Distribution Data - using the advanced mode

To import ArcView GIS water distribution network data into MIKE URBAN, start by creating a new project or opening an existing project. We will create a new project in this tutorial by selecting File | New and defining the project name and the unit system.



1. The Create New Project dialog box will appear as shown in Figure 2.19. Select the subdirectory in which to save the data, specify the filename IMPORT.MDB and select «OK».

Create New Project	<u>?</u> ×
Save jn: 🗀 ImportShp	- 🖬 📩 🖃
File memory	
File name: Import	Save
Save as type: MIKE URBA	N Database (*.mdb)
Working Mode	Coordinate System
C Collection System	To view or change the settings for the coordinate system, press 'Advanced'.
I SW/MM	WARNING! not all settings can be changed later.
🗖 Asset	Edit Coordinate System
Water Distribution	Unit System
Asset	SI_LPS

Figure 2.19 Create new project dialog box

2. The ArcView shapefiles to be imported can be displayed in MIKE URBAN and thus it is possible to check the graphical representation of data, database structure and field attributes. To display the ArcView shapefile, select the Insert Layer tool (Edit|Insert Layer). This will display the Insert Layer dialog box as shown in Figure 2.20. Select Zoom Extent to display the entire layer of the external data.

Insert Layer		×
Look in: 🛛 🛜	C:\Users\mikeadmin\Documents 🔹 🏠 🏥 🖛 🖆 🗊 🚳	9
WD.mdb WD-CS.mdl cs-nodes.sh cs-pipes.sh wd_pipes.sh	P	
Name: Show of type:	wd_pipes.shp Add Datasets and Layers Cancel	

Figure 2.20 Insert Layer dialog box



3. To view the ArcView data attributes, select Tools | Identify to open the Identify tool and click the objects in the Map window to browse the data fields and values.

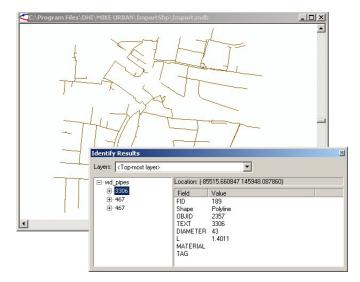


Figure 2.21 Identify tool

4. Next, select File | Import and Export. This will display the Import dialog box as shown in Figure 2.22. In the Import and Export dialog box, select the WD-ImportSHP-Pipes option and then select the source directory in which the source ArcView shapefile is located. The transfer mode allows you to either import the shapefiles as a new project, append it to the existing model data, or update the existing model data from the shapefiles. Select Import and replace existing data.



Import/Export Default Read Only	Settings (WD-ImportSHP-Pipes&Nodes)	x
Import MOUSE	Basic Source Target Bridge Topology	
Import SWMM	Source	
Import EPANET Append MIKE URBAN	C:\Program Files (x86)\DHI\2016\MIKE URBAN\ImportShp	
	Target	
Export MOUSE	RunTime Workspace	
Export SW/MM	Transfer Mode	
Export EPANET full Export EPANET without vertices	Replace existing data	
	Only append to existing data	
WD-ImportSHP-Pipes	Append and update existing data Only update existing data	
WD-ImportSHP-Pipes&Nodes	Only update existing data	
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 XML		
Connector To Shapefiles		
Backup To Simple XML		
Restore From Simple XML		
<u>A</u> pply	V Show Details	

Figure 2.22 Import and Export dialog box

5. The user defined format settings will now appear as shown in Figure 2.23. First, it is necessary to select the source shapefile name such as "WD_Pipes"



(
💷 Import/Export Default Read Only	Settings (WD-ImportSHP-Pipes&Nodes/Pipes)				
Import MOUSE	Basic Assignment Source Target				
Import SW/MM Import EPANET					
Append MIKE URBAN	Source wd_pipes				
	Target mw_Pipe				
Export MOUSE	Table Config. Mode				
Export SW/MM	Replace existing data				
Export EPANET without vertices	Only append to existing data Append and update existing				
	Only update existing data				
WD-ImportSHP-Pipes WD-ImportSHP-Pipes&Nodes	Source Filter				
Pipes	Source Sorting				
Nodes					
CS-ImportSHP-Pipes&Nodes	Source Distinct				
WD-Export to Shapefiles	Load Target Data Before Conversion				
CS-Export to Shapefiles	Save Target Immediately After Conversion				
CS-Update From Pipe Design					
Connector To Project					
Connector To Simple PFS					
Connector To Simple XML					
Backup To Simple XML					
Restore From Simple XML					
Apply	Show Details				
	J				

Figure 2.23 Basic settings for Pipes dialog box

6. The user defined import dialog settings are also used to select which attribute fields contained within the ArcView shapefile correspond to the equivalent MIKE URBAN database fields. We will use the following assignment to use the original GIS data in MIKE URBAN, note that the keyword "shape" is used to import the GIS geometry while the other assignments are used to import the field values. In this example, we will be storing the original GIS ID (OBJID) in both MUID and ASSETID fields of MIKE URBAN, so that the ASSETID will maintain the link to the original GIS ID even if the pipe will be split in MIKE URBAN.

shape = shape MUID = OBJID ASSET = OBJID DESCRIPTION = TEXT DIAMETER = DIAMETER MATERIAL = MATERIAL L = LENGTH



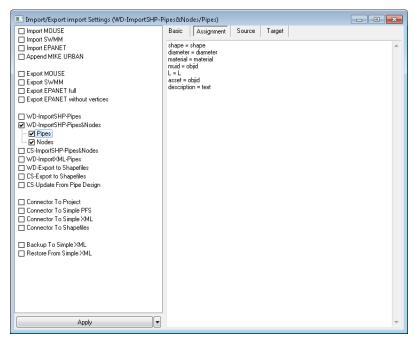


Figure 2.24 Assign Database Attributes for Pipes dialog box

- 7. As discussed previously, water distribution network GIS databases typically do not have a junction node attribute data that cross-reference with the pipe attribute data defining the starting and ending junction nodes. However, the starting and ending junction node information is necessary when defining a water distribution network to be modeled by EPANET. MIKE URBAN handles this by automatically defining junction nodes after importing the pipe data, and will assign a starting and ending node to each imported pipe. This allows MIKE URBAN WATER to easily import any type of line or polyline data to define the water distribution network pipe layout, since the nodal and topological cross-references are automatically established.
- Select "Postprocess Network Topology" (see below) to create the pipe starting and ending nodes automatically and to build-up the network topology.



Import/Export import Settings (WD-ImportSH	P-Pipes&Nodes)	
Import MOUSE Import SWMM Import SPANET Append MIKE URBAN	Basic Source Target	
Export MOUSE	Settings: Name	Value
Export EPANET full	NetworkName JunctionName	mw_net mw_junction
WD-ImportSHP-Pipes WD-ImportSHP-Pipes&Nodes	OrphanName TempName	mw_net_junctions topotemp
Pipes	EdgeName SnapTolerance	mw_pipe 0.1
CS-ImportSHP-Pipes&Nodes WD-ImportXML-Pipes WD-Export to Shapefiles	FinalSnap Prefix	0.1
CS-Export to Shapefiles	RemoveTinyEdges	true
Connector To Project Connector To Simple PFS Connector To Simple XML Connector To Shapefiles		
Backup To Simple XML Restore From Simple XML		
Apply		

Figure 2.25 Topology settings

 Select <OK> to import the ArcView shapfiles. MIKE URBAN will then import the shapefile data (i.e., DBF, SHP, and SHX files) and construct a graphical representation of the water distribution system.

Prepared input and Output Files

Completed input and output files were provided for this lesson. These files are located in TUTORIAL1 directory:

- 1. WD_PIPES.SHP, WD_PIPES.DBF, WD_PIPES.SHX. These files are the input ArcView Shapefiles with the water distribution pipe network. These files are used as the starting files for this lesson.
- 2. WD.MDB. This file is the output file with the imported water distribution network.

Reviewing Import Results

The Map window of MIKE URBAN will display both original ArcView shapefiles (as reference layer) and model pipes i.e. the result of the import procedure, as shown in Figure 2.26.

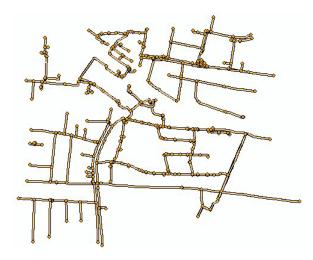


Figure 2.26 Imported model

2.1.3 Importing Wastewater Network Data - using the advanced mode

To import ArcView GIS wastewater network data into MIKE URBAN, start by creating a new project or opening an existing project. We will create a new project in this tutorial by selecting File | New and defining the project name and the unit system. Note that it is also possible to use existing model, such as water distribution model to add the wastewater data to it.



1. The Create New Project dialog box will appear as shown in Figure 2.27. Select the subdirectory in which to save data, specify the filename IMPORT.MDB and select «OK».

🕋 Create New P	roject					×
Look in:	Import	Shp	•	G 🤌 🖻	۶ 🛄 ד	
(Es)	Name		*	Date mod	ified	Туре
Recent Places			No items match you	r search.		
Desktop						
Libraries						
Computer	4					
Network	Database n		import.mdb	,		Create
	Files of type	c	MIKE URBAN Database (*.mdb)		-	Cancel
			Open as read-only			
-Working Mode		Coordi	nate System			
 Collection S 	ystem	To vi	ew or change the settings for the co	ordinate		
MOUSE		syster WAR	m, press 'Edit Coordinate System'. NING! not all settings can be chang	jed later.		
SWMM						
🗖 Asset	hution		Edit Coordinate System			
Model	Duttori	Unit S	ystem			
Asset		SI	-CUSTOM	•		

Figure 2.27 Create new project dialog box

2. The ArcView shapefiles to be imported can be displayed in MIKE URBAN and thus it is possible to check the data graphical representation, database structure, and field attributes. To display the ArcView shapefile, select the Insert Layer tool (Edit|Insert Layer). This will display the Insert Layer dialog box as shown in Figure 2.28. Select Zoom Extent to display the entire layer of the external data.



Insert Layer	
Look in: 🛛 🛜 🛛	C:\Users\mikeadmin\Documents 🔹 📤 🏢 🖛 😂 🐼 🖆 🗊 🚳 🧊
WD.mdb WD-CS.mdl cs-nodes.sh cs-pipes.sh wd_pipes.sh	P P
Name:	cs-nodes.shp; cs-pipes.shp Add
Show of type:	Datasets and Layers Cancel

Figure 2.28 Insert Layer dialog box

3. To view the ArcView data attributes, select View | Info to open the Info tool and click the objects in the Map window to browse the data fields and values.

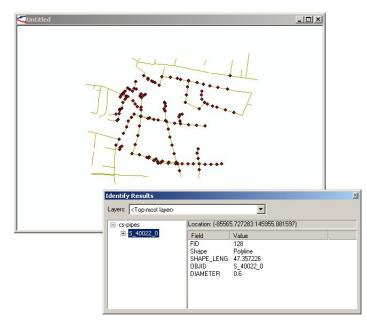


Figure 2.29 Info tool

4. Next, select File | Import and Export. This will display the Import dialog box as shown in Figure 2.30. In the Import and Export dialog box, select the CS-ImportSHP-Pipes-Nodes option and then select the source directory in which the source ArcView shapefile is located. The transfer mode allows you to either import the shapefiles as a new project, append it to



the existing model data, or update the existing model data from the shapefiles. Select Import and replace existing data.

Import/Export Default Read	Only Settings (CS-ImportSHP-Pipes&Nodes)	×
Import MOUSE Import SWMM	Basic Source Target Bridge Topology	
Import EPANET Append MIKE URBAN	C:\Program Files (x86)\DHI\2016\MIKE URBAN\ImportShp	
Export MOUSE	Target RunTime Workspace	
Export SWMM Export EPANET full Export EPANET without vertice	Transfer Mode ◎ Replace existing data ◎ Dnly append to existing data	
WD-ImportSHP-Pipes WD-ImportSHP-Pipes&Nodes WC-ImportSHP-Pipes&Nodes CS-ImportSHP-Pipes&Nodes	 Append and update existing data Only update existing data Delete existing data 	
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		
☐ Backup To Simple XML ☐ Restore From Simple XML		
۰ III ا		
Load Settings	✓ Show Details	

Figure 2.30 Import and Export dialog box

 The user defined format settings will now appear as shown in Figure 2.31. First, it is necessary to select the source shapefile name such as "CS_Pipes"



Import/Export Default Read	Only Settings (CS-ImportSHP-Pipes&Nodes/Pipes)
Import MOUSE Import SWMM	Basic Assignment Source Target
Import EPANET Append MIKE URBAN	Source cs-pipes
Export MOUSE Export SWMM Export EPANET full Export EPANET without vertice	Target msm_Link Table Config. Mode Peplace existing data Only append to existing data Append and update existing Only update existing data
UWD-ImportSHP-Pipes&Nodes	Source Filter
CS-ImportSHP-Pipes&Nodes	Source Distinct
└── ✔ Nodes	Load Target Data Before Conversion Save Target Immediately After Conversion
Connector To Project Connector To Simple PFS Connector To Simple XML Connector To Shapefiles	
Backup To Simple XML Restore From Simple XML	
< □ □ ► Load Settings ▼	✓ Show Details

Figure 2.31 Basic settings for Pipes dialog box

6. The user defined import dialog settings are also used to select which attribute fields contained within the ArcView shapefile correspond to the equivalent MIKE URBAN database fields. We will use the following assignment to use the original GIS data in MIKE URBAN, note that the keyword "shape" is used to import the GIS geometry while the other assignments are used to import the field values. In this example, we will be storing the original GIS ID (OBJID) in MUID. Note that you can also store the original GIS ID (OBJID) in ASSET field in MIKE URBAN.

shape = shape

MUID = OBJID

DIAMETER = DIAMETER

MATERIAL = MATERIAL

LENGTH = SHAPE_LENG



Import/Export Default Read	Only Settings (CS-ImportSHP-Pipes&Nodes/Pipes)
Import MOUSE	Basic Assignment Source Target
Import SWMM Import EPANET	shape = shape
Append MIKE URBAN	diameter = diameter muid = objid
Export MOUSE	length = shape_leng
Export SW/MM	
Export EPANET full	
WD-ImportSHP-Pipes	
CS-ImportSHP-Pipes&Nodes	
Nodes	
WD-ImportXML-Pipes	
CS-Export to Shapefiles	
CS-Update From Pipe Design	
Connector To Project	
Connector To Simple PFS	
Connector To Shapefiles	
Backup To Simple XML	
Restore From Simple XML	
4 III +	
Apply	v

Figure 2.32 Assign Database Attributes for Pipes dialog box

 Next, we will define the settings for importing sewer manholes (nodes). First, it is necessary to select the source shapefile name such as "CS_Nodes"



💷 Import/Export Default Read	Only Settings	(CS-ImportSHP-Pipes&Nodes/Nodes)
Import MOUSE Import SWMM	Basic A	ssignment Source Target
Import EPANET Append MIKE URBAN	Source	cs-nodes
Export MOUSE	Target	msm_Node
Export SWMM Export EPANET full Export EPANET without vertice	Only ap Append	g. Mode existing data pend to existing data and update existing date existing data
WD-ImportSHP-Pipes	Source Filter	
CS-ImportSHP-Pipes&Nodes	Source Sortin	g
Nodes	Source Distin	
WD-ImportXML-Pipes		get Data Before Conversion
CS-Export to Shapefiles CS-Update From Pipe Design	Save ran	get Immediately After Conversion
Connector To Project		
Connector To Simple PFS		
Connector To Shapefiles		
☐ Backup To Simple XML ☐ Restore From Simple XML		
۰	_	
Apply	🔽 Show Del	ails

Figure 2.33 Basic settings for Nodes dialog box

 The user defined import dialog settings are also used to assign which attribute fields contained within the ArcView shapefile correspond to the equivalent MIKE URBAN database fields. We will use the following assignment to import manholes.

shape = shape

MUID = OBJID

DIAMETER = DIAMETER

INVERTLEVEL= INVERTLEVE

GROUNDLEVEL = GROUNDLEVE

💷 Import/Export Default Read	d Only Settings (CS-ImportSHP-Pipes&Nodes/Nodes) 📃 😑	×
Import MOUSE	Basic Assignment Source Target	
Import SW/MM Import EPANET	shape = shape	
Append MIKE URBAN		
	DIAMETER= DIAMETER INVERTLEVEL= INVERTLEVE	
Export MOUSE	GROUNDLEVEL = GROUNDLEVE	
Export SWMM		
Export EPANET without vertic	C.	
	-	
WD-ImportSHP-Pipes		
WD-ImportSHP-Pipes&Nodes ✓ CS-ImportSHP-Pipes&Nodes	8	
Pipes		
✓ Nodes		
WD-ImportXML-Pipes		
CS-Export to Shapefiles		
CS-Update From Pipe Design		
Connector To Project		
Connector To Simple PFS		
Connector To Simple XML		
Connector To Shapefiles		
Backup To Simple XML		
Restore From Simple XML		
<		
Apply		<u> </u>

Figure 2.34 Assign Database Attributes for Nodes dialog box

 Select "Postprocess Network Topology" (see below) to allow MIKE URBAN to create the pipe starting and ending nodes and to build-up the network topology.

Postprocess Networ Settings:	k Topology				
Name	Value				
NetworkName	msm_net				
JunctionName	msm_node				
OrphanName	msm_net_junctions				
TempName	FempName topotemp				
EdgeName	msm_link				
SnapTolerance	0.1				
FinalSnap	0.1				

Figure 2.35 Topology settings

- Select <OK> to import the ArcView shapfiles. MIKE URBAN will then import the shapefile data (i.e., DBF, SHP, and SHX files) and construct a graphical representation of the water distribution system.
- 11. construct a graphical representation of the collection system.



Nodes i.e. sewer manholes were imported from the "cs-nodes.shp" shapefile and they were linked to the adjacent links as a part of the topology post-processing. Note, that it is also possible to create these nodes automatically i.e. without importing them from the shapefile.

In order to create beginning and ending link nodes automatically, check-off importing of "Nodes" in the import task definition.

Prepared input and Output Files

Completed input and output files were provided for this lesson. These files are:

- 1. CS_PIPES.SHP, CS_PIPES.DBF, CS_PIPES.SHX. These files are the input ArcView Shapefiles with the wastewater pipe network. These files are used as the starting files for this lesson.
- CS_NODES.SHP, CS_NODES.DBF, CS_NODES.SHX. These files are the input ArcView Shapefiles with the wastewater manholes. These files are used as the starting files for this lesson.
- CS.MDB. This file is the output file with the imported wastewater distribution network.

Reviewing Import Results

The Map window of MIKE URBAN will display both original ArcView shapefiles (as reference layer) and model pipes i.e. the result of the import procedure, as shown in Figure 2.36.

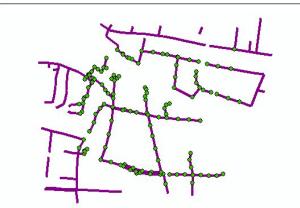


Figure 2.36 Imported model

2.2 Export of data: Exporting MIKE URBAN Data to ESRI Shapefiles - using the advanced mode

To export a water distribution or wastewater network from MIKE URBAN projects to ArcView Shapefiles:

1. Select File | Import and Export to display the Import and Export dialog box, as shown in Figure 2.37. In the Import and Export dialog box, create the new Task "Export-SHP".

Import/Export ImportExport Settings (SHP-ex	port)	
Import MOUSE	Basic Source Target Bridge	
Import SW/MM	.Source:	
Import MIKENET	RunTime Workspace	- 1
Import EPANET		
	.Target:	
Export MOUSE	C:\Program Files\DHI\MIKE URBAN\ImportShp	
Export EPANET	□. Transfer Mode	
	Import and replace existing data	
T WD-ImportSHP-Pipes	C Append into existing data	
WD-ImportSHP-Pipes&Nodes	C .Import and update existing data	
WD-ImportXML-Pipes	U Import and update existing data	
	Selected Only	
Connector To Project	1 .Joideled drilly	
Connector To Simple PFS		
Connector To Simple XML		
Connector To Shape Files		
Lesson1-export		
SHP-export		
Lesson02-export		
Lesson02b-export		
Lesson1-import		
Lesson02-import		
1		
Apply 👻	Show Details	

Figure 2.37 Import and Export dialog box

Define the source and target data; use "RunTimeWorkspace" for the source data and select the directory into which the ArcView shapefiles will ne exported from MIKE URBAN.

2. Click <OK>. MIKE URBAN will export the whole network geometry into shapefiles i.e. both WD and CS mode features. The shapefiles

Once the water or wastewater network has been exported to ArcView GIS, the pipe network layout can be updated (i.e., pipes added, deleted, moved, resized, edited, etc.) within the GIS whenever there are changes to the water distribution network. If an updated analysis of the water or wastewater network system is required, the water distribution network contained within ArcView GIS can be exported back into MIKE URBAN and re-analyzed.

Prepared input and Output Files

Completed input and output files were provided for this lesson. These files are:



1. WD-CS.MDB This file is the input file with the water distribution and wastewater network, which will be used for export to the ArcView shape-files.

2.3 Toolbox

2.3.1 Introduction

Urban stormwater flood modelling can be carried out using a 1D/1D or a 1D/2D approach. The 1D/1D approach has a simplified representation of the overland surface hydraulics compared to the 1D/2D approach.

With the 1D/1D approach, simulation times are considerably shorter compared to the 1D/2D approach and is therefore more suitable for detailed design option runs. However configuring the 1D/1D model is more time-consuming than the 1D/2D model.

A 1D/1D stormwater model is typically made up of three main components: the underground sewer/stormwater system, the overland flow system, and rainfall-runoff hydrology.

The CS Toolbox is a new set of tools developed to specifically address some normally time-consuming aspects of building a 1D/1D stormwater model.

The stormwater tools focus on building the stormwater model within a 1D framework, which include:

- Lateral Snapping of Nodes according to the DEM
- Cross Section Extraction from the DEM
- Auto Connection of Overland Network to Storm Water Network
- Sequential Labelling of Nodes
- Catchment Slope and Length

The 4 first tools can be activated from the menu "Tools | Toolbox" or the "Toolbox" toolbar. The last tool can be found under the Catchment Tools menu.

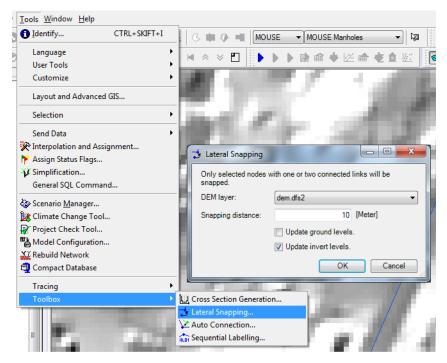
The data for this tutorial can be found in the folder: Examples\ModelManager\CSToolbox.

2.3.2 Lateral Snapping Tool

Overview

When digitising an overland flow path in MIKE URBAN, whether it is along the gutter in a road or the invert of a waterway, it is difficult to locate nodes along the exact invert of the flow path.

The Lateral Snapping tool has been developed for moving selected nodes in MIKE URBAN and snapping them laterally to the lowest DEM value along a Lateral Snapping Alignment.



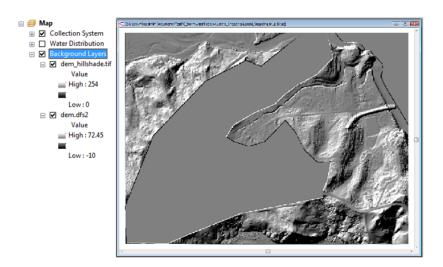
Lateral Snapping: Step-by-Step Procedure

Data:

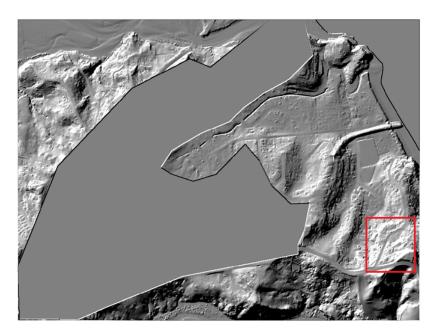
- DEM (Digital Elevation Model) grid file (dem.dfs2)
- Hillshade raster for the DEM or any map that may be used for locating overland flow paths (dem_hillshade.tif)

Procedure:

- 1. Start MIKE URBAN and create a new MOUSE project (leave all the options per default).
- Load background layers. Right click on "Background Layers" at the bottom of the map table of contents and "Insert Layer". Insert the "dem.dfs2" and the "dem_hillshade.tif" files. Arrange them so that "dem_hillshade.tif" is shown above "dem.dfs2."



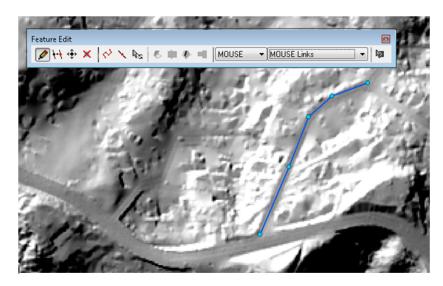
3. Zoom in to the lower-right area of the map where a road could be recognized from the "dem_hillshade.tif" raster as shown in the figure.



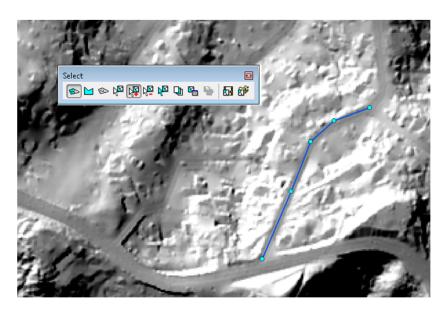
4. Start an editing session and select MOUSE Links as the Active Layer for editing.



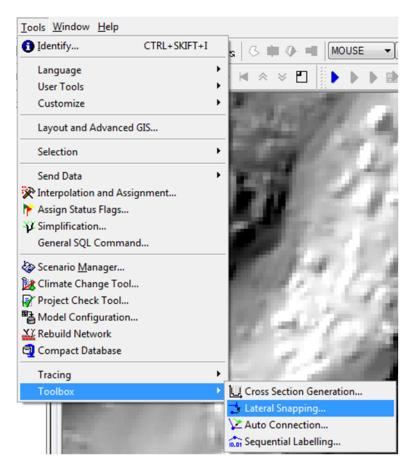
5. Create link features. Place the new links along the road feature seen in the "dem_hillshade.tif" raster. Double-click on the map in order to create intermediate nodes between interconnected link segments.



6. To begin using the Lateral Snapping tool, select the newly created nodes that were generated with the new link features. Note that only the nodes should be selected.



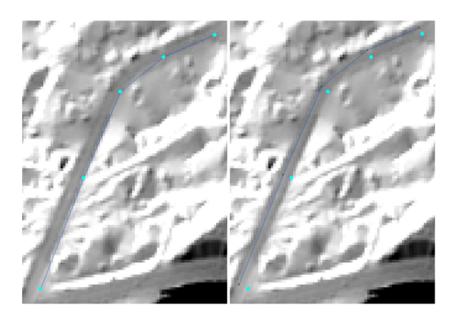
7. Under "Tools | Toolbox", select Lateral Snapping.



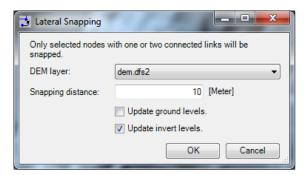
 Select the "dem.dfs2" grid as the "DEM layer" in the dialog (and not the "dem_hillshade.tif" grid as this grid does not represent elevations). Specify the "Snapping distance" (i.e. 10 m). Press OK. Note that the settings used in the tool are stored in the file "StormToolSettings.xml" that can be found in the MIKE URBAN project directory. Press OK.

😼 Lateral Snapping	
Only selected nodes w snapped.	vith one or two connected links will be
DEM layer:	dem.dfs2
Snapping distance:	10 [Meter]
	Update ground levels.
	Update invert levels.
	OK Cancel

9. The selected nodes are then snapped to the lowest point in the DEM within a lateral distance of 10 m.



10. In the Lateral Snapping tool dialog, you may choose to update either the node ground levels or the invert levels. For overland flow path (e.g. street network) nodes, the "Update invert levels" would be more appropriate, while for underground stormwater network nodes the "Update ground levels" option is more fitting.



Please note that you may not have exactly the same value as shown below as you couldn't have drawn the manholes at the exact same location.

Nodes [Base]]						ж
dentification &	connectivity						1
Asset ID:*		Da	ata source:*			Insert	
Node ID:	Node 10	St	atus:*	<null></null>	-	Delete	
Model.*	<null></null>		etwork type:*	<null></null>		Advanced	
	CNOLLS		coordinate:	1752937	•)]
Description:*						Close	ļ
🔲 RM Tail no	ode	Υ	coordinate:	5947113	94		
Tail level:		Lir	nks:	1			
Geometry O		20					
Geometry Q.	H and head loss	2D overland					
Node type:	Manhole	•	Max. Inflov	v			
			Cover				
Diameter:			Type:	Normal			
Ground level:					· · · ·		
Bottom level:	40),06	Buffer pres	sure:	0,00		
			Spill coef:		1,00		
Critical level: Basin geom							
-	cuy		Edit	Creek			
ID;			Edit	Graph			
		ttom level		Diameter	Critical leve		Use I
Node_10	Manhol	40,06	Nul>	<nul></nul>	<nul></nul>	MOUSE Clas	
Node_6 Node_7	Manhole Manhole	57,03 45,02	Null> Null>	<null></null>	<null></null>	MOUSE Clas MOUSE Clas	
Node_8	Manhole	40,55	Null>	<null></null>	<null></null>	MOUSE Clas	
Node_9	Manhole	40,09	Null>	<null></null>	<null></null>	MOUSE Clas	
			-				



2.3.3 Cross-section Generation Tool

Overview

When overland flow paths are defined as MIKE URBAN links, cross sections (CRS) need to be assigned for these overland flow links. Often, a standard road profile will fulfill the modeling requirements. But in other urban flood modeling situations, individual cross sections are required for open spaces, rural areas, park areas, etc.

The Cross Section Generation tool uses cross section alignments drawn in a line feature layer to extract cross sections from a DEM for links intersected by the alignments. It generates cross sections for each link directly into the cross section database as well as set the reference between the link and the generated cross section ID.

Cross-section Generation: Step-by-Step Procedure

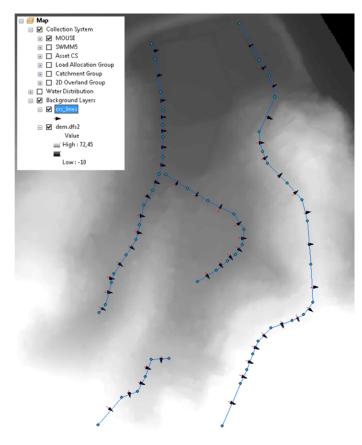
Data:

- A MIKE URBAN project file with overland flow network data (Cross_Section_Generation.mup)
- DEM (Digital Elevation Model) grid file (dem.dfs2)
- ESRI polyline shapefile for cross section alignments (crs_lines.shp)

Procedure:

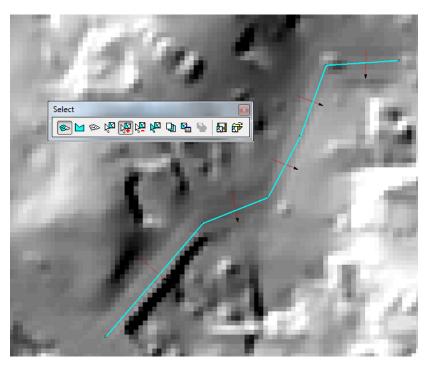
- 1. Open the "Cross_Section_Generation.mup" MIKE URBAN project.
- 2. Insert the cross section lines layer "crs_lines" in the map under "Background Layers." Also add the DEM grid file "dem.dfs2" as a background layer behind the cross section lines.





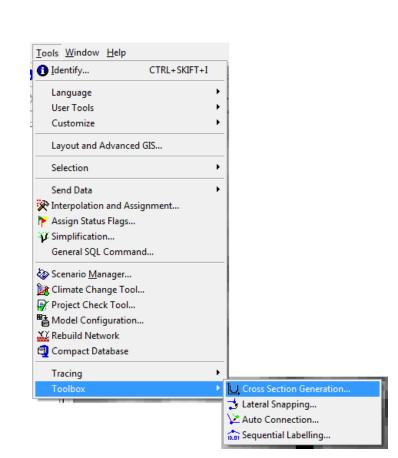
The cross section lines "crs_lines.shp" are polyline features that were created in ArcMap. They represent estimated alignments and widths of cross-sections for overland channel segments.

3. Select the MIKE URBAN links for which you want to extract cross sections.



Note that the tool will generate CRSs for the selected links. If no particular links are selected, the tool will generate CRSs for all links intersecting existing cross section lines.

4. Start an editing session through "Edit | Start Editing". Launch the Cross Section Generation tool through "Tools | Toolbox | Cross Section Generation...".



5. Specify the parameters for the Cross Section Generation tool. Choose the cross section alignment layer "crs_lines" for "Cross section shapes". Choose "dem.dfs2" for "DEM". Specify the number of sample points at which the cross section will be extracted (i.e. 20). Alternatively, you may specify the sample point distance.

Cross Section Generation	
Cross section shapes:	crs_lines
DEM:	dem.dfs2 🔹
Points per CRS:	20
Max distance between CRS points:	10,0 [Meter]
"Wall" height at end of CRS:	1.5 [Meter]
📝 Write CrsID to links (and set Link Sh	ape to "CRS"):
Create slot at the lowest point of the	CRS
Depth of the slot:	0,1 [Meter]
	OK Cancel

Add side walls to the CRS being generated through the "Wall height at the end of CRS" parameter. This will avoid a MIKE URBAN network simulation to stop if the water level in an open channel rises above the defined CRS height. Use a side wall of 1 to 2 meters for an overland flow path such as a road, and 3 to 5 meters for waterways. Adding side-walls adds 2 points to the number of CRS points specified in the tool.

Create a slot at the lowest point of the CRS and specify the depth of the slot as 0.1 m. If a CRS is very flat, too much numerical water will be generated when the link is running dry. Thus, a slot can be inserted at the lowest point of the cross section. Adding a slot adds 2 points to the number of CRS points specified in the tool.

 Press OK to run the Cross Section Generation tool. Cross sections are then extracted into the MIKE URBAN cross section database. The cross section ID (CRS ID) uses the Link ID (or MUID) from which the cross section was extracted.

	1		x
CRS ID:	OL_057_OL_050	Inse	ert
CRS type:	X-Z Open	 Dele 	ete
Description:*		Advan	ced
	Marks	Gra	ph
		CRS g	raph
		Proc.	data
		Clos	se
	<u>.</u>		
X	Z		
0,0000	31,8900		
0,0000	30,3900 30,3225		
0.0410	30.3225		
	30 2150		
1,2831	30,2150		-
1,2831 1 9247 CRS ID *	30 2150 CRS type	Description	+
1,2831 1 9247 CRS ID* OL_050_OL_091	30 2150 CRS type X-Z Open	<null></null>	+
1,2831 1 9247 CRS ID * OL_050_OL_091 OL_055_OL_056	30 2150 CRS type X-Z Open X-Z Open	<nul> <nul></nul></nul>	-
1,2831 1 9247 CRS ID * 0L_050_0L_091 0L_055_0L_056 0L_056_0L_076	30 2150 CRS type X-Z Open X-Z Open X-Z Open X-Z Open	<nul> <nul> <nul></nul></nul></nul>	Ŧ
1,2831 1 9247 CRS ID * OL_050_OL_091 OL_055_OL_056	30 2150 CRS type X-Z Open X-Z Open	<nul> <nul> <nul></nul></nul></nul>	-

CI	RS graph	(OL_057_	OL_050))								x
File	Format											
[m] ²	î		1				 					
33-	1											
32-												
32												
31-												
31-												
30-		- <u> </u>					 			<u> </u>		
30-			1									
29			1									
	1											
28-			1									
	····{····	1	2	3 4	4 .	5 6	 7 8	3 9) 1	0 1	1 1	12 [m]



2.3.4 Auto Connection Tool

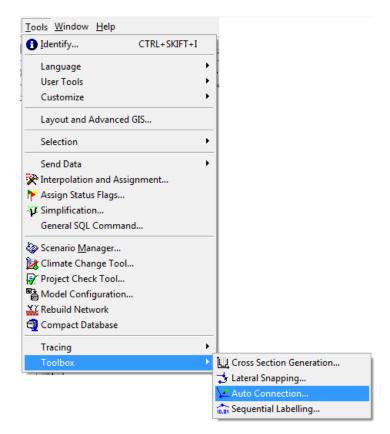
Overview

When overland flow paths have been digitized and snapped to a correct alignment, the overland flow links have to be connected to each other (overland to overland) or connections have to be made between the overland flow network and the underground network.

Manually setting up these connections can be tedious and time consuming. The Auto Connection tool may be used to make this task easier. It has been developed in a generic way so that it can be used for many different types of MIKE URBAN modelling projects and not only in storm water 1D/1D modelling projects.

The tool can generate connections between the same type of network and connections between two different types of network. A network can be defined as i.e. Storm Water, Combined Sewer, Sanitary Sewer or a user-defined network type. The nodes in the model must assign an appropriate network type before proceeding to set up the connections.

The Auto Connection tool is initiated from "Tools | Toolbox | Auto Connection".



Auto Connection: Step-by-Step Procedure

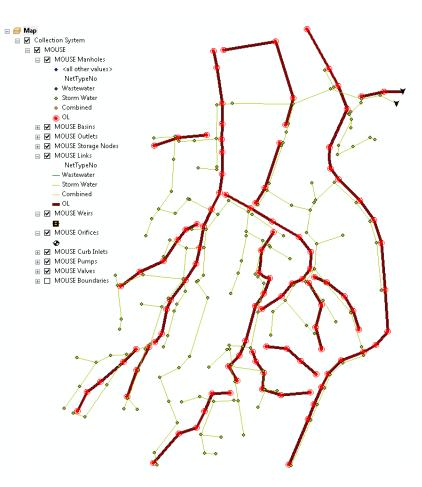
Data:

• A MIKE URBAN project file with underground and overland flow network data (Auto_Connection.mup)



Procedure:

 Open the Auto_Connection.mup MIKE URBAN project. The collection system shown is composed of two types of network—underground and overland.



Information about the network type can be found in the "Network type" and "Description" parameters of nodes and links.

Nodes [Bas	se]						
Identification	n & connectivity						
Asset ID:*			Data source:* StormWater manho			Insert	
Node ID:	1 133	19	Status:*	Importe	-	Delete	
	_					Advanced.	5
Model:*	<nul< td=""><td>L> 🔻</td><td>Network type</td><td>:* Storm</td><td>Water 🔻</td><td>Auvanceu.</td><td>•</td></nul<>	L> 🔻	Network type	:* Storm	Water 🔻	Auvanceu.	•
Description:	:* Under	ground netwo	X coordinate:	175270	09,10	Close	
🔲 RM Tail	node		Y coordinate:	594701	18,65		
Tail level:			Links:	3			
i ali level.			LINKS.	2			
Geometry	Q-H and head	oss 2D overla	bas				
	Q-ri anu neau						
Node type:	Manh	ole 🔻	🔲 Max. Inf	low			
			_				
Diameter:		1,0500	Cover				
Ground leve	el:	29,09	Type:	Norm	al 🔻		
			Buffer pro	essure;	0,00		
Bottom leve	el:	27,63	Spill coef:		1.00		
Critical leve	College Laure Lat			Spin coer,			
	el:*						
-Basin geo							
-				Graph			
Basin geo ID;			Edit	Graph			
-			Edit	Graph			
-			Edit	Graph			
ID;	ometry	Bottom lev			Critical level		Usel
-	ometry	Bottom lev 27,63		Graph Diameter 1,0500	Critical leve <null></null>	ID Weighted Inle	<u>Use I</u>
ID:	Node type *		Ground lev	Diameter			Usel
ID; Node ID * 1 1338	Node type *	27,63	Ground lev 29,09	Diameter 1,0500	<null></null>	Weighted Inle	Usel
ID; Node ID * 1_1338 1_1347	Node type *	27,63 26,30	Ground lev 29,09 28,07	Diameter 1,0500 1,0500	<null> <null></null></null>	Weighted Inle Weighted Inle	Usel
ID; Node ID * 1 1338 1_1347 1_1371	Node type *	27,63 26,30 20,09 19,87	Ground lev 29,09 28,07 22,64 21,63	Diarneter 1,0500 1,0500 1,0500	<null> <null> <null></null></null></null>	VVeighted Inle VVeighted Inle VVeighted Inle	Usel
ID: Node ID * 1 1338 1_1347 1_1371 1_1382 1_1392	Node type * Manhole Manhole Manhole Manhole	27,63 26,30 20,09 19,87 16,07	Ground lev 29,09 28,07 22,64 21,63 18,34	Diameter 1,0500 1,0500 1,0500 1,0500 1,0500 1,0500	<null> <null> <null> <null></null></null></null></null>	Weighted Inle Weighted Inle Weighted Inle Weighted Inle Weighted Inle	Usel
ID: Node ID * 1 1338 1_1347 1_1371 1_1382 1_1392 1_1395	Node type * Manhole Manhole Manhole Manhole Manhole	27,63 26,30 20,09 19,87 16,07 19,40	Ground lev 29,09 28,07 22,64 21,63 18,34 20,89	Diameter 1,0500 1,0500 1,0500 1,0500 1,0500 1,0500	<null> <null> <null> <null> <null> <null> <null></null></null></null></null></null></null></null>	Weighted Inle Weighted Inle Weighted Inle Weighted Inle Weighted Inle	Usel
ID: 1038 1_138 1_1347 1_1371 1_1382 1_1392 1_1395 1_1404	Node type * Manhole Manhole Manhole Manhole Manhole Manhole Manhole	27,63 26,30 20,09 19,87 16,07 19,40 17,50	Ground lev 29,09 28,07 22,64 21,63 18,34 20,89 19,82	Diameter 1,0500 1,0500 1,0500 1,0500 1,0500 1,0500 1,0500	<null> <null> <null> <null> <null> <null> <null> <null> <null></null></null></null></null></null></null></null></null></null>	Weighted Inle Weighted Inle Weighted Inle Weighted Inle Weighted Inle Weighted Inle	Usel
ID: Node ID * 1 1338 1_1347 1_1382 1_1382 1_1395 1_1404 1_1408	Node type * Manhole Manhole Manhole Manhole Manhole Manhole Manhole	e 27,63 26,30 20,09 19,87 16,07 19,40 17,50 14,35	Ground lev 29,09 28,07 22,64 21,63 18,34 20,89 19,82 16,07	Diameter 1,0500 1,0500 1,0500 1,0500 1,0500 1,0500 1,0500 1,0500	<nul> <nul></nul></nul></nul></nul></nul></nul></nul></nul></nul></nul></nul></nul></nul></nul>	Weighted Inle Weighted Inle Weighted Inle Weighted Inle Weighted Inle Weighted Inle Weighted Inle	Usel
ID: 1038 1_138 1_1347 1_1371 1_1382 1_1392 1_1395 1_1404	Node type * Manhole Manhole Manhole Manhole Manhole Manhole Manhole	27,63 26,30 20,09 19,87 16,07 19,40 17,50	Ground lev 29,09 28,07 22,64 21,63 18,34 20,89 19,82	Diameter 1,0500 1,0500 1,0500 1,0500 1,0500 1,0500 1,0500	<null> <null> <null> <null> <null> <null> <null> <null> <null></null></null></null></null></null></null></null></null></null>	Weighted Inle Weighted Inle Weighted Inle Weighted Inle Weighted Inle Weighted Inle	

Overland network nodes and links have been assigned a Network type of "OL" while underground network elements have been assigned a Network type of "Storm Water".

2. Start an editing session. Then, launch the Auto Connection tool through "Tools | Toolbox | Auto Connection".

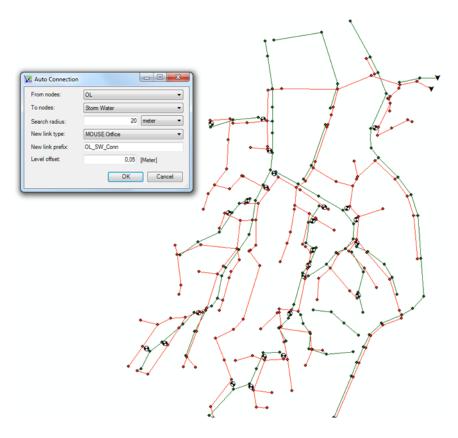
<u>T</u> ools <u>W</u> indow <u>H</u> elp		
1 Identify C	CTRL+SKIFT+I	
Language	+	
User Tools	+	
Customize	•	
Layout and Advanced GIS	S	
Selection	•	
Send Data	•	
🔆 Interpolation and Assignr	ment	
🟲 Assign Status Flags		
🍄 Simplification		
General SQL Command		
😓 Scenario <u>M</u> anager		
💥 Climate Change Tool		
🔐 Project Check Tool		
Model Configuration		
Kebuild Network		
🔁 Compact Database		
Tracing	•	
Toolbox	•	🛄 Cross Section Generation
		📑 Lateral Snapping
		🔀 Auto Connection
		🔝 Sequential Labelling

3. Specify the settings for the Auto Connection tool. Select "OL" for the "From nodes" and "Storm Water" for the "To nodes" to generate connections going from the overland to the underground network.

V Auto Connection	
From nodes:	OL 🔻
To nodes:	Storm Water 💌
Search radius:	20 meter 💌
New link type:	MOUSE Orifice
New link prefix:	OL_SW_Conn
Level offset:	0,05 [Meter]
	OK Cancel

Setting the network type for the "From nodes" and "To nodes" gives more control over the setting-up of connections and also defines the direction of positive flow (From "From nodes" to "To nodes"). For connections from the overland to the underground network the positive flow direction should always be from overland to the underground storm water network.

- 4. Give a value for "Search radius" (e.g. 20 m). The search radius allows the tool to set up connections only between "From node" and "To node" elements at a distance within the search radius from each other. The given value for "Search radius" must be positive.
- 5. Select "MOUSE Orifice" for the "New link type" parameter. Connections between overland and underground networks are usually orifices or storm water inlets. Connections between elements of the overland flow network are usually described by weirs.
- Define a prefix for the IDs of the new connections to be generated by the tool. For example, use "OL_SW_Conn" for connections between the overland and storm water networks.
- 7. Specify a "Level offset" of 0.05 m. It is recommended that a small offset of around 0.05 m is inserted between overland and underground networks. This offset ensures that flow is not always running through the connections.
- 8. Press OK to run the Auto Connection tool. After the new connections are generated, an option for saving a selection file (*.mus) for the connected nodes will be given.



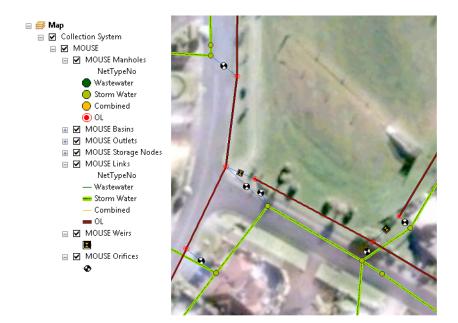
Check the new connections that have been generated by opening the "Orifices" editor through "MOUSE | Orifices".

9. Repeat the process above to make connections between elements of the overland network.

V Auto Connection	
From nodes:	OL 🔹
To nodes:	OL 🔹
Search radius:	20 meter 💌
New link type:	MOUSE Weir 🗸
New link prefix:	OL_OL_Conn
Level offset:	[Meter]
	OK Cancel

The auto connection generation can be controlled by selecting nodes before launching the tool. The tool will make connections for only the selected nodes. If no node is selected, generation of connections will be attempted for all nodes according to the specifications.

An example of connections generated using the Auto Connection tool for a 1D/1D collection system model is shown below.



The green network represents the underground Storm Water network and the red network represents the Overland flow network. The overland network is connected to the stormwater network by orifices while the overland network elements are connected to each other by weirs.

2.3.5 Sequential Labelling Tool

Overview

The Sequential Labelling tool is used for easy and systematic re-labelling of nodes and links in MIKE URBAN. When an overland flow or other type of network is digitised in a MIKE URBAN model, it is often required to provide IDs that are descriptive and intuitive. The IDs are often made up of street name, network type, sub-catchment identifier, etc. depending on the particular naming convention used. Specifying names for model components can be a tedious task and the Sequential Labelling tool is used for this purpose.

A format for the automatically assigned IDs can be specified for selected nodes or links in the map. Label prefixes and suffixes may be given together with sequential numbering parameters.



Sequential Labelling: Step-by-Step Procedure

Data:

 A MIKE URBAN project file with collection system data (Sequential_Labelling.mup)

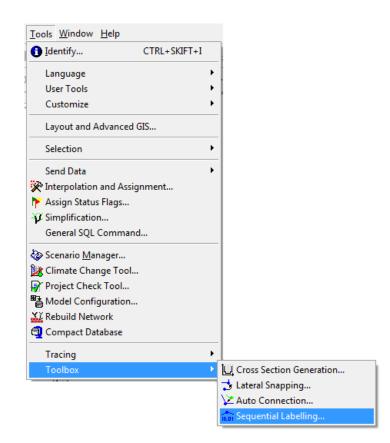
Procedure:

- 1. Load the MIKE URBAN project Sequential_Labelling.mup.
- 2. Start an editing session.
- Select the overland flow network nodes that will be re-labelled. To select these nodes, open the Nodes editor through "MOUSE | Nodes and Structures". Click the "Advanced" button and select the option "Select by attribute." Select items for which "NetTypeNo = 4".

Select by Attribute	X	_
Select from layer: Select method:	Create a new selection	-]
[Data Source] [Description] [SubModelNo] [Net Type No] [Element_S]		-
= <> Like > >= And < <= Or ? • () Not	NULL 2 - Storm Water 4 - OL Get Unique Values Go To:	
SELECT * FROM msm_Nor [NetTypeNo] = 4		1
Clear Venfy	Help Load Save OK Apply Cancel)

The Sequential Labelling tool only works for selected items.

4. Launch the Sequential Labelling tool through "Tools | Toolbox | Sequential Labelling".



5. Specify the settings for the Sequential Labelling tool. Specify a label prefix (e.g. OL_) that will be descriptive for the selected items. Set a start number for the sequential numbering as well as a minimum number of digits for the sequential numbering. A label suffix can also be specified. Check that the example preview of the label is as desired.

💼 Sequential Labe	elling								
Labels will apply to selected elements.									
Map layer:	MOUSE Nodes 💌								
Label Prefix:	0L_								
Start Number:	1								
Minimum Digits:	3								
Label Suffix:	A								
Example:	OL_001_A								
	OK Cancel								

6. Press OK to run the tool.



7. Open the Nodes editor through "MOUSE | Nodes and Structures" to see the results of the sequential labelling operation.

Nodes [Base]					- 0 -	3			
dentification &	connectivity									
Asset ID:*			Data source:*			Insert				
	01.00				Delete					
Node ID:	OL_00		Status:*	<nuli< td=""><td colspan="3"></td></nuli<>						
Model:*	<null< td=""><td>> •</td><td>Network type</td><td>:* OL</td><td>•</td><td>Advanced.</td><td></td></null<>	> •	Network type	:* OL	•	Advanced.				
Description:*	Overlar	nd network	X coordinate:	17524	87,99	Close				
🔲 RM Tail no	ode		Y coordinate:	59470	49,89					
Tail level:			Links:	3	1					
			-	-						
Geometry Q	-H and head lo	ss 2D overla	and							
Node type:	Manho	le 🔻	📃 Max. Inf	low						
Diameter:		5,0000	Cover							
			Type:	Norm	al 🔻					
Ground level:		38,67								
Bottom level:		33,67								
Critical level:	*		Spill coef:		1,00					
Basin geom										
ID;	···· ,		Edit	Graph						
ID;										
Node ID *	lode type *	Bottom lev	Ground lev	Diameter	Critical leve	ID	LA			
OL_001_A	Manhole	33,67	38,67	5,0000	<null></null>	No Cross Se				
OL_002_A	Manhole	24,29	29,29	5,0000		No Cross Se				
OL_003_A	Manhole	10,33	15,33	5,0000		No Cross Se	L			
OL_004_A	Manhole	7,96	12,96	5,0000		No Cross Se	L			
OL_005_A	Manhole	7,42	12,42	5,0000	<null></null>	No Cross Se	L			
OL_006_A	Manhole	6,74	11,74	5,0000		No Cross Se	+			
OL_007_A	Manhole	14,99	19,99	5,0000		No Cross Se	+			
OL_008_A	Manhole	24,72	29,72	5,0000		No Cross Se	+			
OL_009_A	Manhole	24,61	29,61	5,0000		No Cross Se	+			
OL_010_A	Manhole Manhole	16,39 17.88	21,39 22.88	5,0000		No Cross Se No Cross Se				
I OL UTLA	Mannole	17.00	22.00	5.0000	NULLS IN UNITS	NU CIUSS Se				
					-		-11			

8. Follow the same produce to re-label overland flow network links.

2.3.6 Catchment Slope and Length Tool

Overview

The Catchment Slope and Length tool is used for automatically estimating the slope and length of a catchment using catchment delineation information, elevation data from a DEM and flow path lines within a catchment. Catchment length and slope data are needed for some types of rainfall-runoff models,



and the tool estimates hydrological parameters for each catchment in a consistent, documented and reproducible way.

The Catchment Slope and Length tool is initiated from the Catchment Tools menu in MIKE URBAN under MOUSE or SWMM, depending on the collection system model being used, or from the Catchment Tools toolbar.

Nodes and Structures	CTRL+ALT+N		
Pipes and Canals	CTRL+ALT+P		
Weirs	CTRL+ALT+W		
Orifices	CTRL+ALT+O		
Stormwater Inlets		۲	
Pu <u>m</u> ps	CTRL+ALT+U		
Valves			
🖡 Emptying Storage Nodes			
Catchments & catchment p	arameters	►	
Catchment Tools		•	😲 Catchment Delineation Wizard
Runoff Models			🛒 Find Catchment Overlaps
Kunoff Models		-	🕼 Find Catchment Gaps
Boundary Conditions		•	Catchment Connection Wizard
Repetetive Profiles		•	Connect To Node
2D Overland Flow		•	Show All Connected Catchments
2D Overland Tools		•	Show All Disconnected Catchments
Castal		_	Show Connection Location
Control			Show Connected Catchments
Water Quality Long Term Statistics			Catalum ant Day against
			Catchment Processing Catchment Slope and Length
Materials			Catchment Slope and Length
Local Head Losses			
CRS & Topography		•	
Curves & Relations			
Load Allocation		•	

Catchment Slope and Length: Step-by-Step Procedure

Data:

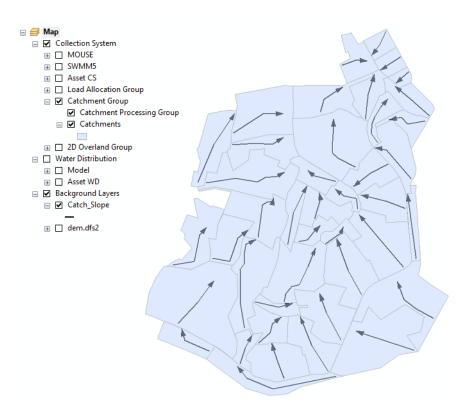
- A MIKE URBAN project with network and catchment data (Catchment_Slope_Length.mup)
- DEM (Digital Elevation Model) grid file (dem.dfs2)
- ESRI line shapefile for flow path/slope lines (Catch_Slope.shp)

Procedure:

- 1. Open the "Catchment_Slope_Length.mup" MIKE URBAN project.
- 2. Add the DEM grid file "dem.dfs2" as a Background Layer.

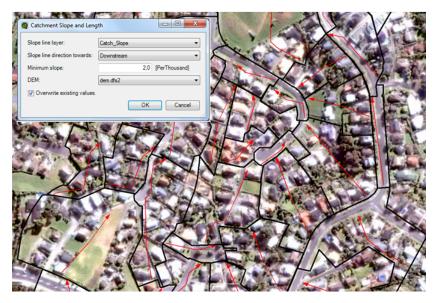


3. Obtain digitized line shapes representing typical flow paths or slope lines within catchments for your model and add it as a background layer in MIKE URBAN. In this example, the "Catch_Slope.shp" file contains the information about catchment slope lines. Add the "Catch_Slope.shp" file as a background layer in the map.



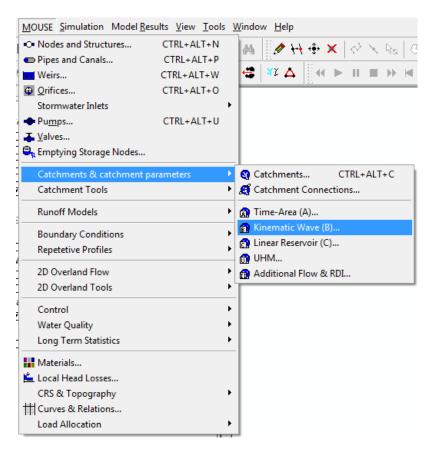
Flow paths can be drawn from the load point or towards the load point but a consistent methodology should be used in a project. A multiple number of slope lines can be defined for each catchment and the slope and length is calculated as an average slope and length of all lines completely within the catchment.

An example from a model is provided below where catchment flow lines have been drawn in a shape file (red lines). In this case the lines have been drawn from the inner catchment towards the load point.



- 4. Start an editing session in MIKE URBAN.
- 5. Ensure that there are records for catchments in the Kinematic Wave (B) parameters table. View this table through "MOUSE | Catchments & catchment parameters | Kinematic Wave (B)".

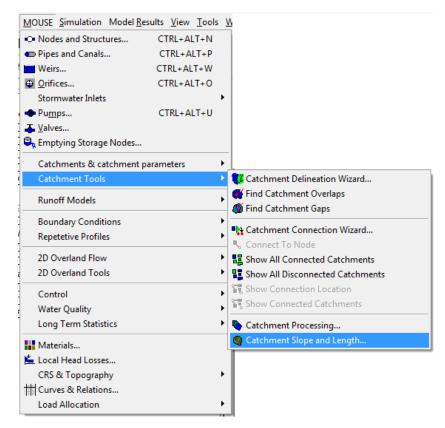




If the Kinematic Wave (B) parameters table is empty, generate records by clicking on "Advanced | Generate model records." This will generate Kinematic Wave (B) parameter records for all the catchments listed in the Catchments table.

ß	Kinematic V	Vave (B) [Base	2]			Į	- O <mark>-</mark>	
	Catchment II	D:	imp1				Insert	
	Catchment a	rea:	1.080				Delete	
	Length:		116.345	Slope:	10	90.39	Advanced	
	-		110,040	Slope.		Select by	attribute	
	- Contributing s		Impervious	Pe		Unselect		
		Ste		Low M				
	Area:	0.00		0.00 0.		Copy to	clipboard	
	Aled.	0,00	100,00	0,00 0,		Paste fro	m clipboard	
	Hydrological p	parameters				Report to	o XML	
	Parameter se	et:	-DEFAULT-			Generate model records		
					\checkmark	Show all		
		parameters				Show selected		
	Manning nur	mber: 80,0) 70,0	30,0 30	-			
					✓	Default I	ayout	
	Catchment	Length	Slope	Parameter	1	ISteep	AIFlat	
Ŀ	imp1	116,345	190,39			0,00	100,00	
Н	imp149	113,490	128,53		_	0,00	100,00	_
Н	imp150	125,167	126,62		-	0,00	100,00	- 11
Н	imp151	206,720	60,71	-DEFAULT-	-	0,00	100,00	- 🏼 🌌 🕐
H	imp152	101,856	33,27	-DEFAULT-		0,00	100,00	- 7
Н	imp153	117,245	70,55			0,00	100,00	_ /
H	imp2	119,068	114,60	-DEFAULT-		0,00	100,00	-
H	imp22 imp23	168,833 82,898	51,34 55.87	-DEFAULT-	-	0,00	100,00	-
H	imp23 imp3	172,572	94,56		+	0,00	100,00	-
Н	imp3 imp42	130,235		-DEFAULT-	+	0,00	100,00	
1	111072	130,235	100,50	-DELAVET-	1	0,00	100,00	
1.1							,	- 38

6. Launch the Catchment Slope and Length tool through "MOUSE | Catchment Tools | Catchment Slope and Length".



7. Specify the settings for the Catchment Slope and Length tool.

🔇 Catchment Slope and Length						
Slope line layer:	Catch_Slope					
Slope line direction towards:	Downstream 🔻					
Minimum slope:	2,0 [PerThousand]					
DEM:	dem.dfs2 💌					
Overwrite existing values.						
	OK Cancel					
	.4					

Choose the "Catch_Slope.shp" layer as the "Slope line layer." The slope line direction is towards the downstream. A minimum slope can be specified. If the calculated slope is less than the specified minimum slope then this minimum slope value is used. Finally, select the "dem.dfs2" file as the DEM layer for the calculation of the slope and length.

8. Press OK to run the tool. The user has the option of applying the tool only for some selected catchments. To do this, select the catchments of interest before launching and running the tool.



3 MOUSE Tutorial

3.1 Background

Modelling of storm runoff and collection network hydrodynamics in MIKE URBAN - MOUSE requires understanding of the information requirements. In this tutorial we will focus on the following elements in defining a MOUSE network and creating a MOUSE model.

- 1. Definition of the network data, i.e. nodes, links, basins...
- 2. Definition of the catchments
- 3. Specifications of the boundary conditions: catchments and network
- 4. Running the simulation
- 5. Results

3.1.1 Data

The available data is a map illustrating roads and houses and the existing network within the area of interest. Some network data is given on the map - layout and ground levels (GL) and bottom levels (BL) for the nodes. As we progress with our model, the necessary additional data will be provided.

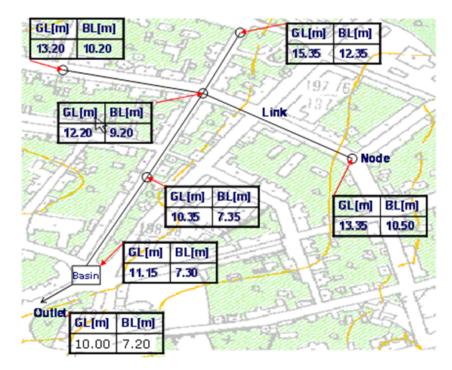


Figure 3.1 Layout of the system and level data for the nodes

The network to be modelled consists of 5 manholes, 1 basin, 1 outlet and 6 links.

3.2 Definition of a MOUSE Network

A MOUSE network within MIKE URBAN can be defined in several ways. In this example we will use graphical data digitalization combined with manual typing of data within the different MIKE URBAN editors.

The model will consist of the following elements:

- Nodes (manholes, basin, outlet)
- Circular Pipes

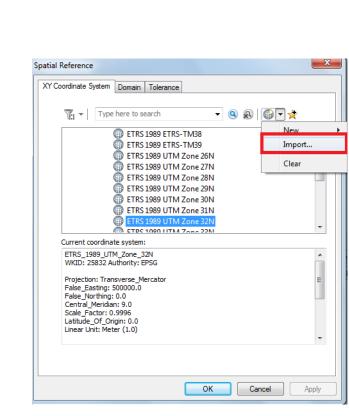
(A detailed description on the MOUSE network data model and the associated editors can be found in the MIKE URBAN CS user guide).

3.2.1 Step 1: Creating a New Project

Start by creating a new project – File | New. Choose a MIKE URBAN Database of the type mdb.

Create New Proj	ject						? ×
Look in	: 🔁 М	DUSEModel	1	•	(-	r 🗄 🕶	
My Recent Documents Desktop My Documents My Computer							
My Network Places	Databa Files of		MyModel MIKE URBAN Dat			•	Create Cancel
Working Mode Collection 3 MOUS SWMM Asset Water Dist Model	System E 1	To vie systen WARI	nate System ew or change the se n, press "Edit Coordin NINGI not all setting Edit Coordina	ttings for the coor nate System'. Is can be change			

Now press 'Edit Coordinate System' and choose to import information used for the coordinate system from the available map (i.e. the EXAM.TIF file) by choosing 'Import'.



In the dialog that comes up map the folder to where the input data for this tutorial is found (Examples\CollectionSystemMOUSE\Tutorial1) and choose the tif-file afterwards:

Browse for Datas	sets or Coordinate Systems		x
Look in: 🛅 1	Tutorial1 🔹 🛃 💌 🔁) ()	9
Exam.tif			
Name:	Exam, tif	Add	
Show of type:			
snow of type:	Datasets and Coordinate Systems	Cancel	

Press 'Add' and then 'OK' on the spatial reference dialog that comes up afterwards.



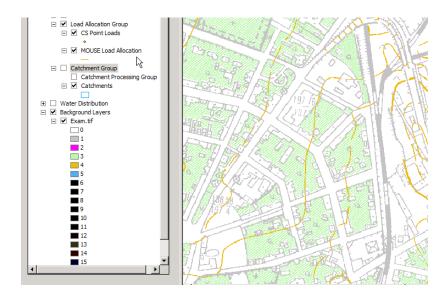
Now you are able to create the project by pressing the 'Create' button on the new project dialog.

3.2.2 Step 2: Loading a Background File

A background image of the modelling area is available as an EXAM.TIF file. TIF is a raster format, and must therefore be accompanied by a 'world file' (the same file name, but the extension TFW). The world file includes the information about the geographical position, orientation and the size of the raster.

> 1.27 0.0 0.0 -1.27 -687998.0 -1056008.0

The TIF file is loaded by using the 'Insert Layer' tool , and browsing to the file location. The file is added as a 'background layer' into the Table of Contents. Please note that you might need to connect to the folder where your layers are stored before your can add these.



3.2.3 Step 3: Graphical Digitalisation of the Network

The Feature Edit toolbar is used for interactively laying out the network components of the collection system.

1 ++ 🔅 🗙	N X Rs	8 🗰 🖗 🖪	MOUSE	▼ MOUSE Manholes ▼
				MOUSE Manholes MOUSE Basins MOUSE Outlets
				MOUSE Storage Nodes MOUSE Links MOUSE Weirs
				MOUSE Orifices MOUSE Curb Inlet MOUSE Pumps
				.MOUSE Valve MOUSE Catchment Connection Coupled2D Connections
				Coupled2D Areas CS Point Loads Catchments

82			
2		7	
	0	•	
	,		
	9	Y	Ÿ.

In order to quickly draw a network layout, click on the 'Create Feature' tool (see left margin) from the 'Feature Edit' toolbar. The tool will perform the action on the selected layer of the currently active network (right and left dropdown box, respectively).



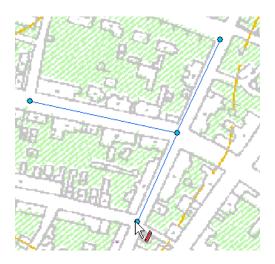
Adding nodes

Use and click once at the map to place the node (according to the figure of the network). Then, move the pointer and click at a new location. Repeat until all wanted nodes are created.



Adding links

The tool inserts both the nodes and links. To add a link, click at the first node to begin, double-click on all intermediate nodes, and double-click on the end node. Any intermediate single click will be recorded as a vertex of the link polyline. To end the actual link series, press the Escape key.

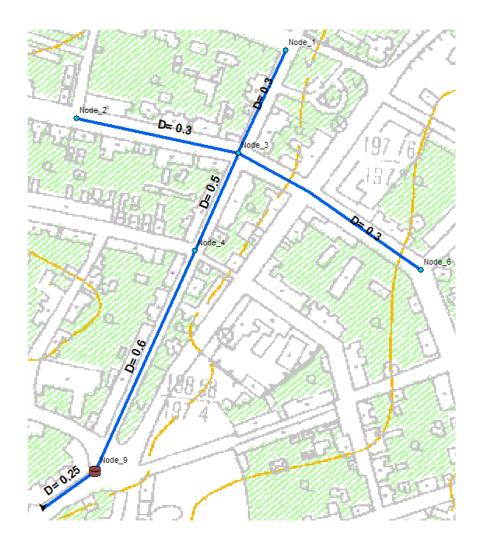


The remaining nodes and links are added using the same procedure. Afterwards, the two nodes are changed into the basin and outlet, respectively,

using the 'Change feature type'

tool.

Your network layout should now look approximately like this:



3.2.4 Step 4: Typing Attributes

Having created the nodes and links graphically you can open the 'MOUSE/Nodes and Structures' and the 'Pipes and Canals' to get an overview of your network and to complete the input data needed.



The 'Identification and Connectivity' part for both nodes and links are created based on your graphical input. You can change the IDs given and add a description if needed.

With respect to the model needs, all active fields WITHOUT a'*' symbol, must be filled out. Some attributes are provided automatically (as default values) during the graphical digitalisation, while the others must be filled in manually. E.g., the data relevant for manholes are:

- Node type (Manhole) automatically provided
- Diameter (1.0 m) to be typed in.
- GL, BL (values taken from the Map seen in Figure 3.1)
- Cover Type (Normal) default
- The Outlet head loss (MOUSE Classic Engelund) default

Identification & connectivity Jnsett Asset ID:* Data source:* Node ID: Node_8 Status:* <null> Description:* <null> Description:* <null> Coordinate: <687646.04 RM Tail node Y coordinate: Image: Status:* RM Tail node Y coordinate: Geometry Q.H and head loss 2D overland Node type: Manhole Wint: Spill coef: 1,00 Critical level:* Basin geometry To: Node 1 Manhole Node 2. Manhole Spill coef: 1,00 Spill coef: 1,00 Spill coef: 1,00 Node 1 Manhole Node 2. Manhole Node 3. Manhole Node 4. Manhole Node 5. Manhole Node 6. Manhole Node 7. Manhole</null></null></null>	Identification							
Asset ID:" Node ID: Node_8 Status:" NULL> Data source:" Model: (NULL> Network type:" (NULL> Delete Model:* (NULL> X coordinate: -687646.04 Close Description:* X coordinate: -1056304.53 Close Tail level: Links: 1 Close Geometry Q-H and head loss 2D overland Node type: Manhole Node type: Manhole Max. Inflow Image: Cover Image: Cover Image: Cover Ground level: Buffer pressure: 0,00 Spill coef: 1,00 Image: Cover Diameter: Image: Cover Spill coef: 1,00 Image: Cover Image: Cover Ground level: Image: Cover Spill coef: 1,00 Image: Cover Image: Cover ID: Image: Cover Image: Cover Image: Cover Image: Cover Image: Cover ID: Image: Cover Image: Cover Image: Cover Image: Cover Image: Cover ID: Image: Cover Image: Cover Image: Cover Image: Cover Image: Cov	a share a contraction i	& connectivity-					Incont	
Node ID: Node_o Status: (NULL> Advanced Modet: (NULL> Network type: (NULL> Advanced Description:* X coordinate: -687646,04 Close RM Tail node Y coordinate: -1056304,53 Close Tail level: Links: 1 Close Geometry Q-H and head loss 2D overland Node type: Manhole Node type: Manhole Max. Inflow Image: Cover Image: Cover Ground level: Spill coef: 1,00 Spill coef: 1,00 Critical level:* Spill coef: 1,00 Spill coef: 1,00 ID: Edit graph MoUSE Class Node_1 Manhole <null> Null> Null> MoUSE Class Node_1 Manhole <null> Null> Null> Null> MoUSE Class Node_1 Manhole <null> Null> Null> Null> MOUSE Class Node_2 Manhole <null> Null> Null> Null> MoUSE Class Node_6 Manhole <null></null></null></null></null></null>	Asset ID:*			Data source:*			Inset	
Intervent gyb. Intervent gyb. Intervent gyb. Intervent gyb. Description:* X coordinate: -687646.04 Close Image: Bit Balance Balan	Node ID:	Node_8	3	Status:*	<nuli< td=""><td>> •</td><td><u>D</u>elete</td><td></td></nuli<>	> •	<u>D</u> elete	
Description:* X coordinate: 687646.04 Close R M Tail node Y coordinate: -1056304.53 Tail level: Links: 1 Geometry Q H and head loss 2D overland Max. Inflow Image: Cover type: Manhole Max. Inflow Node type: Manhole Max. Inflow Image: Cover type: Normal Image: Cover type: Image: Cover type: Normal Image: Cover type:	Model:*	<null< td=""><td>> •</td><td>Network type:</td><td>× <</td><td><u> </u></td><td>Advanced</td><td></td></null<>	> •	Network type:	× <	<u> </u>	Advanced	
Tail level: Links: 1 Geometry Q-H and head loss 2D overland Node type: Manhole Max. Inflow Diameter: Cover Type: Ground level: Buffer pressure: 0,00 Bottom level: Spill coef: 1,00 Critical level:* Spill coef: 1,00 D: Edit Graph Mode_10 Node type Bottom level Graph D: Edit Graph Mode_1 Manhole <null> <null> MOUSE Class Node_1 Manhole <null> <null> MOUSE Class Node_2 Manhole <null> <null> NUll> MOUSE Class Node_6 Manhole <null> <null> NUll> MULI> MOUSE Class Node_6 Manhole <null> <null> <null> MULI> MOUSE Class Node_6 Manhole <null> <null> <null> MULI> <null> <null> <null> Node_6 Basint<null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null>	Description:*	·		X coordinate:	-68764	6,04	Close	1
Tail level: Links: 1 Geometry Q-H and head loss 2D overland Node type: Manhole Max. Inflow Diameter: Cover Type: Ground level: Buffer pressure: 0,00 Bottom level: Spill coef: 1,00 Critical level:* Spill coef: 1,00 D: Edit Graph Mode_10 Node type Bottom level Graph D: Edit Graph Mode_1 Manhole <null> <null> MOUSE Class Node_1 Manhole <null> <null> MOUSE Class Node_2 Manhole <null> <null> NUll> MOUSE Class Node_6 Manhole <null> <null> NUll> MULI> MOUSE Class Node_6 Manhole <null> <null> <null> MULI> MOUSE Class Node_6 Manhole <null> <null> <null> MULI> <null> <null> <null> Node_6 Basint<null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null>	BM Tailu	node		Y coordinate:	10563	04.53		
Node type: Manhole Max. Inflow Diameter: Cover Type: Normal Ground level: Buffer pressure: 0,00 Bottom level: Spill coef: 1,00 Critical level:* Basin geometry ID: Edit Graph ID: ID: Edit Graph MOUSE Class Node 10 Node type Bottom leve Ground leve Note type Node 11 Manhole <null> <null> <null> MOUSE Class Node 2 Manhole <null> <null> <null> MOUSE Class Node 2 Manhole <null> <null> <null> MOUSE Class Node 6 Manhole <null> <null> <null> Null> NoUSE Class Node 3 Basin <null> <null> <null> Null> Null> Null> NUll> Node 3 Manhole <null> <null> <null> NULl> NULl></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null>		·····	_					
Diameter: Cover Ground level: Buffer pressure: 0,00 Bottom level: Spill coef: 1,00 Critical level:* Spill coef: 1,00 ID: ID: Edit Graph Mode ID Node type Bottom leve Ground leve Diameter ID: ID: ID: Edit Graph Node_1 Manhole Null> Null> MoUSE Class Node_1 Manhole Mull> Mull> MOUSE Class Node_4 Manhole Mull> MOUSE Class Node_2 Manhole MUll> MOUSE Class Node_6 Manhole	Geometry Q	I-H and head lo	ss 2D overla	and				
Note ID Node type Bottom level: O,00 Bottom level: Buffer pressure: 0,00 Spill coef: 1,00 Critical level:* Spill coef: 1,00 ID: ID: Edit Graph Mode 1D Node type Bottom level Graph ID: ID: Edit Graph Mode 1 Manhole								
Mode ID Node type Bottom level: Image: State in the state i	Node type:	: Manh	ole 💌	Max. Ir	flow			
Bottom level: 0,00 Spill coef: 1,00 Critical level:* Spill coef: 1,00 ID: ID: Edit Graph Mode ID Node type Bottom level Ground leve Diameter Critical level ID Node 1 Manhole <null> <null> <null> MOUSE Class Node 2 Manhole <null> <null> <null> MOUSE Class Node 6 Manhole <null> <null> <null> MOUSE Class Node 9 Basin «Null» <null> <null> <null> MOUSE Class Node 9 Basin «Null» <null> <null> <null> MOUSE Class Node 9 Basin «Null» <null> <null> <null> MOUSE Class Node 9 Basin «Null» <null> <null> MOUSE Class Node 9 Basin «Null» <null> <null> MOUSE Class</null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null>	Diameter:			Cover -			7	
Bottom level: D Critical level:* Spill coef: Basin geometry D: ID: ID: Edit Graph Node ID Node type Bottom level Spill coef: ID: ID: ID: Moli> ID: ID: ID: <td>Ground lev</td> <td>el:</td> <td><u></u></td> <td>Type:</td> <td>Norr</td> <td>nal 💌</td> <td>[]</td> <td></td>	Ground lev	el:	<u></u>	Type:	Norr	nal 💌	[]	
Node ID Node type Bottom leve Graph ID: Edit graph Mode ID Node type Bottom leve Graph Node_1 Manhole <null> <null> <null> Node_2 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> <null> MOUSE Class Node_7 Basin <null> <null> <null> <null> MOUSE Class Node_7 Manhole <null> <null> <null> MOUSE Class Node_7 Basin <null> <null> <null> MULI> MOUSE Class Node_3 Basin <null> <null> <null> MULI> MOUSE Class</null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null>				Buffer p	ressure:	0,00		
Kode ID Node type Bottom leve Ground leve Diameter Critical level ID Use Node ID Node type Bottom leve Ground leve Diameter Critical level ID Use Node 1 Manhole Null> Null> MOUSE Class Node 2 Manhole Mull> MOUSE Class Node 2 Manhole Mull> MOUSE Class Node 6 Manhole	Bottom lev	el:		Spill coef	:,	1.00		
Basin geometry Edit Graph ID: Edit Graph Node ID Node type Bottom leve Ground leve Diameter Critical level ID Use Node 1 Manhole <null> <null> <null> MOUSE Class Node 2 Manhole <null> <null> <null> MOUSE Class Node 6 Manhole <null> <null> <null> MOUSE Class Node 3 Manhole <null> <null> <null> MOUSE Class Node 3 Manhole <null> <null> <null> MOUSE Class</null></null></null></null></null></null></null></null></null></null></null></null></null></null></null>	Critical leve	-l·*		, spiircoei	1	1,00		
ID: Edit Graph Node ID Node type Bottom leve Ground leve Diameter Critical level ID Use Node 1 Manhole <null> <null> <null> <null> MOUSE Class Node 2 Manhole <null> <null> <null> MOUSE Class Node 6 Manhole <null> <null> <null> MOUSE Class Node 6 Manhole <null> <null> <null> MOUSE Class Node 9 Basin <null> <null> <null> MOUSE Class Node 3 Manhole <null> <null> MUII> MOUSE Class</null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null>			1					
Node_1 Manhole <null> <null> <null> MOUSE Class Node_4 Manhole <null> <null> <null> MOUSE Class Node_2 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_9 Basin <null> <null> <null> MOUSE Class Node_3 Manhole <null> <null> <null> MOUSE Class</null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null>	ID;			Edit	Graph			
Node_1 Manhole <null> <null> <null> MOUSE Class Node_4 Manhole <null> <null> <null> MOUSE Class Node_2 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_9 Basin <null> <null> <null> MOUSE Class Node_3 Manhole <null> <null> <null> MOUSE Class</null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null>								
Node_1 Manhole <null> <null> <null> MOUSE Class Node_4 Manhole <null> <null> <null> MOUSE Class Node_2 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_9 Basin <null> <null> <null> MOUSE Class Node_3 Manhole <null> <null> <null> MOUSE Class</null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null>								
Node_1 Manhole <null> <null> <null> MOUSE Class Node_4 Manhole <null> <null> <null> MOUSE Class Node_2 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_9 Basin <null> <null> <null> MOUSE Class Node_3 Manhole <null> <null> <null> MOUSE Class</null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null>								
Node_1 Manhole <null> <null> <null> MOUSE Class Node_4 Manhole <null> <null> <null> MOUSE Class Node_2 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_9 Basin <null> <null> <null> MOUSE Class Node_3 Manhole <null> <null> <null> MOUSE Class</null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null>								
Node_1 Manhole <null> <null> <null> MOUSE Class Node_4 Manhole <null> <null> <null> MOUSE Class Node_2 Manhole <null> <null> Null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_9 Basin <null> <null> <null> MOUSE Class Node_3 Manhole <null> <null> <null> MOUSE Class Node_3 Manhole <null> <null> <null> MOUSE Class</null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null>								
Node_4 Manhole <null> <null> <null> MOUSE Class Node_2 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_9 Basin <null> <null> <null> MOUSE Class Node_3 Manhole <null> <null> <null> MOUSE Class</null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null></null>	Hede ID	Node type	Pottom law		Diamatar	Critical Jours	(in	lles
Node_2 Manhole <null> <null> <null> MOUSE Class Node_6 Manhole <null> <null> <null> MOUSE Class Node_9 Basin <null> <null> <null> MOUSE Class Node_3 Manhole <null> <null> <null> MOUSE Class</null></null></null></null></null></null></null></null></null></null></null></null>								Use
Node_6 Manhole <null> <null> <null> MOUSE Class Node_9 Basin <null> <null> <null> MOUSE Class Node_3 Manhole <null> <null> <null> MOUSE Class</null></null></null></null></null></null></null></null></null>	Node_1	Manhole	<null></null>	<null></null>	<null></null>	<null></null>	MOUSE Class	Use
Node_9 Basin <null> <null> <null> MOUSE Class Node_3 Manhole <null> <null> <null> MOUSE Class</null></null></null></null></null></null>	Node_1 Node_4	Manhole Manhole	<null> <null></null></null>	<null> <null></null></null>	<null> <null></null></null>	<null> <null></null></null>	MOUSE Class MOUSE Class	
	Node_1 Node_4 Node_2	Manhole Manhole Manhole	<null> <null> <null></null></null></null>	<null> <null> <null></null></null></null>	<null> <null> <null></null></null></null>	<null> <null> <null></null></null></null>	MOUSE Class MOUSE Class MOUSE Class	
Node_10 Outlet <null> <null> <null> <null> MOUSE Class</null></null></null></null>	Node_1 Node_4 Node_2 Node_6	Manhole Manhole Manhole Manhole	<null> <null> <null> <null></null></null></null></null>	<null> <null> <null> <null></null></null></null></null>	<null> <null> <null> <null></null></null></null></null>	<null> <null> <null> <null></null></null></null></null>	MOUSE Class MOUSE Class MOUSE Class MOUSE Class	
	Node_1 Node_2 Node_2 Node_6 Node_9 Node_3	Manhole Manhole Manhole Manhole Basin	<null> <null> <null> <null> <null></null></null></null></null></null>	<null> <null> <null> <null> <null></null></null></null></null></null>	<null> <null> <null> <null> <null></null></null></null></null></null>	<null> <null> <null> <null> <null></null></null></null></null></null>	MOUSE Class MOUSE Class MOUSE Class MOUSE Class MOUSE Class	
	Node_1 Node_2 Node_2 Node_6 Node_9 Node_3	Manhole Manhole Manhole Basin Manhole	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	MOUSE Class MOUSE Class MOUSE Class MOUSE Class MOUSE Class MOUSE Class	
	Node_1 Node_2 Node_2 Node_6 Node_9 Node_3	Manhole Manhole Manhole Basin Manhole	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	MOUSE Class MOUSE Class MOUSE Class MOUSE Class MOUSE Class MOUSE Class	
	Node_1 Node_2 Node_2 Node_6 Node_9 Node_3	Manhole Manhole Manhole Basin Manhole	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	MOUSE Class MOUSE Class MOUSE Class MOUSE Class MOUSE Class MOUSE Class	
	Node_1 Node_2 Node_2 Node_6 Node_9 Node_3	Manhole Manhole Manhole Basin Manhole	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	MOUSE Class MOUSE Class MOUSE Class MOUSE Class MOUSE Class MOUSE Class	

With regard to Links, you need to specify:

- Geometric properties such as shape (Circular) and size (0.3 m)
- Information about material (concrete), hydraulic friction losses (Manning's explicit)

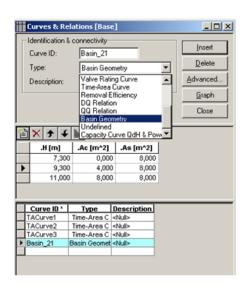
By clicking on the 'Advanced' button and activating the 'Recompute' tool, the link Length, UpLevel, DwLevel and Slope will be displayed in the grey fields. These 'system' values can be overwritten by typing appropriate values in the adjacent fields.

Œ	Pipes and Ca	anals [Base]						Ľ
Г	Identification 8	& connectivity -					lucet	- 1
	Asset ID:			Data source	e:*		<u>I</u> nsert	
	Link ID:	Link 1		Status:*	KN	IULL> 🔻	<u>D</u> elete	•
	Description:*			Network typ	be:* ZN	IULL> 🔻	 	d
	From node:	Node 2	,		1		Close	
	To node:	Node 3			essure main			
		, -	·		casare main			
I	-Geometrical p	roperties						
	Shape:	Circular		Length		_		
	Size:			UpLevel:				
	Width:			DwLevel				
	Height:			Slope:			1	
	CRS ID:			<u>E</u> dit	<u>G</u> raph			
	Topography:			Edjt	Max Dx:		1	
	-Hydraulic fricti					, 		
			e (Smooth)	Formulation			1	
	Material:		e (Sillooth)	Formulation	: Ima	anning Explici 💌	1	
		85.0	_	Eg roughne		0.000500	1	
	Manning:	,		Eq roughne	:55.	J0.000000		
I	- Miscellaneous	;		_				
	Re <u>gu</u> lation	Additi <u>o</u> n	al	Non ret	um valve			
<u> </u>								
	Link ID*	Shape	UpLevel	DwLevel	Length	Size	Material*	Forn
Ľ	Link_1	Circular		<null></null>	<null></null>	<null></null>	Concrete (Sm	
L	Link_2	Circular		<null></null>	<null></null>	<null></null>	Concrete (Sm	
\vdash	Link_3 Link_5	Circular Circular		<null></null>	<null></null>	<null></null>	Concrete (Sm Concrete (Sm	
\vdash	Link_5 Link_6	Circular		<null></null>	<null></null>	<null></null>	Concrete (Sm	
\vdash	Link_0	Circular		<null></null>	<null></null>	<null></null>	Concrete (Sm	
F	_				1	1		
Ľ								

All our pipes are circular. The diameters are given in the illustration above.

We also need to add information about the geometry of the basin. This is done under 'MOUSE/Curves and Relations'. Choose Insert and provide an ID and the type of curve. Fill in values for the elevation H, the cross section area **Ac** and the surface area **As** to define the physics of the structure.





In MOUSE/'Nodes and Structures' select the created basin in the list and under 'Basin Geometry' choose the basin geometry.

•0	Nodes [Base	2]							L.	
	-Identification 8	connectivity-								
	Asset ID:*			Datas	ource:*				lns	ert
	Node ID:	Node_9	3	Status	*	, <n< td=""><td>ULL></td><td>T</td><td><u>D</u>e</td><td>ete</td></n<>	ULL>	T	<u>D</u> e	ete
	Model:*	<null< td=""><td>> -</td><td>Netwo</td><td>ork type:</td><td>× (<n< td=""><td>ULL></td><td>1</td><td><u>A</u>dvar</td><td>iced</td></n<></td></null<>	> -	Netwo	ork type:	× (<n< td=""><td>ULL></td><td>1</td><td><u>A</u>dvar</td><td>iced</td></n<>	ULL>	1	<u>A</u> dvar	iced
	Description:*				rdinate:		7937.61	-	Clo	ise
	□ BM Tail r	node				t geome			×	
	Tail level:			Lin				1.15		
	r an iovei.	J						netry ID	<u> </u>	
	Node type:	Basin	•	1	Bass		eometry l	D		
	Diameter:	1	1.0000	_i	DOSS	5111				
Κ	Ground level:		11.15							
ß	Bottom level:		7.30							
	Critical level:*		1.00							
	-Basin geometr	·		-						
	ID:	Bassin		Ŀ						
	Q-H relation			_						
	Г			1						
	-Outlet head lo	,								
			Classic (Fa	1						
	ID:	,	Classic(En	·						
	🗌 Use loca	al data	Coeff.:		1					
			Eff. flow area:				ОК	Car	ncel	
H	No do 104	No do Anno	Detter level	-		104	0-111-		104	Us
H	Node ID* Node_1	Node type Manhole	Bottom leve 12.35	Groui		ID* <null></null>	<null></null>	al level	ID* MOUSE	
F	Node_10	Outlet	7.20			<null></null>	<null></null>		MOUSE	
F	Node_2	Manhole	10.20			<null></null>	<null></null>		MOUSE	
F	Node_3	Manhole	9.20		12.20	<null></null>	<null></null>		MOUSE	Class
	Node_4	Manhole	7.35		10.35	<null></null>	<null></null>		MOUSE	Class
	Node_6	Manhole	10.50		13.35	<null></null>	<null></null>		MOUSE	Class
Þ	Node_9	Basin	7.30		11.15	Bassin	<null></null>		MOUSE	Class
Ŀ										►

3.3 Definition of Catchments and the Hydrological Model

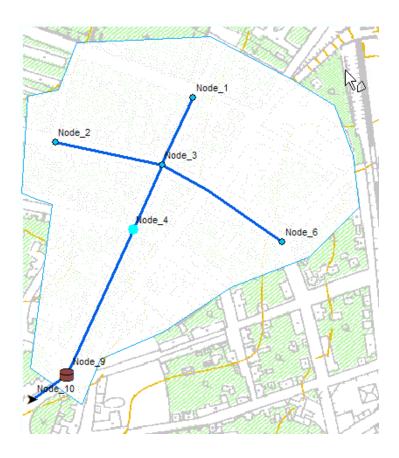
Catchments are essential for any hydrological model. In MIKE URBAN, the geographical extent of a catchment is determined by the catchments polygon perimeter. In this exercise we delineate the catchment area into smaller subcatchments to be able to allocate runoff generated on the surfaces into the nodes in our network.



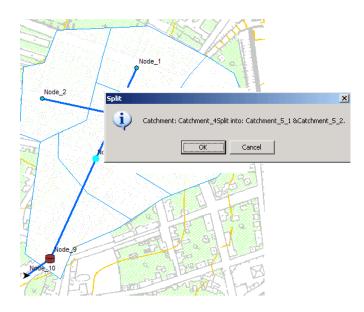
3.3.1 Step 1: Delineate the Catchments



 First, the catchment layer must be activated. Click on the Create feature tool in the toolbar to insert a new catchment. Click on the map and draw a polygon following the contours of your total catchment area. End with double-click.



• Switch to the 'Split Polygon' tool it to split the catchment into smaller sub-catchments describing your system with respect to the network. As shown in the illustration below the total catchment has been split into five smaller catchments. You will be asked to confirm each split – click OK. Make sure that the line crosses both boundaries (this is done by making it too long).



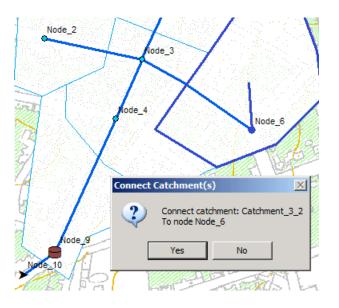
3.3.2 Step 2: Connecting Catchments to the Network

In this exercise we will use the computed runoff as an input load to the network model. To do so we need to connect the created catchments to the network.

This can be done in two ways:

- 1. Manually: Go to MOUSE/Catchments & Catchment Parameters/Catchments. For each catchment, select it in the list, go to MOUSE/Catchment Tools/Connect to Node, and select the desired node on the map.
- 2. With the 'Catchment Connection Wizard', go to MOUSE/Catchment Tools/Catchment Connection Wizard.





3.3.3 Step 3: Reviewing/Editing Catchments

The 'Catchment' editor is primarily used for editing the catchment attributes. It is possible, but not recommendable, to insert catchments through the editor, these are then given as schematized quadratic shape.

The 'catchments' editor can be accessed through the menu: MOUSE/Catchments & Catchment Parameters/Catchment. Having created your catchments graphically this editor provides an overview and allows you to insert further data.

3.3.4 Step 4: Defining a Rainfall-Runoff Model

In MIKE URBAN the user can quickly prepare a precipitation-runoff model set-up of desired level of detail and utilize the computed runoff as a load of the collection network.

It should be noted that runoff computations and its subsequent use as a network load are, in principle two distinct steps in the modelling process.

Steps involved in preparation of a rainfall-runoff model are:

- Definition of MIKE URBAN catchments
- Catchment connections, i.e. specification of the point of runoff inflow into the network. If this is not done before running the runoff simulations, the computed results will not be possible to be connected to the network.
- Specification of the hydrological model parameters



- Definition of the rainfall, i.e. definition of the rainfall boundary condition
- Runoff computations

As we have already defined the catchments and connected them to the network we need to define which hydrological model to use and allocate some precipitation data.

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

- Surface runoff models: These are the most common type in urban runoff analysis. The common characteristic off all the models in this class is that only surface runoff is computed.
- 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.

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

In this exercise we will use the Time Area Method (MOUSE model A) which is a simple surface runoff model with minimum data requirements.

The data to be used in the model can be accessed for editing in the Time-Area editor through: MOUSE / Catchments & catchment parameters / Time-Area (A). This table contains the data records for the Runoff model Time-Area (A) – one record for each catchment. Per default, all the catchments which were created so far get automatically the record in the runoff model table. This is so for all MIKE URBAN tools. However, if catchments polygons are imported from some external data source, the user must insert the records (Advanced/Generate model records) or manually. The system take care automatically that each catchment gets only one record.

The missing parameters are:

Imperviousness – here user must specify the percentage of impervious area.

Parameter set – the name of the parameter set to be used. The set of hydrological parameters to be used in our Time –Area model have been set as default values and for now we will not change these values.



Parameters	Time-Area			
Parameter set	ID:	DEFAULT-		<u>I</u> nsert
Time of conce	entration:	7		<u>D</u> elete
Initial loss:	l.	0,0006		Advanced
Reduction fac	otor:	0,90		Close
 Time-area 	curve:	TACurve1	<u>E</u> c	
C Time-area	coeff.:	1,00	<u>₽</u>	
<u> </u>			<u> </u>	
Parameter	Reduction f	Initial loss	Time of con	Time-area c Time 🔺
-DEFAULT-	0,90	0,0006	7	Time-Area Cu TACur
•				

The type of Time-Area curve used in the runoff calculations is specified through: MOUSE/Curves & Relations

	Curves & Relations										
	-Identification 8	k connectivity -									
	Curve ID:	TACurve1			Insert						
	Туре:	, Time-Area	Curve	-	<u>D</u> elete						
	Description:				Advanced						
	Description.	1									
					<u>G</u> raph						
					Close						
¥) <mark>× +</mark> +										
×	€ 🗡 🗡 🗄 🕹	.a/A [%]									
*											
1	.t/T [%]	.a/A [%]									
	.t/T [%] 0,000 100,000	.a/A [%] 0,000									
	.t/T [%] 0,000 100,000 Curve ID*	.a/A [%] 0,000	Description								
	.t/T [%] 0,000 100,000	.a/A [%] 0,000 100,000			<u></u>						
	.t/T [%] 0,000 100,000 Curve ID*	.a/A [%] 0,000 100,000 Type									

3.3.5 Step 5: Defining the Rainfall Boundary Condition

Prior to running the model we need to address the issue of boundary conditions.

A boundary condition is defined by its type and model connectivity. MOUSE distinguishes between three groups of boundary conditions:

- Catchment Loads and Meteorological Boundary Conditions
- Network Loads



External Water Levels

In this exercise we will specify only one boundary condition: Rainfall.

Adding rainfall as a catchment boundary condition

Go to MOUSE|Boundary Conditions|Catchment Loads.... This will open the 'Catchment loads and meteorological items' editor.

- Press insert to add a new boundary condition
- Insert a proper name (ID)
- Choose the type of boundary connection
- Choose 'Connection type' (we use the option 'All' because this boundary condition applies for all our catchments

Catchment	loads and me	eteorological	items [Base	1	<u> </u>
Boundary ID:	BC	1_Rainfall	_	Insert	1
		Apply		<u>D</u> elete	i
Туре:	Ra	ainfall	- 4	dvanced	i
Connection ty	ype: All		J 🔽	[avigator	i
	rce location			ltems	ĺ
X:				Close	1
Y:	I				1
Individual					
List					
		Distribute			
, Boundary ID	Apply	Туре	Connection	Data sou	rce 🔺 🔺
▶ BC1_Rainfall	True		AI	F	alse

Having defined the boundary conditions, moving to the 'Boundary items' editor allows for a full definition of a boundary item in terms of type, temporal variation, quantity/quality and temporal validity.

The 'Boundary Items' editor has been designed to operate in conjunction with the three boundary condition editors. When opened simultaneously with one of these, this editor displays only the boundary items associated with the currently activated boundary condition.

Click on 'Items' in the 'Catchment Load and...' editor



- Insert a new item and type a descriptive item name
- The Item type is set as default and the Scaling factor to 1 (as default)

🕗 Boundary items [Base]		_0	×
Boundary ID: BC1_Rainfall	Rair	nfall <u>I</u> nsert	1
Item name: Rainfall	Item type	<u>R</u> euse item	1
Scaling factor: 1,000	Mode:		1
Load type:		Advanced	1
Temporal variation Constant			-
Time Series/Result File 💌	🗌 🗖 Gradual start up		
Start from:	Startup time:	60 Close	
Pattern:	. Method: <nl< td=""><td></td><td></td></nl<>		
Value	-		
Time series			
Source type: DFS0	Path: C:\F	rogram Files\D	
	Time series ID: RAII		
	Item ID: Rain	ifall intensity	
Limited validity interval			
Start: 01-01-2005 🔽 00:00:00 😤	End: 01-01-2005	▼ 00:00:00 🚔	
			_
Item name Boundary ID Item type TRAP Rainfall BC1 Rainfall Default Null>	comp TRAP fractic Scaling fact <pre></pre> <pre><!--</td--><td>Load type Temporal v Mot <null> Time Series/R</null></td><td>-</td></pre>	Load type Temporal v Mot <null> Time Series/R</null>	-
	Sridenter 1,000		
			•

The temporal variation of the boundary item (rainfall) is specified using a Time series file.

• Specify the 'Source type' as DSF0 and specify the path to the wanted time series file.

Time Series (Active)				
File Edit Zoom Format E	xport V	Vindow Time	RAIN [mu-m	RAIN [mu-m/s]
	1	16-05-1953 18:00:00.000	0	Untitled
BAIN	2	16-05-1953 18:10:00.000	0	
	3	16-05-1953 18:12:00.000	2.999999998	40
	4	16-05-1953 18:15:00.000	7.00000001	
	5	16-05-1953 18:16:00.000	12.0000002	35
	6	16-05-1953 18:20:00.000	13.99999999	
	7	16-05-1953 18:22:00.000	42.0000008	30
	8	16-05-1953 18:24:00.000	9.999999993	
	9	16-05-1953 18:27:00.000	20.0000000	25
	10	16-05-1953 18:28:00.000	17.99999994	
	11	16-05-1953 18:29:00.000	14.0000002	20
	12	16-05-1953 18:32:00.000	10.0000000	
	13	16-05-1953 18:34:00.000	3.999999997	15 + + + + + + + + + + + + + + + + + + +
	14	16-05-1953 18:40:00.000	5.00000001	
	15	16-05-1953 18:47:00.000	0.999999999	10
	16	16-05-1433 19:00:00.000	0	5
	17	16-05-1953 20:00:00.000	0	
	18	16-05-1953 21:00:00.000	0	
	19	16-05-1953 22:00:00.000	0	
▲				18:00 19:00 1953-05-16

3.4 MOUSE Simulations

Having defined the model data for all catchments, specified the catchment connections and the boundary conditions, we are now ready to execute simulations.

It should be understood that running the rainfall-runoff (i.e. catchment) simulation and hydrodynamic (i.e. network) simulation are two distinct steps in the simulating the rainfall effect on the drainage network.

In step 1, rainfall-runoff is simulated.

In step 2, the results of the rainfall-runoff simulation must be specified as a network boundary condition (i.e. network load) and the hydrodynamic simulation run.

The MOUSE simulations are started from 'Simulation / Run Mouse...'.

The editor is split into five tab sheets:

- General
- Runoff parameters
- Network Parameters
- Network Simulation Summary
- 2D overland parameters

The 'Simulation ID' and 'Scenario ID' are found on all five TAB sheets as a common reference.



Immediately after opening the editor, insert a new simulation and give it a suitable name. Then, proceed to the specification of relevant simulation parameters.

On the General TAB page you specify:

- The Name of the simulation (Simulation ID)
- The simulation period (xx xx xxxx xx xx xxxx) use 'Set max. time' tool
- The simulation type (Network, Runoff or 2D overland)

General	Runoff p	aramota	Network		motoro	Network su		2D overlag	d paramete	
	Runon p	arameters	Netwon	c para	meters	Network sur	mmary	2D Overlan	u paramete	rs
Simulat	ion ID:	5	Simulation_	1-Rund	off					Insert
Scena	rio ID:	E	Base							Сору
										Delete
										Close
Simulatio	n type				Simulation	description				
 Rur 										
Net	work									
O Rur	noff follow	ed by net	work							
Rur	noff + netv	work (simu	ltaneous)							-
Net	work + 2[) overland	ł		•					Þ.
Simulatio	n period							Star	t simulation	
	ſ	Info		Set	max. time				Include to b	atch
Start:	10	6-05-1953		18:00:0		÷			Start simulat	ion
		-03-1333							Batch rur	
Duration				0	0	[dddd] [hh] [mm]	[ss]	Daterritur	
End:	16	6-05-1953		22:00:0	00	÷				
MUID *	Inclu	IdeToB	Descripti	on C	omputat	io Compu	tatio	Computatio	MJLFile	Na RSFDa
	1-		<nul></nul>	_		off 16-05-1	052	16-05-1953	<null></null>	<nul></nul>

The Runoff Simulation

On the Runoff Parameters TAB page specify:

- Model Type (T-A curve (A))
- Simulation time step (60 sec)



omputation			_ [
ieneral Runoff parameter	s Network parameters	Network summary 2D overland pa	arameters
Simulation ID:	Simulation_1-Runoff		Insert
Scenario ID:	Base		<u>D</u> elete
Model type:	T-A Curve (A) 💌		Close
Simulation time step		Additional parameters	
Fixed time step:	60		
Time step dry weather:	60	Surface quality (SRQ)	
Time step wet weather:	300	RDII hot start	
Time step FRC:	60	Hot start file:	
Time step SRC:	4,00		
Overwrite			
Cverwrite			
C Overwrite			
	chude to b Rescription	n] Computatio] Computatio] Con	oputatio MIL Filena Conter
	clude to b Descriptio False <null></null>	n Computatio Computatio Com Network 01-01-2009 01-0	nputatio MJL Filena Conter 2-2009 «Null» «Null»

Return to the General TAB and press 'Start Simulation'. The system starts with export of model data to MOUSE.

🔀 59% - Converting Task "Export MOUSE"	_ 🗆 🗙
Converting Task "Export MOUSE"	

After completed export, the system may come up with some warning prompts. It is always a good idea to read the warnings and, if necessary, make appropriate modifications.

If something is not OK with the model data, the system will issue an Error. Upon studying the reason for error, it must be corrected, and the simulation restarted.



🧱 MOUSE Simulat	tion Launcher - Runoff Model	_ 🗆 🗙
Simulation Charts	Help	
Project Name :	NING_COURSES\simple_model_building\Simulation_1-Ru	noffBase.mex
Started at :	2005-06-14 16:31:45 Spee	d Reduction
Current time :	2005.06.14 16:31:45	
Estimated end at :	Information	≤ o Limit
Simulation start :	Calculation ended. Show summary report ? (Y/N)	nterval
Simulation time :		
Simulation end :	Yes No	Time Step
	0.0	
Finished		

Each simulation generates a Summary containing some useful information. The summary can be examined immediately, or later. It is saved in the project directory in html format.

The result file from Runoff simulation gets the extension CRF (meaning: catchment result file). When the simulation is finished, the result file is automatically loaded into the memory and is ready for the presentation.

The Network Simulation

Create a new simulation job (to do this you mus press 'Start Editing' in the toolbar), give it an appropriate name, and on the General TAB page, select it as 'Network'.

Then, on the Network Parameters TAB page specify:

- Simulation mode (Normal)
- Runoff input, tick on, and specify the *CRF file previously generated by the runoff simulation.
- Dynamic simulation
- Hot Start (no), tick off)
- Processes (no), tick off
- Results (standard and overwrite)

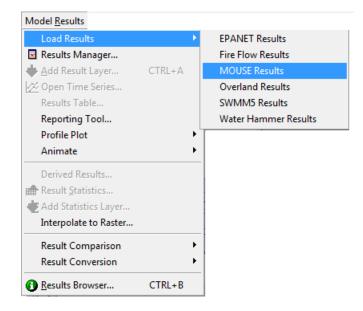
Þ	Computation						_ []	×
ſ	General Runoff parame	ters Network	parameters	Network summa	ary 2D overla	nd parameters	1	
	Simulation ID: Scenario ID: Simulation mode	Simulation Base	_2-Network	R.	unoff input		<u>I</u> nsert <u>D</u> elete Close	
	Normal		1 -		Runoff	ulation_1-Rund		
	O LTS Dynamic simulation Model type: Dy	namic Wave		dit Fil ot start Hot start	e: Jour	alauuni i munc		
	M Time step: 10	in. Max. I	Factor F ,30 H Sec.	ile: 🛛 🗌	1-2009 💌		*	
	Processes F BTC Additional parame	eters	Г	Ilutants Transport (Al Water quality	ם ד	ediments Transport (S Graded sedir		
	Results Standard User-specifield	Conte	TS statistics nts:	□ 0v	verwrite			
				Computatio			Overw 🔺	
	Simulation_1-Runoff Simulation 2-Network	False False			01-01-2009	01-02-2009		
•	ormalation_2-rectivery	i alse	DINUILE	NOLWOIN	01-01-2003	01202-2003		

On the Network Summary tab page specify the type of results you would like to be available in the summary report presented after the network simulation.

3.5 Result Presentation

With MIKE URBAN you can present your results in a number of ways:

- By the use of Time series
- By the creation of different Profile plots
- Create animations on the horizontal plan
- Result comparisons
- Create Thematic maps and Statistical analysis



3.5.1 Step 1: Loading Results

Per default, after a completed simulation, the system automatically loads the result file into the memory for the presentation.

The selection of result files to be loaded into the computer memory for the presentation in MIKE URBAN is done through the menu (Model Results / Load Results / MOUSE Results... This will access the result files stored in your specified working directory on your computer.

A dialog with choices of which data types are to load from the result file appears. If 'OK' is pressed with everything selected all items in the result file will be included.

Results Manager	×
Simulation_1-RunoffBase.CRF Simulation_2-NetworkBase.PRF	Additional Close
,	

The result manager provides an overview of the files loaded and allows removing redundant files.

The following figures illustrate some way of presenting results. Further information in presenting results can be found in the MIKE URBAN Model Manager user guide.

3.5.2 Step 2: Viewing Result Time Series

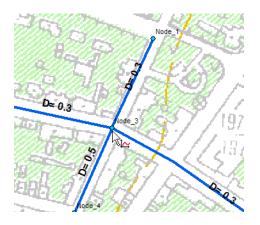
Go to Model Results / Open Time Series (or click on the Time Series

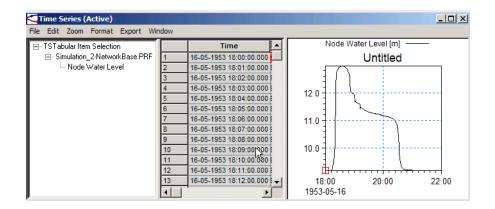
button in and choose the result file and data type from the result selection dialog.

🔀 Result Sele	ction	×
Result File:	Simulation_2-Network	(Base.PF 💌
Data Type:	Node Water Level	~
🗖 List	Cancel	Ok

When choosing 'List' in the above dialog for this example a list of all nodes in the system will appear and you can choose the relevant node for which you wish to see the water level time series.

If you click 'Ok' the cursor will change and include a TS icon and you can point on a node, link etc. (depending on what type of results you selected) to display the relevant time series. The cursor changes colour when approaches to an element containing the selected result type.



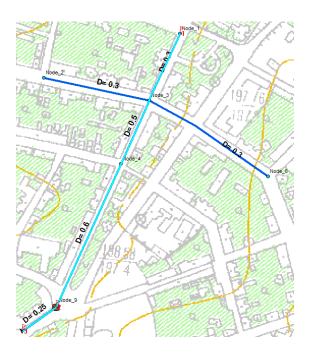


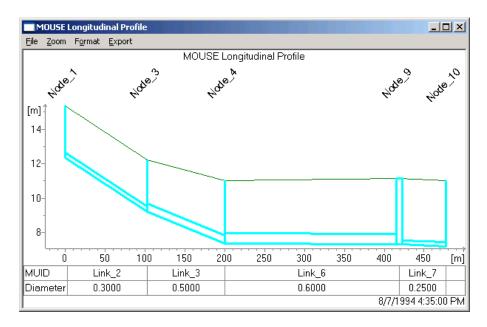
3.5.3 Step 3: Profile Plots and Animations

Initially, the profile plots display the network geometry only. Subsequently, the profile plot may be configured to include the results.

The profile plot is drawn between specified flags. Go to Model Results / Profile Plot and choose: 'Define Path Flags'

The cursor changes to a small flag: point to the first node in the profile and a small flag will be placed. Continue until the path of your profile is unambiguously defined. Then, the profile is plotted by choosing 'Create Profile Plot'.

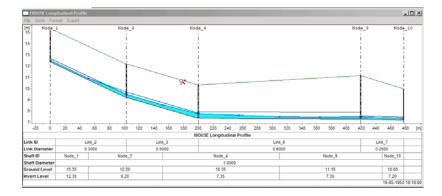




If results are loaded in, the longitudinal profile will be appearing with results added to the profile plot. For a MOUSE result file the water level will be added per default. If you wish to show other results or change the layout of the profile go to the profile plot properties dialog (right-click on the profile to get to the local menu that provides easy access to the property page).

<mark>圣</mark> LongitudinalProf									<u>- 🗆 ×</u>
Axis Symbols and	Fonts Graph	ical Items] Tabula	ar Items	Labels	Multiple I	_inks F	Raster Item	ns
	[Add		Rem	ove	Move	Up	Move D)own
Default Name	Display Name	Тур	be	Line	Style	Line Wid	lth	Line Colo	r 🗌
Link Water L I	∟ink Water	Filled Ar	nimat	So	id	1pt			
Legend Placeme No legend Inside profile a C Outside profile	s area Ho	elect Foni rizontal rizontal	t for Lege Right		Vertical	Тор	T		
Save	Load		OK		Apply		Cancel	F	lelp

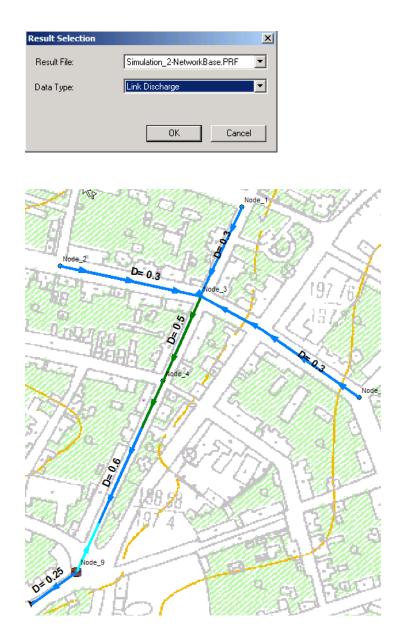
The profile plot may be animated. The actual time of the animation is seen in the lower right corner of the longitudinal profile.





3.5.4 Step 4: Adding Result Layers

'Add Result Layer' allows for the presentation of results as layers in the map.



By appropriate symbology, the results may be displayed with various effects. The results in the map can be animated.



3.5.5 Step 5: Result Statistics

'Result Statistics' allows for the calculation of some basic statistics over the result time series, such as minimum and maximum values.

	esult File: Simulation_ ata Type: MOUSE Re		▼ Calcu	late	Advanced
	Statistics Name	Min	Max	Accumulated	
	Time Dive				
•	Time Step				
•	Volume			Г	
•				Г	
	Volume		-		
	Volume Link Velocity			Г	

The computed statistics get saved in the database and can be presented in the map as a statistics layer, e.g. with graduated symbols.





4 SWMM Tutorial

4.1 Background

This example demonstrates how to model a system that conveys both sanitary wastewater and stormwater through the same pipes. Systems like these are known as combined sewer systems and are still quite common in older communities and cities. During periods of moderate to heavy rainfall the capacity of these systems to convey and properly treat the combined flow can be exceeded, resulting in what are known as Combined Sewer Overflows (CSOs). CSO discharges can cause serious pollution problems in receiving waters. Contaminants from these discharges can include conventional pollutants, pathogens, toxic chemicals and debris.

4.2 Data

The data for this tutorial is found in the Examples\CollectionSystemSWMM\Example3 folder of your MIKE URBAN installation. In this folder you will find the following files:

- A mdb-file with a wastewater model (Example3.mdb) and the corresponding project file (Example.mup)
- An txt-file with a 2 year storm event (TS.txt)

These files are the only files necessary to run through this tutorial.

In this exercise we will be adding 7 catchments and a rain gauge to the existing wastewater model.

4.2.1 Step 1: Opening the project

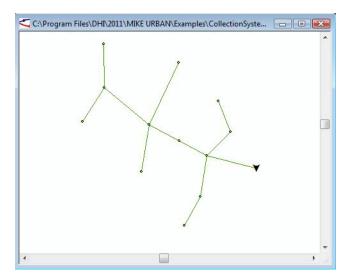
Go to File|Open and open the relevant model. This can be done be either opening the mdb-file (Example3.mdb) or the project file (Example3.mup). If you open the project file you will need to indicate that you would like to open this file by selecting files of type 'MUP' and press the button 'Open':



😋 Open					×	
Look in:)) example3		87	- G 🜶 🖻 🖽 -		
(Per	Name	^	Date modified	Туре	Size	
Recent Places	Example3.	mup	23-11-2010 20:57	MUP File	87	
Desktop						
Henriette Tamasauskas						
Computer	•		m		•	
	File <u>n</u> ame:	File name: Example3.mup			Open	
2	Files of type: MIKE URBA		BAN Project (*.mup)	•	Cancel	
				Op	oen GDB/SDE	

Figure 4.1 Opening the wastewater model

The advantage by opening up a project file is that the working mode is set to SWMM 5 and hence dialogs etc. are set to work with the SWMM 5 model. The model looks like seen in Figure 4.2 to begin with.





Now we will start by adding the raingauge.



4.3 Inserting a Raingauge and editing the parameters

The first step is add the raingauge and then the rainfall data. The rainfall data for a raingauge can be either a user-defined time series or it can come from an external file (several formats can be used).

In this example, the rainfall data will consist of a synthetic design event that represents a 2 year storm of 2-hour duration. The 2 year storm is found in the TS.txt file. The time series (as found in the text-file) is seen in Figure 4.3. The txt file follows the 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.

File	Edit	Format	View	Help		
TS2 TS2 TS2 TS2 TS2 TS2 TS2 TS2 TS2 TS2	2007 2007 2007 2007 2007 2007 2007 2007	1 1 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	$\begin{array}{c} 1 \\ 0 \\ 1 \\ 0 \\ 2 \\ 0 \\ 0 \\ 2 \\ 0 \\ 0 \\ 2 \\ 0 \\ 0$	0.35 0.30 0.20 0.19 0.18 0.17 0.16 0.15 0.15 0.15 0.14 0.14 0.13		*
*					F	

Figure 4.3 The time series as it looks in Notepad

The graphical representation of the rainfall is seen in Figure 4.4.

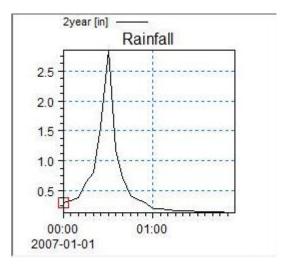


Figure 4.4 Graphical representation of the rainfall

4.3.1 Step 1: Inserting the rain gauge

When you insert a rain gauge you would like to see it appearing on the map. Hence make sure to toggle on the display of the rain gauge in the TOC (as seen in Figure 4.5).





Start an edit session (Edit|Start Editing). Using the 'Feature Edit' toolbar insert a 'SWMM5 Raingauge' at an approximate location north of the wastewater system, by using the 'Create Feature' cursor (see Figure 4.6).







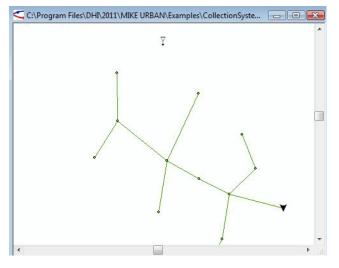


Figure 4.7 Raingauge is placed north of the system

4.3.2 Step 2: Editing the rain gauge parameters

When a raingauge is designated as receiving its rainfall data from a file, you must supply the name of the file and the name of the recording station referenced in the file. In the text file supplied the recording station is identified as 'TS2'

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 raingauge properties. For the other file types allowed these properties are defined by their respective file format and are automatically recognized by SWMM.

Rain gauge data is supplied in the table below.

Table 4.1 Rain gauge data

Raingauge Data	
Name	Raingauge
Rainformat	Intensity
Time Interval	0.05



Table 4.1Rain gauge data

Raingauge Data	
Snow Catch Factor	1
Data Source	File

The rain gauge data from the table above is entered in SWMM5|Boundary Conditions|Raingauges. As the time interval in TS.txt is entered in a 5 minutes time interval and the time interval in the dialog is entered in hours you will need to write the 5 minutes as a decimal number (i.e. 5/60 = 0.08333).

After entering the rain gauge parameters along with the reference to the rainfall data file the raingauge dialog will look like in Figure 4.8. Make sure to save your data by either using the appropriate toolbar or Edit|Save Edits which will save the data but leave MIKE URBAN open for editing.

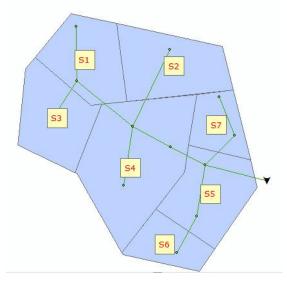
	Raingauges	connectivity				_		
	Rain gauge I	D:	Raingauge					Insert
Description:				-				Delete
	X coordinate:	:	649.945	-				Advanced
	Y coordinate:		1531.468					Close
	Raingauge pro	perties format						
	File:		TS.txt					
			Station name:		TS2			
			Unit type:	() in	() mm		
	Timeseries	s ID:		-				
	Format:			INTEN	ISITY	•		
	Time interval:				0.08	333		
	Snow catch de	eficiency corre	ection factor, SC	CF:	1.00)		
	MUID	TypeNo	FileNameSe	Station	Nam	S	cf	FormNo
ľ			TS.bxt	TS2			1.000	INTENSITY

Figure 4.8 Rain gauge input

4.4 Adding catchments and editing the catchment parameters

For the developed site, seven catchments will be created. The catchment layout and ID's are shown in the figure below.







4.4.1 Step 1: Adding the catchments

The catchments can be drawn by using the 'Edit Feature' toolbar. Click on the Create feature tool in the toolbar to insert a new catchment (the catchment layer must be activated first), see Figure 4.10.



Figure 4.10 Feature Edit Toolbar

 Click on the map and draw a polygon following the contours of your total catchment area. The catchment layout and ID's are shown in the figure below.

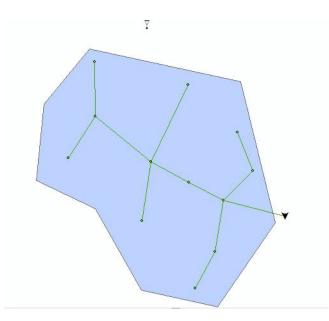


Figure 4.11 Start by drawing the contours of your total catchment area

Switch to the 'Split Polygon' tool is to split the catchment into smaller sub-catchments describing your system with respect to the network. As shown in the illustration below the total catchment has been split into seven smaller catchments. You will be asked to confirm each split – click OK.

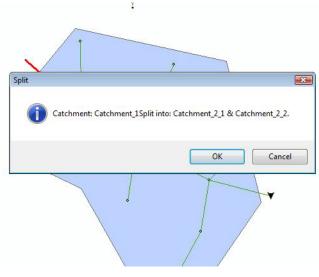


Figure 4.12 Using the Split Polygon tool

• When creating the catchments they will automatically receive an ID. In order to rename the catchments to the ID's as shown in Figure 4.9. go to SWMM5|Hydrology|Catchments to open the Catchments dialog and renaming the Catchment ID (see Figure 4.13). You can use the select tools to select the individual catchments (remember to set the 'Set Selectable Layers' to include catchments) and rename them afterwards.

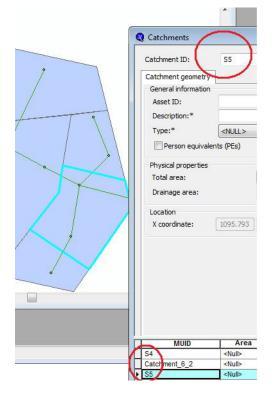


Figure 4.13 Renaming the catchments

Save your edits (Edit|Save Edits).

4.4.2 Step 2: Assigning catchment parameters

In the Catchments dialog the other parameters for the catchments need to be inserted. The catchment areas (Total Area) are automatically computed while the catchments are being inserted. The entire area is expected to drain - hence the Drainage Area is set equal to the Total Area. This can easily be done by using the field calculator. Right click in the heading of the Drainage Area column as seen in Figure 4.14. Make sure to have either all catchments - or none - selected as the Field Calculator works on selected data sets.



Catchment ID:	S1	Catchment area
Catchment geometr	у	
General informatio	n	
Asset ID:		System:*
Description:*	-	Sub-system:*
Type:*	<null></null>	▼ Status:*
Person equiva	alents (PEs)	
Physical properties		
Total area:	3.833	Max. level:*
Drainage area:		Min. level:*
Location		
X coordinate:	421.915 421.91	15 Y coordinate:
MUID	Area	X I Y I
S1	Area I	
	Area	X Y

Figure 4.14 Activating the field calculator

In the field calculator set the drainage area (Area) equal to the total computed area (Area_C) and press 'OK'. Save your edits (Edit|Save Edits).



Parser			
) Python		
elds:		Type:	Functions:
OBJECTID	*	G Number	Abs ()
SHAPE		-	Atn () Cos ()
MUID	E	String	Exp()
Area		O Date	Fix ()
x			Int() Log()
Y			Sin ()
Area_C			Sqr ()
x_c			Tan ()
Y_C	+		
Show Codeblock		G	
rea =		*	* / & + - =
[Area_C]			
	Clear	Load	Save Help



Save your edits (Edit|Save Edits).

4.4.3 Step 3: Assigning routing parameters

Table 4.2 lists the Width, slope, imperviousness, roughness coefficients, depression storage and percentage of impervious area without storage parameters which are to be inserted in the 'Routing' dialog found at SWMM5|Hydrology|Routing. The Catchment ID, Loading Point ID and Raingauge ID must be specified here.

All subcatchments:

- Load to Node
- Subarea Routing: Outlet

Sub- catch- ment ID	Load Point ID	Width (ft)	Grou nd Slope (%)	Per- cent Imper- vious	Impervi- ous Man- ning	Pervi- ous Manni- uing	Impervi- ous Depres- sion Stor- age (in)	Pervi- ous Dep Stor- age (in)	% of imper vious area with- out dep stor- age
S1	M1	1587	2	56.8	0.015	0.24	0.06	0.3	25
S2	M4	1653	2	63	0.015	0.24	0.06	0.3	25
S3	M3	1456	3.1	39.5	0.015	0.24	0.06	0.3	25
S4	M6	2331	3.1	49.9	0.015	0.24	0.06	0.3	25
S5	M10	1670	2	87.7	0.015	0.24	0.06	0.3	25
S6	M12	690	2	95	0.015	0.24	0.06	0.3	25
S7	M8	907	3.1	0	0.015	0.24	0.06	0.3	25

Table 4.2 Catchment routing data

Press 'Insert' on the Routing dialog and then enter the data from the table. You can use the field calculator for speeding up the entry of data that is the same for all the catchments.



I de altre altre a	ase] & connectivity							
Catchment I		S1	Ta				_	Insert
				y.				Delete
Load to: Node 🔻		•		_		_		
Load point II	ad point ID: M1		De	scription:			F	Advanced
Outlet catchment ID		Ra	in gauge ID:	F	laingauge		Geocode	
Hydrology &	hydraulic prope	rties						Close
Width:		1587.00	Imp	perv. d. storage	e: 0	.06		
Ground slope	e:	2.00	Pe	rv.d.storage:	0	.30		
Imperviousne	BSS:	56.80	%	DCIA w/o d. sto	prage: 2	5.00	-	
Imperv. man	ning:	0.0150	Subarea routing:		6	UTLET	•	
Pervious ma	nning:	0.2400		Percent routed:		00.00	1	
Snowpack II	D:	-			Curb length: 0		_	
CatchID *	Description	Tag	RaingageID	LoadToNo	OutletNodel	Width	Slope	Impervious
	<null></null>	<null></null>	Raingauge	Node	M1	1587.00	2.00	56.80
		<null></null>	Raingauge	Node	M4	1653.00	2.00	63.00
S1 S2	<null></null>							
S2 S3	<null></null>	<null></null>	<null></null>	Node	M3	1456.00	3.10	
S2 S3 S4	<null> <null></null></null>	<null> <null></null></null>	Raingauge	Node	MG	2331.00	3.10	49.90
S2 S3 S4 S5	<nul> <nul> <nul></nul></nul></nul>	<null> <null> <null></null></null></null>	Raingauge Raingauge	Node Node	M6 M10	2331.00 1670.00	3.10 2.00	49.90 87.70
	<null> <null></null></null>	<null> <null></null></null>	Raingauge	Node	MG	2331.00	3.10	49.90 87.70

Figure 4.16 The Routing dialog after data has been entered

4.4.4 Step 4: Assigning catchment infiltration data

Table 4.3 contains the max and min infiltration rates, decay rate and DWF regeneration parameters which are to be inserted in the 'Catchment Infiltration' dialogue found at SWMM5|Hydrology|Infiltration.

Sub-catch- ment ID	Max infiltration rate (in/h)	Min infiltration rate (in/h)	Decay rate	DWF Generation (h)
S1	4.5	0.2	6.5	7
S2	4.5	0.2	6.5	7
S3	4.5	0.2	6.5	7
S4	4.5	0.2	6.5	7
S5	4.5	0.2	6.5	7
S6	4.5	0.2	6.5	7
S7	4.5	0.2	6.5	7

Table 4.3 Catchment Infiltration Data



Using the field calculator is a huge advantage when entering the infiltration data. Use Edit|Save Edits to save data.

	n & connectivity	2				
Catchmer	it ID:	S1				Delete
nfiltration pr Horton	roperties					Advance
Max infiltration rate:		4.50	Decay rate:		6.500	Close
Min infiltration rate:		0.20	DWF regene	eration:	7.00	0030
Max infiltr	ation vol.:					
Green am	pt					
Soil capil	ary suction:		Conductivity:			
Initial soil	moisture:]			
Curve nur	nber					
Curve nu	mber:		Conductivity	Conductivity:		
DWF reg	eneration:		j			
CatchID *	MaxRate	MinRate	DecayRate	HRegen	MaxInFil	- 1
1	4.50	0.20	6.500	7.00	<null></null>	
2	4.50	0.20	6.500	7.00	<null></null>	
3	4.50	0.20	6.500	7.00	<null></null>	
4	4.50	0.20	6.500	7.00	1.6	
5	4.50	0.20	6.500	7.00	<null></null>	
6	4.50	0.20	6.500	7.00	<null></null>	
7	4.50	0.20	6.500	7.00	<null></null>	

Figure 4.17 The Infiltration dialog after data has been entered

4.5 Simulation

To simulate the combined system go to Simulation|Run SWMM5.

You will need to set up the 'General', 'Network' and 'Runoff' tab. Only one simulation needs to be run, we will call this simulation "2Y_Combined_System".

Table 4.5 contains the information needed to set up the 'General' tab.

	-
General	
Start Date/Time	01/01/2007 00:00:00
End Date/Time	01/01/2007 12:00:00
Start Reporting	01:00:00

Table 4.4 General settings

Table 4.4 General settings

General	
Reporting Time Step	00:01:00
Flow Units	CFS

General Runoff Netwo	rk Summary				
Identification & connectiv	vity			Close	
Simulation ID:	2_Year_Combine	ed_System		Close	
Scenario ID:	Base				
Description:					
Simulation period			Start simulation		
Start analysis on:	01-01-2007	00:00:00	Single run		
End analysis on:	01-01-2007	12:00:00	Single run		
Start sweeping:	01-01-2007	00:00:00			
End sweeping:	01-01-2007	12:00:00			
Start reporting:	01-01-2007	00:00:00			
Reporting time step:		00:01:00 💠			
Units selection					
OFS (cubic feets peed)	r second)	CMS (cubic meters per	second)		
○ GPM (gallons per minute)					
MGD (millions gallon	s per day) 💿	MLD <mark>(</mark> million liters per d	ay)		

Figure 4.18 General settings for the simulation

Table 4.5 contains the information needed to set up the 'Runoff' tab.

Table 4.5 Runoff settings

Runoff	
Infiltration Method	Horton
Wet Weather Timestep	00:01:00
Dry Weather Timestep	01:00:00

Identification & connectiv	rity				1	Close	
Simulation ID:	2_Yea	_Combin	ed_System		1	Close	
Scenario ID:	Base						
Infiltration method		Runoff t	ime steps				
Horton		Wet w	eather:	00:01:00	*		
🔘 Green ampt		Dry w	eather:	01:00:00	4		
Curve number		Antec	edent dry days:				
Runoff interface files							
🕅 Rainfall data	O Use						
Save runoff data	U Savi						
🕅 Save RDII data							

Figure 4.19 Runoff settings for the simulation

Table 4.6 contains the information needed to set up the 'Network' tab. Within the 'Routing Simulation' option the 'Model Type' must be set to DYNWAVE or the error message 'ERROR 115: adverse slope for Conduit xxx' will be received. Under Steady or Kinematic Wave routing, all conduits must have positive slopes. Adverse slopes are permitted under Dynamic Wave routing.

Table 4.6 Network settings

Network	
Model Type	DYNWAVE
Force Main Solution	Hazen-Williams
Time step	00:00:15
Dynamic Wave Time step	15
Safety Factor	0.75



eneral Runoff	Network Si	ummary						
Identification & con	nectivity				Network	k settings		Close
Simulation ID:		2_Yea	ar_Com	nbined_System				Close
Scenario ID:		Base			Link offs	set:	e 🔘 Elevation	
Routing simulation				Dynamic wave options				
Model type:	DYNWA	VE	•	Inertial terms:	None	Partial	C Full	
Force main soln:	Hazen-V	Villiams	•	Normal flow limited:	Slope	C Froude	Obtemport Both	
Time step:	00:00:15	ł	\$ -	Skip steady state		Safety factor:	0.75	
🔲 Ignore rainfall .	/ runoff			Allow ponding at jun	ctions	Time step:	15	
						Surface area:	0.00	
Routing interface fi	les							
🔄 Hotstart data		O Us						
Inflow		0.00						
Outflow								
📃 Use RDII data								
Use runoff dat	a							

Figure 4.20 Network settings for the simulation

Once the simulation has been set up completely, simply click 'Single Run' on the 'General' tab to run.

4.6 Simulation Results

After the simulations of the combined system has been completed, the result can be viewed using MIKE URBAN. To load results go to Model Results|Load Results|SWMM5 Results and choose the result file generated 2_Year_Combined_SystemBase.out.

To produce a discharge hydrograph go to Model Results|Open Time Series and choose 'Link: Discharge', click on a specific pipe to produce the discharge hydrograph. Link with ID L12 was used below.



le <u>E</u> dit <u>Z</u> oom F <u>o</u> rmat	Export	Window			
- TSTabular Item Selection		Time	L12 [ft^3/s]		L12 [ft^3/s]
	1	01-01-2007 00:01:00.00	0.00011389		Discharge
L12	2	01-01-2007 00:02:00.00	0.00030700		
10.01.0.44	3	01-01-2007 00:03:00.00	0.00139963		
	4	01-01-2007 00:04:00.00	0.01067527		40
	5	01-01-2007 00:05:00.00	0.10467635		
	6	01-01-2007 00:06:00.00	0.39900419		
	7	01-01-2007 00:07:00.00	0.65625411		30 -
	8	01-01-2007 00:08:00.00	0.84633284		30
	9	01-01-2007 00:09:00.00	0.97814589		
	10	01-01-2007 00:10:00.00	1.06155562		
	11	01-01-2007 00:11:00.00	1.11977040		20
	12	01-01-2007 00:12:00.00	1.17007851		
	13	01-01-2007 00:13:00.00	1.22250950		
	14	01-01-2007 00:14:00.00	1.31088626		
	15	01-01-2007 00:15:00.00	1.49302828		10 10
	16	01-01-2007 00:16:00.00	1.80012416		
	17	01-01-2007 00:17:00.00	2.31383728		
	18	01-01-2007 00:18:00.00	2.97543334		
	19	01-01-2007 00:19:00.00	3.73784899		04:00 08:00 12:0
(III)	20	01-01-2007 00:20:00.00	4.52795028	-	2007-01-01



Now you can continue to view results - or change the storm event to anlayze the system under different events.

5 2D Overland Flow Example

5.1 Introduction

The files found in the folder Examples\CollectionSystemMOUSE\2DOverlandFlow constitute a MIKE URBAN data set allowing for running a combined 1D pipe flow and 2D overland flow simulation.

The model includes the following model components:

- Storm water drainage network, which discharges through an outlet to the coast line in the northeastern corner of the model areas
- Catchments and corresponding rainfall-runoff models providing inflow to the storm water drainage network
- Digital elevation model
- Definition of 2D model area and resolution
- Definition of couplings between the pipe flow and overland flow model

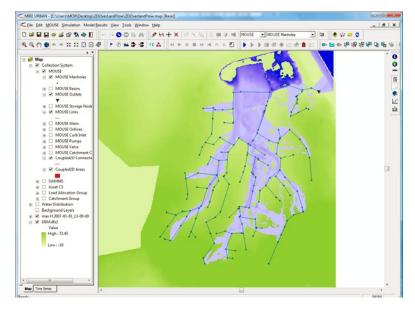


Figure 5.1 The model with DEM and flooding results

5.2 Running the model

The 2D model and model couplings have been defined in the data set provided. To run the combined pipe flow and overland flow models you should do the following:



- 1. Start MIKE URBAN and open the .mdb file
- 2. Add the DEM.dfs2 file as a raster background layer
- 3. Choose MOUSE > Simulation and click the "Start simulation" button



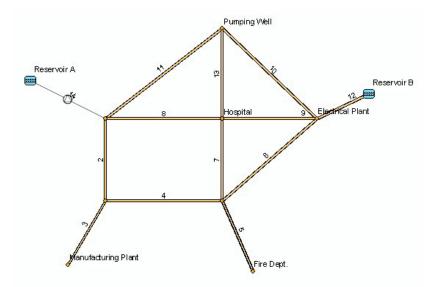
6 Water distribution tutorials

6.1 Lesson 1 - Defining a Pipe Network System

This lesson takes you step-by-step, illustrating how to use MIKE URBAN WD DISTRIBUTION (WD) to define the input data for a pipe network system. This lesson discusses the steps required to begin a project, define a pipe network system, perform an analysis, and display the graphical results.

Before you begin to define your pipe network using MIKE URBAN WD, it is generally more efficient to gather all of the materials you will need to define the model. For example, create a schematic diagram of the pipe network system to be analysed. Then, list all of the junction nodes, pipes, pumps, valves, storage tanks, and other components that make up the water distribution model.

Figure 6.1 shows a schematic diagram of the pipe network to be created in this lesson. The pipe network system shown in Figure 6.1 consists of two reservoir nodes, 8 junction nodes, 13 pipes, and a booster pump. Reservoir A is the designated pressure source, so water is pumped into the water distribution system from reservoir A. To simplify data input, all pipes, pumps, tank nodes, and junction nodes should be numbered in the schematic diagram, as shown in Figure 6.1. In subsequent lessons, additional components will be added to this schematic diagram.





The pipe network system, as shown in Figure 6.1, is intended to represent a typical water distribution system for a small city. Each pipe shown in the figure represents a large water main underneath a major city street. Thus, these pipes are labelled with their ID numbers in the schematic diagram. At junction nodes, demands are entered as multiple junction demand (positive demands for outflow and negative demands for inflow).

To save time with this lesson, we have prepared data files that have already been set up in order for you to follow through the lesson. For a list of all the files in this lesson, see the section titled Prepared Input and Output Files.

6.1.1 Defining the Project

To begin a new project, select File | New. The program will display the Create New Project dialog box, as shown in Figure 6.2. This dialog box allows you to specify the project name (database name), the working mode of the program (Collection System, Water Distribution), and the project units. After specifying the project name, select EPANET | Project Options. The program will then display the Project Options dialog box, as shown in Figure 6.3. This dialog box allows you to specify project configuration information for the pipe network system to be modeled. Project configuration data includes analysis type, friction loss formulation, simulation options, analysis options, and flow-rate units.

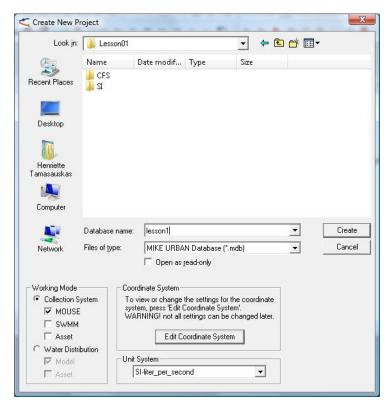


Figure 6.2 The Create New Project Dialog Box

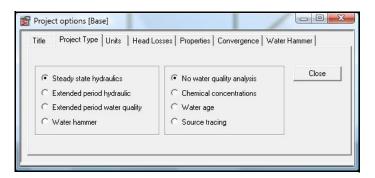


Figure 6.3 The Project Options Dialog Box

The Project Options dialog box, when first displayed, will contain default values. This lesson uses these default values, so no changes are necessary to these settings in the Project Options dialog box.

Dynamic Data Input

Many of the data input dialog boxes in MIKE URBAN WD are dynamic, and thus change based on the settings specified in the Project Options dialog

box. For example, the friction loss formulation (i.e., Hazen Williams, Darcey Weisbach, or Chezy Manning) selected within the Project Options dialog box determines what units are displayed for the Roughness data entry field displayed in the Pipe Editor dialog box.

6.1.2 Defining Junction Node Data

After defining the initial project configuration values, the next step is to define the junction node data for the pipe network system. Table 6.1 lists elevation, demand, and label data used to define the eight junction nodes contained in the pipe network system shown in Figure 6.1.

Table 6.1 Junction node data for the pipe network system shown in Figure_1.1

Node	Elevation (ft)	Demand (cfs)	Description
1	97 (30 m)	0.0	
2	102 (31 m)	0.0	
3	96 (29 m)	0.8 (25 l/s)	Manufacturing Plant
4	100 (30 m)	0.0	
5	120 (37 m)	0.5 (12 l/s)	Fire Station
6	110 (34 m)	1.0 (30 l/s)	Electrical Plant
7	104 (32 m)	1.1 (30 l/s)	Hospital
8	96 (29 m)	-1.2 (-35 l/s)	Pumping Well

Each junction node has a unique numerical ID that identifies it. For each junction node, an elevation and water demand is defined. Typically, junction nodes are labeled with location names corresponding to building names. Therefore, junction nodes 3, 5, 6, 7, and 8 are given label descriptions that indicate their location and may suggest the type of demand that can be expected at each location.

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

We will use the Horizontal Plan window to interactively layout a schematic representation of the water distribution network. Once the network has been graphically laid out, we will then further define the actual junction node, pipe,



and other related data using the network component editors in MIKE URBAN WD.



Figure 6.4 Use the Add Junction tool for interactively laying out the network junction nodes

In order to quickly draw a network layout, click on the Create Feature tool from the Feature Edit floating toolbar as shown in Figure 6.5. The icon will become active (it will appear pressed down) and you can then point and click within the Horizontal Plan window to graphically add junction node locations. Using this tool and referring to Figure 6.1, start by adding junction node 1 and continue through to junction node 8. Place these nodes in the sequential order as they are shown in Figure 6.1 so that the node IDs match the rest of this lesson's input. When finished, click again on the Add Junction tool (it will pop back up) to end insertion of junction nodes.

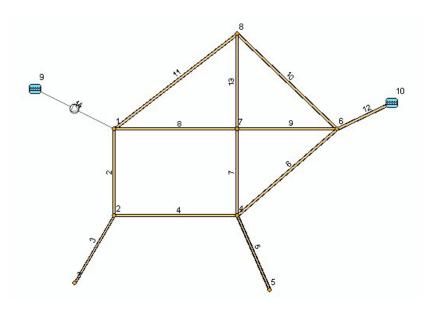


Figure 6.5 Node and pipe IDs

Once the junction nodes have been graphically laid out in the Horizontal Plan window, additional data can be specified to further define the junction nodes that comprise the model. This is done by using the Junction Editor dialog box, as shown in Figure 6.6. To display the Junction Editor, select EPANET | Junctions.

Junction d	ata							- 0
unction dat	Emitter data	Air-valve data	al					
	ion & connectivi	là					Insert	
Asset ID:			Data	source:			Delete	
Junction I	D:	1	Statu	5:	<null></null>	-		_
X coordin	ate:	42.84					Advanced.	
Y coordin-	ate:	98.65	_				Close	
Descriptio	n:		Press	ure zone:				_
– Model dat	-	·						
	a	Junction	▼ State:		unmarked			
					unmarked			
Type:		Junction			- Internet			
Demand o	oefficient:		Eleva		97.000			
			Eleva		97.000			
Demand o			Eleva	tion:	97.000			
Demand o			Eleva	tion:	97.000			
Demand o			Eleva	tion:	97.000			
Demand o Minimal pr	essure:	Category	Eleva	tion:	97.000			
Demand of Minimal pr	essure:		Eleva	tion:	97.000		 	
Demand of Minimal pr	essure:		Eleva	tion:]]	
Demand of Minimal pr Demand	essure:	Category Data source	Pattern*	tion: ce elevation:	Surface Elev		Statello	
Demand of Minimal pr Demand	essure:	Category Data source <null></null>	Pattern* Description <null></null>	tion: ce elevation:	Surface Elev	<null></null>	unmarked	<nu< td=""></nu<>
Demand of Minimal pr Demand	essure:	Category Data source <null> <null></null></null>	Pattern* Pescription	tion: ce elevation:	Surface Elev <null> <null></null></null>	<null> <null></null></null>	unmarked unmarked	l ≺Nu ≺Nu
Demand d Minimal pr Demand	essure:	Category Data source <null> <null> <null> <</null></null></null>	Pattern' Pescription <null> Manufacturin</null>	tion: ce elevation:	Surface Elev <null> <null></null></null>	<null> <null> <null></null></null></null>	unmarked unmarked unmarked	<nu <nu <nu< td=""></nu<></nu </nu
Demand of Minimal pr Demand	essure:	Category Data source <null> <null> <null></null></null></null>	Pattern* Pattern* Description <null></null>	tion: ce elevation: Elev 97 102 96 100	Surface Elev <null> <null> <null></null></null></null>	<null> <null> <null> <null></null></null></null></null>	unmarked unmarked unmarked unmarked	<nu <nu <nu <nu< td=""></nu<></nu </nu </nu
Demand d Minimal pr Demand MUID'	essure:	Category Data source <null> <null> <null> <null></null></null></null></null>	Pattern* Pattern* Pescription <null> <null> Manufacturin Fire Dept.</null></null>	tion: ce elevation: Elev 96 100 96 100 120	Surface Elev <null> <null> <null> <null> <null></null></null></null></null></null>	<null> <null> <null> <null> <null></null></null></null></null></null>	unmarked unmarked unmarked unmarked unmarked	<nu <nu <nu <nu< td=""></nu<></nu </nu </nu
Demand of Minimal pr Demand MUID'	essure:	Category Data source <null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	Pattern* Pat	tion: ce elevation: Elev 97 102 96 100 120 120	Surface Elev <null> <null> <null> <null> <null> <null> <null></null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	unmarked unmarked unmarked unmarked unmarked unmarked	<nu <nu <nu <nu <nu <nu< td=""></nu<></nu </nu </nu </nu </nu
Demand d Minimal pr Demand MUID'	essure:	Category Data source <null> <null> <null> <null></null></null></null></null>	Pattern* Pattern* Pescription <null> <null> Manufacturin Fire Dept.</null></null>	tion: ce elevation:	Surface Elev <null> <null> <null> <null> <null></null></null></null></null></null>	<null> <null> <null> <null> <null></null></null></null></null></null>	unmarked unmarked unmarked unmarked unmarked	<nu <nu <nu <nu <nu <nu <nu< td=""></nu<></nu </nu </nu </nu </nu </nu

Figure 6.6 The Junction Editor dialog box

Each junction node has a unique numerical ID that identifies it. The data used to define each junction node includes node elevation, multiple water demand, pattern ID, pressure zone, X–Y coordinates, and a text description. These data requirements are described in detail in the section titled Junction Editor in Chapter 4. In this example we will only define the junction node data that is shown in Table 6.3.

To define additional data for junction node 1 (see the schematic layout shown in Figure 6.1 and the node data in Table 6.1), select node 1 from the list of junction nodes in the Junction Editor. For the Elevation data entry specify 97, and for the Demand data entry specify 0. Because junction node 1 does not have a demand pattern, pressure zone ID, or a text description, the remaining data entries can remain blank.

Once the data for junction node 1 has been defined, repeat this process for the other remaining junction node data in Table 6.1. Note that data can be defined in either the data entry fields or directly in the table.

Horizontal Plan Options

To display node ID numbers in the Horizontal Plan window, select the Properties of Junctions Layer (by clicking with the right mouse button while in the



Layer Manager Window). The Layer Properties dialog box will be displayed, as shown in Figure 6.7.

eneral Selection Labels Symbology	
Layer Name: Water Junctions	Visible
Description:	~
	-
Credits:	
Scale Range	
You can specify the range of scales at which this layer will be sho	own:
Show layer at all scales	
C Don't show layer when goomed:	191
Out beyond: <a>None> (minimum scale)	
	6 G 5
In beyond: None> _ (maximum scale)	52 AT

Figure 6.7 The Layer Properties dialog box is used to adjust the display settings for horizontal plan (map) window

From the Layer Properties dialog box, select the Labels tab and then turn on the Label Features in this layer checkbox. Next, select Junction ID from the list of available label fields in Text String group box. Selecting «OK» will cause the node ID numbers to be displayed in the Horizontal Plan window.

From the Layer Properties dialog box it is also possible to select the font used for displaying the node labels, label placement options, label styles, and many other settings. Note that you can format pipe labels in a similar way.

Displaying Multiple Dialog Boxes

To allow you to be more efficient while defining your water distribution model, MIKE URBAN WD allows you to open multiple dialog boxes simultaneously. For example, you can keep the Layer Properties dialog box displayed while you make changes to the display settings, and examine the effects of the defined changes by clicking on «Apply». Any number of dialog boxes can be opened simultaneously. The header bar of the currently active dialog box will be highlighted.

In this lesson, if desired, you can simultaneously display the Junction Editor, Pipe Editor, Tank Editor, and Pump Editor dialog boxes. To open these dialog boxes, select Junctions, Pipes, Tanks, and Pumps from the Edit Menu. If necessary, once these dialog boxes are displayed, you can minimize any of these dialog boxes by clicking on the minimize icon in the dialog box upper



right corner. This will shrink the dialog box to a small task button bar on the MIKE URBAN WD desktop work space.

Network Definition Methods

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

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

Many times, existing water distribution systems do not have a detailed map that can be used to graphically construct a network model. For these situations, MIKE URBAN WD allows the user to develop a model by simply defining water distribution components (i.e., pipes, junction nodes, pumps, values, tanks, and reservoirs) using the network component editor dialog boxes.

6.1.3 Defining Tank Data

The next step is to define the reservoir data for the pipe network system. Table 6.2 lists the tank data used to define the reservoirs contained in the pipe network system shown in Figure 6.1.

Node (ID)	HGL (ft)	Description
9	100 (30 m)	Reservoir A
10	200 (60 m)	Reservoir B

Table 6.2	Tank node data for the pipe network system	as shown in Figure_1.1
-----------	--	------------------------

As was performed in defining the junction node data, we will graphically define the reservoir nodes in the Horizontal Plan window using the Add Tank tool. Select the Add Tank tool from the Components floating toolbar. The icon will become active (it will appear pressed down) and then point and click within the Horizontal Plan window to graphically add tank node locations. Place these reservoir nodes in the sequential order as they are shown in Figure 6.1 so that the node IDs match the rest of this lesson's input. When finished, click again on the Add Tank tool to end insertion of tank nodes.



Once the reservoir nodes have been graphically laid out in the Horizontal Plan window, additional data can be specified to further define the reservoirs that comprise the model. This is done using the Tank Editor dialog box, as shown in Figure 6.8. To display the Tank Editor, select EPANET Tanks.



Figure 6.8 Adding a reservoir (tank) to the pipe network system

Each tank has a unique numerical ID that identifies it. The data used to define each tank node includes constant water elevation, which is a mandatory field, and several optional fields such as a text description, pressure zone, X-Ycoordinates, and phase. These data requirements are described in detail in the section titled Tank Editor in Chapter 2. In this example use the Tank Editor to define the reservoir data shown in Table 6.2.

Converting Junction Node to Tank Node

You can easily convert an existing junction node to a tank node using MIKE URBAN WD. To convert a junction node to a tank node, select the Modify Feature Type tool from the Components floating toolbar, select Tanks from the list box, and click on the existing junction node in the Horizontal Plan window. MIKE URBAN WD will display a query dialog, asking you whether to convert the selected junction node to a tank. Note that this method can be used to convert from a junction node to tank node (by using the Add Tank tool) or from tank node to a junction node (by using the Add Junction tool).

6.1.4 Defining Pipe Data

The next step in defining this network model is to define the pipe data. Table 6.3 lists the pipe data used to define the pipes contained in the pipe network system shown in Figure 6.1.

Table 6.3Pipe data for the pipe network system

Pipe	Nodes		Length	Diameter	Friction	Minor	Label			
(ID)	Start	End	(feet)	(inches)	Loss (C-M)	Loss				
2	1	2	5200 (1580 m)	12 (300 mm)	80 (1.5mm)	0.0	Randall St			
3	2	3	1000 (300 m)	12 (300 mm)	120 (1.5 mm)	0.0	Randall St			
4	2	4	8000 (2430 m)	12 (300 mm)	100 (1.5 mm)	0.0	Regent St			

Table 1:

5	4	5	9000 (2740 m)	12 (300 mm)	110 (1.5 mm)	0.0	Park St
6	4	6	3400 (1030 m)	12 (300 mm)	120 (1.5 mm)	0.1	Washington Ave
7	4	7	3450 (1050 m)	12 (300 mm)	120 (1.5 mm)	0.0	Park St
8	1	7	4000 (1220 m)	8 (200 mm)	80 (1.5 mm)	0.0	University Ave
9	7	6	2500 (760 m)	12 (300 mm)	100 (1.5 mm)	0.8	University Ave
10	8	6	3000 (910 m)	8 (200 mm)	100 (1.5 mm)	0.0	State St
11	1	8	5400 (1640 m)	12 (300 mm)	90 (1.5 mm)	0.0	University Bay Dr
12	10	6	700 (210 m)	15 (400 mm)	100 (1.5 mm)	0.1	University Ave

Table 1:

We will begin by graphically defining the pipes in the Horizontal Plan window using the Add Pipe tool. Select Pipes from the list box from within the Feature Edit floating toolbar and select Create Feature tool. The icon will become active (it will appear pressed down). Using Figure 6.1 as a guide for laying out the pipe network, point and click within the Horizontal Plan window on a node. A rubberbanding line representing the pipe will appear between the cursor and the node. Point and click on another node to define the pipe. To end insertion of a pipe, press «Enter» or double-click with your mouse cursor and the rubberbanding line will end. Place these pipes in the sequential order as they are shown in Figure 6.1 so that the pipe IDs match the rest of this lesson's input. Continue laying out pipes until you have completed the pipe network system, as is shown in Figure 6.1.

Note that while you are defining a pipe using the rubberbanding line, pressing «Enter» will end the line at the last junction node, double-clicking with the left mouse button will end the pipe at the current cursor position by inserting a junction node, pressing «Backspace» will delete the last inserted junction node and pipe segment, and pressing «Esc» will abort the pipe insertion.

As you click on or near an existing node, MIKE URBAN WD will snap to the node and treat this as a starting node. A rubber-banding line will then be drawn from this node, representing the pipe, while you select the ending node. Then, if you click on or near an existing node, it will snap to that node and treat the selected node as the ending node drawing in the pipe. If you click anywhere else, it will place an ending node at the selected location.



While selecting the starting node, if you click with the Add Pipe tool in the Horizontal Plan window somewhere else other than a node or a pipe, MIKE URBAN WD will place a starting node at the clicked location. Additional information on how to use the Add Pipe tool to graphically layout a pipe network is discussed in Water Distribution User Guide.

Once the pipes have been graphically laid out in the Horizontal Plan window, additional data can be specified to further define the pipes that comprise the model. This is done using the Pipe Editor, as shown in Figure 6.9. To display the Pipe Editor, select EPANET | Pipes.

e	Pipes									
1	-Identification	& connec	tivity-							
	Asset ID: CHARID			RID) Data source:				Insert	
	Pipe ID:				Status:		<null></null>	•	Delete	
	From node: 8 To node: 6								Advanced	
				St		Pressure zone ID:				
	- Geometrical r	nonerties								
	Length:			.00 76.552	Diameter:		8.000			
п П	-Hydraulics &	friction los	ses							
	Material:		MAT	ERIAL	Formulatio	m:	Hazen-Williams			
	Construction year:				_ Roughnes	is:	100.000			
	Wall thickness:		Ē		Loss coeffici		0.000			
	Wave speed:			Pressure normal:						
1	- Miscellaneou	0								
		Demand coeff. 1: 1.000 State: unmarked								
			1.000	-	Check	1	Closed			
						, vaive.	I_ Closed	_		
	Street name:	Street name: STREETNAME								
-	Pipe ID*	Descri	ption	CDate	Constructio	Asset ID	Data source	Dem	and coeff. 1	
•	10	State St		1/1/2005	<null></null>	CHARID	<null></null>			
	11	Universi	ity Ba	1/1/2005	<null></null>	CHARID	<null></null>			
-	12	Universi	ity Av	1/1/2005	<null></null>	CHARID	<null></null>			
	13	Park St		1/1/2005	<null></null>	CHARID	<null></null>			
	2	Randall St		1/1/2005	<null></null>	CHARID	<null></null>	÷		
	3	Randall	St	1/1/2005	<null></null>	CHARID	<null></null>			
	4	Regent	St	1/1/2005	<null></null>	CHARID	<null></null>	5		
	5	Park St		1/1/2005	<null></null>	CHARID	<null></null>			
	6		ghton	1/1/2005	<null></null>	CHARID	<null></null>			
	1-	0.00		A M MOODE		OLIADID				



Each pipe has a unique numerical ID that identifies it. Each pipe definition includes the two connecting nodes, identified as starting and ending nodes. The specified flow direction is defined as going from the starting node to the ending node. To reverse the order of the nodes, choose «Swap·Nodes» from Advanced.

The data used to define each pipe includes length, diameter, friction loss coefficient (roughness or roughness coefficient), minor loss coefficient, and

demand coefficients. If a friction loss coefficient is not available, a value can be interpolated using the data tables accessed through the MIKE URBAN WD Options Menu. Minor loss coefficients can be specified if effects of fittings along a pipeline are to be considered (see the Water Distribution user guide for further). In this example we will use the Pipe Editor to define the pipe data shown in Table 6.3.

Note that the pipe ID is automatically assigned when the pipes are drawn in the Horizontal Plan window. Therefore in your example, if you did not follow the layout of the pipe network as shown in Figure 6.1, the pipe IDs may differ than what is shown in Table 6.3. Therefore, when defining the additional pipe data for this lesson, only consider the starting and ending nodes when identifying what data is to be entered into the Pipe Editor.

6.1.5 Defining Pump Data

The function of a booster pump is to overcome the friction resistance and head loss in transporting water from one location to another. There are three types of booster pumps available in MIKE URBAN WD, including flow pumps, power pumps, and head pumps.

Flow Pumps

A flow pump is a booster pump whose flowrate is constant. Data required to fully describe a constant flow pump include its controlled pipe and specified flowrate (in cfs, gpm, or mgd). Flow pumps are defined as a Flow Control Valve (FCV) in the Valve Editor.

Power Pumps

A power pump is a booster pump whose pump power is constant. Data required to fully describe a power pump include its controlled pipe and specified power (in HP).

Head Pumps

Head pumps can be defined as constant head pumps, exponential pumps, and exponential extended pumps.

A constant head pump is a booster pump whose head is constant regardless of operating flowrate. The parameters of a constant head pump include its controlled pipe and constant pump head (in feet or meters).

An exponential standard pump is a booster pump whose pump head is described by a pump characteristic curve fitted to three data points. Each data point consists of a pump head and a corresponding flowrate. The pump curve is represented by an exponential formula fitted to these three points. The parameters of an exponential pump include its controlled pipe and three data points. A standard pump curve with no extended flow range (where the



cut-off head is 133% of the design head and the maximum flow is twice the design flow) is used if only a single operating point is specified.

An exponential extended pump is a booster pump whose pump curve is represented by an exponential formula fitted to three points. The parameters of an exponential pump include its controlled pipe, three data points and maximum flow in extended flow range.

Illustrations of the input requirements for the various types of pump curves are shown in Figure 6.10.

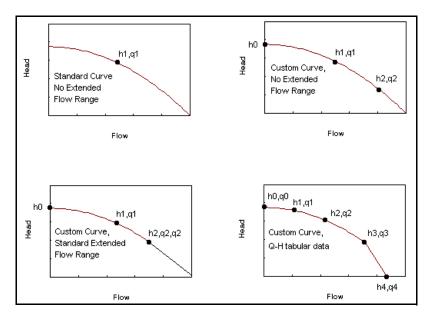


Figure 6.10 Input requirements for the various types of pump curves

The pipe network system, as shown in Figure 6.1, contains a booster pump on pipe 1. This pump is an exponential pump. The pump can be added to the pipe network either graphically using the Add Pump tool from the Components floating toolbar or by defining the starting and ending nodes in the Pump Editor dialog box. In this example, we will use the Pump Editor to define the pump. To display the Pump Editor, select Edit | Pump Editor.



-	Pumps							_	
	Asset ID: Pump ID: From node: To node:	& connectivity- CHARII 14 9 1			urce:	<pre></pre>		<u>I</u> nser Delet Advance	e
	Description: Model data Pump type Constant po Shutoff: Design:	3-point c wer: Head (m) 300.000 [250.000	urve The flow (m/s)	Relative Pump st Energy (Energy (speed: ate: usage price: price pattern:	unmarked	IN		
	High end: Q-H Curve: Plot PumpID*	AssetID	Data source	Water h	ettings	Variable :	speed drive	Shutoff	Desig
	14	CHARID	<null></null>	University Av	EFCURVE	EPATTERN	<null></null>	300	
4									Þ

Figure 6.11 The Pump Editor dialog box

To add a new pump in the Pump Editor, click on «Insert». A new pump will then be added into the pump table, allowing you to define the necessary pump data. In the Pump Type frame select the 3-point pump curve option, used to define an exponential pump. An exponential pump is defined by a head versus flowrate curve. Therefore, three data points must be entered in the Head and Flow data entries. For the first data point, enter 300ft (90m) for the shutoff head value. Define the second data point by entering 250ft (75m) for the design head and 8cfs (225l/s) for the design flow. Finally for the high end values, enter 160ft (50m) as the head and 10cfs (285l/s) for the flow. Next, define the starting and ending pump nodes, looking at the node IDs shown in Figure 6.1. Note that the order of these two nodes define the pumping direction. To reverse the direction of the pump (and the order of the nodes), choose «Swap·Nodes» from Advanced.

When you are finished defining the pipe network system, save the completed pipe network system as LESSON1A.MDB by using File | Save As. For more details, see the section titled Saving Your Data in this lesson. In this lesson we will save each modification to the pipe network system as a separate file so that they can be analyzed, reviewed, and compared later in this lesson.

Note

All pumps are assumed to be operating continuously throughout a simulation unless the Pump Status is set to CLOSED in the Pump Editor or the Control Editor. In addition, the program automatically prevents reverse flow through a



pump, and issues a warning message when a pump is operating outside of its normal operating range. Additional information on defining pumps can be found in the Water Distribution User Guide.

Graphically Defining Pumps

You can also define pumps graphically from the Horizontal Plan window. To insert a pump between two existing junction nodes, select the Add Pump tool from the Components floating toolbar. Then select the beginning and ending junction nodes. MIKE URBAN WD will then insert a pump between these two nodes. If you want to insert a pump into a pipe, select the Add Pump tool and click on the pipe that you want to insert a pump into. MIKE URBAN WD will display a query dialog box, asking if you want to replace the pipe with a pump or insert a pump into the pipe. For more details on inserting a pump, see the Water Distribution User Guide.

6.1.6 Opening a MIKE URBAN WD Data File

If you prefer, instead of interactively entering the data for this lesson, you can load in an existing MIKE URBAN WD data file that has the completed pipe network data used for this lesson. To open this data file, select File | Open. MIKE URBAN WD will display an Open dialog box. For this lesson, select LESSON1.MDB from the LESSONS\LESSON1 subdirectory by double clicking on the filename or highlighting the filename and selecting «OK». MIKE URBAN WD will then load this data file. This data file contains all the data used to define the pipe network system for this lesson, as shown in Figure 6.1.

6.1.7 Saving Your Data

To save the pipe network system you have defined, select File | Save or File | Save As. Select the appropriate subdirectory and then enter a filename of up to eight characters.

6.1.8 Performing the Analysis

After you have finished defining the pipe network model, you can perform an analysis of the pipe network system. Select Simulation | Run Simulation. MIKE URBAN WD will then display Run Simulation dialog from where it is possible to run the simulation or check the project for errors. If no errors were reported, MIKE URBAN WD will then perform a hydraulic analysis of the pipe network model. If an error is reported during the analysis, it will be necessary to correct the input model to remove the error. However, it is normal for warnings to be reported during the analysis. The user should check the analysis output to make certain that any reported warnings or status messages do not pose a threat to the validity of the analysis results.



6.1.9 Prepared Input and Output Files

Completed input and output files were provided for this lesson. These files are:

 LESSON1.MUP, LESSSON1.MDB, LESSON1.RES. These files are the input and output files with the basic network components defined. These files are used as the starting files for this lesson.

These files can be found in the LESSONS\LESSON01\CFS or LES-SONS\LESSON01\SI subdirectory and can be used to perform the analysis and view the analysis results, without having to interactively enter the data for this lesson.

6.1.10 Viewing the Analysis Results

After the analysis has been successfully performed, you next need to load the analysis results into MIKE URBAN WD. Select Model Results| Load Simulation Results|EPANET Results. MIKE URBAN WD will display a Load Analysis Results dialog box. From this dialog box, select the analysis results file LES-SON1.RES. MIKE URBAN WD will then load this analysis results file into memory.

EPANET Analysis Results

To view the analysis results generated by the EPANET Analysis Engine click on Summary button in the Simulation dialog. MIKE URBAN WD will display a file viewer, as shown in Figure 6.12, displaying the EPANET analysis results. If there are any warning messages during the analysis, they will be displayed in the EPANET analysis results.

LESSON1A.SUM - Note ile Edit Format View		
Page 1	Ten Jan 09 16	E4-22-2004
raye I	FIL JAN 05 16	.54.37 2004
***********	******	*****
*	EPANET	*
*	Hydraulic and Water Quality	*
*	Analysis for Pipe Networks	*
*	Version 2.0	*
***********	***************************************	*******
Turnet Data 1	F: 1 -	
Input Data I	File	TND
	unctions 8	. THE
	eservoirs	
	anks 0	
	ipes 12	
	umps 1	
	alves0	
	rmulaDarcy-Weisbach	
	imestep 1.00 hrs	
	ccuracy 0.001000	
	als 40	
	lysis None	
	avity	
Relative Kin	nematic Viscosity 10.76	
	emical Diffusivity 1.00	
Demand Mult:	iplier 1.00	
Total Durat:	ion 0.00 hrs	
Reporting C:	riteria:	
No Nodes		
No Links		
Analysis begun 1	Fri Jan 09 16:54:37 2004	
Analysis ended 1	Fri Jan 09 16:54:37 2004	
energen z szonató (berenzőn) i		

Figure 6.12 The EPANET Analysis Results

Analysis Results Table

To review the loaded analysis results in a tabular format, select Model Results| Results Statistics. MIKE URBAN WD will then display the Results Statistics dialog box, as shown in Figure 6.14. From this dialog box, choose «Calculate» to generate the results statistics display the results for all of the nodes and links within the system. MIKE URBAN WD will save the results (minimum and maximum values, or actual time level values) to the geodatabase database; the result table can be opened from within e.g. Microsoft Access or ESRI ArcGIS.

To open the node results using Microsoft Access, open the MDB file in Microsoft Access, and open "mw_ResJunction" table. The table contains the junction node results.



esult File: Lesson1b.res ata Type: EPANET Resul		- Calı	culate	Advanced.
 Statistics Name	Min	Max	Current	Accumulated
Link: Starting/Ending HGL				
Link: Friction Factor				
Link: Reaction Rate				
Link: Setting				
Link: Status				
Link: Quality				
Link: Headloss/1000				
 Link: Velocity				
 Link: Flow				
Node: Quality				
Node: Pressure				
Node: HGL				
Node: Demand			E	

Figure 6.13 The Results Statistics Dialog box

Your analysis results in case of SI \project units, should be similar to those as shown in Figure 6.14.

MUID	Demand_Max	Demand_Min	Grade_Max	Grade_Min	Pressure_Min	Pressure_Max
7	30	30	70.77263	70.77263	38.77262	38.77262
9	-250.8012	-250.8012	30	30	0	(
8	-35	-35	72.86816	72.86816	43.86815	43.86815
10	188.8012	188.8012	60	60	0	(
5	12	12	69.09694	69.09694	32.09694	32.09694
6	30	30	61.82301	61.82301	27.82301	27.82301
3	25	25	81.56897	81.56897	52.56897	52.56897
4	0	0	69.69219	69.69219	39.69219	39.69219
1	0	0	96.46494	96.46494	66.46494	66.46494
2	0	0	81.81463	81.81463	50.81463	50.81463

Figure 6.14 The hydraulic analysis results in the Results Table

Component Browser

The Browser allows you to graphically select any network component from the Horizontal Plan window by simply clicking with the Select tool, and will then display that component's input attributes and analysis results. This allows you to quickly examine the pipe network system at the component level (i.e., pipe, junction node, valve, pump, tank, and reservoir), check what is defined for the model, and determine the computed analysis results. For example, selecting a pipe from the Horizontal Plan window will display in the Component Browser the pipe ID, asset ID, description, street name, diame-



ter, material, construction year, length, roughness coefficient, flowrate, velocity, and headloss. Minimum and maximum values can be displayed for extended period simulation results. Figure 6.15 displays the Browser.

Browser			
LESSON1A.RES		<u> </u>	
Primary		∇	
🔽 Minimum	🔽 Maximum		
Pipe ID	8		
Asset ID			
Description	University Ave		
Street Name			
Diameter	8.000000		
Length	4000.000000		
Material			
Year			
Roughness	80.000000		
Pressure Zone			
Check Valve			
Time	1/1/2000	1/1/2000	1/1/2000
Flow	1.564	1.564	1.564
Velocity	4.480	4.480	4.480
Headloss/1000	23.300	23.300	23.300
Quality	0.000	0.000	0.000
Status	3.000	3.000	3.000
Setting	80.000	80.000	80.000
Reaction Rate	0.000	0.000	0.000
Friction Factor	0.050	0.050	0.050
Starting/Ending H	321.982	321.982	321.982

Figure 6.15 The Browser allows you to examine the input attributes and analysis results for any network component

Horizontal Plan Graphical Plots

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

MIKE URBAN WD allows extensive customization of the Horizontal Plan graphical plot. From the Layer Manager (displayed on the left part of the application window), select Properties by simply pressing the right mouse button while the cursor is located on the selected layer.) This will display the Layer Properties dialog box, as shown in Figure 6.16.



neral Label Appearance Graduated Color Flow Arrows	
Show Flow Arrows	
Proportions Width: 8.000000 Units: • Points (pixels) Length: 12.00000 • Map Points	
Color C Use color of link Use this color	
Position C In the middle of link C In the middle of link's section	
In the middle of link's sector In the middle of each segment In the middle of link's central segment	
Showing Mode	
C Always show arrows Do not show arrows if too many arrows in window	
Always show arrows Do not show arrows if too many arrows in window Maximal arrows count: 100 Minimal Flow Value To Draw Arrow Minimal Value: 0.005000	

Figure 6.16 The Layer Properties dialog box allows you to customize the Horizontal Plan graphical plot

From the Layer Properties dialog box, select the Flow Arrows tab. In this section, check on Show Flow Arrows. Note that the Minimal Value data entry field defines the minimum flow quantity for which a flow direction arrow will be displayed. Note that you can use this capability to identify those pipes that carry the majority of pipe flow by setting the threshold value to a value equal to one-half the maximum flowrate in the network.

Selecting «Apply» displays these changes on the displayed Horizontal Plan window. You should see the flow direction arrows displayed on the pipe network in the Horizontal Plan window, as shown in Figure 6.17.



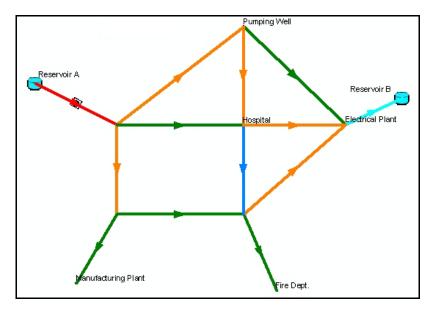


Figure 6.17 Computed flow direction arrows plotted on the pipe network in Horizontal Plan window

Profile Plots

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

Profile plots can have two separate vertical axes to allow plotting of variables from two separate unit families, such as flow and pressure. Profile plots can be plotted along any user-specified route. Profile plots can be plotted with an envelope to show the minimum and maximum values reached during an extended period simulation.

To define a profile plot path, select Model Results| Profile Plot | Define Path Flags. MIKE URBAN WD will then allow you to graphically select from the Horizontal Plan window the profile path to use by setting flags along the path. Simply click on the junction nodes that make up the path that you want the profile plot to display. When finished defining the profile plot path, choose Create Profile Plot. This will display the profile as seen in Figure 6.18.



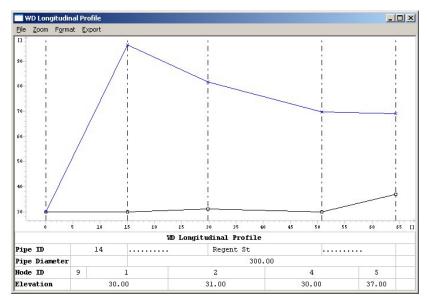


Figure 6.18 Profile plots allow to graphically plot the analysis results along any pipeline path

Further discussion on plotting profile results is provided in the Model Manager User Guide.

Prepared input and Output Files

Completed input and output files were provided for this lesson. These files are:

- 1. LESSON1A.MDB. This files is the input model of the water distribution network. This file is used as the starting file for this lesson.
- LESSON1B.MDB. This files is the output model of the water distribution network.

6.2 Lesson 2 - Pressure reducing valve static analysis

This lesson takes you step-by-step, illustrating how to use MIKE URBAN WD to define a pressure reducing valve (PRV), perform a steady state analysis, and view the analysis results for the defined pipe network system. Also presented is a brief review of the analysis results.

This lesson also simulates the effect of various modifications not related to maintaining the pressure at junction node 2. These modifications include a junction node demand change, peak demand change, and pipe status change. The effect of these modifications will be cumulative. We will save the analysis results after each modification has been applied so that the analysis results can be later viewed and compared to each other.



A schematic diagram for the pipe network to be analyzed in this lesson is shown in Figure 2.1. The pipe network system consists of two reservoirs, 8 junction nodes, 13 pipes, a booster pump, and a pressure reducing valve. Water is distributed from reservoir A to the pipe network system and to the supplementary reservoir B by the pressure from the booster pump. To simplify data input, all pipes, pumps, reservoirs, and junction nodes are numbered in the diagram.

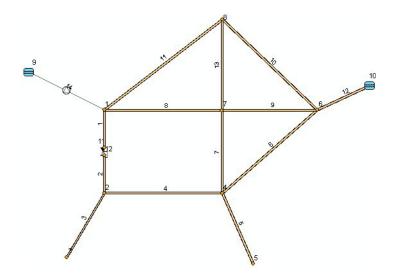


Figure 6.19 A schematic diagram of the pipe network system

In this lesson a pressure reducing valve (PRV) is installed between junction nodes 1 and 2. The PRV is intended to regulate the pressure downstream of the PRV.

To save time with this lesson, we have prepared data files that have already been set up in order for you to quickly follow through the lesson. For a list of all the files in this lesson, see the section titled Prepared Input and Output Files on 5-27.

Begin this lesson by loading LESSON2A.MDB from the LESSONS\LES-SON02 subdirectory. This file contains the pipe network system.

To load the file:



1. Select File | Open. The program will then display the Open dialog box, as shown in Figure 6.20.

Open Project					? ×
Look jn:	🗀 SI		•	- 🗈 📸 🎫	
My Recent Documents	계Lesson2a.mdb				
My Documents					
My Computer					
My Network	File name:	Lesson2a.mdb		Ţ	<u>O</u> pen
Places	Files of type:	MIKE URBAN Database (*.	mdb)		Cancel

Figure 6.20 The Open dialog box

 Select LESSON2A.MDB from the listing of files or type LESSON2A.MDB in the File Name box and then select «Open» to open the selected file. If the filenames for the lessons are not listed, make certain you are in the LESSONS\LESSON02\SI or LESSONS\LESSON02\CFS subdirectory.

6.2.1 Defining a Pressure Reducing Valve

The basic network file (LESSON2A.MDB) comes with a pressure reducing valve (PRV) already defined to reduce the time required to complete this lesson. However, this section explains the steps used to specify a PRV for the existing network system. Therefore, in this lesson you do not have to insert a PRV—it has been done for you. As shown in Figure 5.2.1, this PRV will regulate the pressure at the downstream node 2. The booster pump feeding node 1 may cause the pressure at junction node 2 to be too high. To prevent this, a pressure reducing valve has been installed between nodes 1 and 2.

A pressure reducing valve is used to regulate the pressure at the downstream node of the pipe it is installed. With a PRV, the pressure at the downstream node will not be higher than a constant specified pressure (the pressure setting of the PRV). For a more detailed discussion of PRVs, see the Water Distribution User Guide.



Inserting a Pressure Reducing Valve

When you want to insert a valve into an existing pipe, you must insert it as a new component. This can be easily done within the Horizontal Plan window using the Add Valve tool from the Components floating toolbar. To insert a valve:

- 1. Open the Horizontal Plan window by selecting View | Horizontal View.
- 2. Split the pipe by using Split tool from the Components toolbar.
- 3. Split the pipe once more to create a sections for 3 pipes.
- 4. Select the middle section by Select Feature tool and delete it by using the Delete Selected tool. It is also possible to press key.
- Select the PRV's valve tool from the Components floating toolbar and click on the beginning pipe node in the Horizontal Plan window where you want to place the valve and double click on the ending node. This will create a new PRV valve.

Note

When inserting a new valve, a junction node will be automatically inserted before and after the valve in case that there are now existing nodes within the snap node tolerance radius. In order to see these inserted junction nodes, you may have to zoom in and stretch the distance between the new junction nodes since the nodes might be initially seen as overlapping each other. To zoom in and stretch the distance:

- 1. Choose the Zoom tool from the Command toolbar. Then, from within the Horizontal Plan window, click and drag a zoom window around the valve. You should now see the two junction nodes.
- 2. Choose the Select tool from the Components floating toolbar. Then, click and drag the two junction nodes on either side of the valve further apart. If the junction nodes cannot be stretched further apart, then the network is locked against geometry changes. If this is the case, select Edit | Project Lock to unlock the project. The project lock is used to prevent unintentional moving of network components.
- 3. After stretching the nodes apart, return to your previous view by choosing the Zoom Previous tool in the Command toolbar and then click in the Horizontal Plan window.

Inserting a Valve Between Two Junction Nodes

You can graphically add a valve between two existing junction nodes. To do this, select the Valve tool (PRVs, PBVs, TCVs, PBVs, FCVs, GPVs,) from the Components floating toolbar. Then, select the beginning and ending junction nodes. MIKE URBAN WD will then insert a valve between these two nodes.



Defining PRV Properties

The properties of a PRV include valve diameter, the pressure head setting, and minor loss coefficient. Figure 5.2.1 shows the pressure reducing valve that was inserted.

Once the PRV has been inserted into the pipe network, the next step is to define the properties of the PRV. This is done using the Valve Editor, as shown in Figure 2.3. To display the Valve Editor, select EPANET| Valves.

	Valves							_ O ×
1	-Identification	& connectivity	-	5				
	Asset ID:			Data source:				nsert
	Valve ID:	16		Status:	<nu< th=""><th>LL> 💌</th><th><u> </u></th><th>elete</th></nu<>	LL> 💌	<u> </u>	elete
	From node:	17		🖹			Adv	anced
	To node:	18						Close
	Description:	PRV	Valve	Pressure zone	ID:			
1	- Model data -							
	Valve type:	PRV	•	Valve state:	<nu< th=""><th>LL> 💌</th><th></th><th></th></nu<>	LL> 💌		
	Settings:	30			n c	losed:		
	Valve curve:				_ ₩a	ter hammer —	a 2	
	Diameter: [m	m] 300				Settings		
	Loss coeffici	ent: 0.6						
231		-						
	Valve ID	AssetID	Data source		Diameter	Loss coeffi		Setting
•	16	<null></null>	<null></null>	PRV Valve	300	0.6	<null></null>	
	1							
4								<u> </u>

Figure 6.21 The Valve Editor dialog box

In this lesson, we have set the valve as a pressure reducing valve in the Valve Type frame, defined the valve diameter as 12 inches (300mm), set the pressure setting to 46 psi (30m), and set the minor loss coefficient to 0.600. Select «Close» to store these values and close the Valve Editor dialog box.

When you are finished defining the PRV, save the completed network with a PRV as LESSON2B.MDB by using File | Save As. In this lesson we will save each modification to the pipe network system as a separate file so that they can be analyzed, reviewed, and compared later in this lesson.

6.2.2 Defining a Junction Node Demand Change

In this section, we will use a junction node demand change to simulate a marked increase in demand at a particular junction node. Junction node 3 of the pipe network system shown in Figure 2.1 represents a large manufacturing plant. On a particular day of the month a large portion of the manufacturing equipment is flushed and cleaned. Obviously this will require more water than usual. To simulate this unusually high demand and its effects on the rest of the pipe network system, a junction node demand change can be applied.

Junction data	Emitter data	Air-valve data						
- Identificati	on & connectivi	tv					1	-6
Asset ID:			Data s	ource:			<u>Insert</u>	
Junction I	D:	3	Status:		, <null></null>	-	<u>D</u> elete	
X coordina	ate:	6.403	_		1		Advanced.	
Y coordina	ate:	4.000	_				Close	-
Description		Manufacturing	Plan Pressu	re zone:				_
Model data Type:	oefficient:	Junction	State:	on: [m]	unmarked 29.000	•		
Minimal pre	essure: [m]			e elevation: (m				
Minimal pre	essure: [m]	Category	Surface					
Minimal pre	essure: [m]	Category						
Minimal pre	essure: [m]		Surface Pattern*					
Minimal pre	essure: [m]		Surface Pattern* <null></null>			PZonelD	Statello	De
Minimal pro	essure: [m]	<null></null>	Surface Pattern* <null></null>	Elev	Surface Elev	PZonelD ⊲Null>	Statello unmarked	
Minimal pro	essure: [m]	<null> Data source</null>	Surface Pattern* <null> Description</null>	Elev 30	Surface Elev			≺Nu
Minimal pre	essure: [m]	<null> Data source</null>	Pattern* <pre> Pattern* </pre> <pre> Pattern* </pre>	Elev 30 30	Surface Elev	<null></null>	unmarked	<nu <nu< td=""></nu<></nu
Minimal pre	essure: [m]	<null> Data source <null> <null></null></null></null>	Pattern* <null> Description <null></null></null>	Elev 30 30	Surface Elev <null> <null></null></null>	<null> <null></null></null>	unmarked unmarked	<nu <nu <nu< td=""></nu<></nu </nu
Minimal pre	essure: [m]	<null> Data source <null> <null> <null> <null> <null></null></null></null></null></null></null>	Pattern* <null> Description <null> <null></null></null></null>	Elev 30 30 30 31	Surface Elev <null> <null> <null> <null></null></null></null></null>	<null> <null> <null></null></null></null>	unmarked unmarked unmarked	≺Nu ≺Nu ≺Nu

Figure 6.22 The Junction Editor dialog box

To simulate the increased demand of the flushing, select Edit | Junction Editor and select junction node 3, as shown in Figure 6.22, and change the demand to 1.05 cfs (30l/s). Select «Close» to apply the change to the network and close the Junction Editor dialog box.

When you are finished defining the junction node demand change, save the completed network with a PRV and junction node demand change as LES-SON2B.MDB by using File | Save As. As explained earlier, we will save each modification to the pipe network system as a separate file so that they can be analyzed, reviewed, and compared later in this lesson.

6.2.3 Defining a Global Demand Change

In this section, we will define a peak demand on the network system to simulate a severe demand condition. This is done by applying a global water demand change. This global demand change is used to simulate a peak-hour demand. A steady state analysis of the network at peak-hour conditions can be used to verify the efficiency of the pipe network and identify ineffective areas based on the PRV that was previously installed. A global demand factor of 1.5 is the multiplication factor applied to the demands at all junction nodes to simulate this severe demand condition.

To change global demands at all junction nodes by a factor of 1.5:



- 1. Select EPANET | Multiple Demands Editor to open the Multiple DemandEditor dialog box.
- Choose «Demand» column in the Multiple Demand Editor dialog and select Field Calculator from the right mouse click pop-up menu to display the Field Calculator dialog box, as shown in Figure 6.24. Note that you can select only such records, where you want to apply the global demand change, Figure 6.23.

F	Multiple der Junction ID: Demand: Demand coe	7	0.000	Descripti Demand Demand	category:			Insert Delete
F	Junction ID*		Description <null></null>	Demand cat <null></null>	Demand pat <null></null>	Demand co <null></null>	í.	Close
F	5 6 7	45	<null> <null> <null></null></null></null>	<null> <null> <null></null></null></null>	<null> <null> <null></null></null></null>	<null> <null> <null></null></null></null>		
E	8	1	<null></null>	<null></null>	<null></null>	<null></null>		

Figure 6.23 Select records, where you want to apply the global demand change

- 3. In the Field Calculator dialog, use the wizard to define the following statement: Demand = Demand * 1.5.
- Select «OK» to apply the defined global change and close the Global dialog box. If you do not want to apply the defined global change to the network system, choose «Cancel».



Eields:	Туре:	Functions:
Category Demand Description gF JunctionID OBJECTID PatternID smF Sqn	⊂ <u>N</u> umber ⊂ String ⊂ <u>D</u> ate	Abs() Atn() Cos() Exp() Fix() Int() Log() Sin() Sin() * / &
Demand =	□ <u>A</u> dvanced	+ - =
[Demand] *1.5		<u>L</u> oad
		<u>S</u> ave
		Help
		OK
		Cancel

Figure 6.24 The Field Calculator dialog box

When you are finished defining the global junction node demand change, save the completed network with as LESSON2C.MDB by using File | Save As.

Field Calculator

Using the Field Calculator, a simple SQL statements can be quickly constructed. Statements can be saved and loaded using Save and Load from the Field Calculator dialog.

However, for complex SQL commands, a SQL statement must be entered using ArcMap or Microsoft Access.

6.2.4 Defining a Pipe Status Change

In a pipe network system, a pipe can be either open or closed. In MIKE URBAN WD, a pipe status change allows the user to simulate the pipe network system under either condition. A pipe status change is only applicable to pipes that do not have check valves, as check valves already control the pipe's open and closed status.

This lesson will use a pipe status change to simulate the effect of closing one pipe in the network system. Pipe 7 in the network system is the pipe that runs under Park Street. Due to road construction on Park Street, pipe 7 is closed. An analysis of the network in this condition will show the effect of closing pipe 7 on the network system.

	Pipes							_ 🗆 ×
ſ	-Identification	& connectivity-						. 1
	Asset ID:			_ Data sour	ce:			Insert
				Status:		<null></null>		Delete
						JUNULLY	-	Advanced
	From node: 4			N				
	To node:	7		📐				Close
	Description:	Park	St	Pressure a	one ID:			
	-Geometrical p	properties						
	Length: [m]	1050	.00 14.696	Diameter:	[mm]	300.000 0		
L L	Hydraulics & f	friction losses —						
	Material:			- Formulatio	in:	Darcy-Weisbac	h	
	Construction	vear:		Roughness: [mm] 1.50		1.500		
	Wall thickne	ee: [mm]		Loss coefficient:				
				Pressure normal: [m]				
	Wave speed	i: (m/sj		Pressure r	normai: (mj	1	- 5	
-	Miscellaneou	s						
	Demand coe	eff. 1: 1.000)		unmark	ed	-	
	Demand coe			Check	1	Closed		
			· .		e ranno.	1. 0.0004		
	Street name:							
			00-4-	Constructio	1	In the second	Description	Description
-	Pipe ID 13	Description Park St	CDate <null></null>	<null></null>	Asset ID <null></null>	Data source <null></null>	Demand co 1	Demand 🔺
_	2	Randall St	<null></null>	<null></null>	<null></null>	<null></null>	1	
	2	Randall St	<null></null>	<null></null>	<null></null>	<null></null>	1	
F	3	Randall St	<null></null>	<null></null>	<null></null>	<null></null>	1	
	4	Regent St	<null></null>	<null></null>	<null></null>	<null></null>	1	
	5	Park St	<null></null>	<null></null>	<null></null>	<null></null>	1	
	6	Washinghton	<null></null>	<null></null>	<null></null>	<null></null>	1	
•	7	Park St	<null></null>	<null></null>	<null></null>	<null></null>	1	
	8	University Av	2.5 C	<null></null>	<null></null>	<null></null>	1	-
4	ĥ	hare a s	A1.0	L & L & U		A. U.		•

Figure 6.25 The Pipe Editor dialog box

To change the pipe status, select EPANET | Pipe Editor. Choose pipe 7 and close the valve by selecting the Closed check box in the Pipe Status frame. Select «Close» to apply the changes to the network and close the Pipe Editor dialog box.



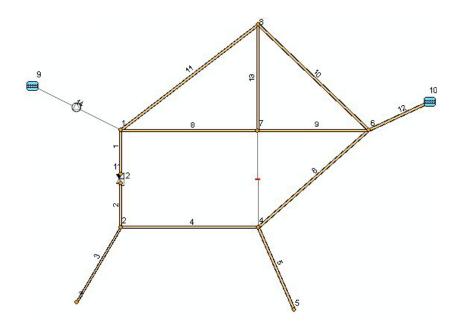


Figure 6.26 Schematic diagram with the closed pipe

When you are finished defining the pipe status change, save the completed network with a PRV, junction node demand change, peak demand change, and a pipe status change as LESSON2D.GDB by using File | Save As. Once again, we will save each modification to the pipe network system as a separate file so that they can be analyzed, reviewed, and compared later in this lesson.

6.2.5 Prepared input and Output Files

Completed input and output files were provided for this lesson. These files are:

- 1. LESSSON2A.MDB, LESSON2A.RES. These files are the input and output files with the basic network components and the PRV defined. These files are used as the starting files for this lesson.
- 2. LESSON2B.MDB, LESSON2B.RES. These files are the input and output files with the PRV and a junction node demand change defined.
- 3. LESSON2C.MDB, LESSON2C.RES. These files are the input and output files with the PRV, junction node demand change, and peak demand change defined.
- 4. LESSON2D.MDB, LESSON2D.RES. These files are the input and output files with the PRV, junction node demand change, peak demand change, and pipe status change defined.

These files can be found in the LESSONS\LESSON02 subdirectory and can be used to perform the analysis and view the analysis results, without having to interactively enter the data for this lesson.

To analyze the pipe network models you have defined, refer to the section titled Performing an Analysis in Lesson 1.

6.2.6 Reviewing the Analysis Results

The analysis results from the various modifications of the network system can be viewed individually or can be compared with each other. To view the analysis results individually, see the section titled Viewing the Analysis Results in Lesson 1. Also, note that there are additional ways to display the analysis results, such as the Component Browser window, Profile Plot window, and Map window. These are also discussed in Lesson 1.

In this lesson, a pressure reducing valve is used to regulate the pressure at junction node 2 (LESSON2A). The pressure at all the junction nodes are between 38 psi (25 m) and 117 (80 m) psi (within a reasonable operating range). The high pressure at node 1 (117 psi, 80 m) is caused by the pressure from the booster pump and is the first node that water from reservoir 1 is sent to. The pressure at junction node 2 (the junction node regulated by the PRV) is 45 psi (29.8 m). Thus, the PRV effectively reduced the pressure at node 2 to 45 psi (app.30m).

The first cumulative effect applied was a junction node demand change (LES-SON2B). The demand at node 3 was changed to model increased water usage at the manufacturing plant. The demand at node 3 is increased to 1.05 cfs (30 l/s).

The peak demand change was the second cumulative effect applied to the network (LESSON2C). Here a global demand factor of 1.5 is used to model the peak-hour demand situation. The demand at each junction node in the system was increased by this factor. For example, the demand at junction node 6 was raised from 1.0 cfs (30 l/s) to 1.5 cfs (45 l/s). This had a slight effect of lowering the pressure at this node from 40.85 psi (27 m) to 40.10 psi (26.60 m).

The last cumulative effect applied is a pipe status change. Pipe 7 is closed to model road construction on Park Street (LESSON2D). From the analysis results it can be seen that pipe 7 has no flow going through it, indicating that pipe 7 is closed.

6.2.7 Comparing the Analysis Results

The Compare Alternatives feature can be used to compare analysis results for two or more simulations. This feature is used to compare the analysis result files by computing the difference for every component of the network



from two analysis result files for a parameter modification to the same network. In this case, we will be comparing LESSON2B.RES to LES-SON2C.RES to see how a peak demand change effects the pipe network.

In order to use the Compare Alternatives feature, we have to have two analysis result files. MIKE URBAN WD will subtract the two analysis results files from each other. Note that it is only possible to subtract two analysis result files if the number of nodes, pipes, and time-steps (if performing an extended period simulation) are the same. In this lesson we will compare the pipe network without the global demand change defined (LESSON2B.RES) and the pipe network with a global demand change defined (LESSON2C.RES). Comparison of the other network modifications is left to the reader.

To compare these two analysis result files:

- 1. Select Model Results Result Comparison | EPANET Results. The File Open dialog box will appear; in this dialog box the first result file is selected. Select «OK» when finished.
- 2. Another File Open dialog box will appear; in this dialog box the second result file is selected. Select «OK» when finished.

The results from LESSON2C.RES will be subtracted from LESSON2B.RES. For more details on comparing alternatives, including other comparisons that can be performed, see the section titled Displaying and Outputting Analysis Results on -205.

Viewing the Comparison Results

To view the results after running the Compare Alternatives feature, use standard ways of the results post-processing such as Results Browser window, Profile Plot window, and Map window.

Since the results for LESSON2C.RES were subtracted from LES-SON2B.RES, a positive value in the Analysis Results Table indicates that the value in LESSON2B.RES is higher than in LESSON2C.RES. A negative value indicates that the value in LESSON2B.RES is lower than in LES-SONCB.RES. A zero value indicates that there is no difference.

Compared Results of LESSON2B.RES and LESSON2C.RES

The peak-hour demand also slightly reduces the booster pump head and the pressures at all the network junction nodes. Note that the inflow at junction node 8, the pumping well, is also increased. This means that the peak-hour demand requires an increase in water drawn from the pumping well.

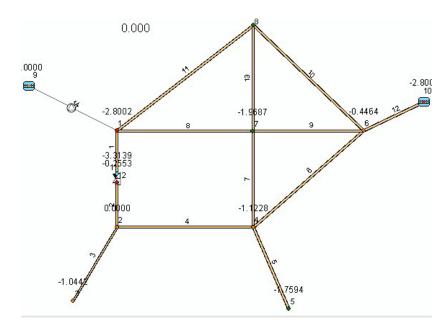


Figure 6.27 The comparison results (pressure differences in this case) are displayed in the Map window

6.3 Lesson 3 - Extended Period Analysis

This lesson takes you step-by-step, illustrating how to use MIKE URBAN WD to define an extended period analysis, extended period control rules (i.e., control of storage tanks and pumps), and perform an extended period analysis for the defined pipe network system. Also presented is a brief review of the analysis results.

A schematic diagram for the pipe network to be analyzed in this lesson is shown in Figure 6.28. The model consists of one reservoir, one node, one tank, 8 junction nodes, 13 pipes, and a booster pump. Reservoir A is the designated pressure source, so water is pumped into the water distribution system from reservoir A. To simplify data input, all pipes, pumps, tank nodes, and junction nodes should be numbered in the schematic diagram, as shown in Figure 6.28. In subsequent lessons, additional components will be added to this schematic diagram.

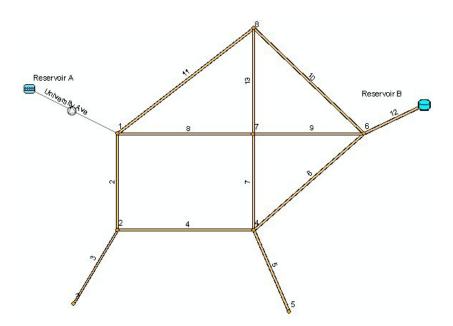


Figure 6.28 A schematic diagram of the pipe network system

To save time with this lesson, we have prepared data files that have already been set up in order for you to quickly follow through the lesson. For a list of all the files in this lesson, see the section titled Prepared Input and Output Files.

Begin this lesson by selecting File | Open and choosing LESSON3.MDB from the LESSONS\LESSON3 subdirectory. This file contains the pipe network system.

6.3.1 Defining an Extended Period Analysis

The basic network file (LESSON3A.MDB) has been defined with the extended period input data already specified to reduce the time required to complete this lesson. However, this section explains the steps used to define an extended period analysis project. The project configuration includes specifying the project type and extended period time parameters. To define the project type, select EPANET | Project Options. The program will then display the Project Options dialog box, as shown in Figure 6.29.



e Project Type Units Head L	.osses Properties Convergence Wa	ter Hammer
Steady state hydraulics Extended period hydraulic Katended period water quality Water hammer	 No water quality analysis Chemical concentrations Water age Source tracing 	Close

Figure 6.29 The Project Options dialog box

From the Project Options dialog box, select the extended period Hydraulics option. Select «OK» to apply this change and close the Project Options dialog box.

An extended period analysis requires that extended period time parameters be defined. To edit the extended period time parameters, select EPANET | Extended Period| Time Settings to display the Time Editor dialog box, as shown in Figure 6.30.

Analysis duration:	24	hrs	_ _	Close
Hydraulic time step:	10	min	•	
Pattern time step:	1.000	hrs	•	
Report time step:	1.000	hrs	-	
Report start time:	0.000	hrs	•	
Quality time step:	0.100	hrs	-	
Start clocktime:	12 ÷ 0	÷ AM	-	
Statistics:	without statistics		-	

Figure 6.30 The Time Editor dialog box

The Analysis Duration, which has been defined as 24 hours, is the overall time the simulation will take. The Hydraulic Time Step specifies how often a new hydraulic computation of the pipe network system is to be computed. Here, it has been defined as 10 minutes. The Pattern Time Step is defined as 1.00 hour. The Pattern Time Step defines and specifies the length of time between each pattern change (i.e., the period of time over which water demands and constituent source strengths remain constant). The Report Time Step specifies the interval of time in which network conditions are reported in the analysis results, and has been defined as 1 hour. The remaining default values will be used for the rest of the parameters. For a more detailed discussion on extended period time parameters, see the Water Distribution User Guide. Choose «OK» to apply these changes and close the Time Editor dialog box.



6.3.2 Defining and Applying a Demand Pattern

Data in an extended period analysis can change during a simulation (e.g., demand, external flow to a storage tank, etc.). Demand changes can be applied globally to all junction nodes or to specific junction nodes.

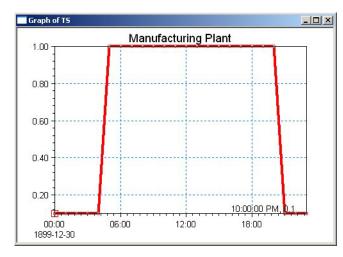
In this section, we will use a demand curve to simulate a simplified diurnal demand curve that corresponds with the data defined in the Time Editor. At each time step, a new demand factor is applied to the original demands defined at the junction nodes. These factors are defined in the Diurnal Profile Editor. To display the Pattern Editor, select EPANET | Extended Period | Repetitive profiles| Diurnal Patterns. The program will then display the Diurnal Profile Editor dialog box, as shown in Figure 6.31.

Ī	l 08:00:00 Diurnal patt	Categ	:00:00 Jory	-	1.000		
	07:00:00		8:00:00		1.000		
	06:00:00		:00:00		1.000		
-1	05:00:00		00:00		1.000		
	04:00:00	05	5:00:00		0.100		
I,	03:00:00	04	:00:00		0.100		
U.	02:00:00	03	3:00:00	1	0.100		
	01:00:00	02	2:00:00	1	0.100		
	00:00:00	01	:00:00		0.100		
1	From	1	Го	M	ultiplier		
			Dis	tribut	B	<u>G</u> raph	
	Description:		Manu	ufactu	ring Plant	Close	
	Category:		-		Industry		Advanced
	Time step;		1.00		hrs 💌	<u>D</u> elete	
	Diurnal patter	n ID:	P1			<u>Insert</u>	

Figure 6.31 The Diurnal Profile dialog box

To define a demand pattern:

- 1. Select «Insert» and enter a Pattern ID of P1.
- Select «Distribute» to fill in the upper grid with the diurnal profile multipliers. Note that this data has already been specified for you to save time. From Figure 6.32 it can be seen that the 24 hour period has been broken up into 1 hour demand pattern time-steps, which was specified in the Time Editor.
- 3. Enter the multipliers for each time step according to Figure 6.31. A graph of the defined demand pattern can be shown by selecting «Graph».





 Select «Close» to save the demand pattern and close the Multipliers dialog box.

This demand diurnal profile curve can now be combined with other diurnal profiles to define a Cyclic Pattern. To define the cyclic pattern:

- 1. Select EPANET | Extended Period| Repetitive Profiles | Cyclic Profiles to open the Cyclic Profile dialog box.
- 2. Create the new cyclic profile and select the already defined diurnal profile. Note that more diurnal profiles can be used to compose the cyclic pattern, each of which can be used in different week days. The first profile defined will be used for the first 24 hours of the simulation, the next profile for the following 24 hours and so forth (see Figure 6.33).

Cyclic profile II): P1	<u>I</u> nsert
Description:	Manufacturing Plan	<u>D</u> elete
Profile composi	ion	Advanced
Diurnal pattern	P1	Close
Calendar:	I	<u><u> </u></u>
Diurnal patt P1	Calendar*	_
Cyclic profil	escription	
P1 1	lanufacturin	

Figure 6.33 A demand diurnal pattern curve



This demand pattern curve can now be defined over the entire network or at any junction node in the network. In this section, we will apply this demand pattern for the entire network by specifying a global change. To define the demand pattern for the entire network:

- 1. Select EPANET | Junction to open the Junction Editor dialog box.
- 2. Select the junction node 3 (Manufacturing Plant) and define the demand pattern P1.

Field Calculator

MIKE URBAN allows you to use the Field Calculator to assign attribute values globally. Open any editor and select the records you want to update. If you don't select any, the calculation will be applied to all records. Right-click the field heading for which you want to make a calculation and click Field Calculator. Use the Fields list and Functions to build a calculation expression. You can also edit the expression in the text area. Optionally, you can type a value to set to the field.

6.3.3 Defining Storage Tank Data

To make certain that the network demand is always met, a storage tank is installed behind the distribution network, as shown in Figure 6.28. As was previously explained, this lesson comes with the storage tank already defined to reduce the time required to complete this lesson. Therefore, in this lesson you do not have to define a storage tank (it has already been done for you). However, this section explains the steps used to define a storage tank for the existing network system. To define a storage tank:

- 1. Use the Map window.
- 2. Select the Add Water Tanks tool from the Editing floating toolbar and click on the position in the Map window where you want to place the tank.

After defining the storage tank and the connecting pipe to the pipe network system, we must define the tank and pipe parameters. To define the tank parameters

- 1. Select EPANET | Tanks to display the Tank Editor dialog box.
- 2. Select tank 10 and enter 180 feet (55 m) for the Base Elevation. In the Type frame, select the Variable tank with the Circular size type option. Enter 80feet (25m) for the Diameter. Enter 30 feet (10 m) for the Maximum Level and 16 feet (5 m) the Initial Level and 3 feet (1 m) for the Minimum Level. When finished, the tank definition should be as shown in Figure 6.34. Select «Close» to store these values and close the Tank Editor dialog box.

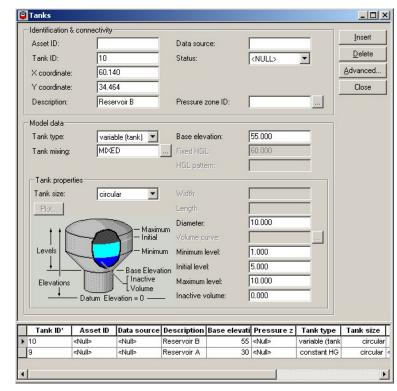


Figure 6.34 The Tank Editor dialog box

6.3.4 Defining Control Rules

Typically during an extended period simulation, the pipes, pumps, and valves contained in a network will change their status (i.e., open or closed) as storage tanks recharge and discharge water and pressures change throughout the network system. Therefore, it is necessary to define extended period control rules to control these systems. Note that this has already been done for you to save time. However, this section explains the steps used to define the extended period control rules for the pipes and pumps in the water distribution network system.

As was explained earlier, the network has one pumping station. In this scenario, we will define the controls which will shut down the pump at 4 p.m. to determine whether the storage tank capacity is sufficient to provide alternative water supply.

To define these controls:

Select EPANET | Extended Period| Simple Control and define the following statement: LINK 14 (Pump) will be closed at clocktime 4.pm, as shown in Figure Figure 6.35.

Control ID:	1	_			Inser
Link type:	Pump	-			<u>D</u> elet
Link ID:	14	N			Advance
Description:	Pump is clos	sed at 4pm.			Close
Setting	Condition		Control node	Control leve	
C Open	C If node	below			
Closed	C If node	above			
C Value	At time		16	Hours	•
	C At cloc	ktime		<null></null>	-
Control ID *	Control lev	Condition	Description	Link ID *	Link type
1	≺Null≻	At Time	Pump is clos	14	Pump

Figure 6.35 The Simple Control dialog box

6.3.5 Performing an Extended Period Analysis

After you have finished defining the pipe network model, you can perform an extended period analysis of the pipe network system. Performing an extended period analysis is exactly the same as a steady state analysis. To perform an extended period analysis, select Simulation | Run Simulation. MIKE URBAN WD will then display a Run Simulation dialog box, from within which it is possible to check the model for errors and run the analysis. Run Simulation dialog box is shown in Figure 6.36.

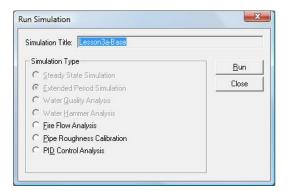


Figure 6.36 The Run Simulation dialog box

Use the Check Data to run a check of the project. MIKE URBAN WD will perform several tests on the pipe network model. If a modeling input error is reported, you will need to correct the input data defining the model.

If no errors were reported, you can select Run to perform an extended period analysis of the pipe network model. If an error is reported during the analysis, it will be necessary to correct the input model to remove the error. However, it is normal for warnings to be reported during the analysis. The user should check the analysis output to make certain that any reported warnings or status messages do not pose a threat to the validity of the analysis results.

Note that the simulation results are automatically loaded after the successful simulation run.

6.3.6 Prepared Input and Output Files

Completed input and output files were provided for this lesson. These files are:

 LESSSON3A.MDB, LESSON3A.RES, LESSON3B.MDB, LESSON 3B.RES. These files are the input and output files with the basic network components, extended period hydraulics, demand pattern, tank, and extended period control rules defined. These files are used as the starting files for this lesson.

These files can be found in the LESSONS\LESSON3 subdirectory and can be used to perform the analysis and view the analysis results, without having to interactively enter the data for this lesson.

6.3.7 Viewing the Extended Period Analysis Results

After the extended period analysis has been successfully performed, you next need to load the extended period analysis results into MIKE URBAN WD before you can view the results. Viewing extended period analysis results is slightly different from viewing steady state analysis results.

The extended period analysis results can be viewed from the Result Browser window, Results Statistics, Time Series Plot, Profile Plot, and Map window. To load the analysis results, select Model Results Load Simulation Results and select Water Distribution type of results.

EPANET Analysis Results

To view the analysis results generated by the EPANET Analysis Engine, click on the Summary button from the simulation dialog. MIKE URBAN WD will display a file viewer, as shown in Figure 5.12, displaying the EPANET analysis results. If there are any warning messages during the analysis, they will be displayed in the EPANET Analysis Summary.

Energy	Usage:					
 Pump	Usage Avg. Factor Effic.	Kw−hr ∕m3	Avg. Kw	Peak Kw	Cost ∕day	
	65.97 75.00					
			Demand Total (Charge: Cost:	0.00 0.00	
Node Re Node	sults at 0:00:00 Deman L⁄:		Pressure m	NON mg/L		
1 2 3 4 5 6 7 8 9 10	0.0 0.0 2.5 0.0 12.0 30.0 30.0	0 87.76 0 87.76 0 71.36 0 70.77 0 62.12	56.76 58.76 41.36 33.77 28.12 40.17	0.00 0.00 0.00 0.00 0.00	Reservoir Tank	
Link Re	sults at 0:00:00	hrs:				
Link	Flo L⁄s	w Velocity s m⁄s	Headloss ⁄1000m	-		
10 11 12 13 2	36.8 116.5 -204.1 114.6 79.2	8 1.17 7 1.65 0 1.62 9 0.91	13.39 15.12 10.10	-		



Further discussion on displaying analysis results is provided in the section titled Viewing the Analysis Results contained on -160.

Results Statistics

To review an extended period analysis results in a tabular format using the Results Statistics, follow these steps:

1. Select Model Results Statistics to display the Results Statistics. Select Calculate to calculate the predefine types of the results statistics. Close the dialog when done and select Model Results | Add Statistics Layer to add the statistics layer to the table of contents. Click on the statistics



layer properties and select Open Attribute Table, as shown in Figure 6.38.

Gra	de Grade_Max	Grade_Min	Grade_Avg	Pressure	Pressure_	Pressure_	Pressure_
	118.3986	62.06111			30.06111	86.39861	
	30	30			0	0	
	118.832	62.18679			33.18679	89.83202	
	65	60			5.000002	9.999999	
	117.5387	61.44838			24.44838	80.53875	S
	118.1032	62.09252			28.09252	84.10319	
	117.9467	62.07094			33.07094	88.9467	
	118.134	62.04363			32.04363	88.13401	
	119.9227	62.11794			32.11794	89.92274	
	118.1924	62.07517			31.07517	87.19236	

Figure 6.38 The Results Statistics

2. In the Statistics Results Table, you are only able to view the results for a single time step at a time or for the min and max values. The results shown in Figure 6.38 are for the start of the extended period simulation. To display the results at a different time step, select Model Results | Animate | Options to display the Time Step dialog box, as shown in Figure 6.39. Alternatively, you can click on the Time Step icon in the Component Browser.

Result Animation P	roperties]	x
File Name	Lesson3b.res	▼ OK]
From Time	00:00:00 12/30/0/	Cancel	
To Time	00:00:00 07/20/36]	
Current Time	00:00:00 12/30/0/]	
Animation Interval	1000 ÷		

- Figure 6.39 Animation The Time Step dialog box allows you to select a different time step in which to display results
- 3. From the Animation Option dialog box, choose day 0 at time 8:00 hours and then select «OK» to display the results at this time step in the Map window, as shown in Figure 6.40.

esult Animation P	roperties		[
File Name	Lesson3b.res	•	ОК
From Time	0 d. 0:00:00]	Cancel
To Time	0 d. 0:00:00	1	
Current Time	0 d. 1:00:00 0 d. 2:00:00 0 d. 3:00:00		
Animation Interval	0 d. 4:00:00 0 d. 5:00:00	1	
	0 d. 6:00:00 0 d. 7:00:00		
	0 d. 8:00:00		
	0 d. 9:00:00		

Figure 6.40 The select time level dialog box

Further discussion on displaying results in the Analysis Results Table is provided in the section titled Analysis Results Table contained on -160.

Results Browser

The Results Browser allows you to graphically select any network component from the Map window by simply clicking with the mouse cursor, and will then display that component's input attributes and analysis results. This allows you to quickly examine the pipe network system at the component level (i.e., pipe, junction node, value, pump, tank, and reservoir), check what is defined for the model, and determine the completed analysis results.

To view the extended period analysis results in the Component Browser:

- 1. Select Model Results | Results Browser and choose any component in the Map window. The analysis results for the chosen component will appear in the Results Browser, as shown in Figure 6.41.
- To change the time step that is displayed in the Component Browser, select the current time level from the Animation floating toolbar or by selecting Model Results | Animation | Options. The results for the selected time step will be displayed in the Results Browser.

Lesson3b.res		-	
EPANET Primary		~	
🔽 Minimum	Maximum		
Property	Value	Minimum	Maximum
Pipe ID	8		
Asset ID			
Description	University Ave		
Street Name			
Diameter	200.000000		
Length	1220.000000		
Material			
Year			
Roughness	1.500000		
Pressure Zone			
Check Valve	0		
Time	0 d. 0:00:00	0 d. 0:00:00	1 d. 0:00:00
Link: Flow	47.800	-3.504	47.800
Link: Velocity	1.522	0.052	1.522
Link: Headloss/10	22.072	0.047	22.072
Link: Quality	0.000	0.000	0.000
Link: Status	3.000	3.000	3.000
Link: Setting	0.005	0.005	0.005
Link: Reaction Rat	0.000	0.000	0.000
Link: Friction Fact	0.037	0.032	0.068
Link: Starting/Endi	99,102	62.118	119.923



Time Series Plot

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

To view the extended period analysis results in a Time Series Plot:

1. Select Time Series from the floating to select the result item, as shown in Figure 6.42.

Result File:	Lesson3b.res	
Data Type:	Link: Flow	

Figure 6.42 The Time Series Result item selection dialog box

2. Click on the network element in the Map window to create the time series plot, as shown in Figure 6.43

	Time	8 [l/sec]	8 [l/sec]
1	0	47.8004455	Flow
2	3600	47.36633300	
3	7200	46.90881347	
4	10800	46.45448303	40
5	14400	10.41200542	
6	18000	10.41316890	30
7	21600	10.41316986	
8	25200	10.41316890	
9	28800	10.41316986	20
10	32400	10.41316890	
11	36000	10.41316986	10
12	39600	10.41316890	
13	43200	10.41316890	
14	46800	10.41316986	0 1
15	50400	10.41316890	
10	£4000	10 41040000	00000



Map Window Thematic Plots

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

To display the extended period analysis results in the Map window:

- 1. Display the Map window.
- Select Model Results | Add Results Layer to specify the type of the result item to be used for the thematic mapping, such as "Link: Flow". Once specified, the new layer "Link: Flow" will be added to the table of content.
- Right click the layer "Link: Flow" to display the Properties dialog, from which it is possible to define the number of color intervals, type of the thematic map, labelling, displaying flow arrows and many other settings. From the Layer Properties dialog box, choose the values that you want to display.



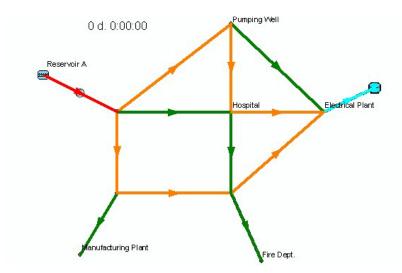


Figure 6.44 Flow arrows displayed for the result layer Link: Flow

6.3.8 Reviewing Extended Period Analysis Results

An extended period analysis is a series of steady state analyses conducted over an extended time period. Each individual analysis is referred to as a time step. MIKE URBAN WD assumes constant pipe flowrates and no change in external conditions between time steps. Thus, the results computed at each time step are based upon the previous time step's computed results and the changes applied.

The diurnal demand curve for the junction node 3 (Manufacturing Plant) in this lesson was defined using the Diurnal Profile editor. The calculated demand curve in begins with a demand factor of 0.1 at 0 hours and ends again with a demand factor of 0.1 at 24 hours. The peak network demand is specified from 5a.m. (5 hours) till 7pm (19 hours) when the demand in is increased by a factor of 1.0.

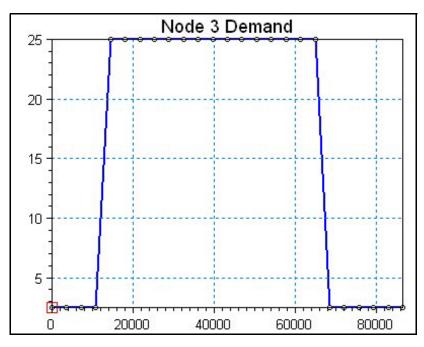


Figure 6.45 The calculated node demand for the node 3 (Pattern P1)

The storage tank that is located at the other side of the distribution zone is initially partly filled and is open. As shown in Figure 6.46, after 4 hours the tank is completely full and it is closed. Note that the graph in this figure changes linearly from a value at hour 0, to a another value at hour 1. This is caused by performing a network simulation a 1 hour report time step. A shorter report time step would allow us to see more detailed change in the graph as the hydraulic time step is 10 minutes. At 16 hours, the tank begins to drain in order to supply the network demand as the pumping station is closed, as shown in Figure 6.46.

Although as illustrated in this lesson, a storage tank can become closed due to overfilling or overdrawing. When a tank is closed, it will be reopened when water begins flowing into an empty tank or flowing out of a full tank 4 p.m. (16 hours) and stays closed for the rest of the 24 hours simulation.

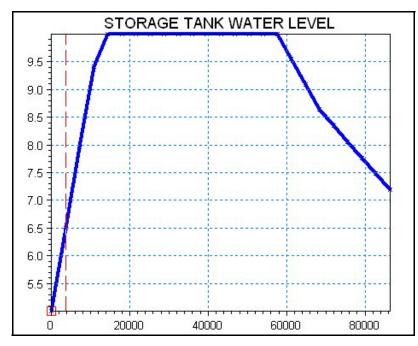


Figure 6.46 Extended period storage tank level results

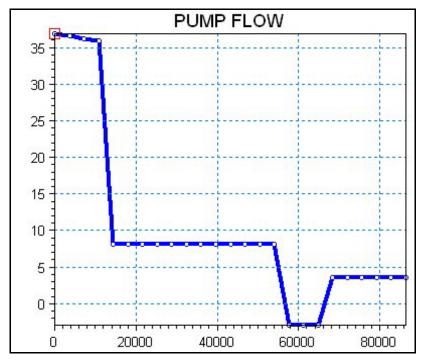


Figure 6.47 Extended period flow results for pump 14



6.4 Lesson 4 - Fire Flow Analysis

This lesson takes you step-by-step, illustrating how to use MIKE URBAN WD to define and perform a fire flow steady state analysis for the defined pipe network system. Also presented is a brief review of the analysis results.

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

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

In this lesson, we will model a fire flow using both of these methods.

To save time with this lesson, we have prepared data files that have already been set up in order for you to quickly follow through the lesson. For a list of all the files in this lesson, see the section titled Prepared Input and Output Files.

6.4.1 Specifying a Design Fire Flow Rate

Specifying a design fire flow rate is the easiest method for simulating a fire flow. Using this method, we will determine the fire flow pressure at junction node 5 (Fire Department) that provides the required fire flow.

A schematic diagram for the pipe network to be analyzed in this lesson, is shown in Figure 6.48. The pipe network system consists of 2 reservoirs, 8 junction nodes, 12 pipes, and a booster pump.

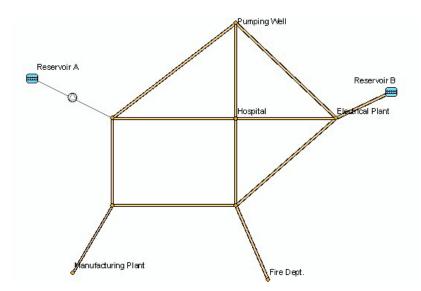


Figure 6.48 A schematic diagram of the pipe network system used to define a fire flow analysis

Begin this section by selecting File | Open and selecting LESSON4A.MDB from the LESSONS\LESSON4 subdirectory. This file contains the network system already defined to reduce the time required to complete this section.

Junction node 5 (Fire Hydrant at the Fire Department) is the junction node that the constraint flow will be defined at since it is the farthest junction node in the pipe network system from the water source, which therefore will have the greatest head loss and thus is considered a critical location. The design fire flow rate method involves finding a pressure at junction node 5 which provides the required fire flow rate. To compute the available fire flow pressure:

- 1. Start editing by selecting Edit | Start Editing.
- 2. Select the junction node 5 using the Select tool.
- 3. Select EPANET| WD Tools| Fire Flow Analysis to display the Fire Flow Settings dialog. Select Calculate Available Pressure for Design Flow Set and set the design fire flow to 1.25 cfs (35 l/s). This is your new constraint flowrate. Select "Use only selected junction nodes" to run the fire flow analysis for the selected node.
- 4. Select «Close» to close the Fire Flow Settings dialog.
- Select Simulation | Run Simulation to display the Run Simulation dialog. Check on "Fire flow analysis" and select «Run» to run the fire flow analysis.
- Select «Close» when the fire flow analysis is completed to run the fire flow results. Note that you can load the fire flow analysis results at any time by selecting Model Results | Load Simulation Results | Fire flow results.



 Select Model Results | Results Browser to display the Results Browser window and click the junction node 5 to view the simulated results. It can be seen that a demand of 1.25 cfs (35 l/s) requires a pressure of 33.6 psi (22m) at junction node 5.

Lesson04a.ffr		
EPANET Primary		
🥅 Minimum	🔲 Maximum	
Property	Value	
Junction ID	5	
Asset ID		
Description	Fire Dept.	
Elevation	37.000000	
Estate height		
Pressure Zone		
Time	0.000	
Fire Flow Pressur	21.936	
Fire Flow	35.000	
Status	0.000	

Figure 6.49 Results browser is used to view the simulated fire flow results

Determining the fire flow pressure that provides a required fire flow rate can either be done for the selected junction node, such as junction node 5 or for more of selected junction nodes or the whole network respectively.

- 1. Repeat the previous procedure but select Use All Junction Nodes in the Fire Flow Analysis dialog. Select «Calculate» to run the fire flow analysis for each node in the network.
- 2. From the analysis results, corresponding pressure can be determined for each junction node. In order to view the fire flow analysis results in the horizontal plan view, color nodes by fire flow pressure and adjust the corresponding color legend. The fire flow pressure is retrieved from the separate fire flow analysis for each node. Therefore, other results than fire flow pressure are not available for nodes and pipes.

When you are finished computing the fire flow pressure, save the completed network as LESSON4A.MDB by using File | Save As so that the results can be analyzed and reviewed later in this lesson.

Tabular results

The simulated fire flow results are reported to the *.FFA text file. Open this text file using any file processing program such as Windows Notepad to see the detailed list of the fire flow results for each simulated junction node.

6.4.2 Specifying a Design Fire Flow Pressure

Another method for modeling a fire flow is to specify a design fire flow pressure and to determine the maximum flow rate at junction node 5 to maintain a minimum residual pressure at that node. This maximum flow rate then corresponds to the fire flow.

Begin this section by selecting File | Open and selecting LESSON4A.MDB from the LESSONS\LESSON4 subdirectory. This file contains the network system already defined to reduce the time required to complete this section.

The design fire flow pressure method involves finding a fire flow rate at junction node 5 which is available under the required fire flow pressure. To compute the available fire flow rate:

- Select EPANET| WD Tools | Fire Flow Analysis to display the Fire Flow Analysis Editor. Select junction node 5. Select Calculate Available Flow for Design Pressure Set and set the design fire flow pressure to 20 psi (15m). This is your new constraint pressure. Select «Close» to close the fire flow analysis dialog.
- 2. Select Simulation | Run Simulation to run the fire flow analysis for the selected node.
- 3. Display the analysis results for junction node 5 in the Results Browser. It can be seen that a maximum available flow of 2.9 cfs (51.1/s) maintains the minimum residual pressure of 20 psi (15 m) at junction node 5.

Determining the fire flow pressure that provides a required fire flow rate can either be done for the selected junction node, such as junction node 5 or for more of selected junction nodes or the whole network respectively.

- 1. Repeat the previous procedure but select Use All Junction Nodes in the Fire Flow Analysis dialog. Select «Calculate» to run the fire flow analysis for each node in he network.
- 2. From the analysis results, corresponding pressure can be determined for each junction node. In order to view the fire flow analysis results in the horizontal plan view, color nodes by demand and adjust the corresponding color legend. The fire flow rate is retrieved from its separate fire flow analysis for each node. Therefore, other results than fire flow pressure are not available for nodes and pipes.

When you are finished computing the fire flow pressure, save the completed network as LESSON6B.GDB by using File | Save As so that the results can be analyzed and reviewed later in this lesson.

Prepared Input and Output Files

Completed input and output files were provided for this lesson. These files are:

 LESSSON4A.MDB, LESSON4A.RES. These files are the input and output files for computing the available fire flow pressure. These files illustrate how to model a fire flow using a design fire flow rate.



• LESSSON4B.MDB, LESSON4B.RES. These files are the input and output files for computing the available fire flow rate. These files illustrate how to model a fire flow using a design fire flow pressure.

These files can be found in the LESSONS\LESSON4 subdirectory and can be used to perform the analysis and view the analysis results, without having to interactively enter the data for this lesson.

6.4.3 Reviewing the Analysis Results

In this lesson, a fire flow was simulated at junction node 5 by using two different methods—defining a design fire flow rate flow and defining a design fire flow residual pressure. In this lesson, the fire flow was simulated at the same junction node so that the results from the two methods can be compared to each other.

Using the first method (computing an available pressure for the design fire flow rate), it was found that a demand of 1.25 cfs (35 l/s) requires a residual pressure of 33.6 psi (22m).

Using the second method (computing a available flow for the design fire flow pressure), it was found that a maximum demand of 2.9 cfs (51.1 l/s) requires a residual pressure of 20.0 psi (15 m).

It can be seen that the results from both methods of computing fire flow bring consistent results. The method that is most suitable to use depends on the circumstances of the situation.

Determining the fire flow rate satisfying the minimum residual pressure calculates the maximum available flow. However, the actual fire flow rate which can be withdrawn from the network depends on other important parameters, such as the fire hydrant size. To simulate this, it is possible to define the fire hydrant connecting pipe size, its length and the friction losses. More realistic results are achieved by this approach and can be compared to the fire flow tests.

It is also possible to compute a Discharge versus Pressure curve for fire flow at junction node 5.

Although not illustrated in this lesson, an extended period simulation of the pipe network system can also be performed while computing a fire flow analysis.

6.5 Lesson 5 - Water Quality–Source Tracing Analysis

This lesson takes you step-by-step, illustrating how to use MIKE URBAN WD to define and perform a source tracing water quality analysis for the defined pipe network system. Also presented is a brief review of the analysis results.

Source tracing is a method of tracking water in a pipe network system. Water is tracked from a single, selected source node (i.e., junction node, tank, or reservoir) and traced throughout the entire pipe network system. The analysis results from the source trace analysis are shown in percentages at each node in the network, showing the amount of water from the selected source node, in comparison to all the other potential sources of water into the pipe network system. This method is especially useful for a water network distribution system in which there are more than one source supplying a demand area and an analysis is required to determine the distribution of the flow from those sources.

A schematic diagram for the pipe network to be analyzed in this lesson is shown in Figure 7.1. The pipe network system consists of 1 tank, 587 junction nodes, and 607 pipes. The total network demand is 3.5 cfs (100 l/s).

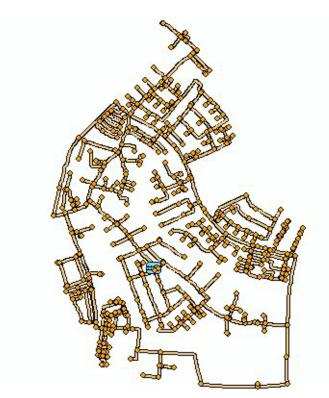


Figure 6.50 A schematic diagram of the pipe network system

To save time with this lesson, we have prepared data files that have already been set up in order for you to quickly follow through the lesson. For a list of all the files in this lesson, see the section titled Prepared Input and Output Files.



Begin this lesson by selecting File | Open and choose LESSON5A.MDB from the LESSONS\LESSON05 subdirectory. This file contains the pipe network system.

6.5.1 Defining a Source Tracing Analysis

The network file (LESSON5A.MDB) has been defined with all the water quality input data already specified to reduce the time required to complete this lesson. However, this section explains the steps used to define the water quality source tracing data input.

Before a source tracing analysis can be performed, the project type must be properly defined. To define the project type, select EPANET | Project Options to display the Project Options dialog box, as shown in Figure 6.51. Select the Extended Period Water Quality and Source Tracing options. When finished, the option settings should appear as shown in Figure 6.51. Select «Close» to apply these changes and close the Project Options dialog box.

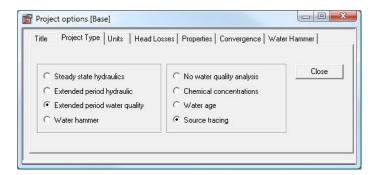


Figure 6.51 Project Options dialog box

Since this project is an extended period analysis, the extended period time parameters will have to be defined. To define the extended period time parameters, select EPANET | Extended Period | Time Settings to display the Time Settings dialog box, as shown inFigure 6.52. Define the extended period time parameters as in Figure 6.52. Select «Close» to apply these changes and close the Time Settings dialog box.



Analysis duration:	1.000	days	-	Close
Hydraulic time step:	15.000	min	•	
Pattern time step:	1.000	hrs	•	
Report time step:	1.000	hrs	•	
Report start time:	0.000	hrs	-	
🔽 Quality time step:	0.100	hrs	-	
Start clocktime:	12 ÷ : 0	÷ AM	•	
Statistics:	without statistics		-	



For more details on extended period time parameters, see the Water Distribution User Guide.

6.5.2 Defining the Source Node

The next step is to define the source node (i.e., junction, or tank) in which the water originates from so that it can be traced. To define the source node:

1. Select EPANET | Water Quality | Trace Node. The Trace Node dialog box, as shown in Figure 6.53, will appear.

Node type:	Tank	<u>A</u> dvanced
lode (D:	2000114	Close



 Specify the node which will act as the source node by selecting «...». The Select Node dialog box will appear. In the Select Node dialog box, select the tank ID=2000114, Select «OK» to close the Select Node dialog box.

Choose «Close» in the Trace Node dialog box to apply the selected trace node to the network model and close the Trace Node dialog box.

When you are finished defining the source node, save the completed network as LESSON5A.MDB by using File | Save or by selecting Edit | Save Edits so the results can be analyzed and reviewed later in this lesson.

6.5.3 Prepared Input and Output Files

Completed input and output files were provided for this lesson. These files are:



• LESSSON5A.MDB, LESSON5A.RES. These files are the input and output files with the water quality source tracing already defined.

These files can be found in the LESSONS\LESSON5 subdirectory and can be used to perform the analysis and view the analysis results, without having to interactively enter the data for this lesson.

6.5.4 Performing Source Node Water Results

To analyze the extended period water quality model you have defined, select Simulation | Run Simulation. This will open the Run Simulation dialog box, as shown in Figure 6.54.

Run Simulation	3
Simulation Title: Lesson5a	
Simulation Type C Steady State Simulation Extended Period Simulation Water Quality Analysis Fire Flow Analysis Fire Flow Analysis PID Control Analysis	Run Close

Figure 6.54 Run Simulation dialog box

Select «Run» to perform the analysis and close the Water Quality Simulation dialog to load the simulation results.

6.5.5 Percentage of Source Node Water Results

After performing the analysis, you are ready to view the analysis results. We will use only one viewing method to reduce the time required for this lesson. For more details on viewing extended period analysis results, see the section titled Viewing the extended period Analysis Results in Lesson 4. In this lesson, we will display the analysis results in the Horizontal Plan window.

To display the percentage of water that was received at each node or a link (average percentage) in the pipe network system from the single selected source node in comparison to all the other potential source (input) nodes:

- Select Model Results | Load Simulation Results | EPANET Results and in the Open dialog box choose LESSON5A.RES. Select «Open» to load the analysis results. Note, that the simulation results are automatically loaded after running the analysis.
- 2. Select Model Results | Add Result Layer to add the results of the water quality analysis in link to the table of contents. Select "Link: Quality", as shown in Figure 6.55.



Result File:	Lesson5a.res	
Data Type:	Node: Quality	

Figure 6.55 Results Selection dialog box

3. Right-click the "Link: Quality" result layer in the table of contents to display the Layer Property options dialog box to set the thematic mapping style for the results layer, as shown in Figure 6.56.

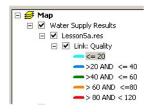


Figure 6.56 Result layer thematic mapping

4. The Map window will display the analysis results for the selected result layer, as shown in Figure 6.57.





Figure 6.57 Analysis results on day 0 at 0:00 hours

The results shown in Figure 6.57 are for the start of the extended period simulation. Note that the percentage of source node water is 0% at all the nodes in the model. To display the results at a different time step, select Model Results | Animate or use the Animation floating toolbar to advance in time or to play the animation, as shown in Figure 6.58.



Figure 6.58 Animation allows you to animate the results in time or to select a different time step in which to display results

5. From the Animation toolbar, use Step forward to advance in time and choose day 0 at time 2:00 hours, and 24:00 hours to display the results at this time step in the Map window. The results for this time step are displayed in Figure 6.59. Observe the percentage of source node water (from Tank A, node 2000114) that has been distributed to the pipes in the pipe network system.





Figure 6.59 Analysis results on day 0 at 2:00 hours





Figure 6.60 Analysis results on day 0 at 24:00 hours

6. Select Create Time Series to display the history of a source trace analysis for the pipe 4610 or any other selected pipe. The results for this pipe are displayed in Figure 6.61. Note that the source trace analysis results provide also a good understanding what are the travel times in the water distribution system.

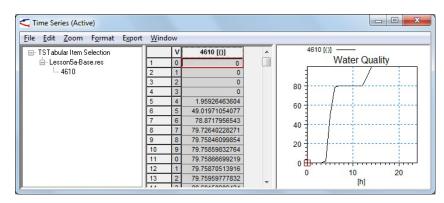


Figure 6.61 Source trace time series for the pipe 4610



6.5.6 Forward and Backward Tracing of Flow

In this section we will demonstrate how to perform forward and backward tracking of flow to and from any selected node. Note that this capability is unique to MIKE URBAN WD and does not require any specialized model setup in order to function. For example, this can be performed with a standard, steady state simulation model.

In this section we will track graphically where the water from reservoir 2000114 goes to in the pipe network system. To track the water through the pipe network system:

- 1. Go to Tools | Tracing | Define Path Flags and put a flag in th the reservoir 2000114
- 2. Go to Tools | Tracing | Forward Flow Tracing
- 3. Select the item from the dialog that comes up

A flow path in the Horizontal Plan window will be displayed, as shown in Figure 6.62, illustrating where the water from reservoir 2000114 goes.



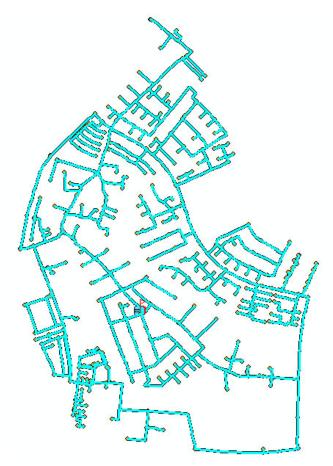


Figure 6.62 Forward tracing of water from reservoir 2000114

Note that this same procedure can be used for backward tracing of water to determine from which sources a node receives water from.

- 1. Select Tools | Tracing | Backward Flow Tracing.
- 2. In the Map window, select junction node 11498.

A flow path in the Horizontal Plan window will be displayed, as shown in Figure 6.63, illustrating from where the water goes to the junction node 11498.

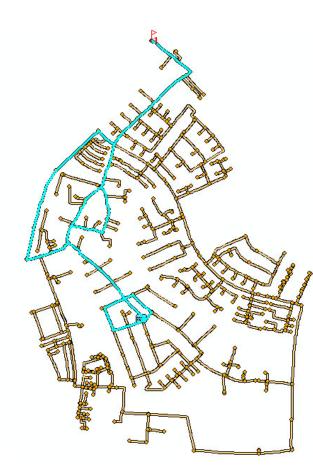


Figure 6.63 Backward tracing of water to junction node 11498

6.5.7 Reviewing the Analysis Results

The initial condition for the source tracing analysis were not defined i.e. water quality 0% is assumed at each node and link of the model network. From the tracing results shown in Figure 6.60 at time 24 hours (day 1, 0:00 hours), it can be seen that the water in the pipe network system has not yet been distributed within the pipe network system. The percentage of water from source node 2000114 for every junction node is not 100% as it had to be as there is only 1 water source for the distribution zone and because each junction node has a positive node demand (consumption).

Note that only one source node can be defined for a source trace water quality simulation. Therefore, if source tracing for more than one source node is required, a source trace water quality model simulation for each source node is required.



6.6 Lesson 6 - Water Quality - Water Age Analysis

This lesson takes you step-by-step, illustrating how to use MIKE URBAN WD to define and perform a water age analysis for the defined pipe network system. Also presented is a brief review of the analysis results.

A schematic diagram for the pipe network to be analyzed in this lesson is shown in Figure 6.64. The pipe network system consists of 1 tank, 587 junction nodes, and 607 pipes. The total network demand is 3.5 cfs (100 l/s).

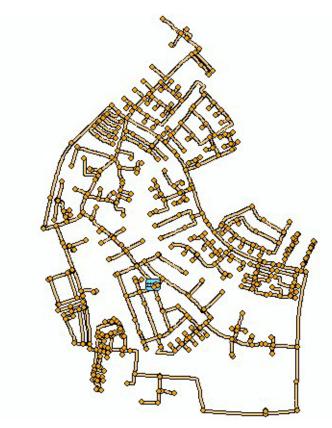


Figure 6.64 A schematic diagram of the pipe network system

To save time with this lesson, we have prepared data files that have already been set up in order for you to quickly follow through the lesson. For a list of all the files in this lesson, see the section titled Prepared Input and Output Files.

Begin this lesson by selecting File | Open and choose LESSON6A.GDB from the LESSONS\LESSON6 subdirectory. This file contains the pipe network system.



6.6.1 Defining a Water Age Analysis

The network file (LESSON6A.MDB) has been defined with all of the water age input data already specified to reduce the time required to complete this lesson. However, this section explains the steps used to define the water age input data.

Before a water age analysis can be performed, the project type must be properly defined. To define the project type, select EPANET | Project Options to display the Project Options dialog box, as shown in Figure 6.65. Select the Extended Period Water Quality and Water Age options. When finished, the option settings should appear as shown in Figure 6.65. The default settings will be used for the remaining parameters. Select «Close» to apply these changes and close the Project Options dialog box.

e Project Type Units Head L	osses Properties Convergence Water Hammer
Steady state hydraulics Extended period hydraulic Extended period water quality Water hammer	No water quality analysis Chemical concentrations Water age Source tracing

Figure 6.65 Project Options dialog box

Since this project is an extended period analysis, the extended period time parameters will have to be defined. To define the extended period time parameters, select EPANET | Extended Period | Time Settings to display the Time Settings dialog box, as shown in Figure 6.66. Define the extended period time parameters as in Figure 6.66. Select «Close» to apply these changes and close the Time Editor dialog box.

Note that the simulation duration is set to 3 days with the report time step of 4 hours as we do not know what would be the maximum water age in most of network junction nodes.



Analysis duration:	3	days	· _	Close
Hydraulic time step:	15	min	•	
Pattern time step:	1.000	hrs	-	
Report time step:	4	hrs	•	
Report start time:	0.000	hrs	•	
🔽 Quality time step:	0.100	hrs	-	
Start clocktime:	12 🕂 : 0	÷ AM	•	
Statistics:	without statistics		-	



For more details on extended period time parameters, see the Water Distribution User Guide.

When you are finished defining the project options and extended period time parameters, save the project as LESSON6A.MDB by using File | Save so the results can be analyzed and reviewed later in this lesson.

Initial Water Quality

The initial water quality conditions allows you to define the initial water quality i.e. at time level 0:00 hrs. at each node. Typically, we do not know what is the initial water age so that we level the initial condition undefined. Water entering the network at the inflow nodes such as storage tanks or reservoirs has the water age of 0:00 hrs. In order to define the entry water age for any of the network nodes, use the Initial Water Quality editor.

6.6.2 Prepared Input and Output Files

Completed input and output files were provided for this lesson. These files are:

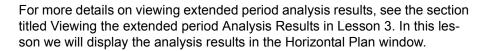
• LESSSON6A.MDB, LESSON6A.RES. These files are the input and output files with the water age input data already defined.

These files can be found in the LESSONS\LESSON6 subdirectory and can be used to perform the analysis and view the analysis results, without having to interactively enter the data for this lesson.

To analyze the extended period water quality model you have defined, select Simulation | Run Simulation. Refer to the section titled Performing an extended period Analysis in Lesson 3 or 5 for more details.

6.6.3 Water Age Results

After performing the analysis, you are ready to view the analysis results. We will use only one viewing method to reduce the time required for this lesson.



To display the water age at each node in the pipe network system:

- 1. Run the water quality simulation to select Model Results | Load Simulation Results|EPANET Results and in the Open dialog box choose LES-SON6A.RES. Select «Open» to load the analysis results.
- 2. Select Model Results | Add Result Layer to add the results of the water quality analysis in link to the table of contents. Select "Link: Quality", as shown in Figure 6.67.

Result File:	Lesson5a.res		
Data Type:	Node: Quality		
□ List	Cancel	Ok	

Figure 6.67 Results Selection dialog box

3. Right-click the "Link: Quality" result layer in the table of contents to display the Layer Property options dialog box to set the thematic mapping style for the results layer, as shown in Figure 6.68.

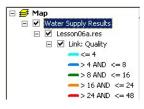


Figure 6.68 Result layer thematic mapping

4. The Map window will display the analysis results for the selected result layer, as shown in Figure 6.69.





Figure 6.69 Analysis results on day 0 at 0:00 hours

The results shown in Figure 6.57 are for the start of the extended period simulation. Note that the water age is 0 hrs. at all the nodes in the model. To display the results at a different time step, select Model Results | Animate or use the Animation floating toolbar to advance in time or to play the animation, as shown in Figure 6.70.



Figure 6.70 Animation allows you to animate the results in time or to select a different time step in which to display results

5. From the Animation toolbar, use Step forward to advance in time and choose day 1 at time 0:00 hours (24:00 hours), and day 1 at time 8 hour (32:00 hours) to display the results at this time step in the Map window. The results for this time step are displayed in Figure 6.71. Observe how the water is aging in the distribution system.

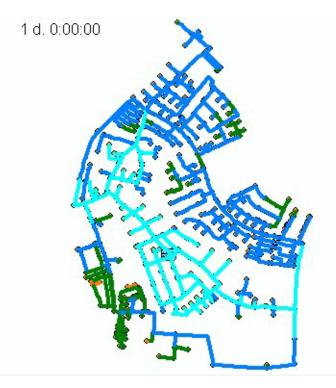


Figure 6.71 Analysis results on day 1 at 0:00 hours (24 hours)

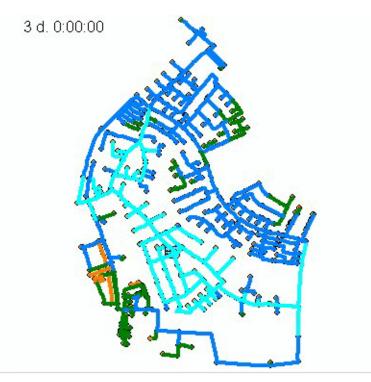


Figure 6.72 Analysis results on day 3 at 0:00 hours (72 hours)

6. Select Create Time Series to display the history of a source trace analysis for the junction node 20987 or any other selected junction node. The results for this pipe are displayed in Figure 6.73. Note that the water age changes in time due to the diurnal profile assigned to each junction node in the model. The average water age at this junction node is 12-19 hours.

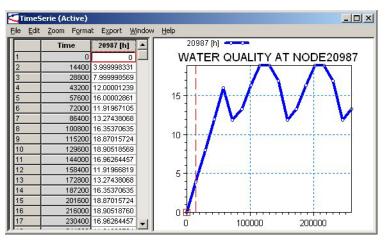


Figure 6.73 Water age time series for the junction node 20987



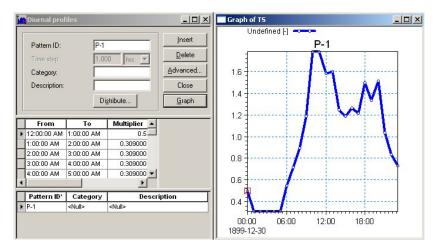


Figure 6.74 Diurnal Profile P-1 assigned to each junction node do model residential demand

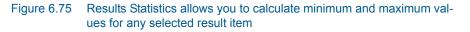
Results Statistics

MIKE URBAN allows you to calculate results statistics for any calculated attribute such as node pressure, link flow, node water quality, and others. The results statistics includes calculation of maximum and minimum values, as well as storing the actual values in the model database for external post-processing using ESRI ArcMap, Microsoft Access and other tools.



1. In order to calculate result statistics, select Model Results | Results Statistics, select which result statistics that you wish to compute and press "Calculate" in the Results Statistics dialog, as shown in Figure 6.75.

	esult File: Lesson6a-Base ata Type: EPANET Result		Cal	culate	Advance Close
1	Statistics Name	Min	Max	Current	Accumulated
	Link: Starting/Ending HGL				
	Link: Friction Factor				
	Link: Reaction Rate				
	Link: Setting				
	Link: Status				—
	Link: Quality				E
	Link: Headloss/1000				
	Link: Velocity				
	Link: Flow				—
	Node: Quality				E
	Node: Pressure				
	Node: HGL				Г
	Node: Demand		Г		F



2. Select Model Results | Add Statistics Layer to add the maximum water age at junction nodes to the layer of contents, as shown in Figure 6.76.

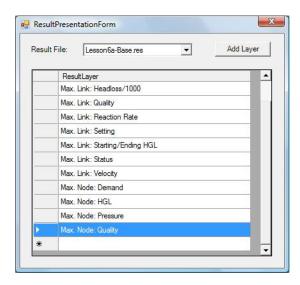


Figure 6.76 Add Statistic Layer

The maximum water age at each junction node will then be displayed in the Map window as a separate layer, as shown in Figure 6.77.



Figure 6.77 Maximum water age at each junction node



6.6.4 Reviewing the Analysis Results

Water age is the time that water has been in the system, reported at each junction node. This includes the time it takes the water to travel from the source to each junction node.

From the water age results shown in Figure 6.69 at time step day 0, 0:00 hours, the water age in the system is 0:00 hours at every node. The water has not yet been distributed within the pipe network system.

As shown in Figure 6.71, the water age is ranging from 4 to 8 hours in most of the junction nodes and stays at the same level till the end of the whole simulation. As for the junction node 20987, the average water age is 12-19 hours; similar water age results would be reported for any other junction node within the water distribution network.

Results statistics is used to display the maximum water age at each node or link. This helps to identify the dead-end nodes and pipes in the system as well as display the average maximum water age in the network.

6.7 Lesson 7 - Water Quality - Constituent Chlorine Analysis

This lesson takes you step-by-step, illustrating how to use MIKE URBAN WD to define and perform a constituent chlorine analysis for the defined pipe network system. Also presented is a brief review of the analysis results.

A constituent analysis is used to simulate the growing or decaying of constituents from an initial source in the pipe network system over a period of time through the entire pipe network system. In this lesson we will be simulating chlorine decay in the pipe network system.

A schematic diagram for the pipe network to be analyzed in this lesson is shown in Figure 6.78. The pipe network system consists of 1 tank, 587 junction nodes, and 607 pipes. The total network demand is 3.5 cfs (100 l/s).



Figure 6.78 A schematic diagram of the pipe network system

To save time with this lesson, we have prepared data files that have already been set up in order for you to quickly follow through the lesson. For a list of all the files in this lesson, see the section titled Prepared Input and Output Files.

Begin this lesson by selecting File | Open and choose LESSON7A.MDB from the LESSONS\LESSON7 subdirectory. This file contains the pipe network system.

6.7.1 Defining a Constituent Analysis

The network file (LESSON7A.GDB) has been defined with all of the constituent input data already specified to reduce the time required to complete this lesson. However, this section explains the steps used to define the constituent analysis input data.

Before a constituent analysis can be performed, the project type must be properly defined. To define the project type, select EPANET | Project Options to display the Project Options dialog box, as shown in Figure 6.79. Select the Extended Period Water Quality and Chemical Concentrations options. When finished, the options settings should appear as shown in Figure 6.79. Select «Close» to apply these changes and close the Project Options dialog box.

le Project Type Units Head L	osses Properties Convergence Wa	ter Hammer
Steady state hydraulics Extended period hydraulic Extended period water quality Water hammer	 No water quality analysis Chemical concentrations Water age Source tracing 	Close

Figure 6.79 The Project Options dialog box

Since this project is an extended period analysis, the extended period time parameters will have to be defined. To define the extended period time parameters, see the section titled Defining a Water Age Analysis in Lesson 5. We will be using the same extended period time parameters as in Lesson 6 (simulation duration 3 days, hydraulic time step 15 minutes, and report time step 4 hours).

6.7.2 Defining Constituent Data

In this lesson we will simulate a chlorine decay in the network system over a 24 hour period. Tank A is the initial constituent source. Initially, the chlorine in the pipe network system is not known. The chlorine at the constituent source (Tank A) has a concentration of 1.00 mg/l before being distributed through the entire pipe network system. Therefore, we will not define the initial chlorine concentration in the pipe network system, and we will only specify the chlorine concentration in the constituent source, and the constituent reaction rates.

To define the initial chlorine concentration in the pipe network system:

1. Select EPANET | Water Quality | Initial Water Quality to display the Initial Water Quality Editor as shown in Figure 6.80.

Node ID:		Insert
Quality:		Delete
	,	
		Close
Node ID *	Quality	
1		





- 2. As the initial chlorine concentration is not known at each node or pipe, leave the editor empty i.e. the initial chlorine concentration of 0 mg/l will be used per default.
- 3. Select «Close» to close the Initial Water Quality Editor.

To define the constituent source:

1. Select EPANET | Water Quality | Point Constituent Sources to display the Point Constituent Source Editor as shown in Figure 6.81.

- Source node Node type: Node ID:	Tank 2000		
Source type: Concentration Cyclic profile I	n	centration 💌	<u>A</u> dvanced Close
Node ID*	Concentrati	Cyclic profil	Source type
2000114	1	<null></null>	Concentration



 In the Source Node frame, set the Node Type to Tank and Node ID to 2000114. Set the Concentration to 1.00 mg/l. Choose «Close» to apply these changes and close the Point Constituent Source Editor dialog box.

To define the reaction rates:

1. Select EPANET | Water Quality | Reaction Rates| Global Settings to display the Reaction Rate Editor dialog box as shown in Figure 6.82.

Global settings		
Bulk reaction rate coefficient:	0.0000e+000	- <u>A</u> dvanced.
Pipe wall reaction rate coefficient:	0.0000e+000	Close
New bulk reaction rate coefficient:		-
Time when new bulk reaction rate is used:		-
 Bulk reaction order 	1.00	
Pipe wall reaction order	1.00	
Limiting potential		
Roughness correlation		

Figure 6.82 Reaction Rate Editor dialog box



- 2. Set the Bulk Reaction Rate Coefficient in the Global Settings frame to -0.5. The bulk reaction rate is defined as how rapidly constituent grows or decays over a period of time. In this lesson the units for the reaction rate coefficient time period is per day. The minus value indicates a decay.
- Set the Pipe Wall Reaction Rate coefficient to -1.0. The pipe wall reaction rate is defined as the rate at which a constituent reacts with the wall of a pipe. In this lesson the units are ft/day (m/day). The minus value indicates a decay.
- 4. Choose «Close» to apply these changes and close the Reaction Rate Editor dialog box.

6.7.3 Prepared Input and Output Files

Completed input and output files were provided for this lesson. These files are:

 LESSSON7A.MDB, LESSON7A.RES. These files are the input and output files with the constituent input data already defined.

These files can be found in the LESSONS\LESSON7 subdirectory and can be used to perform the analysis and view the analysis results, without having to interactively enter the data for this lesson.

To analyze the extended period water quality model you have defined, select Simulation | Run Simulation. Refer to the section titled Performing an extended period Analysis in Lesson 5 or 6 for mode details on how to run the extended period analysis.

6.7.4 Constituent Chlorine Decay Results

After performing the analysis, you are ready to view the analysis results. We will use only one viewing method to reduce the time required for this lesson. For more details on viewing extended period analysis results, see the section titled Viewing the extended period Analysis Results in Lesson 5. In this lesson we will display the analysis results in the Horizontal Plan window.

To display the chlorine concentration at each node in the pipe network system:

- 1. Run the water quality simulation to select Model Results | Load Simulation Results| EPANET Results and in the Open dialog box choose LES-SON7A.RES. Select «Open» to load the analysis results.
- 2. Select Model Results | Add Result Layer to add the results of the water quality analysis in link to the table of contents. Select "Link: Quality", as shown in Figure 6.83.



Result File:	Lesson5a.res	1	
Data Type:	Node: Quality		



3. Right-click the "Link: Quality" result layer in the table of contents to display the Layer Property options dialog box to set the thematic mapping style for the results layer, as shown in Figure 6.68.

-	•	Les	son07a.res	
	Ξ	•	Link: Quality	
			<= 0.125000	
				<= 0.375000
			-> 0.375000 AND	<= 0.625000
				<= 0.875000
			-> 0.875000 AND	<= 1.125000

Figure 6.84 Result layer thematic mapping

4. The Map window will display the analysis results for the selected result layer, as shown in Figure 6.85.



Figure 6.85 Analysis results on day 0 at 0:00 hours



The results shown in Figure 6.85 are for the start of the extended period simulation. Note that the chlorine concentration is 0 hrs. at all the nodes in the model. To display the results at a different time step, select Model Results | Animate or use the Animation floating toolbar to advance in time or to play the animation, as shown in Figure 6.70.





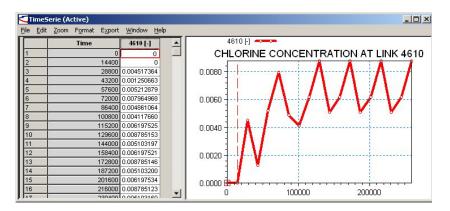
5. From the Animation toolbar, use Step forward to advance in time and choose day 0 at time 4:00 hours to display the results at this time step in the Map window. The results for this time step are displayed in Figure 6.87. Observe how the water is aging in the distribution system.



Figure 6.87 Analysis results on day 0 at 4:00 hours

6. Select Create Time Series to display the history of a source trace analysis for the pipe 4610 or any other selected junction node. The results for this pipe are displayed in Figure 6.88. Note that the water age changes in time due to the diurnal profile assigned to each junction node in the model. The average water age at this junction node is 0.005-0.009 mg/l.







Result Statistics

MIKE URBAN allows you to calculate results statistics for any calculated attribute such as node pressure, link flow, node water quality, and others. The results statistics includes calculation of maximum and minimum values, as well as storing the actual values in the model database for external post-processing using ESRI ArcMap, Microsoft Access and other tools.



1. In order to calculate result statistics, select Model Results | Results Statistics and select "Calculate" in the Results Statistics dialog, as shown in Figure 6.89.

 Ilt File: Lesson7a-Base Type: EPANET Result			culate	Advance Close
 Statistics Name	Min	Max	Current	Accumulated
Link: Starting/Ending HGL				
Link: Friction Factor				
Link: Reaction Rate				П
Link: Setting				
Link: Status				Г
Link: Quality				E
Link: Headloss/1000				
Link: Velocity				
Link: Flow				Γ
Node: Quality				Г
Node: Pressure				Г
Node: HGL				Г
 Node: Demand		Г		F

Figure 6.89 Results Statistics allows you to calculate minimum and maximum values for any selected result item

2. The select Model Results | Add Statistic Layer to add the maximum chlorine concentration at junction nodes to the layer of contents.

The maximum chlorine concentration at each junction node will then be displayed in the Map window as a separate layer, as shown in Figure 6.90.

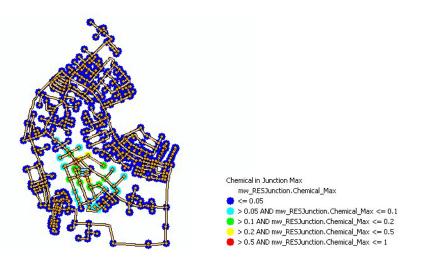


Figure 6.90 Maximum chlorine concentration at each junction node

6.7.5 Reviewing the Analysis Results

The results of extended water quality simulation show a decay of chlorine within the water distribution network. The chlorine concentration is ranging from 0.1-0.5 mg/l in the zone area close the to water source (tank) where the chlorine is injected to the network. Looking further away from the chlorine source, we observe a chlorine concentration below 0.1 mg/l.

6.8 Lesson 8 - Distributed Demands and Pressure Zones

This lesson takes you step-by-step, illustrating how to use MIKE URBAN WD to define distributed demands, pressure zones, import a background image, and perform a steady state analysis for the defined pipe network system. Also presented is a brief review of the analysis results.

Distributed demands are used to compute the demand for each junction node in the network system when only the total demand in a pipe network system or a particular pressure zone of the pipe network system is known.

A schematic diagram for the pipe network to be analyzed in this lesson is shown in Figure 6.91. The pipe network system consists of 1 tank, 587 junction nodes, and 607 pipes. The total network demand is 3.5 cfs (100 l/s) and the leakage is estimated as 10% i.e 0.35 l/s (10 l.s).





Figure 6.91 A schematic diagram of the pipe network system

To save time with this lesson, we have prepared data files that have already been set up in order for you to quickly follow through the lesson. For a list of all the files in this lesson, see the section titled Prepared Input and Output Files.

Begin this lesson by selecting File | Open and choose LESSON8A.MDB from the LESSONS\LESSON08 subdirectory. This file contains the pipe network system.

6.8.1 Distributed Demands

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

In this section the total network demand will be distributed to the pipe network system for each pressure zone using the method of distributed demands. In this lesson the demands at each of the junction nodes in the pipe network system is unknown only the total demand for each pressure zone is known. The total demand in the whole network is 3.5 cfs (100 l/s) and the leakage is 0.35 cfs (10 l/s).

Using the distributed demand feature, MIKE URBAN WD can compute the water demand for each node in the network system based upon the total network demand using one of two methods: the Method of Reduced Pipe Lengths, Method of Equivalent Pipe Lengths, and the Method of Two Coefficients. This is useful when assigning the nodal water demand for a large network, since the software will automatically proportion the total network demand based upon the selected method. These methods are used to mimic the amount of actual demand along each pipe, based upon the pipe length or a pre-defined demand coefficient.

For the three methods of computing distributed demands:

- 1. The Method of Reduced Pipe Lengths uses demand coefficient 1 (or demand coefficient 2,3,4) defined at each pipe and the contributing pipe length.
- 2. The Method of Equivalent Pipe Lengths uses demand coefficient calculated automatically for each pipe based on the actual pipe perimeter versus a perimeter of pipe with the diameter of 6 in (150 mm) and the contributing pipe length.
- 3. The Method of Two Coefficients uses demand coefficients 1 and 2 (or any other combination of demand coefficient 1,2,3,4) defined at each pipe.

For more information on distributed demands, see the Water Distribution User Guide.

In this lesson, we will use the Method of Reduced Pipe Lengths to compute the residential demands and the Method of Equivalent Pipe Lengths to compute the leakage demand.

To distribute the demands among the junction nodes:

- 1. Select EPANET | Distributed Demands to display the Distributed Demand dialog box.
- 2. Define the total network demand to be distributed as 3.15 cfs (or 90 l/s), which is the corresponding residential demand after subtracting the leakage. Select "Method of Reduced Pipe Length" and enter the multiple junction demand category name as "Residential". Select "Compute" to distribute the specified network demand to each junction node within the distribution zone as shown in Figure 6.92.

pe demand coefficients Node demand	coefficients	
Total network water demand: Node pressure zone ID: Pipe pressure zone ID: Use pipe pressure zone demand	90	Compute <u>R</u> eset Close
Distribution method Method of 2 coefficients Method of reduced pipe lengths Method of equivalent pipe lengths Select pipe demand coefficient 1: Select pipe demand coefficient 2:	Coeff 1	
.Target demand Multiple junction demand:	Residential	1

Figure 6.92 The Distributed Demands dialog box - Method of Reduced Pipe Lengths is used for distributing residential demand

3. Define the total network demand to be distributed as 0.35 cfs (or 10 l/s), which is the estimated network leakage. Select "Method of Reduced Pipe Length" and enter the multiple junction demand category name as "Leakage". Select "Compute" to distribute the specified network demand to each junction node within the distribution zone in Figure 6.93.

Total network water demand: Node pressure zone ID: Pipe pressure zone ID:	10 	<u>C</u> ompute <u>R</u> eset Close
C Use pipe pressure zone demand		
Distribution method]	
C Method of 2 coefficients		
C Method of reduced pipe lengths		
Method of equivalent pipe lengths		
Select pipe demand coefficient 1:	Coeff 1	
Select pipe demand coefficient 2:	Coeff 1	
Target demand		
Multiple junction demand:	Leakage	

Figure 6.93 The Distributed Demands dialog box - Method of Equivalent Pipe Lengths is used for distributing demand corresponding to leakage



6.8.2 Prepared Input and Output Files

Completed input and output files were provided for this lesson. These files are:

 LESSSON8A.MDB, LESSON8B.MDB. These files are the input and output files with the model network components including the results of the demand distribution.

These files can be found in the LESSONS\LESSON08 subdirectory and can be used to perform the analysis and view the analysis results, without having to interactively enter the data for this lesson.

6.8.3 Reviewing the Demand Distribution Results

To view the computed distributed demand values, select EPANET | Junctions to open the Junction Editor. The computed demand values are shown in the demand field Figure 6.94.

	lata							
Junction dat	ta Emitter data	a Air-valve data	.]					
	1		1					
	tion & connectiv						Insert	
Asset ID:		10337	Data so	urce:			Delete	
Junction	ID:	10337	Status:		<null></null>	Delete		
X coordin	nate:						Advanced.	
Y coordin	nate:		_				Close	
Descriptio	on:		Pressure zone:					_
– Model da								
- Model da Type:	ita	Junction	▼ State:		unmarked			
1.1		Junction			-	<u> </u>		
Demand	coefficient:		Elevatio	n:	29.699			
Minimal p	ressure:		Surface	elevation:				
							-	
× 🛧 .	€ 🛓							
X 🛧 . Demand	↓ L Description	n Category	Pattern*				_	
and the second s	Description	Category Residential	Pattern* <null></null>				_	
Demand 0.3478	Description							
Demand 0.3478	Description	Residential	<null></null>					
0.3478	Description	Residential	<null></null>					
Demand 0.3478 0.03456 MUID*	Asset ID	Residential Leakage Data source	«Null» «Null»	Eley	Surface Elev		Statello	Der
Demand 0.3478 0.03456 MUID* 0337	Description 5 <null> 7 <null> 10337</null></null>	Residential Leakage Data source	«Null» «Null»	29.699	<null></null>	≺Null>	unmarked	<null< td=""></null<>
Demand 0.3478 0.03456 MUID* 0337 0480	Description 5 <null> 7 <null> 10337 10480</null></null>	Residential Leakage Data source <null></null>	«Null» «Null»	29.699 14.273	<null> <null></null></null>	<null> <null></null></null>	unmarked unmarked	<nul <nul< td=""></nul<></nul
Demand 0.3478 0.03456 MUID* 0337 0480 0483	Description 5 <null> 7 <null> 10337 10480 10483</null></null>	Residential Leakage Data source <null> <null></null></null>	«Null» «Null»	29.699 14.273 18.517	<null> <null> <null></null></null></null>	<null> <null> <null></null></null></null>	unmarked unmarked unmarked	<nul <nul <nul< td=""></nul<></nul </nul
Demand 0.3478 0.03456 MUID* 0337 0480 0483 0484	Description 7 <null> 7 <null> 10337 10480 10483 10484</null></null>	Residential Leakage Data source <null> <null> <null></null></null></null>	«Null» «Null»	29.699 14.273 18.517 22.915	<null> <null> <null> <null></null></null></null></null>	<null> <null> <null> <null></null></null></null></null>	unmarked unmarked unmarked unmarked	<null <null <null <null< td=""></null<></null </null </null
Demand 0.3478 0.03456 MUID* 0337 0480 0483 0484 0486	Description 7 <null> 7 <null> 10337 10480 10483 10484 10486</null></null>	Residential Leakage Data source <null> <null> <null> <null> <null></null></null></null></null></null>	«Null» «Null»	29.699 14.273 18.517 22.915 23.014	<null> <null> <null> <null> <null></null></null></null></null></null>	<null> <null> <null> <null> <null></null></null></null></null></null>	unmarked unmarked unmarked unmarked unmarked	<nul <nul <nul <nul <nul< td=""></nul<></nul </nul </nul </nul
Demand 0.3478 0.03456 0000 0000 0000 0480 0483 0484 0486 0486 0487	Asset ID 10480 10480 10483 10484 10486 10487	Residential Leakage Data source <null> <null> <null> <null> <null> <null> <null></null></null></null></null></null></null></null>	«Null» «Null»	29.699 14.273 18.517 22.915 23.014 33.891	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	unmarked unmarked unmarked unmarked unmarked unmarked	<nul <nul <nul <nul <nul <nul< td=""></nul<></nul </nul </nul </nul </nul
Demand 0.3478 0.03456 00337 0480 0483 0483 0483 0484 0486 0487 0492	Asset ID 10480 10337 10480 10483 10484 10486 10487 10489	Residential Leakage Data source <null> <null> <null> <null> <null> <null> <null> <null> <null> <null></null></null></null></null></null></null></null></null></null></null>	«Null» «Null»	29.699 14.273 18.517 22.915 23.014 33.891 33.841	<null> <null> <null> <null> <null> <null> <null></null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null> <null></null></null></null></null></null></null></null>	unmarked unmarked unmarked unmarked unmarked unmarked unmarked	<nul <nul <nul <nul <nul <nul< td=""></nul<></nul </nul </nul </nul </nul
Demand 0.3478 0.03456 0.03456 0.03456 0337 0480 0483 0483 0484 0484 0486 0487 0482 0489	Asset ID 10480 10337 10480 10483 10484 10486 10486 10487 10492 10496	Residential Leakage Data source <null> <null></null></null></null></null></null></null></null></null></null></null></null></null>	«Null» «Null»	29.699 14.273 18.517 22.915 23.014 33.891 33.841 32.722	<null> <null> <null> <null> <null> <null> <null> <null> <null></null></null></null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null> <null> <null> <null></null></null></null></null></null></null></null></null></null>	unmarked unmarked unmarked unmarked unmarked unmarked unmarked unmarked	<null <null <null <null <null <null <null< td=""></null<></null </null </null </null </null </null
Demand 0.3478 0.03456	Asset ID 10480 10337 10480 10483 10484 10486 10487 10489	Residential Leakage Data source <null> <null> <null> <null> <null> <null> <null> <null> <null> <null></null></null></null></null></null></null></null></null></null></null>	«Null» «Null»	29.699 14.273 18.517 22.915 23.014 33.891 33.841	<null> <null> <null> <null> <null> <null> <null> <null> <null></null></null></null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null> <null></null></null></null></null></null></null></null>	unmarked unmarked unmarked unmarked unmarked unmarked unmarked	<null <null <null <null <null <null <null <null <null< td=""></null<></null </null </null </null </null </null </null </null

Figure 6.94 The Junction Editor - the computed demand values are shown in the demand field



To view the list of all multiple demand values, select EPANET | Multiple Demands to open the Multiple Demand Editor. The computed demand values are shown in the demand field Figure 6.95.

	Demand: Demand coef		337 348	Descripti Demand Demand	category:	Residential	 <u>D</u> elete <u>A</u> dvanced
T	Junction ID*	Demand	Description	Demand cat	Demand pat	Demand co	 Close
•	10337	0.34785	<null></null>	Residential	<null></null>	<null></null>	
ľ	10337	0.034567	<null></null>	Leakage	<null></null>	<null></null>	
1	10480	0.452613	<null></null>	Residential	<null></null>	<null></null>	
1	10480	0.10422	<null></null>	Leakage	<null></null>	<null></null>	
1	10483	0.155324	<null></null>	Residential	<null></null>	<null></null>	
1	10483	0.014307	≺Null>	Leakage	≺Null>	≺Null>	
1	10484	0.251147	≺Null>	Residential	<null></null>	<null></null>	
1	10484	0.057735	<null></null>	Leakage	<null></null>	<null></null>	
1	10486	0.107347	<null></null>	Residential	<null></null>	<null></null>	
1	10486	0.021972	<null></null>	Leakage	<null></null>	<null></null>	

Figure 6.95 The Multiple Demand Editor - the computed demand values are shown in the demand field

Demand Statistics

MIKE URBAN can generate statistical information for multiple junction node demands. Demand statistics is generated for each pressure zone as well as for the complete network. Additionally, demand statistics dialog box allows the user to redistribute node demands by changing the calculated statistical results. The Demand Statistics dialog box is reached by selecting EPANET | Demand Statistics as shown in Figure 6.96.

	Registribute Close									
-	RecTypeNo	PZonelD*	Category	MinDemand	MaxDemand	AvaDemand	SumDeman	NewAyaDe	NewSumDe	CatTypeN
•		<null></null>	Leakage	10		10		<null></null>	<null></null>	<null></null>
	Data	<null></null>	Residential	90	90	90	2340	<null></null>	<null></null>	<null></null>
	Zone	<null></null>		100	100	100	2600	<null></null>	<null></null>	<null></null>
	Network		-	100	100	100	2600	<null></null>	<null></null>	<null></null>

Figure 6.96 Demand statistics is generated for each pressure zone as well as for the complete network

Note that it is possible to redistribute the zone or network demand by specifying the new average or new summary demand by changing the appropriate grid fields and by selecting "Redistribute", as shown in Figure 6.97.

	<u>R</u> efresh Registribute Close									
-	RecTypeNo	PZonelD*	Category	MinDemand	MaxDemand	AvgDemand	SumDeman	NewAvgDe	NewSumDe	CatTypeN
•	Data	<null></null>	Leakage	10	10	10	260	15	<null></null>	<null></null>
	Data	<null></null>	Residential	90	90	90	2340	<null></null>	<null></null>	<null></null>
	Zone	<null></null>		100	100	100	2600	<null></null>	<null></null>	<null></null>
	Network			100	100	100	2600	<null></null>	<null></null>	<null></null>

Figure 6.97 The zone or network demand can be redistributed from within Demand Statistics dialog box

6.9 Lesson 9 - Demand Allocation

This lesson illustrates how to use MIKE URBAN WD to allocated demand (consumption data) from X,Y demand points and how to geocode (assign) such data to the appropriate model junctions or pipes and how to aggregate demand values into node demands or pipe demand coefficients.

To save time with this lesson, we have already prepared data files in order for you to quickly follow through the lesson. For a list of all the files in this lesson, see the section titled Prepared Input and Output Files.

This lesson uses the pipe network system constructed in Lesson 1 of this chapter. The layout of this pipe network system is shown in Figure 6.98.

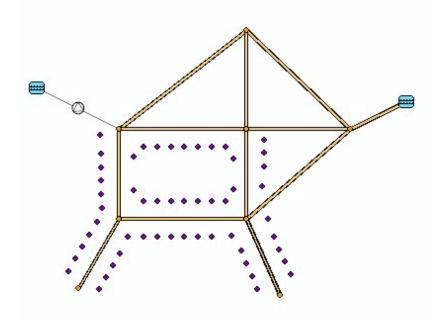


Figure 6.98 The pipe network system used in this lesson

Begin this lesson by loading LESSON10a.MDB from the LESSONS\LES-SON10 subdirectory. This file contains the pipe network system to be used in this lesson.

6.9.1 Demand Points Import

The demand points used in this lesson were imported from X,Y points, corresponding with the parcel centre points. Such data can be imported into MIKE URBAN using File | Import/Export. The mandatory fields are Reference_ID, X,Y, and ActDemand. The reference field is use to maintain the link to the original demand sources.

Once the data is imported to MIKE URBAN, it is displayed in the Map window. Note that it is possible to select different symbology for displaying demand points by selecting "Properties" for "Water Demand Allocation" layer within the Demand Allocation Group in the table of contents.

To display the parcel data, use "Add Data" tool and select "Parcel_Polygons.shp" from /Shapefiles subdirectory located under Lesson10 directory, as shown in Figure 6.99.

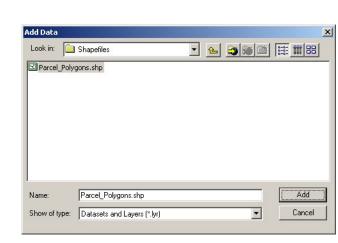


Figure 6.99 Use Add Data tool to load and display background shapefiles

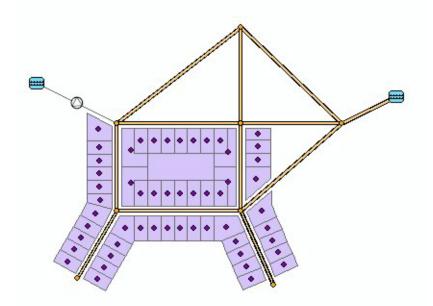


Figure 6.100 Demand points are displayed along with background Shapefiles in the Map window

6.9.2 Demand Geocoding

In this section we will geocode - assign the demand points to the appropriate junctions or pipes in the model database.

To geocode demand points, use the following steps:



1. Select EPANET | Demand Allocation to open the Demand Allocation Editor, as shown in Figure 6.101.

	Reference ID): 1		Desc	ription:	P-1		<u>I</u> nsert	~~~
	Pressure zon	e ID:		Dema	and category:	Resident	ial	<u>D</u> elete	•
	X coordinate:	. 1	00.186	Dema	and pattern:			Advance	d
	Y coordinate:	: 8	8.985	Categ	Category type:				
	Junction ID:	ĺ	1	Addre	ess:	y Ave	Aggregatio	regation	
	Pipe ID:			N Own	er:				0000
			0.005 Equivalent person:						<u>Place</u>
								Close	
	Minimum demand:			E GUIV	alent tenement				_
	Average dem					,			
	Average dem	nand:							
		nand:				1			
	Average dem	nand:							
	Average dem Maximum der	nand:	Pipe ID*	Demand	Minimum d		Maximum d	Demand pat	Þ
	Average dem Maximum der	nand:	Pipe ID* <null></null>		Minimum d		Maximum d	Demand pat	
	Average dem Maximum der Reference I	nand: mand: Junction ID*		Demand 0.0053	Minimum d	Average de			D
	Average dem Maximum der Reference I 1	nand: [mand: [Junction ID*	≺Null>	Demand 0.0053	Minimum d <null> <null></null></null>	Average de <null></null>	<null></null>	<null></null>	
	Average dem Maximum der Reference I 1 10	nand: mand: Junction ID* 0	<null> <null></null></null>	Demand 0.0053 0.006	Minimum d <null> <null></null></null>	Average de <null> <null></null></null>	<null> <null></null></null>	<null> <null></null></null>	
	Average dem Maximum der Reference I 1 10 11	mand: Junction ID* 0 0 0 0	<null> <null> <null></null></null></null>	Demand 0.0053 0.006 0.0064	Minimum d <null> <null> <null></null></null></null>	Average de <null> <null> <null></null></null></null>	<null> <null> <null></null></null></null>	<null> <null> <null></null></null></null>	
	Average dem Maximum der Reference I 1 10 11 12	mand: Junction ID ⁺ 0 0 0 0 0	<null> <null> <null> <null></null></null></null></null>	Demand 0.0053 0.006 0.0064 0.0067	Minimum d <null> <null> <null> <null> <null></null></null></null></null></null>	Average de «Null» «Null» «Null» «Null»	<null> <null> <null> <null></null></null></null></null>	<null> <null> <null> <null></null></null></null></null>	
	Average dem Maximum der Reference I 1 10 11 12 13	Junction ID* 0 0 0 0 0 0	<null> <null> <null> <null> <null></null></null></null></null></null>	Demand 0.0053 0.0064 0.0064 0.0067	Minimum d <null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	Average de <null> <null> <null> <null> <null></null></null></null></null></null>	<null> <null> <null> <null> <null></null></null></null></null></null>	<null> <null> <null> <null> <null></null></null></null></null></null>	
	Average dem Maximum der Reference I 1 10 11 12 13 14	mand: Junction ID* 0 0 0 0 0 0 0 0 0	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	Demand 0.0053 0.0064 0.0064 0.0067 0.0071 0.0074	Minimum d <null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	Average de <null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	<null> <null> <null> <null> <null> <null></null></null></null></null></null></null>	

Figure 6.101 The Demand Allocation Editor

- 2. Press the Geocode button from the Demand Allocation Editor.
- 3. Select Geocode Demands to Nodes if you want to assign demand points to junction nodes and use the demand data to develop node demands. Use Geocode Demands to Pipes if you want to assign demand points to pipes and use the demand data to develop pipe demand coefficients. In case of geocoding demands to nodes, it is possible to find the nearest node directly or it is possible to find the nearest pipe first, test the pipe for its attributes and geocode the demand point to the nearest node on such pipe.
- 4. Define the snap tolerance radius; if the actual distance between the demand point and the nearest node or pipe is greater than the snap tolerance radius, the demand point will not be used.
- 5. Define the maximum pipe diameter if you want to restrict demand assignment to pipe smaller than 300mm (12 inches), for example.
- 6. Define pipe attributes if you want to use only some pipes for the demand geocoding. The pipe selection can be done on the database level i.e. based on their attribute values and it is also possible to combine both pipe and demand attributes such as using only such demand points, which are within the same pressure zone as pipes, and similar.

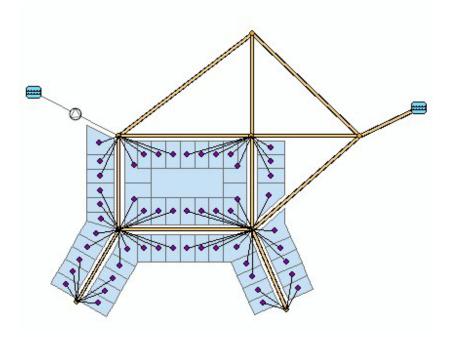


Figure 6.102 Demand points are geocoded to nodes

6.9.3 Demand Aggregation

In this section of the lesson we will aggregate the demand-consumption data into junction node demands. It is also possible to aggregate the demand-consumption data into the pipe demand coefficients.

To aggregate demand points, use the following steps:

- 1. Choose the Aggregate tool from the Demand Allocation Editor.
- Select Aggregate Demands to Node Demands if you want to aggregate the demand data into node demands (base demand or multiple demand). Select Aggregate Demands to Pipe Demand Coefficients if you want to aggregate demand data into pipe demand coefficients.
- 3. Select target demand in case of aggregating demands into node demands. Define the multiple demand category if you want to aggregate demands into specific demand category such as residential, for example. If the specified demand category does not exist, it will be automatically created; if exists, the demand value be increased by the new demand data.
- 4. Select Use Demand Category if you want to aggregate demand data into the same category as it is specified in the category type of each demand point. This is very suitable in cases where demand points belong to different demand categories such as residential, industrial, leakage, and



similar. Their values will be automatically aggregated by the category and the new (or existing) multiple demands will be created.

5. Select Reset Existing Node Demands or Reset Existing Pipe Demand Categories if you want to overwrite the existing node demands.

Aggregate demands to node demands Assign demands to multiple demands	<u>C</u> ompute Close
Aggregate demands to pipe demands coefficients Select pipe demand coefficient: Coeff 1	
Reset existing node demands Reset existing pipe demands coefficients	
Use demand category	
Target demand Multiple junction demand Residential	

Figure 6.103 The Demand Aggregation dialog box

Demands are aggregated into the multiple junction demand and their values can be reviewed using Junction Editor or Multiple Demand Editor.

6.9.4 Prepared Input and Output Files

Completed input files were provided for this lesson. These files are:

- LESSON10A.MDB, LESSON10B.MDB. These files are the initial project files used in this lesson.
- PARCEL_CEN.SHP, PARCEL_CEN.SHX, PARCEL_CEN.DBF. These are the ESRI Shapefiles containing the demand points.

6.10 Lesson 10 - Water Hammer Analysis with Surge Protection

This lessons illustrates how surge protection can be modelled using MIKE URBAN and how surge protection can be used to protect the pumping station against water hammer.

6.10.1 Without surge protection

The water supply pipeline consists of an upstream reservoir with a pump station that sends water to the upstream reservoir. The system characteristics:

- Pipeline length: 38 km
- Pipe diameter: 500 mm



- Celerity of the wave speed: 1200 m/s
- Steady state discharge: 300 l/s
- Steady state velocity: 1.55 m/s
- Pump operating point: 300 l/s @ 80m

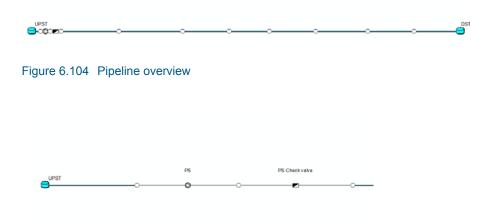


Figure 6.105 Pump station detail

There is a check valve downstream the pump that will automatically close when the differential head across the valve be negative or when the pipeline flow direction would reverse. The valve closure would start reflecting the pressure wave caused by the pump fail back to the downstream reservoir and will protect the pump for high pressures. However, the pipeline would not be protected for both maximum and minimum pressures and the maximum pressure might exceed the pressure rating of the pipeline and the minimum pressure might fall below the pipeline centreline and below minimum pressure (-7m) that holds the water column together.

The simulation will consist of a pump fail and we will investigate the pressure extremes along the pipeline.



1 1 102	V T V			11.00.00		-w-	🗸 🗸 🔫	N 10		- 1 %
📼 Pipes [B	ase]						—	[\times
- Identificatio	n 9. connod	His side a								
Identificatio	n & connect	livity		_				1		Insert
Asset ID:				Data s	source:					
Pipe ID:	-	2		Status	5:	<null< td=""><td>> ~</td><td></td><td>ĺ</td><td>Delete</td></null<>	> ~		ĺ	Delete
From node		2		k				1	Adv	anced
From node	-	_								
To node:		3		📐						Close
Description				Press	ure zone ID:					
· · · ·	-									
Geometrica	l properties									
Length:	-	500.0	0 499.99	9 Diame	eter:	500.0	0			
	-									
Hydraulics 8	friction los	ses								
Material:	-	Steel		Form	ulation:	Darcy	-Weisbach			
Constructio	Construction year: 1998		Roughness:			0.11	0.11			
Wall thickn	ess:			Loss coefficient:		0.000	0.000			
Wave spee	d.	1200.	00	Press	ure normal:	250.0	0	1		
Bulk coeff.	: L	0.000	00e+000	Wall coeff.:		0.000	0e+000			
Miscellaneo										
			0.0					-		
Pipe ID * ▶ 2	Descript	ion	CDate		Asset ID <null></null>		Demand c			
► 2 3	<null> <null></null></null>		1/1/2005 1/1/2005		<null></null>	<null> <null></null></null>	1.000		.000	0.00
4	<null></null>		1/1/2005		<null></null>	<null></null>	1.000		.000	0.00
5	<null></null>		1/1/2005		<null></null>	<null></null>	1.000		.000	0.00
6	<null></null>		1/1/2005		<null></null>	<null></null>	1.000		.000	0.00
7	<null></null>		1/1/2005	1998	<null></null>	<null></null>	1.000	1	.000	0.00
8	<null></null>		1/1/2005	1998	<null></null>	<null></null>	1.000	1	.000	0.00
9	<null></null>		1/1/2005	1998	<null></null>	<null></null>	1.000	1	.000	0.00
AIRCHMB	<null></null>		1/1/2005	1998	<null></null>	<null></null>	1.000	1	.000	0.00
PS-IN	<null></null>		1/1/2005	1998	<null></null>	<null></null>	1.000	1	.000	0.00
PS-OUT	PS Check	valve	1/1/2005	1998	<null></null>	<null></null>	1.000	1	.000	0.00
<										>

Figure 6.106 Pipe data

In order to define the pump fail, we will select the "Pump Trip Off" for the pump Operation schedule type, we will specify the operation schedule that will the pump follow till the time of the pump fail, we will specify the rotational speed of the pump, we will specify the combined (pump and motor) moment of inertia and the pump torque. Note, that you need to collect such data from the pump manufacturer.

Settings	- 0	×
Pump settings		
Operation schedule type:	Pump TripOff V	(
Operation schedule:	PUMP-ON	
Rotational pump speed:	1800	
Moment of inertia:	25.000	
Pump torque:	PUMP-TORQUE	
Motor torque:		
Pump startup time:		
Pump tripoff time:	10	
Q-H curve for rpm=0:		

Figure 6.107 Pump data

Curves and re	elations [Base]	_		×	
Identification & co	onnectivity				
Curve ID:	PUMP-ON		Insert		
Туре:	Pump operational schedule	• ~	Delete		
Description:	Pump is ON		Advance	d	
Plot			Close		
Particular Particul					
.T [s]	.Rpm [RPM]				
• 0.000	1800.000				
60.000	1800.000				
300.000	1800.000				
3600.000	1800.000				
Curve ID *	Type Descriptio				
► PUMP-ON	Pump oper Pump is O				
PUMP-TORQUE	Pump torq <null></null>				

Figure 6.108 Curves data (pump schedule)

Curves and re	elations [Base	e]	_			×			
-Identification & co	onnectivity								
Curve ID:	PUMP-TOF	RQUE			Insert				
Type:	Pump torqu	le	~		Delete	•			
Description:				ŀ	Advance	d			
Plot					Close				
<pre> X → ↓ </pre>									
.Q [L/s]	.T [N*m]								
• 0.000	1200.000								
100.000	1500.000								
200.000	1800.000								
300.000	2000.000								
400.000	1600.000								
500.000	1000.000								
600.000	0.000								
Curve ID *		/pe	Descriptio						
PUMP-ON		erational sc							
PUMP-TORQUE	F	oump torque	<null></null>						

Figure 6.109 Curves data (pump torque)

Once the data is all defined, we will set the project type to "Water hammer" in the Project Options editor and we will define the time settings. Note, that the value of the hydraulic time needs to be very small. Based on the length of pipes the time step typical values are 0.01 seconds (for pipe networks in towns), 0.05 seconds (for long distance pipes). More information is available in MIKE URBAN Water Distribution Book, section Water Hammer.

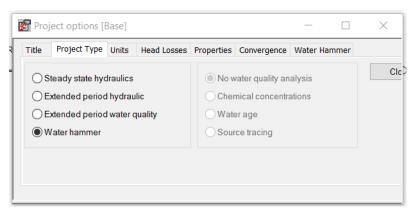


Figure 6.110 Project options

Analysis duration: 300.00 sec Hydraulic time step: 0.05 sec Pattern time step: 1.00 hrs Report time step: 0.25 sec Report start time: 0 hrs Quality time step: 0.1 hrs Start rules clocktime: 12 : 0 AM Statistics: without statistics

Figure 6.111 Time settings

Next, we will run the simulation and load the simulation results. In order to review the simulation results, create a profile plot and add Node HGL into the profile plot as animated, minimum, and maximum time series.

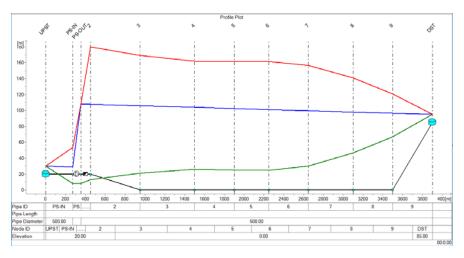
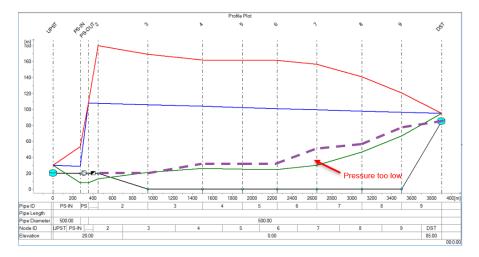


Figure 6.112 Profile plot with HGL levels

We can see from the profile plot that the maximum values of hydraulic grade line will exceed 160m and the minimum values will drop below atmospheric right downstream the pump station. As shown below, based on the actual pipeline elevation minimum pressures night drop below the pipeline centreline along the pipeline and cause water column separation.









6.10.2 With surge protection (air-chamber)

In order to protect the pipeline against low and high pressures, we will add a pressurized vessel (air chamber, pneumatic tank) to the pump station, downstream the pump check valve. The volume of the air chamber needs to be big enough to supply the pipeline with water till the pipeline flow stops.



Figure 6.114 Pump details with air-chamber

Air-tank ID:	AIRCHMB	Description:	Air Chamber	
State:	unmarked \sim	Base elevation:	20.00	
Pressure zone:		X coordinate:	999.652]
Polytropical expa	insion: 1.20	Y coordinate:	74.438	
уре				
Tank size:	circular \vee	Width:	0.00	
Plot		Length		
		Diameter:	2.50	
	- Maximum - Initial	Volume curve:	AirChamber	
Levels	- Minimum	Minimum level:	0.00	1
	Base Elevation	Initial level:	2.00	Ī
Elevations Datum	Elevation = 0	Maximum level:	3.00	
ir tank ID Deseri	ntial Rass alou Drassura d	Tank size Diamatar	nitial leve Maximum Minim	ım I
RCHMB Air Cha		circular 2.50		0.00

Figure 6.115 Air chamber editor

We will repeat the simulation and display the profile plot again.

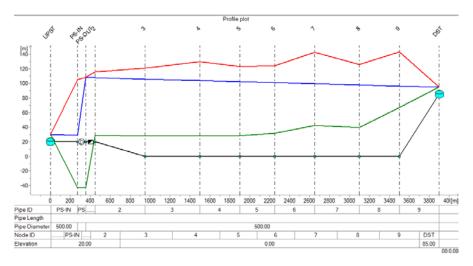


Figure 6.116 Profile plot levels

The maximum pressures were decreased by the air chamber and the minimum pressure improved too. When the pumps and the pump discharge head drops down the pressure is immediately recovered by the air chamber and that can, as illustrated above, worsen the condition near between the pump



and the pump check valve; the pump check valve might be forced to slam close and there might also be a short interval of reverse flows (before the check valve closes) that might get across the pump. It is important to select the pump check valve carefully. Anti-slam check valves and surge anticipation check valves are available by numerous manufacturers.

Air-vessels and air-cushion surge chambers are very important in suppressing transient pressures. They are among the most common devices used for this purpose, and perhaps the major problems that system engineers face are questions how large the initial air-volume should be, and what the total capacity of the air-vessel (or vessels) should be in order to provide a given degree of protection. More information on the air-vessel design including charts for initial air-vessel size estimates are given in Thorley book⁽¹⁾.

¹ Thorley A. R. D, Fluid Transients in Pipeline Systems, John Wiley & Sons; 2nd Edition, 2004, ISBN-10: 1860584055.



Water distribution tutorials

INDEX



Α

Aggrega	tion											. 224
Air-Chan	nber	-										. 227
Analysis												. 143
Analysis	Res	sul	ts	Т	at	ble	è					. 145

В

Backward Tracking						. 192
Browser						. 146

С

Comparing the Analysis Results .		. 160
Comparison Results		. 161
Constituent Analysis		. 206
Constituent Chlorine Analysis		. 205
Constituent Chlorine Decay Results		. 209
Constituent Data		. 207
Control Rules		. 168

D

Defining a Constituent Analysis 2 Defining a Global Demand Change 1 Defining a Junction Node Demand Change 154	55
Defining a Pipe Status Change 1 Defining a Source Tracing Analysis 1 Defining a Water Age Analysis 1 Defining an Extended Period Analysis 1 Defining and Applying a Demand Pattern 165	85 96 63
Defining Constituent Data	07
Defining Control Rules	68
Defining Junction Node Data	
Defining Pipe Data	
Defining Pump Data	
	67
Defining Tank Data	
	30
	86
Demand Aggregation	24
	20
Demand Geocoding	22
	65
Demand Points Import	21
Demand Statistics	
Design Fire Flow Pressure	81
Design Fire Flow Rate	
Displaying Multiple Dialog Boxes 1	

Distributed Demands
E EPANET Analysis Results 144, 170 Extended Period Analysis 162, 169
FField Calculator157, 167Fire Flow Analysis179Fire Flow Pressure181Fire Flow Rate179Flow Pumps140Forward Tracking192
Geocoding
H Head Pumps
Inserting a Valve
L Lessons
M Map Window
Power Pumps140Prepared Input and Output Files144Pressure Reducing Valve152, 153Pressure Zone214Profile Graphical Plots149PRV Properties154
R Results Browser

S

Save					143
Source Node					186
Source Node Water Results					187
Source Tracing Analysis .			18	3,	185
Storage Tank					167

Т

Thematic Plot					175
Throttled Air Chamber					228
Time Series Plot					174

V

Viewing Analysis Results								144
--------------------------	--	--	--	--	--	--	--	-----

W

Water Age Analysis		195, 196
Water Age Results		197
Water Hammer Analysis		225, 226
Water Quality		183, 195, 205

