MIKE URBAN Tutorials

Step-by-Step Training Guide

PLEASE NOTE

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.

1. 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».

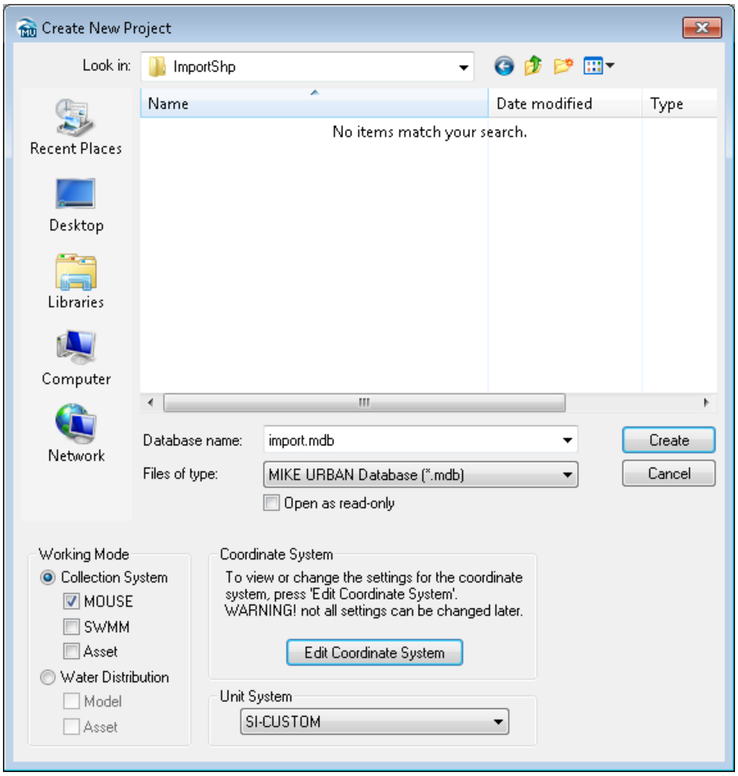


Figure 2.2 Create new project dialog box

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

3. 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.

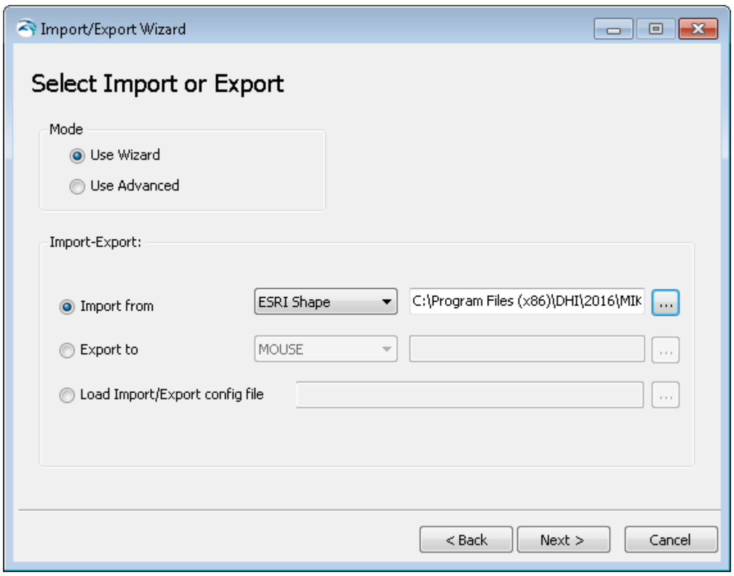


Figure 2.4 Specifying format and files to import from

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.

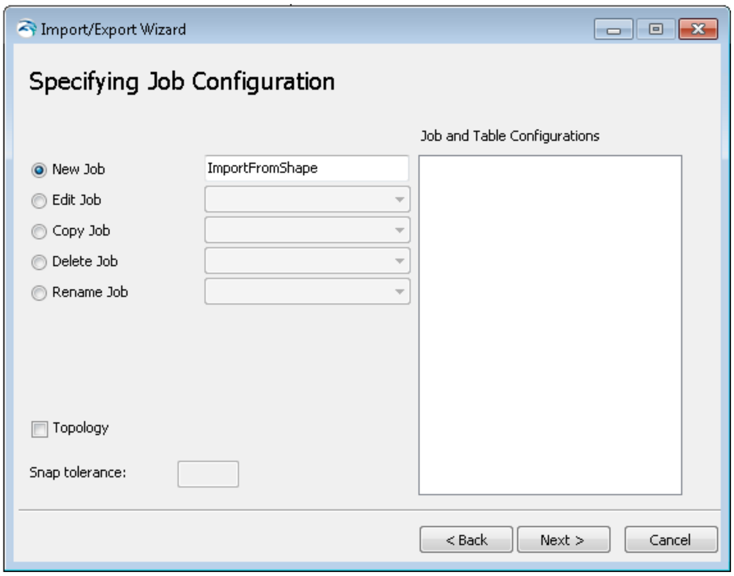


Figure 2.5 Specifying the job configuration

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.

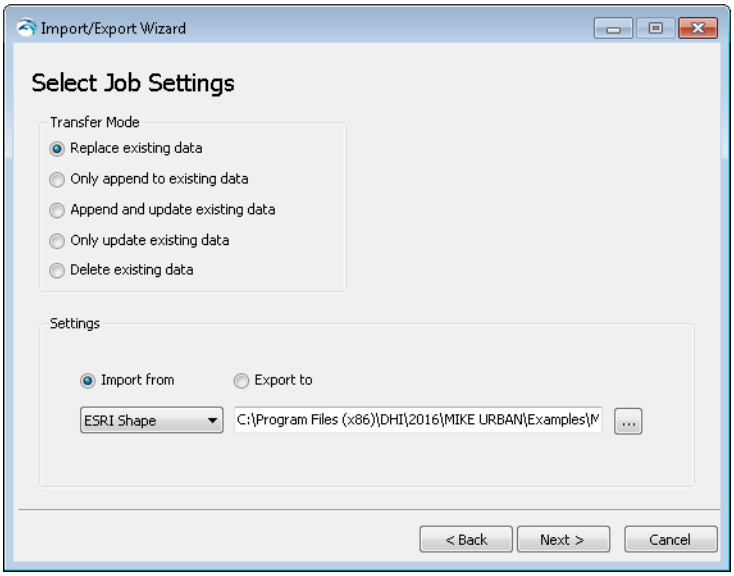


Figure 2.6 Specifying the transfer mode

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.

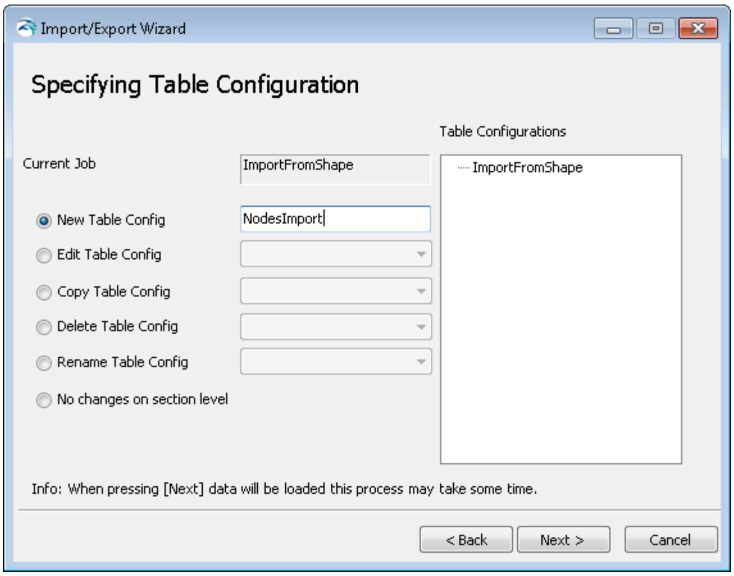


Figure 2.7 Specifying the table configuration

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.

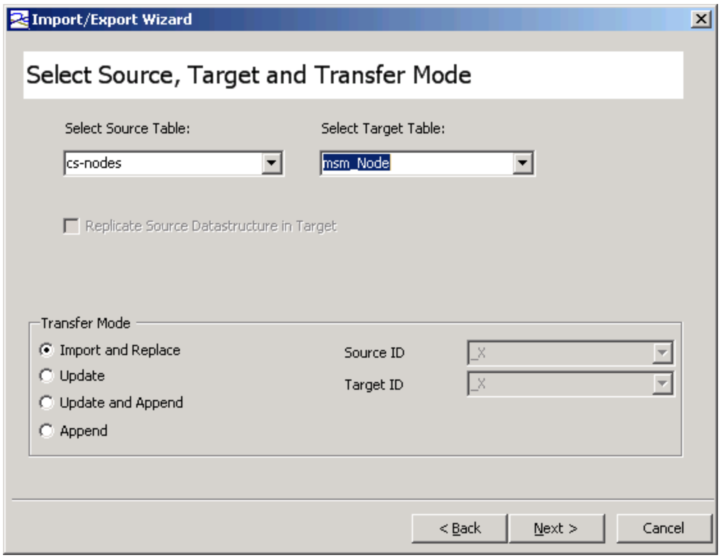


Figure 2.8 Selecting source and target tables

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 assign-ments can be changed. For the other fields the assignment needs to be done as seen in Figure 2.9. Press 'Next' to continue.

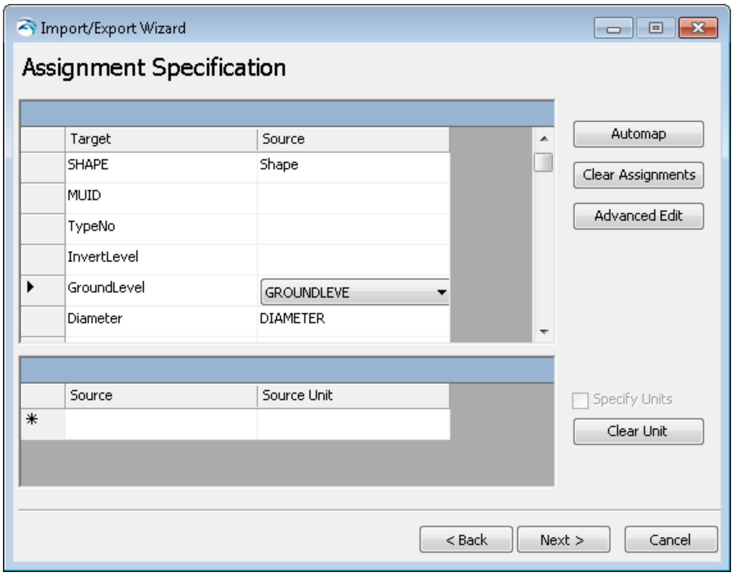


Figure 2.9 Assignment specification for the nodes table

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’.

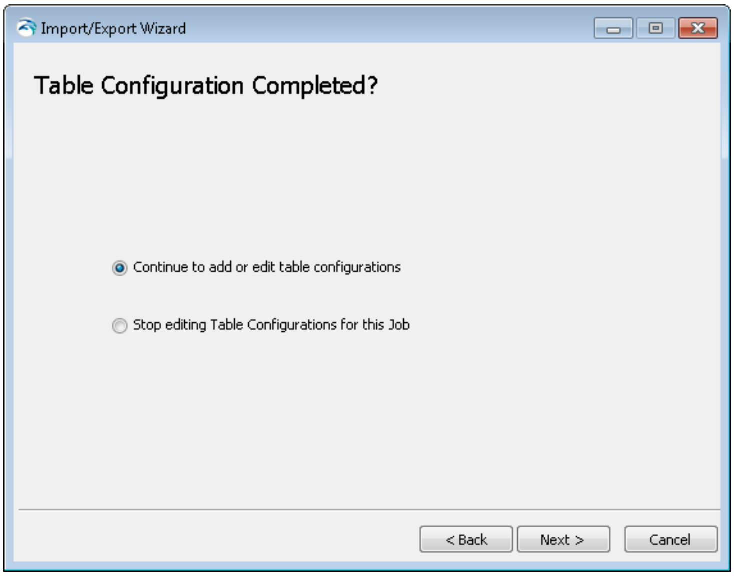


Figure 2.10 Continue to add table configurations

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

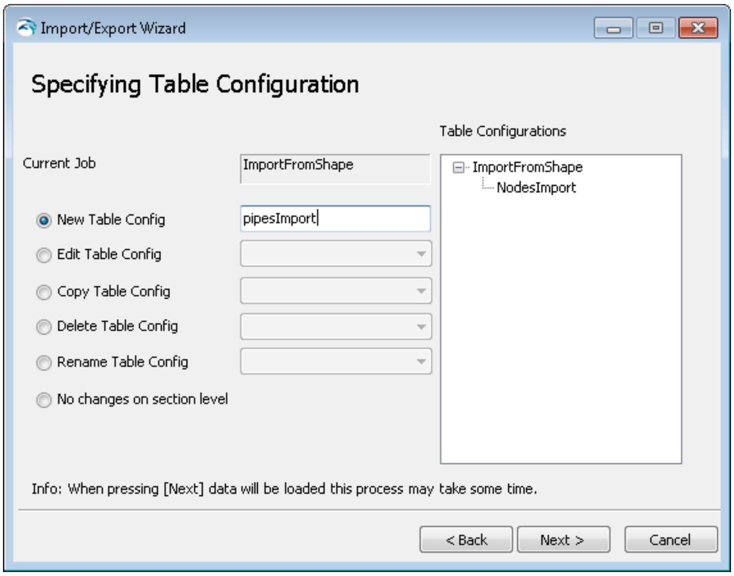


Figure 2.11 Adding a new table configuration for the pipes

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.

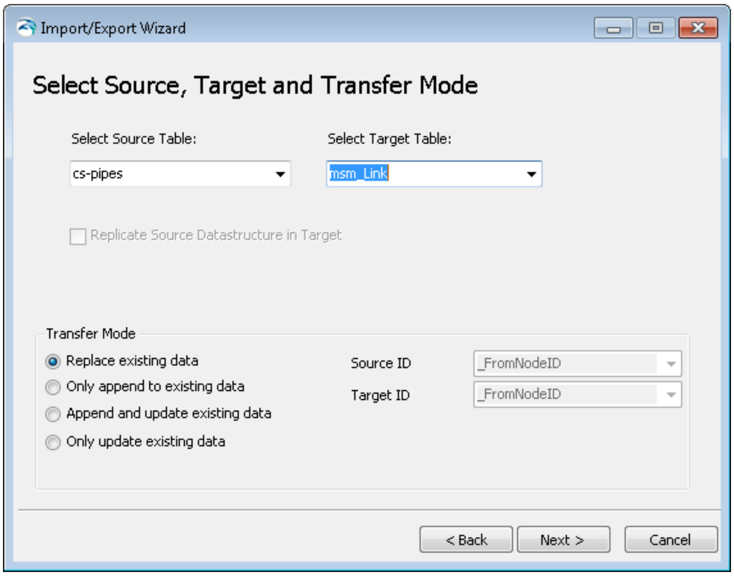


Figure 2.12 Selecting source and target tables

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.

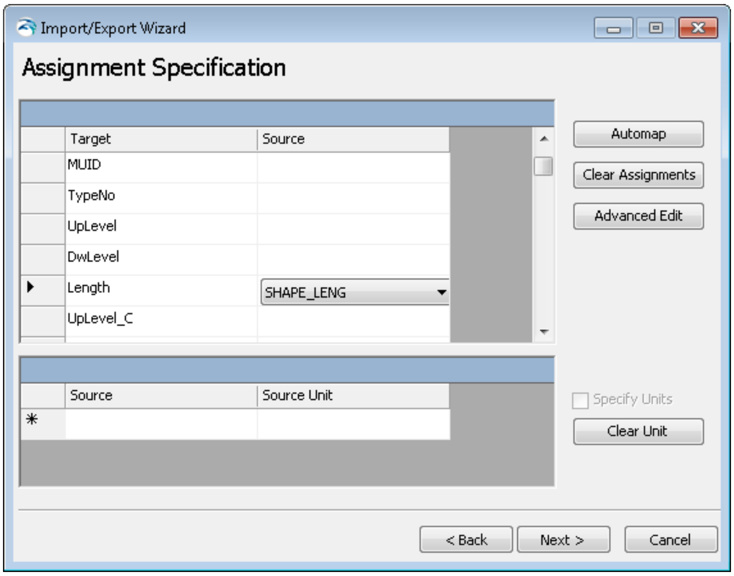


Figure 2.13 Assignment specification for the pipes table

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'.

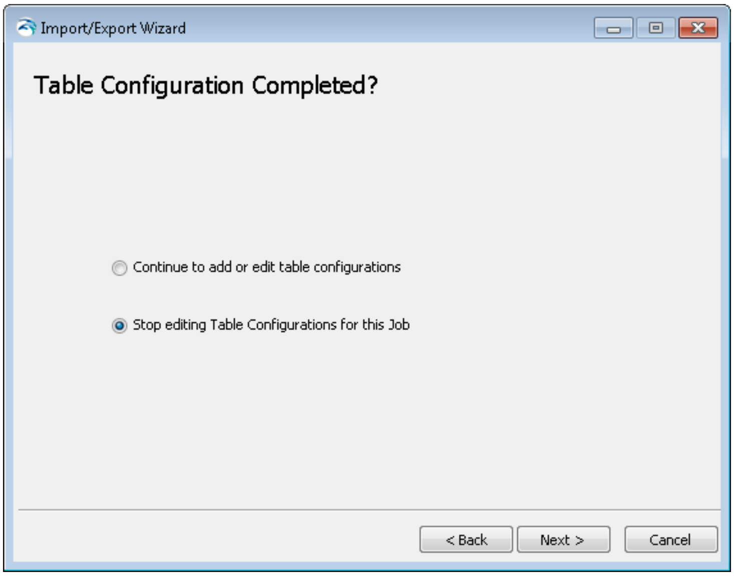


Figure 2.14 Stop adding table configurations

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.

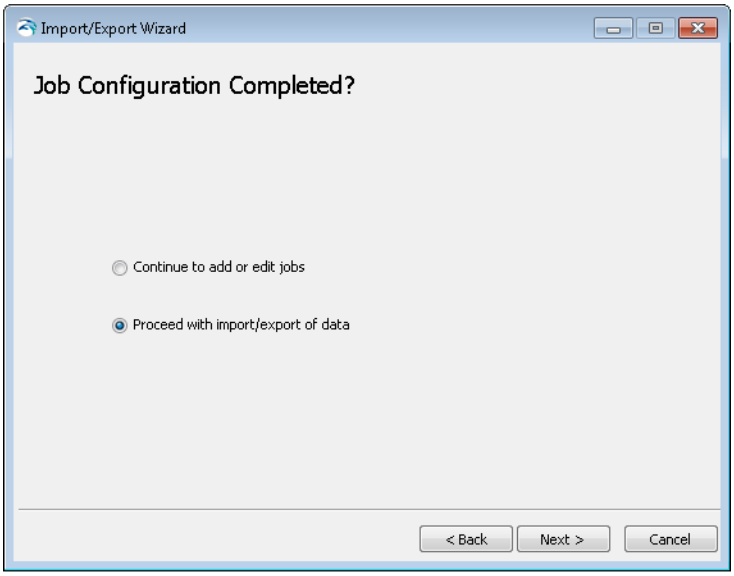


Figure 2.15 The job configuration is complete

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.

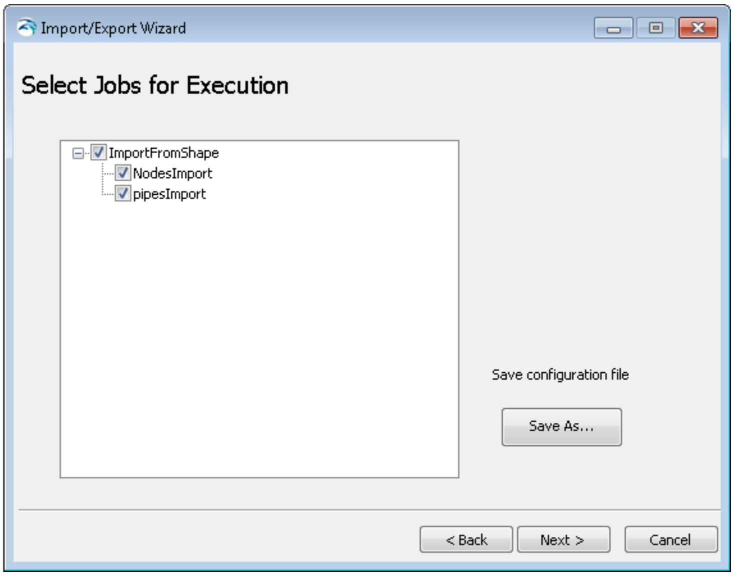


Figure 2.16 Selecting jobs for execution and saving of configuration file

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.

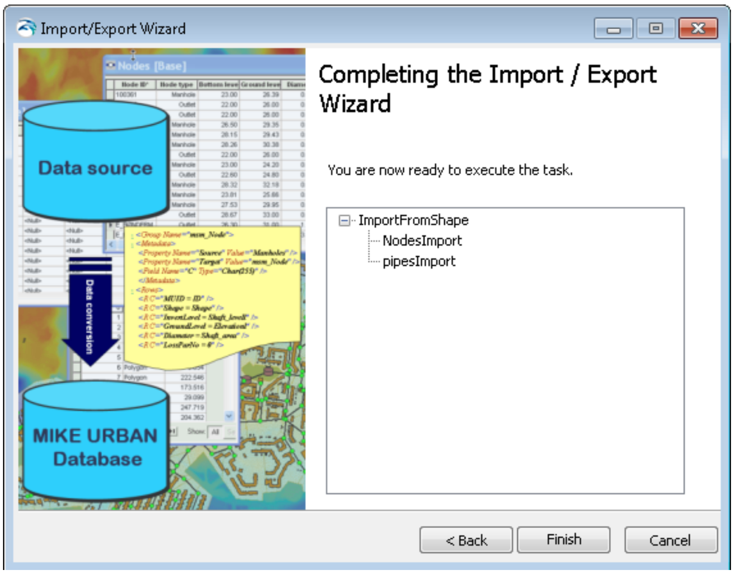


Figure 2.17 Final wizard page, press ‘Finish’ to execute import

The result of the import is seen in Figure 2.18.

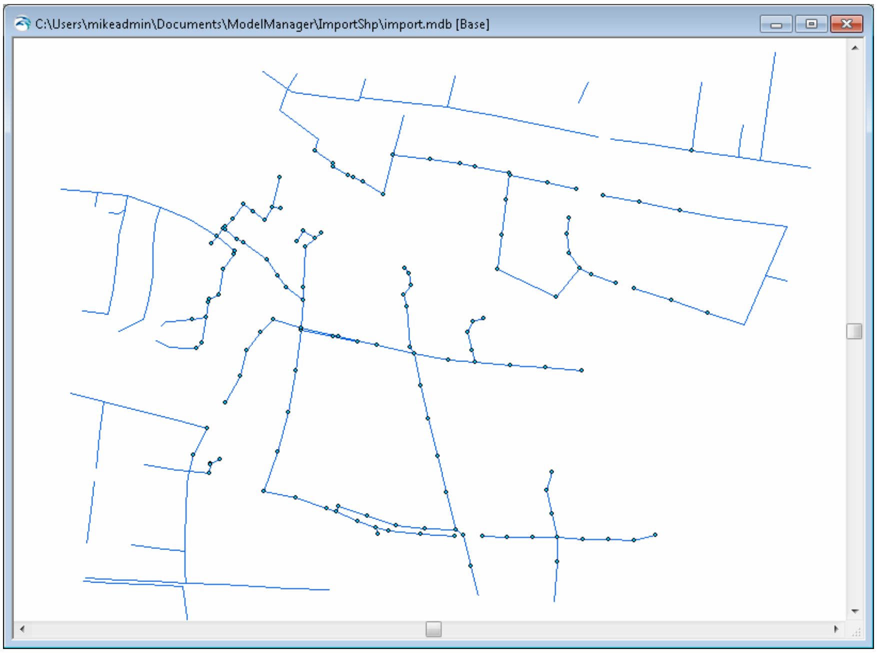


Figure 2.18 The result from the import.

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».

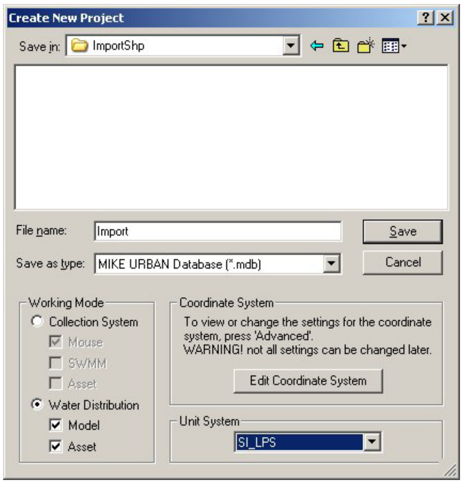


Figure 2.19 Create new project dialog box

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.

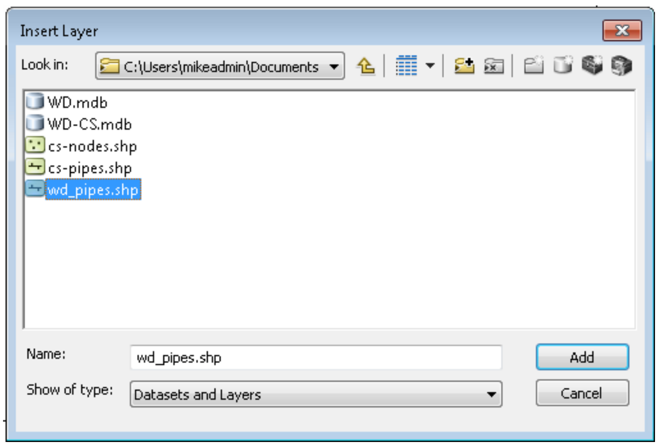


Figure 2.20 Insert Layer dialog box

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.

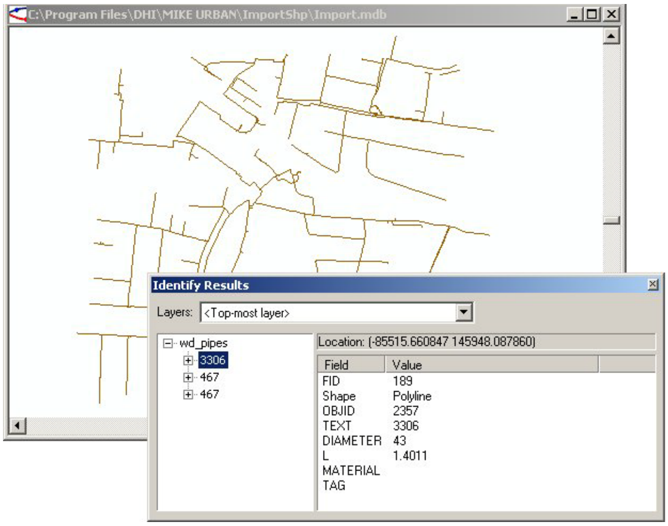


Figure 2.21 Identify tool

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.

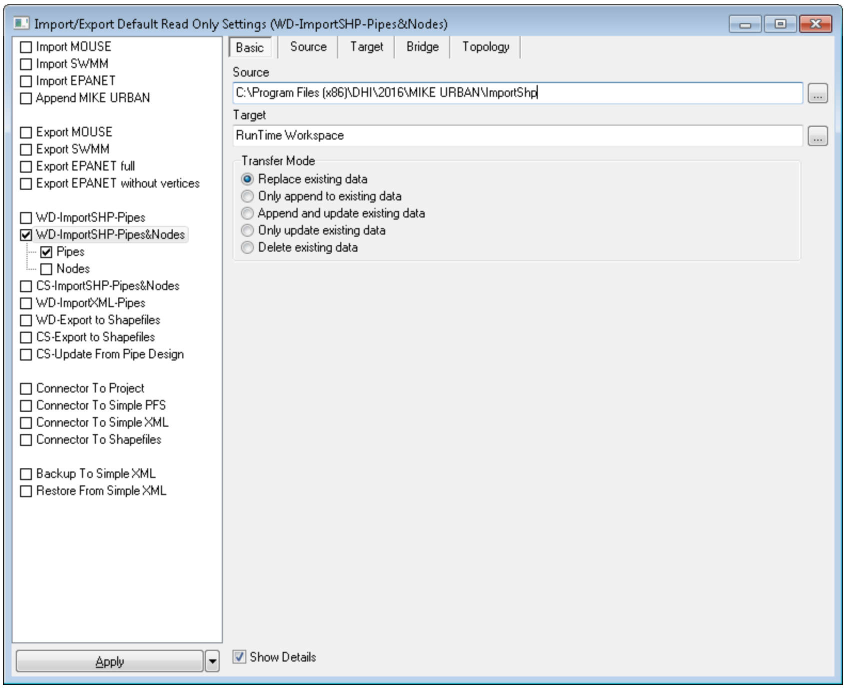


Figure 2.22 Import and Export dialog box

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“

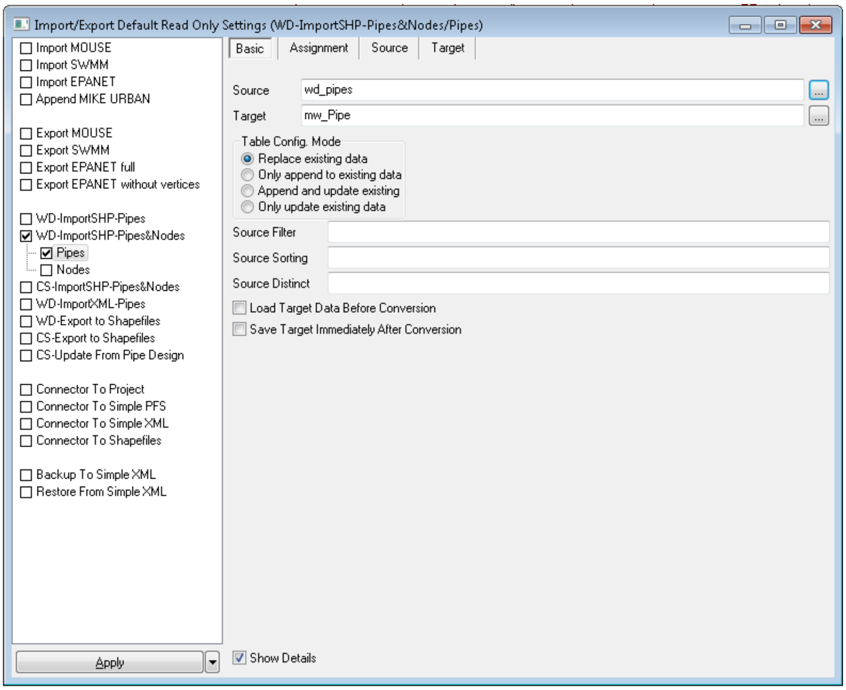


Figure 2.23 Basic settings for Pipes dialog box

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

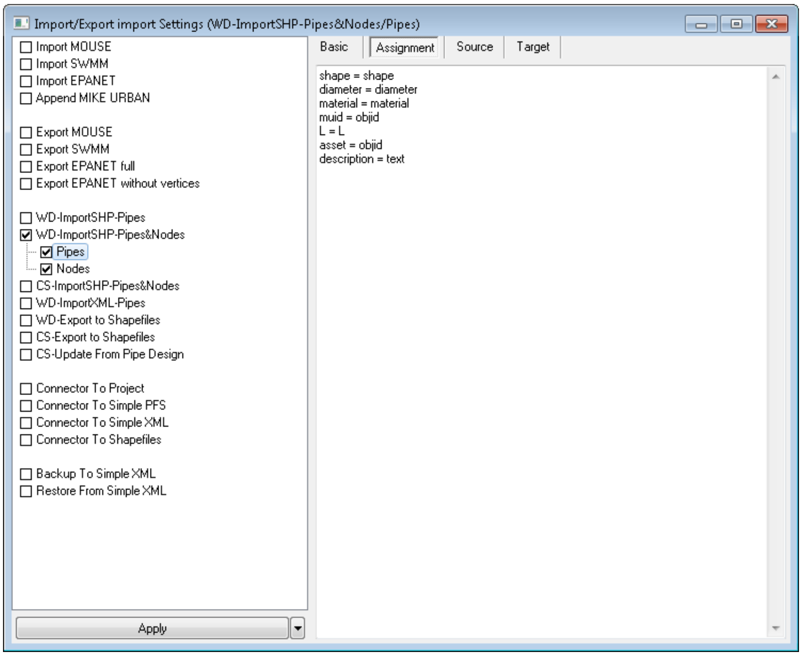


Figure 2.24 Assign Database Attributes for Pipes dialog box

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.
8. Select “Postprocess Network Topology“ (see below) to create the pipe starting and ending nodes automatically and to build-up the network topology.

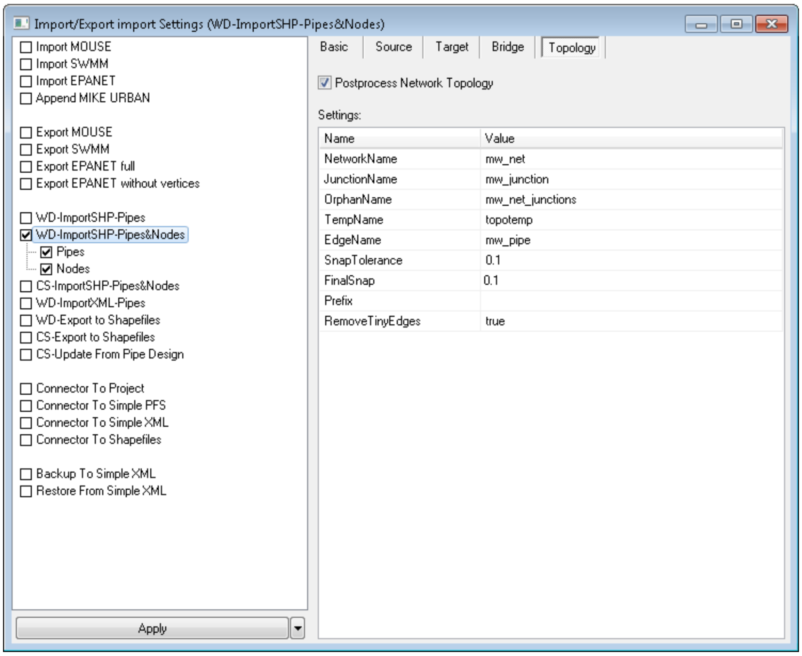


Figure 2.25 Topology settings

9. 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.

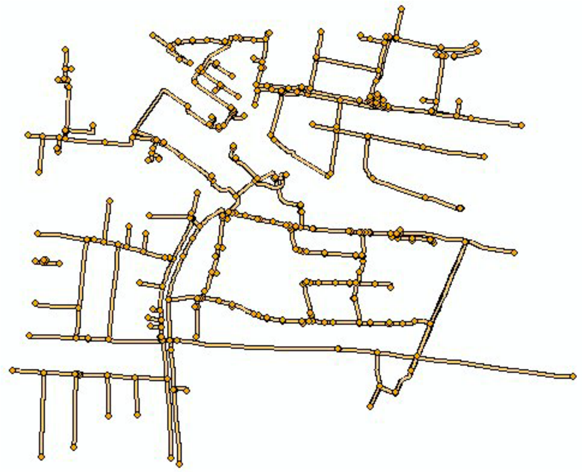


Figure 2.26 Imported model

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».

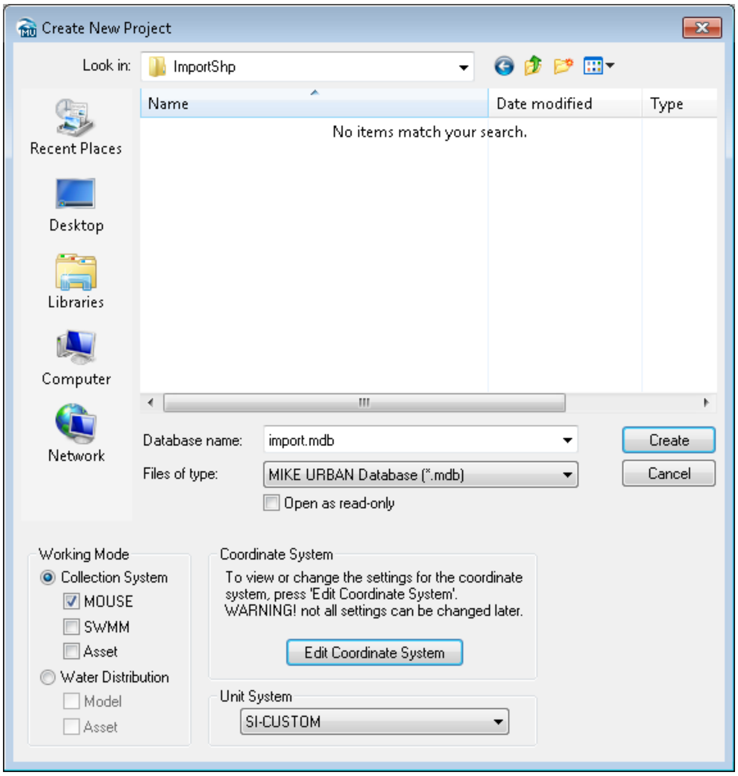


Figure 2.27 Create new project dialog box

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.

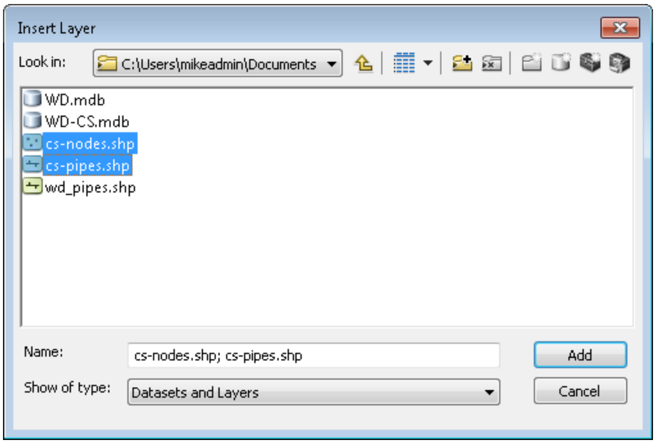


Figure 2.28 Insert Layer dialog box

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.

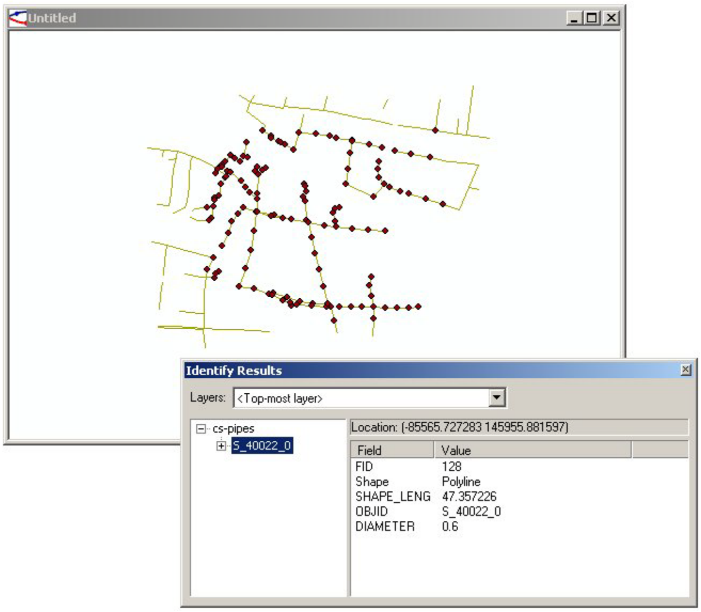


Figure 2.29 Info tool

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.

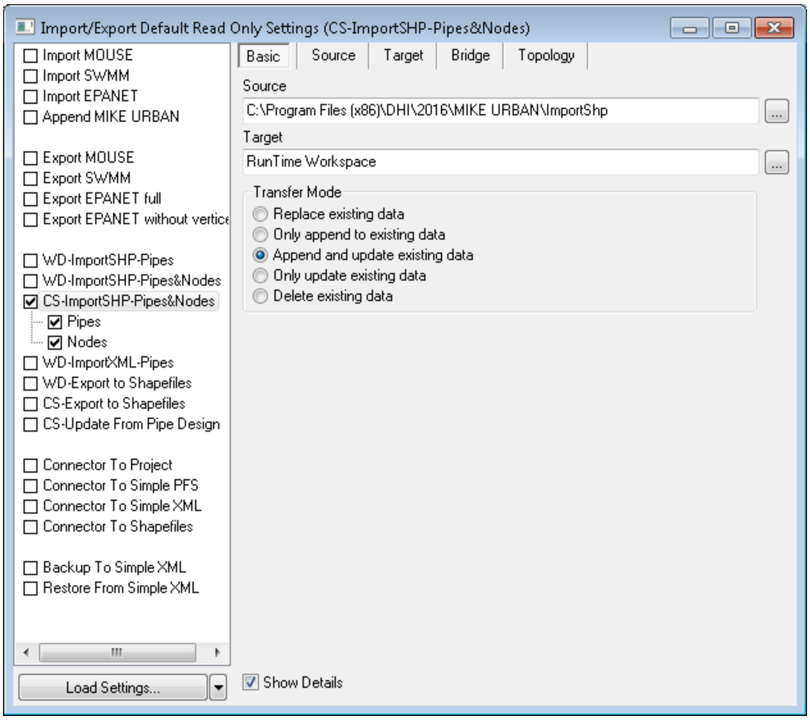


Figure 2.30 Import and Export dialog box

5. 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“

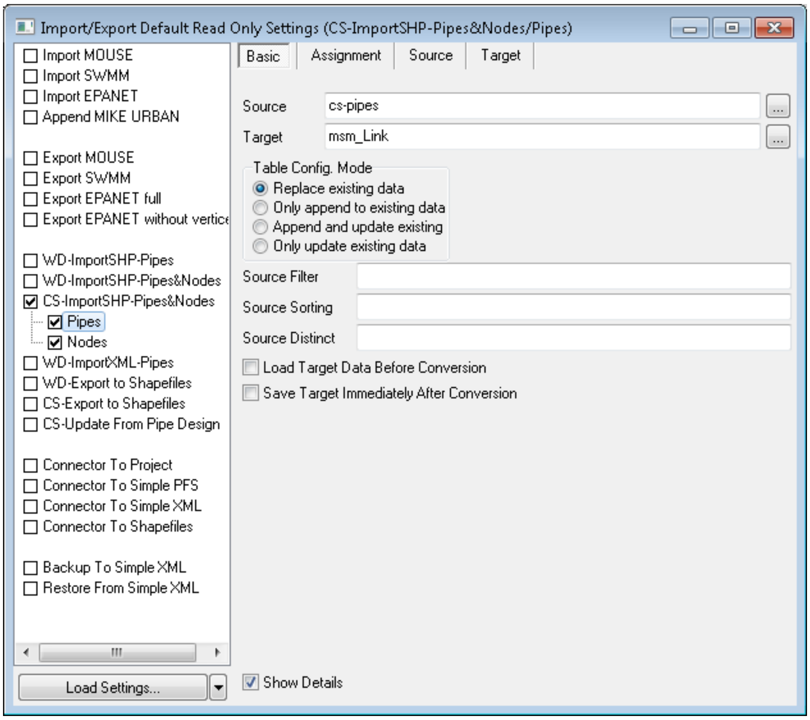


Figure 2.31 Basic settings for Pipes dialog box

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

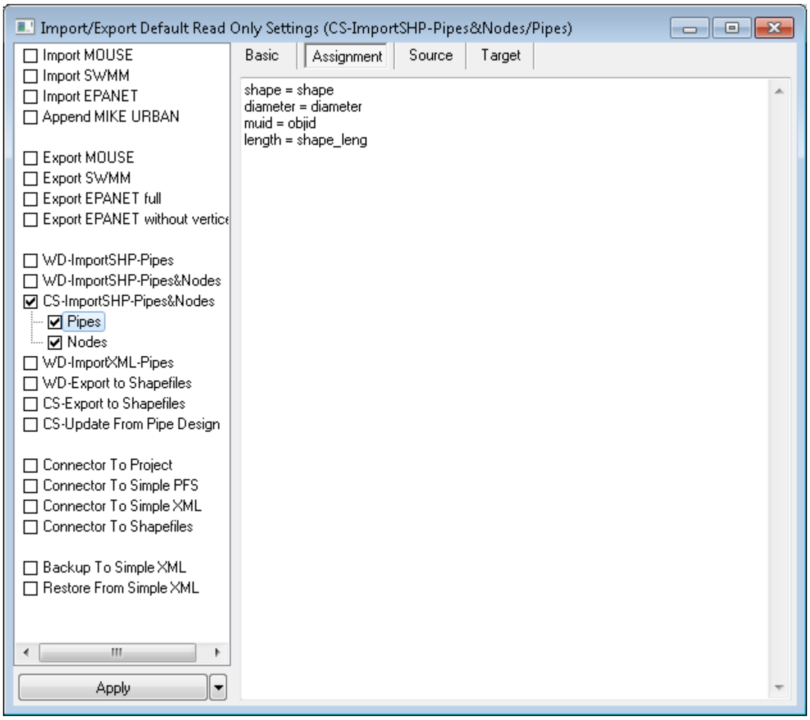


Figure 2.32 Assign Database Attributes for Pipes dialog box

7. 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“

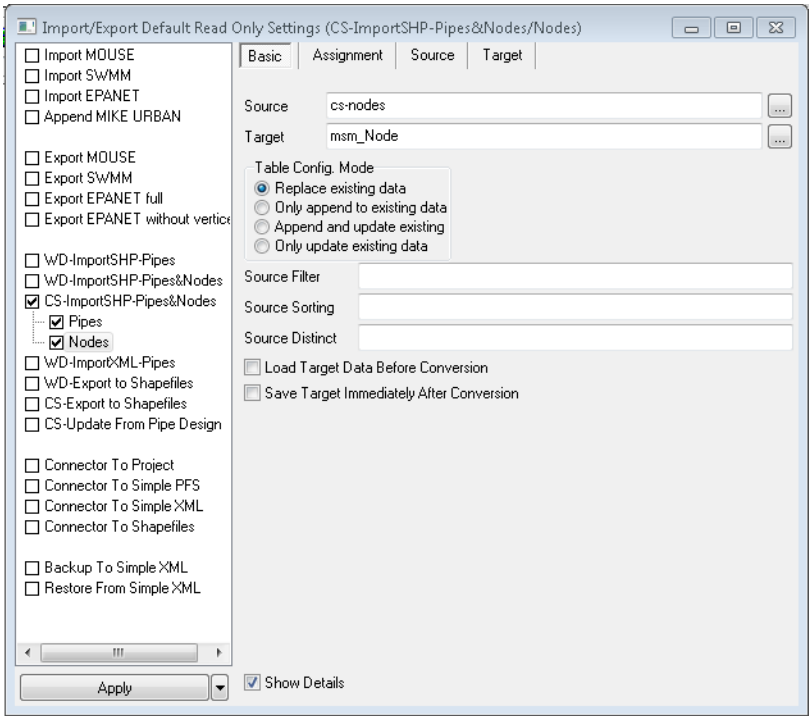


Figure 2.33 Basic settings for Nodes dialog box

8. 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.

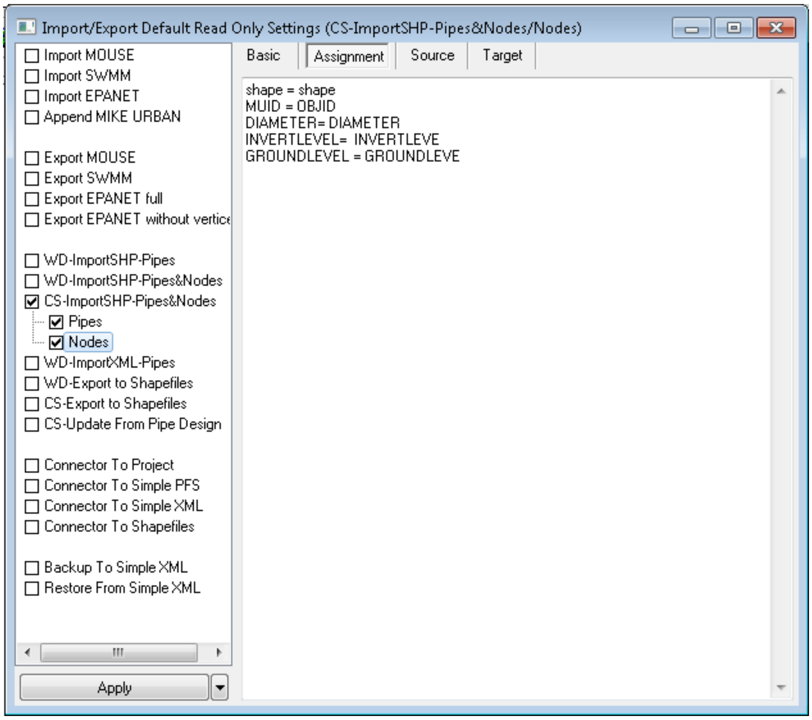
   shape = shape

   MUID = OBJID

   DIAMETER = DIAMETER

   INVERTLEVEL= INVERTLEVE

   GROUNDLEVEL = GROUNDLEVE



å

Figure 2.34 Assign Database Attributes for Nodes dialog box

9. 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.

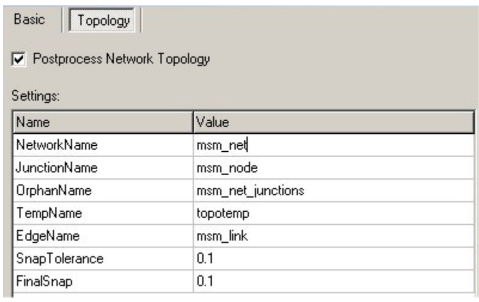


Figure 2.35 Topology settings

10. 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.

Automatic creation of nodes

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.
2. 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.
3. 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“.

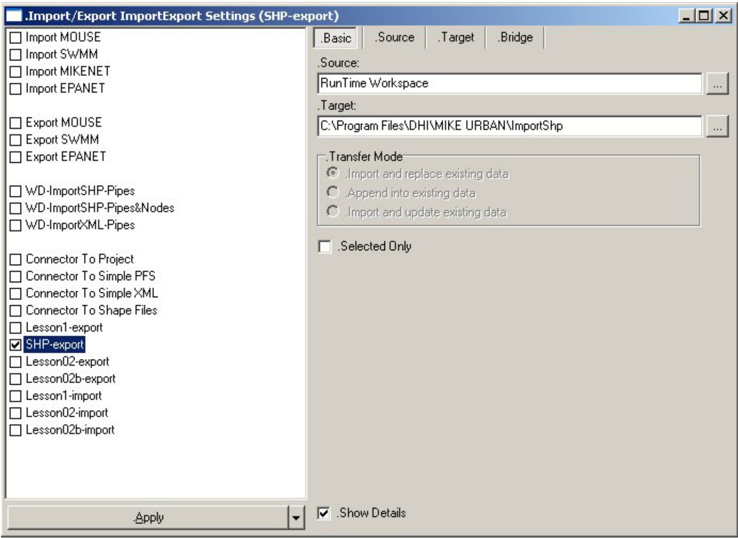


Figure 2.37 Import and Export dialog box

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.

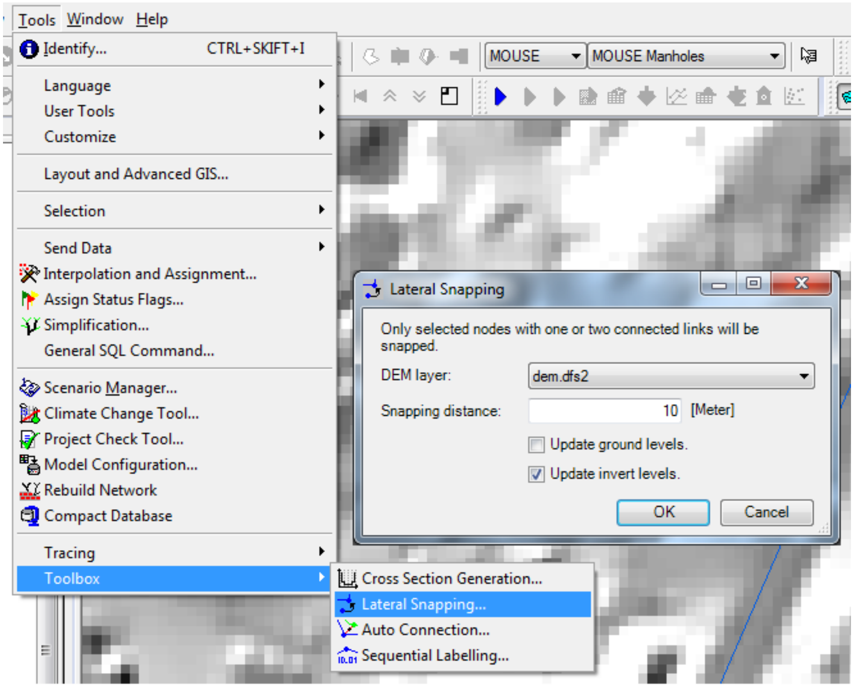
The data for this tutorial can be found in the folder: Examples\ModelManager\CSToolbox.

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.



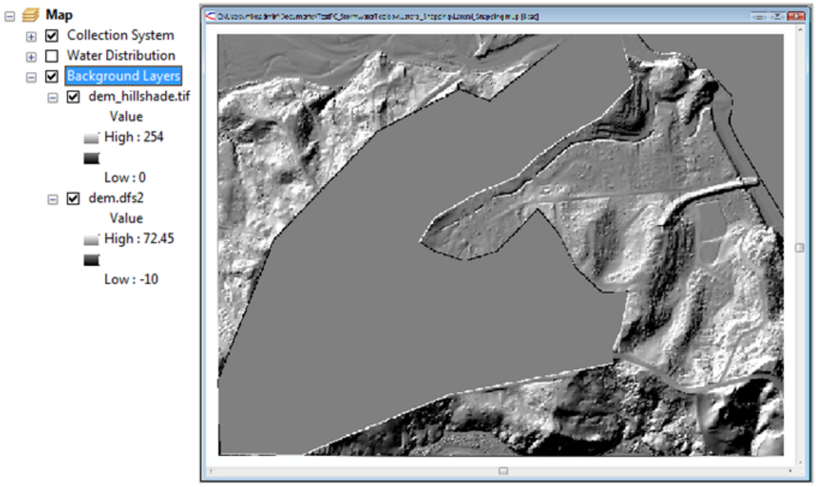
Lateral Snapping: Step-by-Step Procedure

Data:

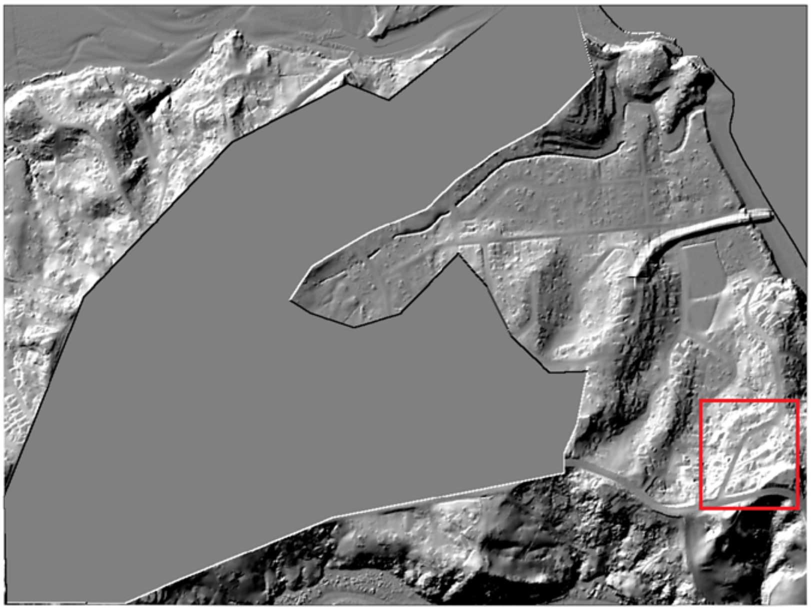
- DEM (Digital Elevation Model) grid file (dem.dfs2)
- Hillshade raster for the DEM or any map that may be used for locating overland flow paths (dem_hillshade.tif)

Procedure:

1. Start MIKE URBAN and create a new MOUSE project (leave all the options per default).
2. 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.”

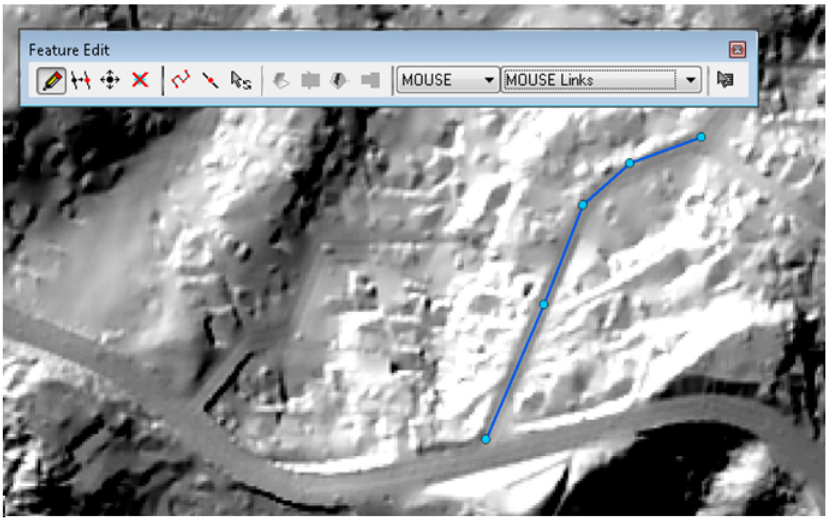


3. Zoom in to the lower-right area of the map where a road could be recognized from the “dem_hillshade.tif” raster as shown in the figure.

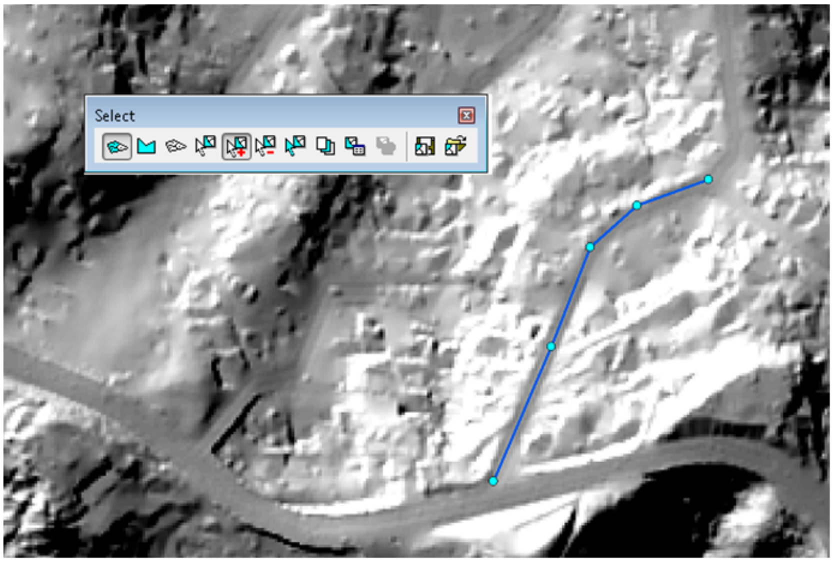


4. Start an editing session and select MOUSE Links as the Active Layer for editing.

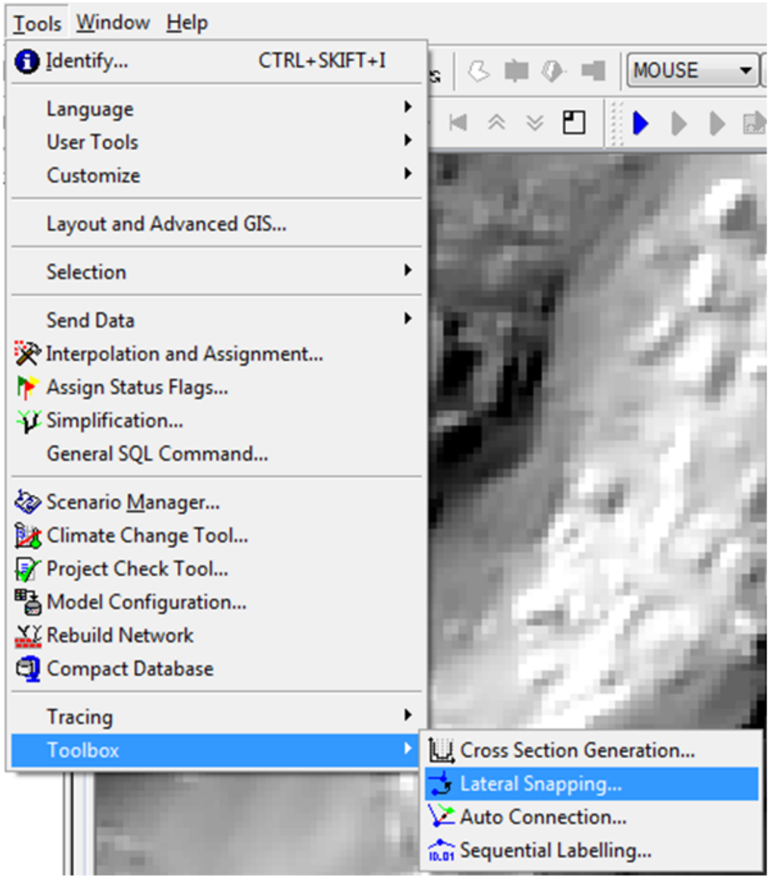
5. Create link features. Place the new links along the road feature seen in the “dem_hillshade.tif” raster. Double-click on the map in order to create intermediate nodes between interconnected link segments.



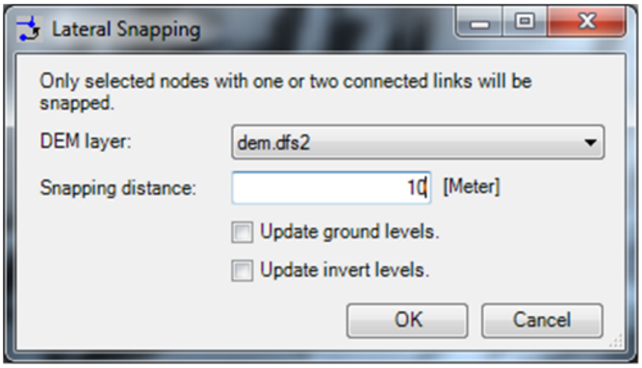
6. To begin using the Lateral Snapping tool, select the newly created nodes that were generated with the new link features. Note that only the nodes should be selected.



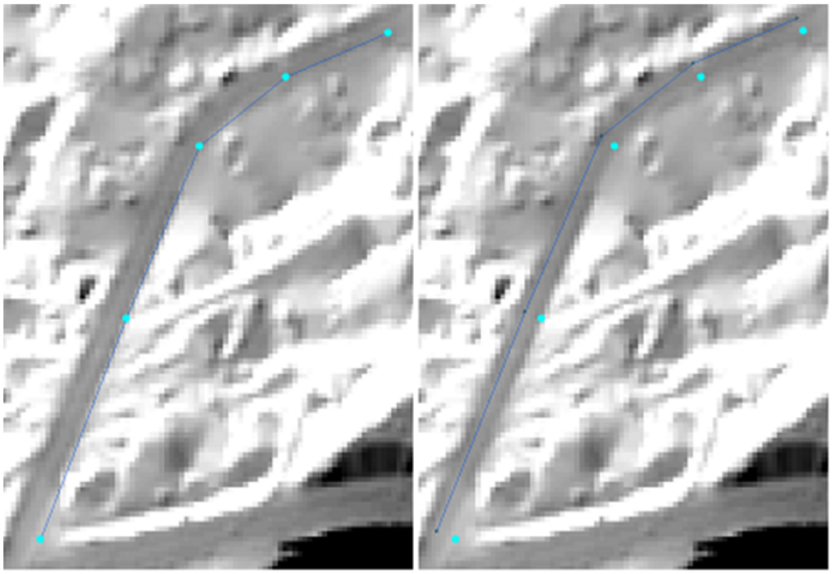
7. Under "Tools | Toolbox", select Lateral Snapping.



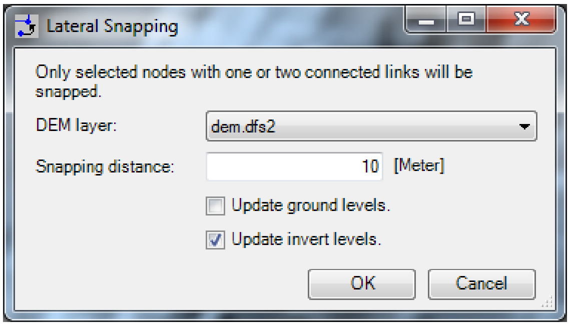
8. Select the “dem.dfs2” grid as the “DEM layer” in the dialog (and not the “dem_hillshade.tif” grid as this grid does not represent elevations). Specify the “Snapping distance” (i.e. 10 m). Press OK. Note that the settings used in the tool are stored in the file “StormToolSettings.xml” that can be found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



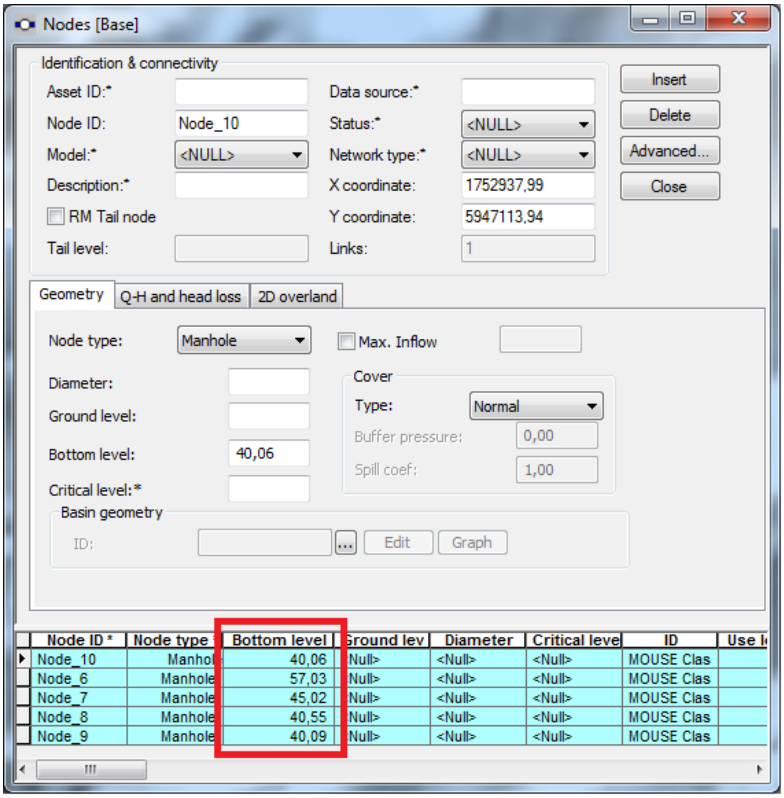
9. The selected nodes are then snapped to the lowest point in the DEM within a lateral distance of 10 m.



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.



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.

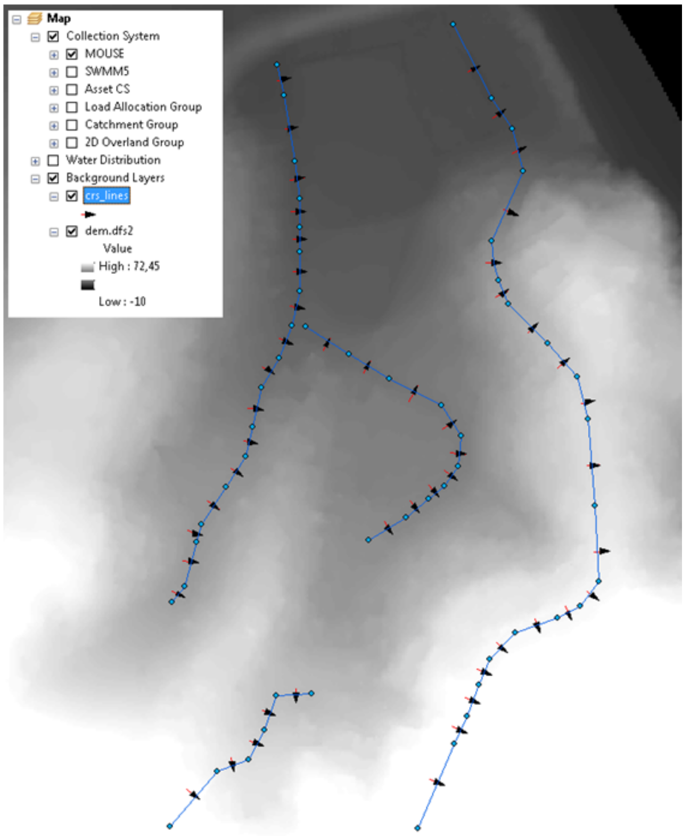
Cross-section Generation: Step-by-Step Procedure

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)

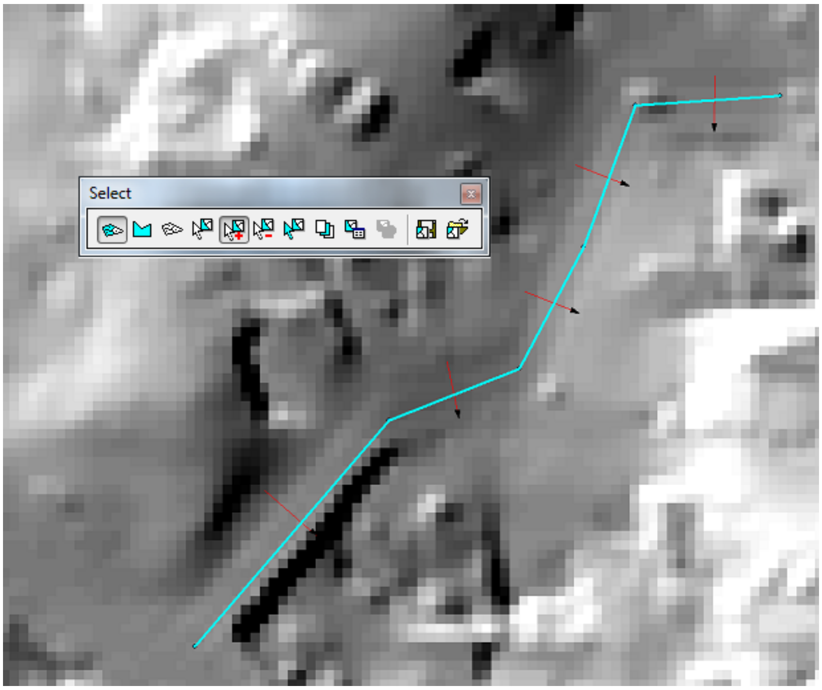
Procedure:

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.



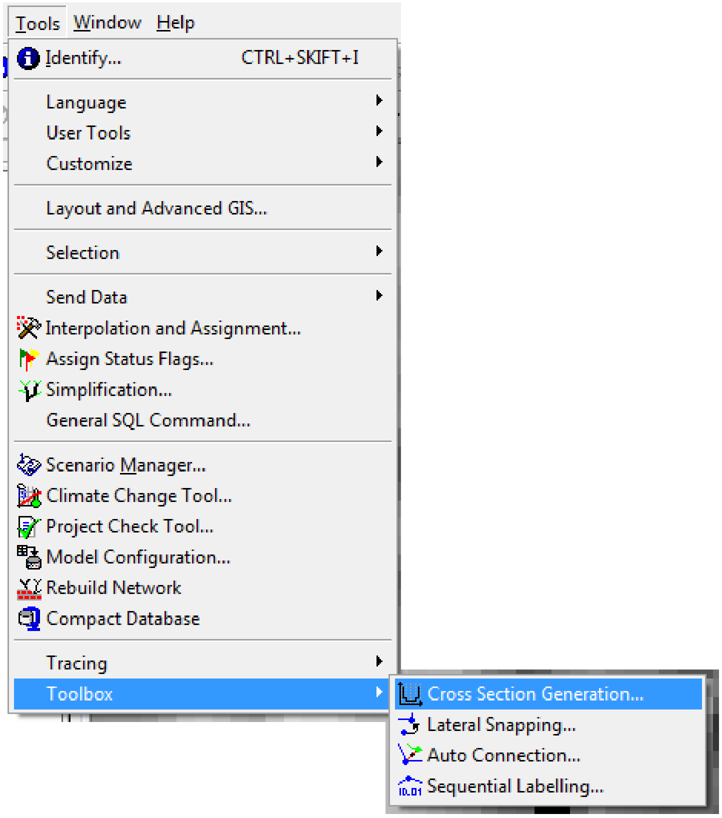
The cross section lines “crs_lines.shp” are polyline features that were created in ArcMap. They represent estimated alignments and widths of cross-sections for overland channel segments.

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

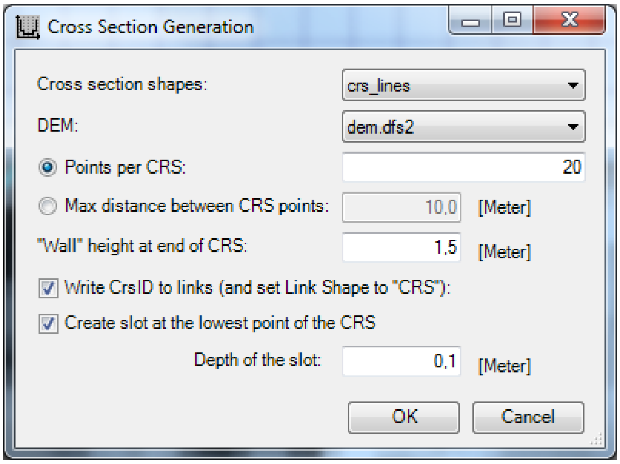


Note that the tool will generate CRSs for the selected links. If no particular links are selected, the tool will generate CRSs for all links intersecting existing cross section lines.

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



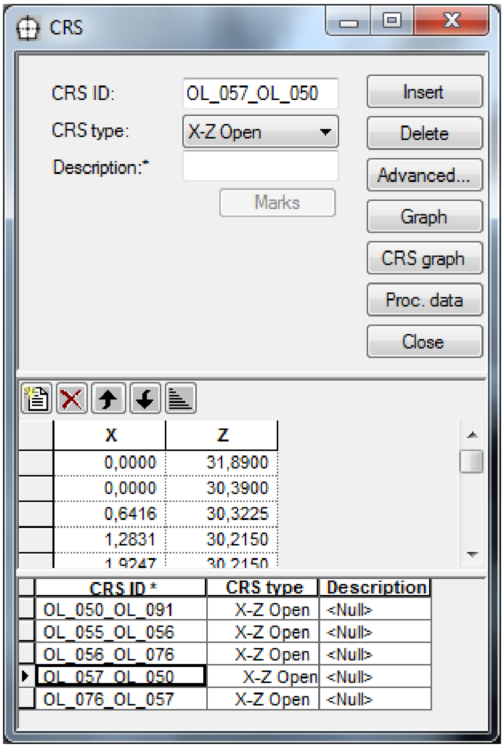
5. Specify the parameters for the Cross Section Generation tool. Choose the cross section alignment layer “crs_lines” for “Cross section shapes”. Choose “dem.dfs2” for “DEM”. Specify the number of sample points at which the cross section will be extracted (i.e. 20). Alternatively, you may specify the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

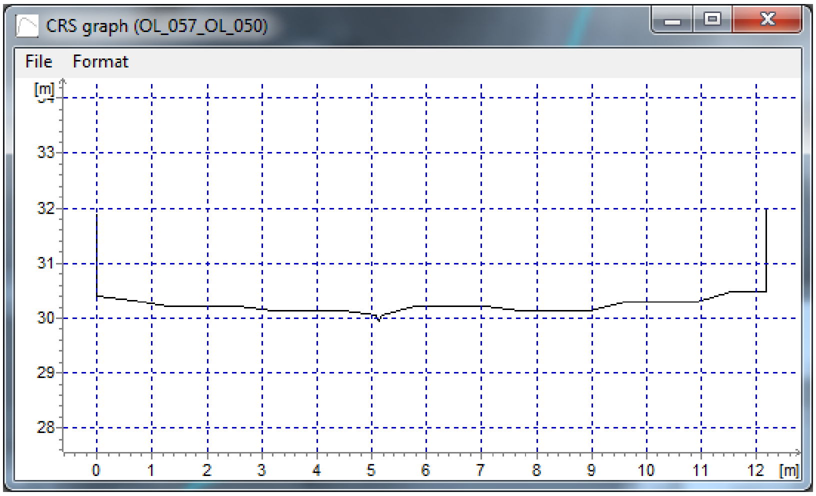


Add side walls to the CRS being generated through the “Wall height at the end of CRS” parameter. This will avoid a MIKE URBAN network simulation to stop if the water level in an open channel rises above the defined CRS height. Use a side wall of 1 to 2 meters for an overland flow path such as a road, 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.

6. 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.


CRS
CRS ID:
OL_057_OL_050
CRS type:
X-Z Open
Description:*
Marks
Insert
Delete
Advanced...
Graph
CRS graph
Proc. data
Close
X
Z
0,0000
31,8900
0,0000
30,3900
0,6416
30,3225
1,2831
30,2150
CRS ID *
CRS type
Description
OL_050_OL_091
X-Z Open
<Null>
OL_055_OL_056
X-Z Open
<Null>
OL_056_OL_076
X-Z Open
<Null>
OL_057_OL_050
X-Z Open
<Null>
OL_076_OL_057
X-Z Open
<Null>


CRS graph (OL_057_OL_050)
File
Format
[m]
33
32
31
30
29
28
0
1
2
3
4
5
6
7
8
9
10
11
12
[m]

2.3.4 Auto Connection Tool

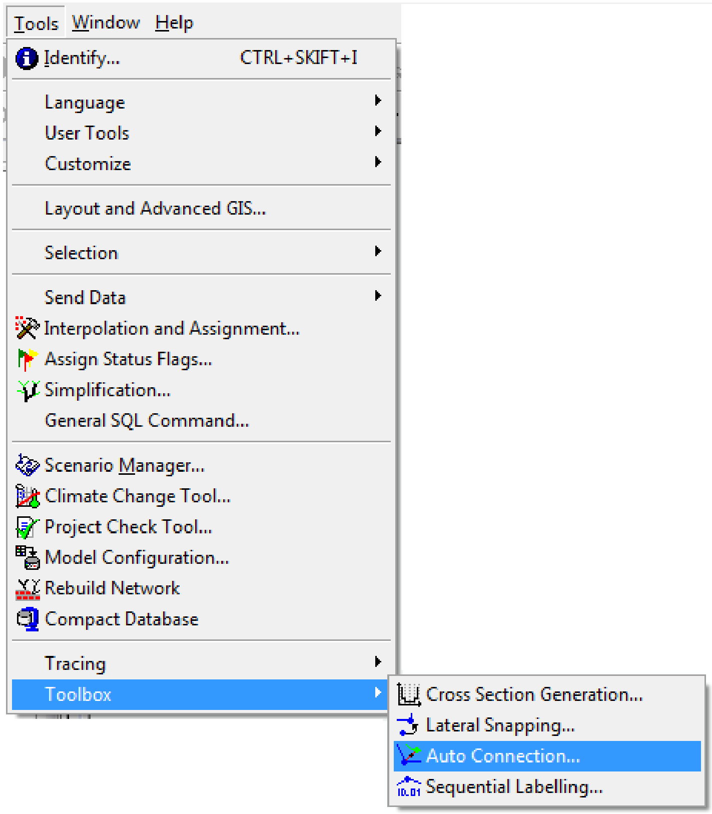
Overview

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”.



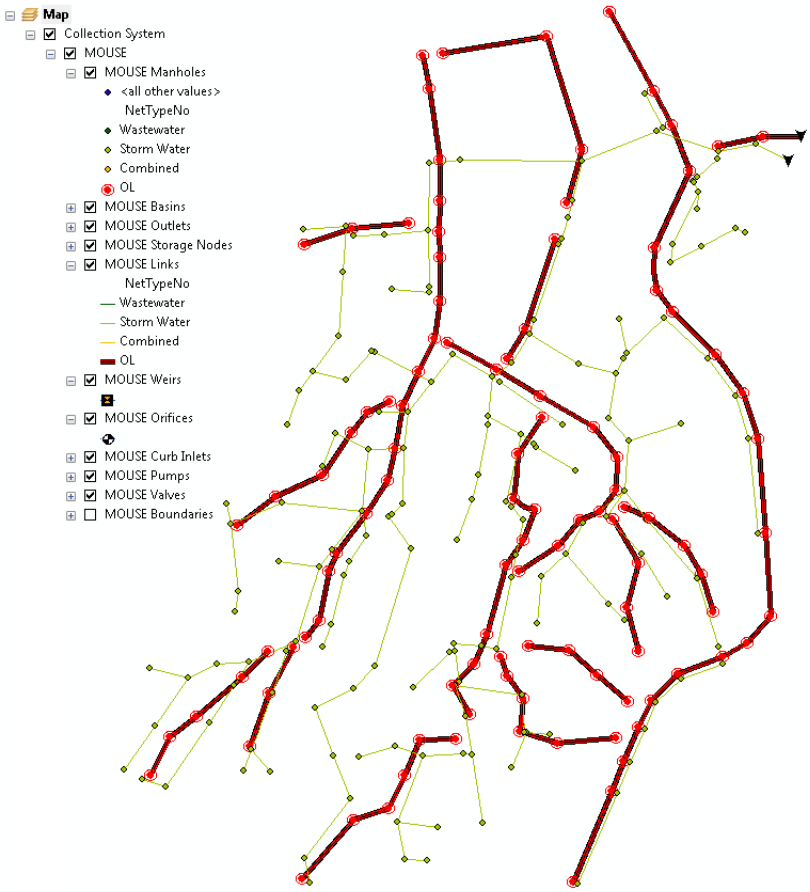
Auto Connection: Step-by-Step Procedure

Data:

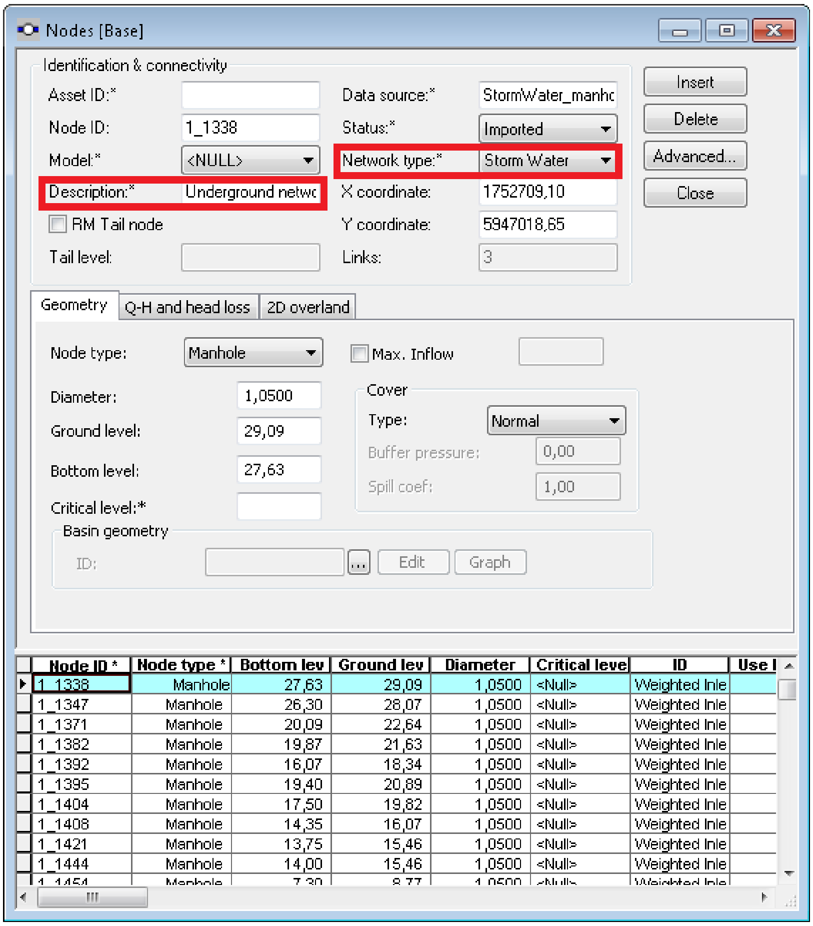
- A MIKE URBAN project file with underground and overland flow network data (Auto_Connection.mup)

Procedure:

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

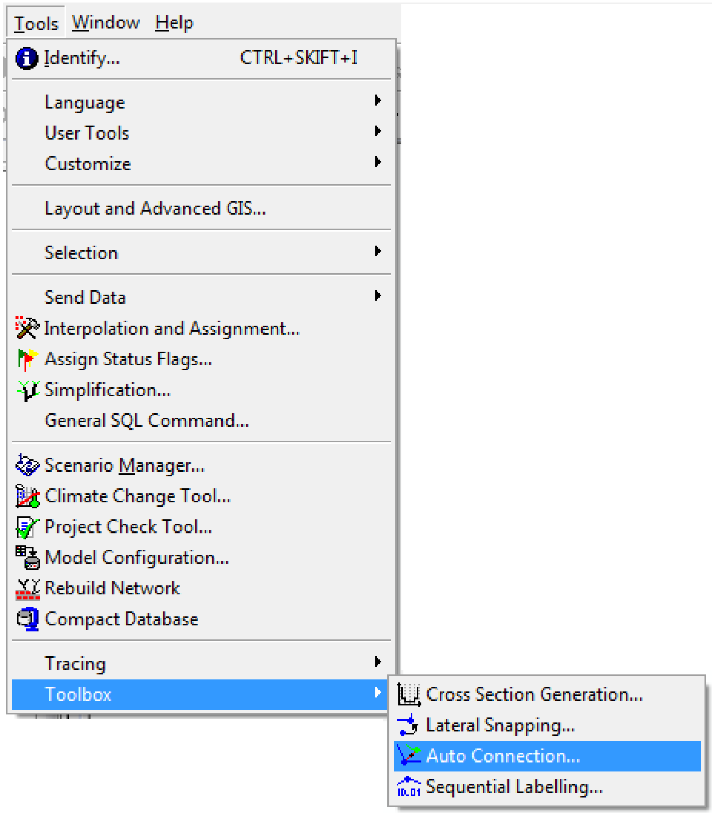


Information about the network type can be found in the "Network type" and "Description" parameters of nodes and links.

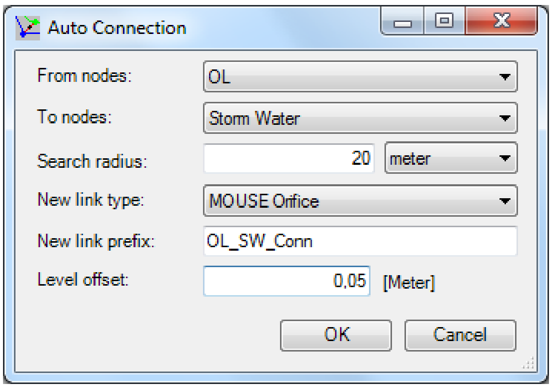


Overland network nodes and links have been assigned a Network type of “OL” while underground network elements have been assigned a Network type of “Storm Water”.

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

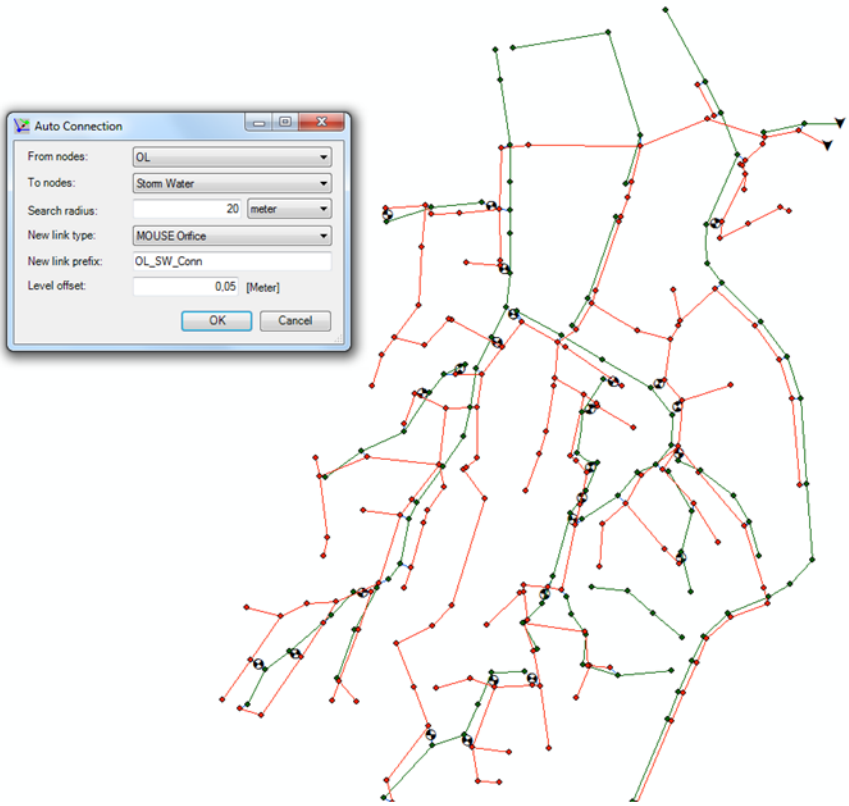


3. Specify the settings for the Auto Connection tool. Select “OL” for the “From nodes” and “Storm Water” for the “To nodes” to generate connections going from the overland to the underground network.



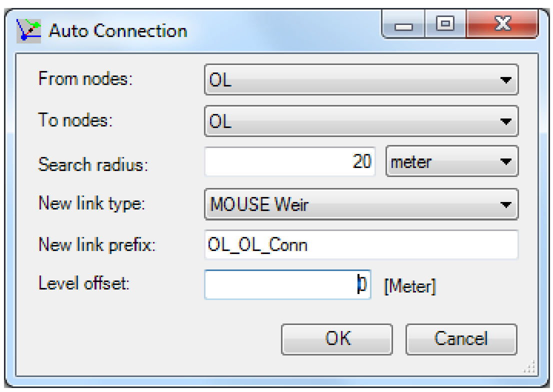
Setting the network type for the “From nodes” and “To nodes” gives more control over the setting-up of connections and also defines the direction of positive flow (From “From nodes” to “To nodes”). For connections from the overland to the underground network the positive flow direction should always 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.
6. 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.



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.

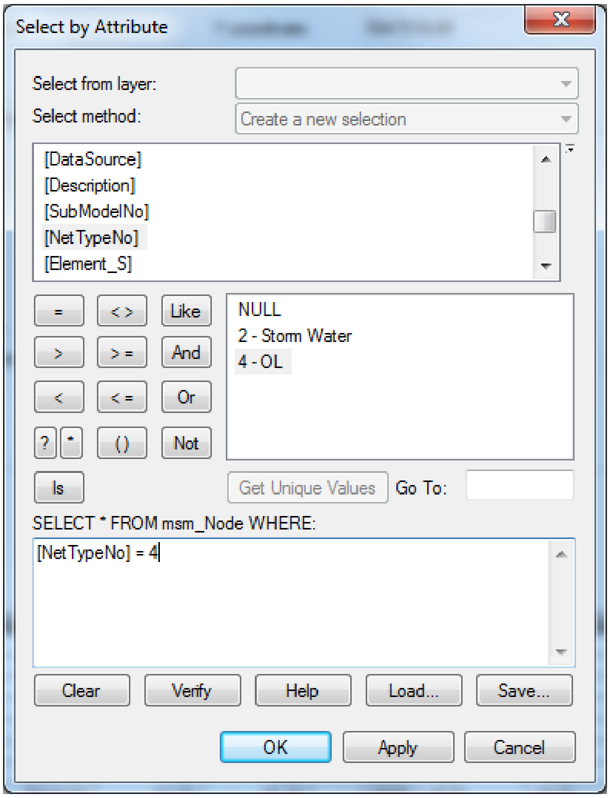
Sequential Labelling: Step-by-Step Procedure

Data:

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

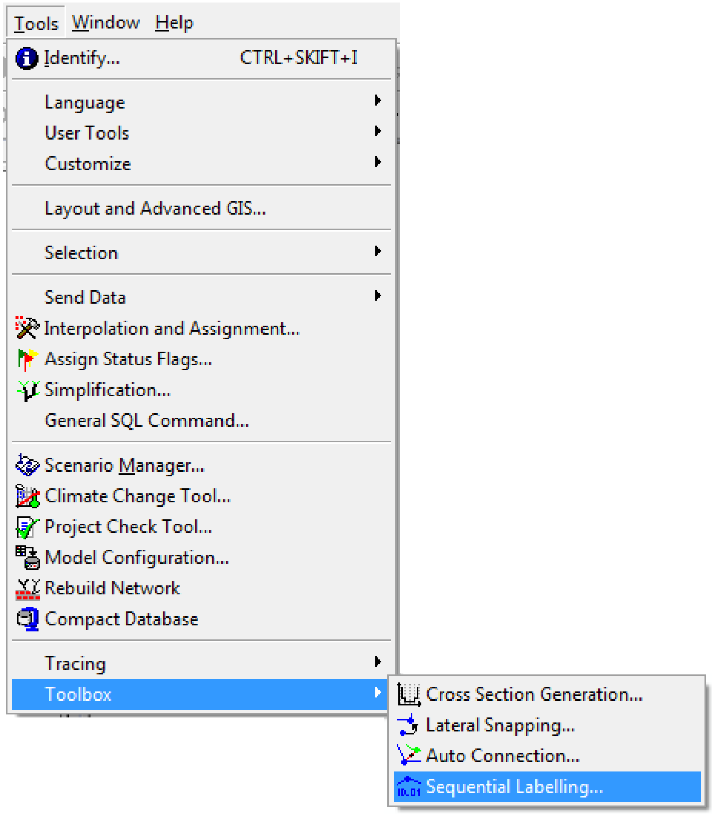
Procedure:

1. Load the MIKE URBAN project Sequential_Labelling.mup.
2. Start an editing session.
3. 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".

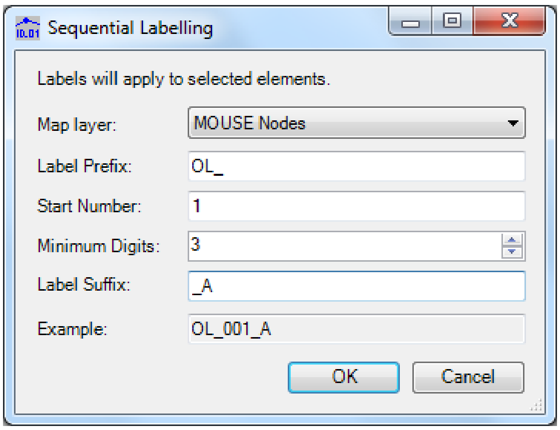


The Sequential Labelling tool only works for selected items.

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

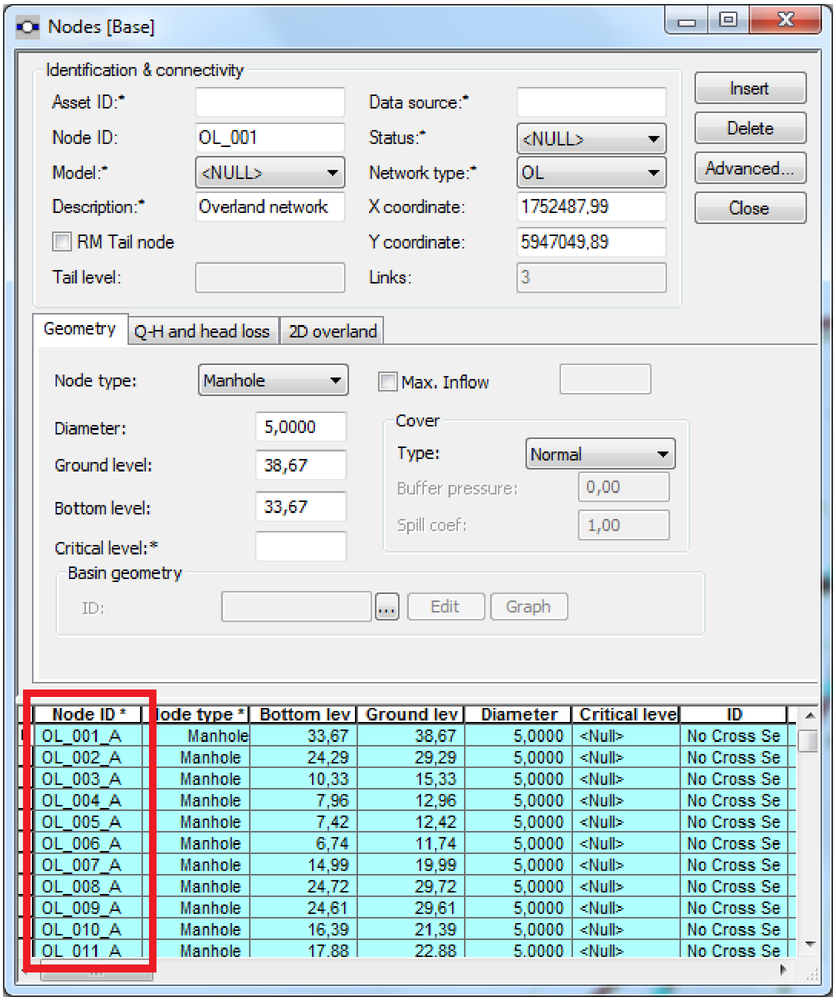


5. Specify the settings for the Sequential Labelling tool. Specify a label prefix (e.g. OL_) that will be descriptive for the selected items. Set a start number for the sequential numbering as well as a minimum number of digits for the sequential numbering. A label suffix can also be specified. Check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



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.



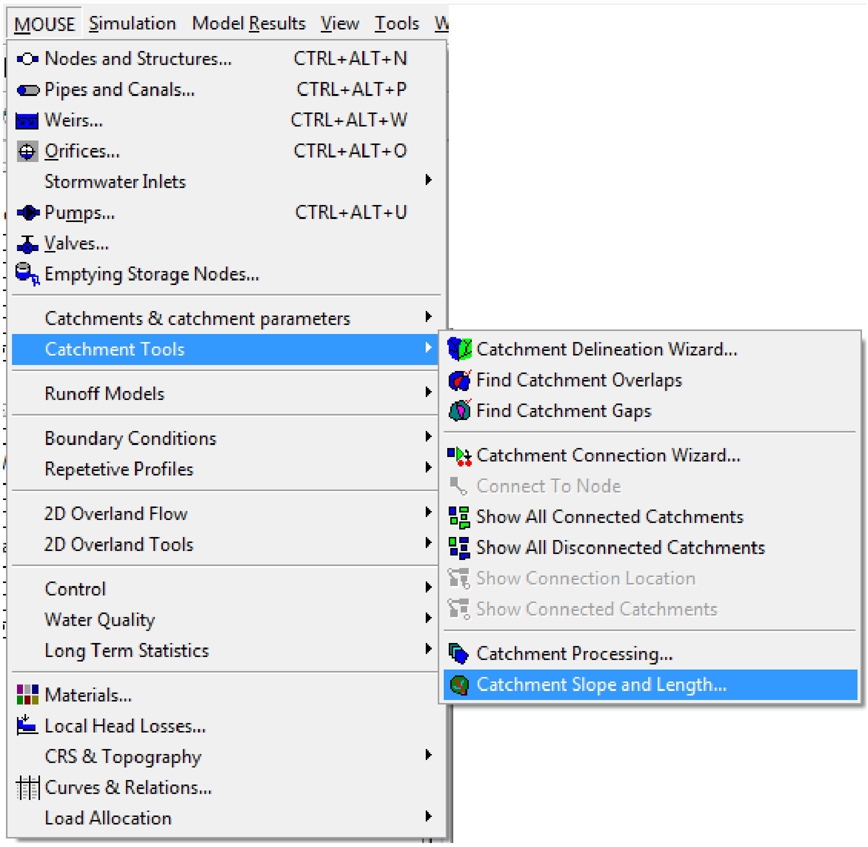
8. Follow the same produce to re-label overland flow network links.

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.



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)

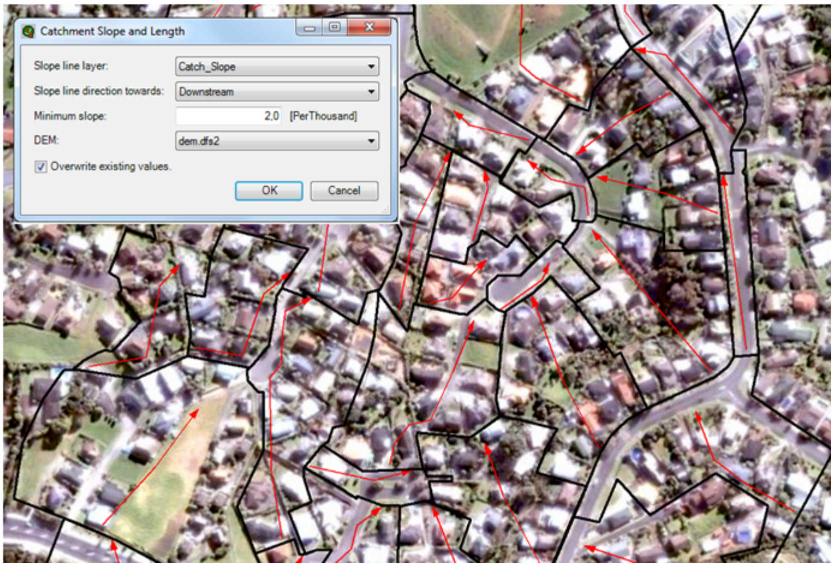
Procedure:

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

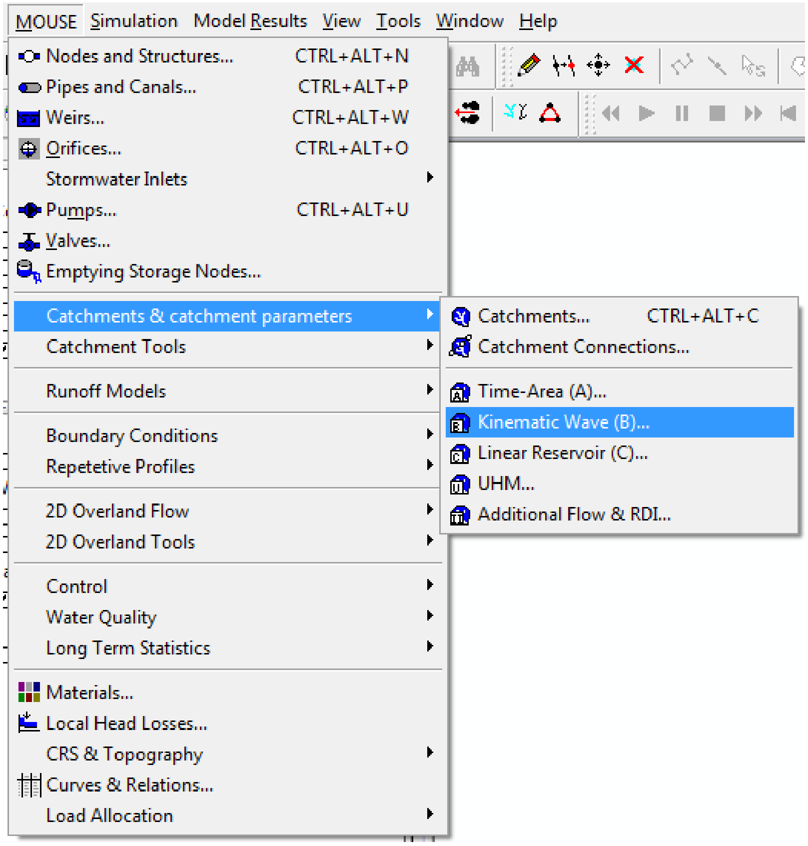
3. Obtain digitized line shapes representing typical flow paths or slope lines within catchments for your model and add it as a background layer in MIKE URBAN. In this example, the “Catch_Slope.shp” file contains the information about catchment slope lines. Add the “Catch_Slope.shp” file as a background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

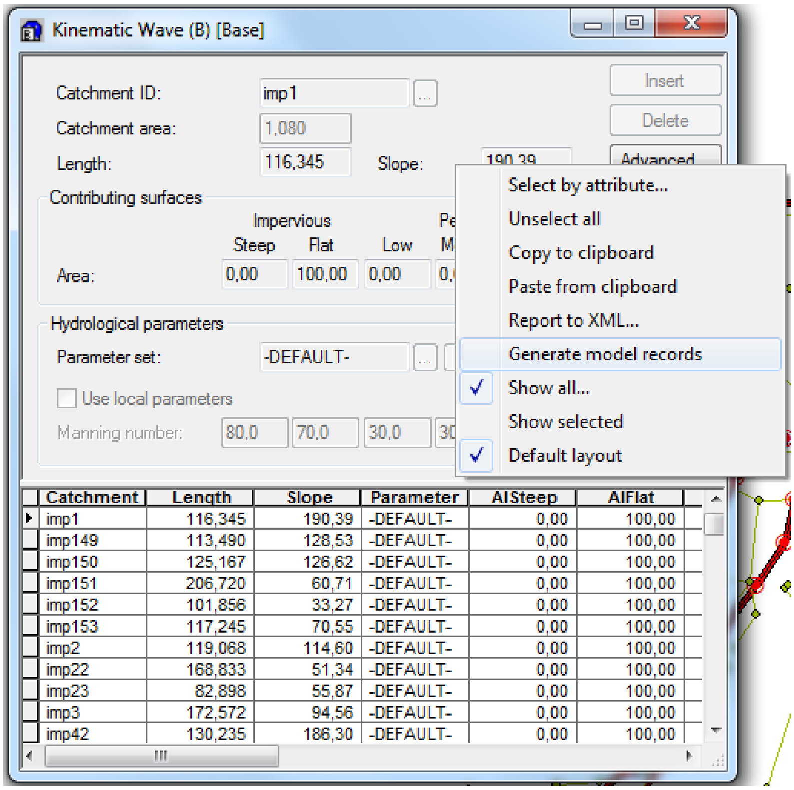
An example from a model is provided below where catchment flow lines have been drawn in a shape file (red lines). In this case the lines have been drawn from the inner catchment towards the load point.



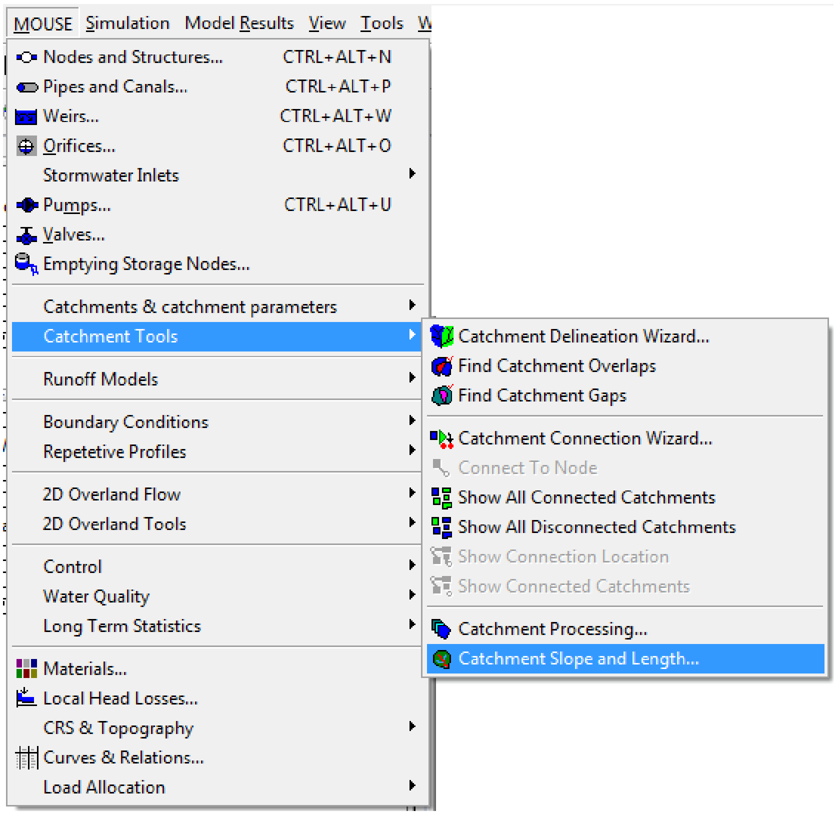
4. Start an editing session in MIKE URBAN.
5. Ensure that there are records for catchments in the Kinematic Wave (B) parameters table. View this table through “MOUSE | Catchments & catchment parameters | Kinematic Wave (B)”.



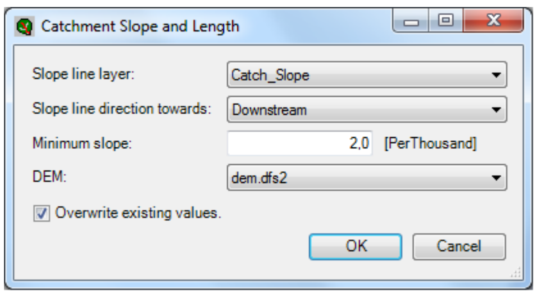
If the Kinematic Wave (B) parameters table is empty, generate records by clicking on “Advanced | Generate model records.” This will generate Kinematic Wave (B) parameter records for all the catchments listed in the Catchments table.



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.



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.

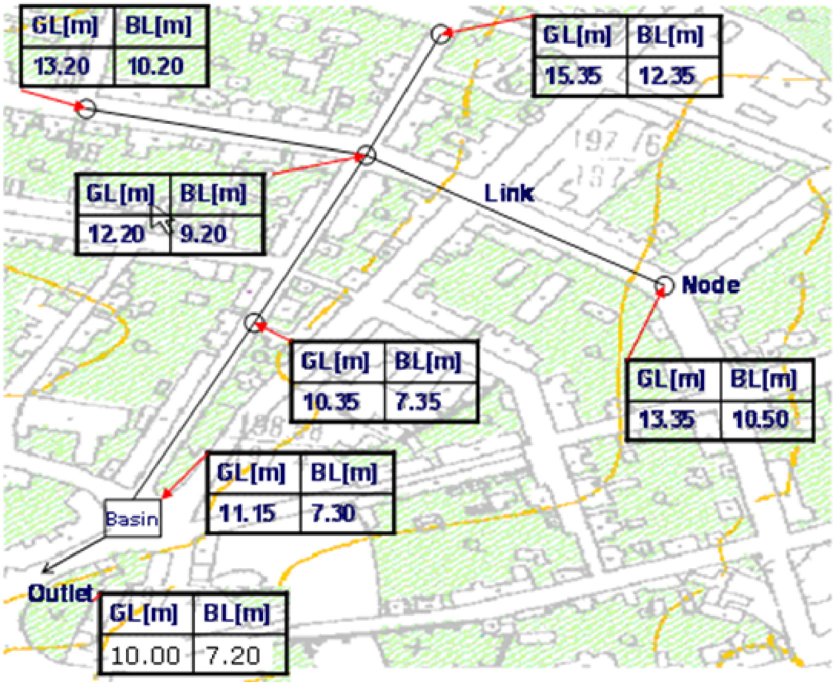


Figure 3.1 Layout of the system and level data for the nodes

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.

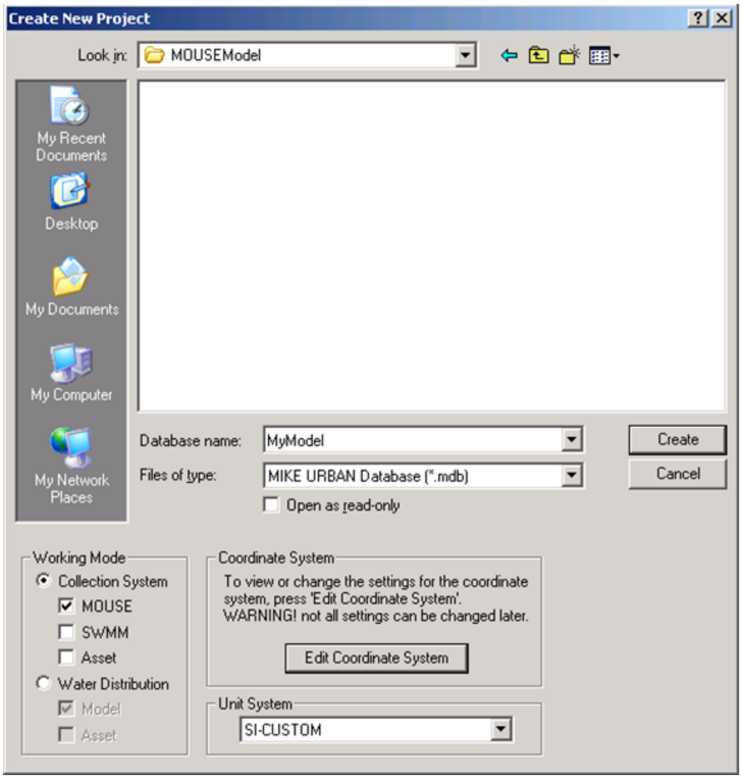
The model will consist of the following elements:

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

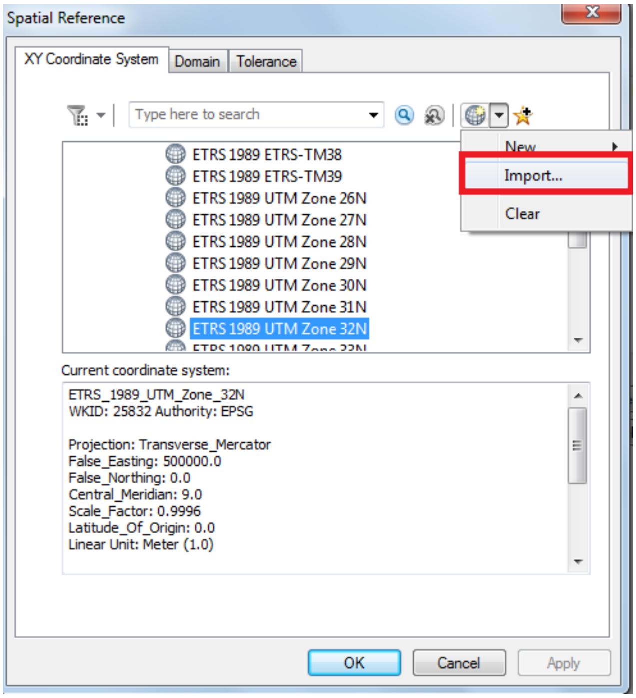
(A detailed description on the MOUSE network data model and the associated editors can be found in the MIKE URBAN CS user guide).

3.2.1 Step 1: Creating a New Project

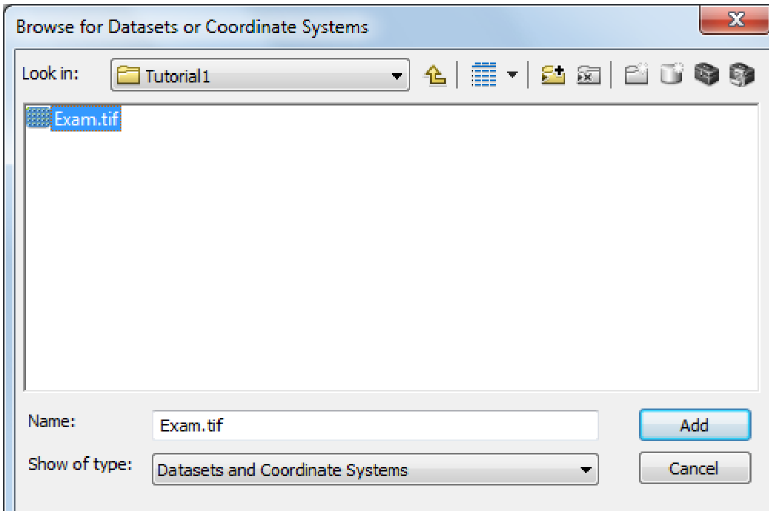
Start by creating a new project – File | New. Choose a MIKE URBAN Database of the type mdb.



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:



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

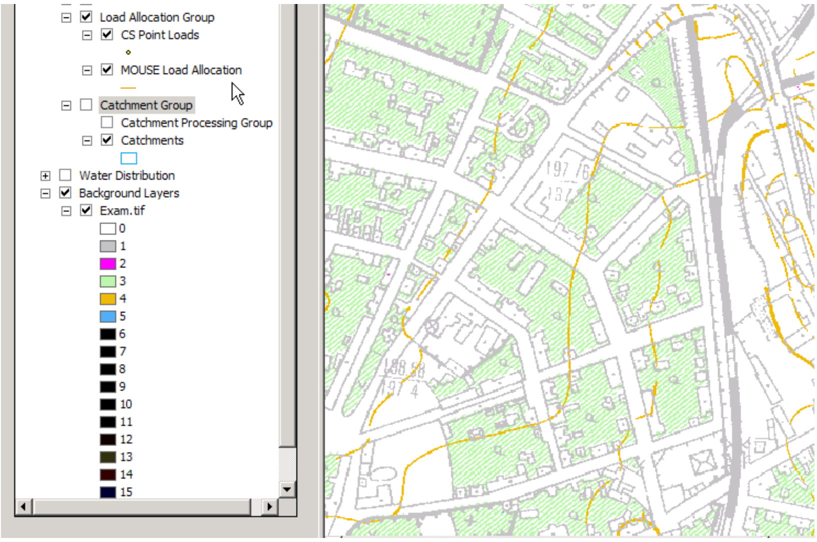
Now you are able to create the project by pressing the 'Create' button on the new project dialog.

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.

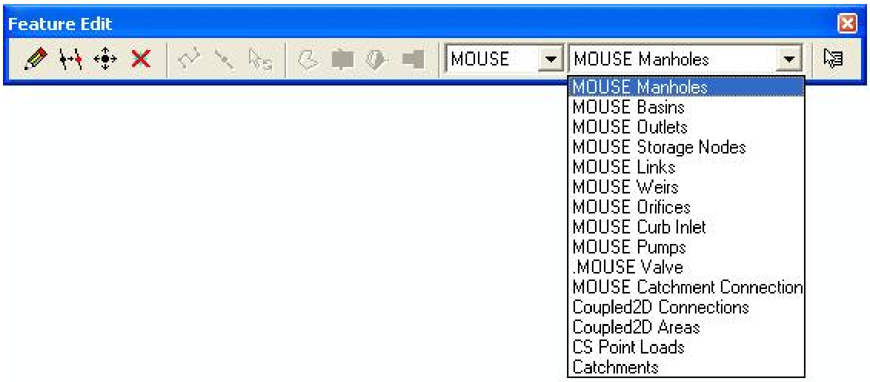
```
1.27
0.0
0.0
-1.27
-687998.0
-1056008.0
```

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.



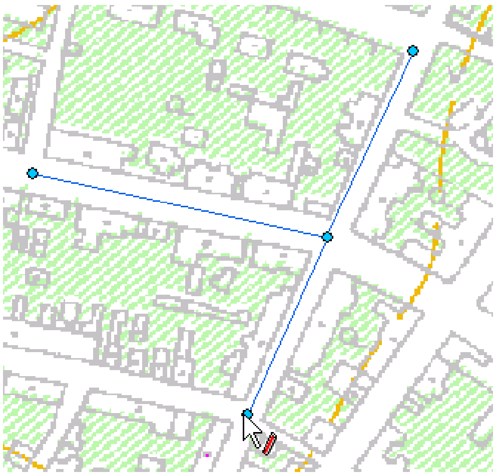
In order to quickly draw a network layout, click on the ‘Create Feature’ tool (see left margin) from the ‘Feature Edit’ toolbar. The tool will perform the action on the selected layer of the currently active network (right and left dropdown box, respectively).

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.

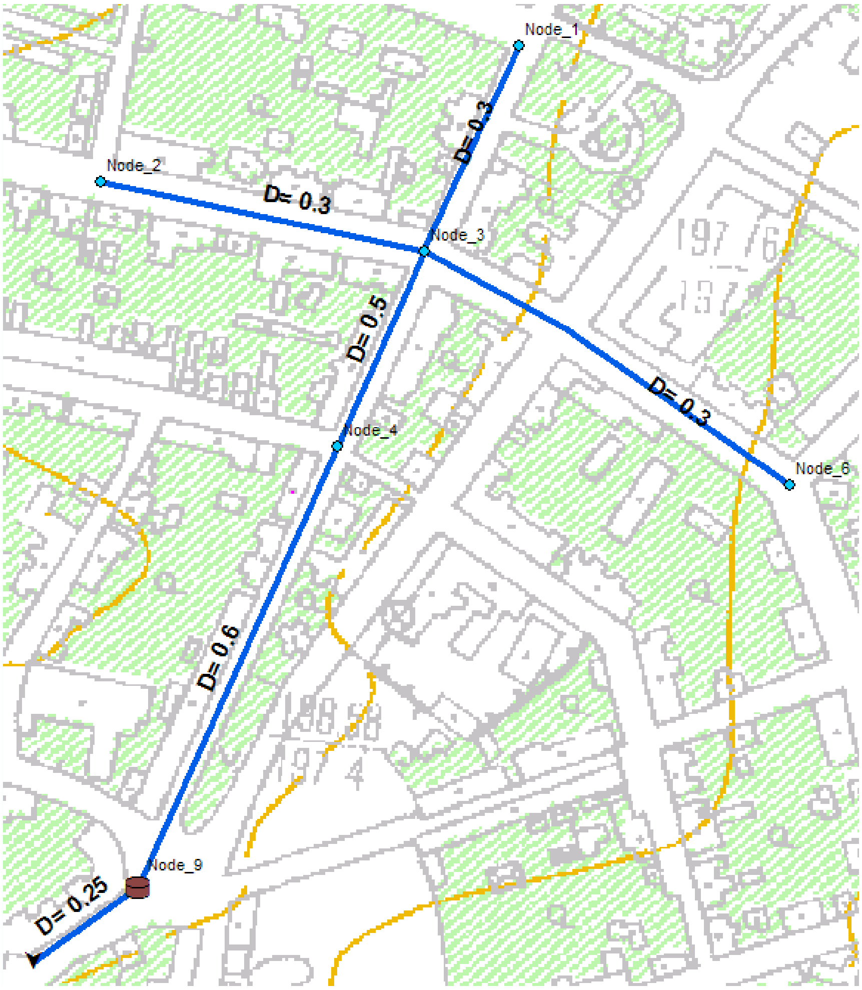
Adding links

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:



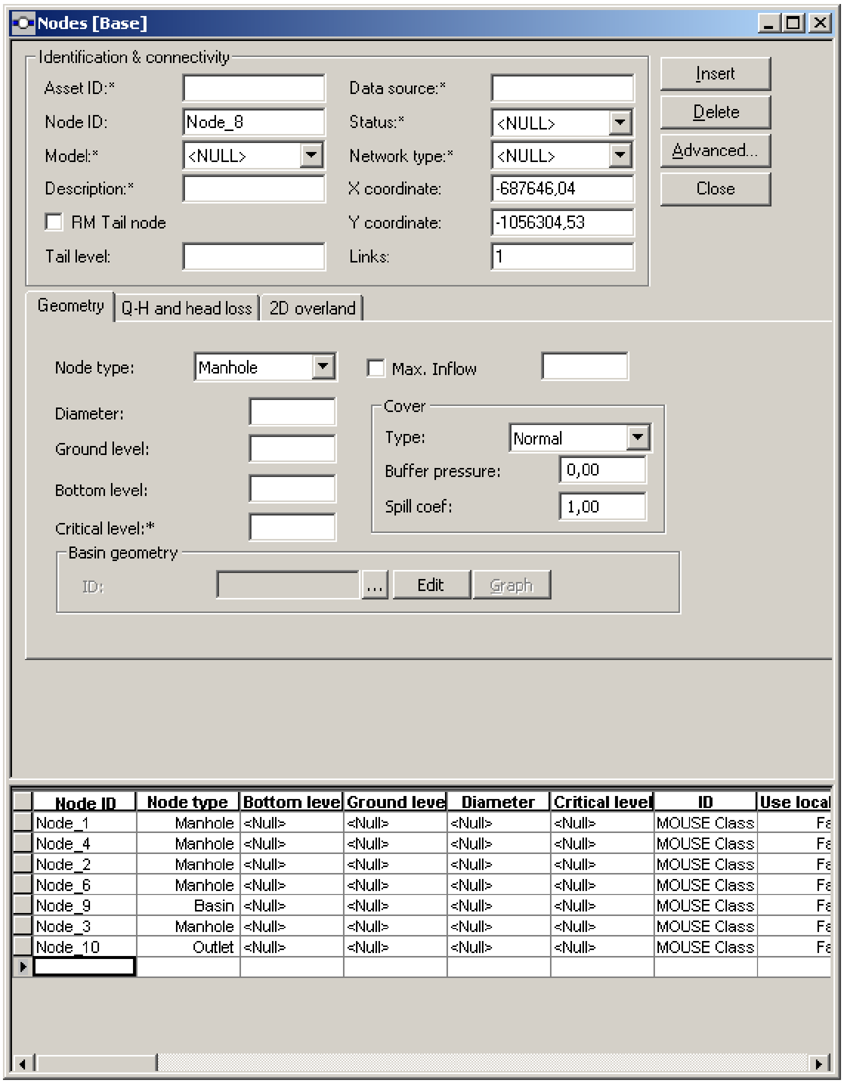
3.2.4 Step 4: Typing Attributes

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:

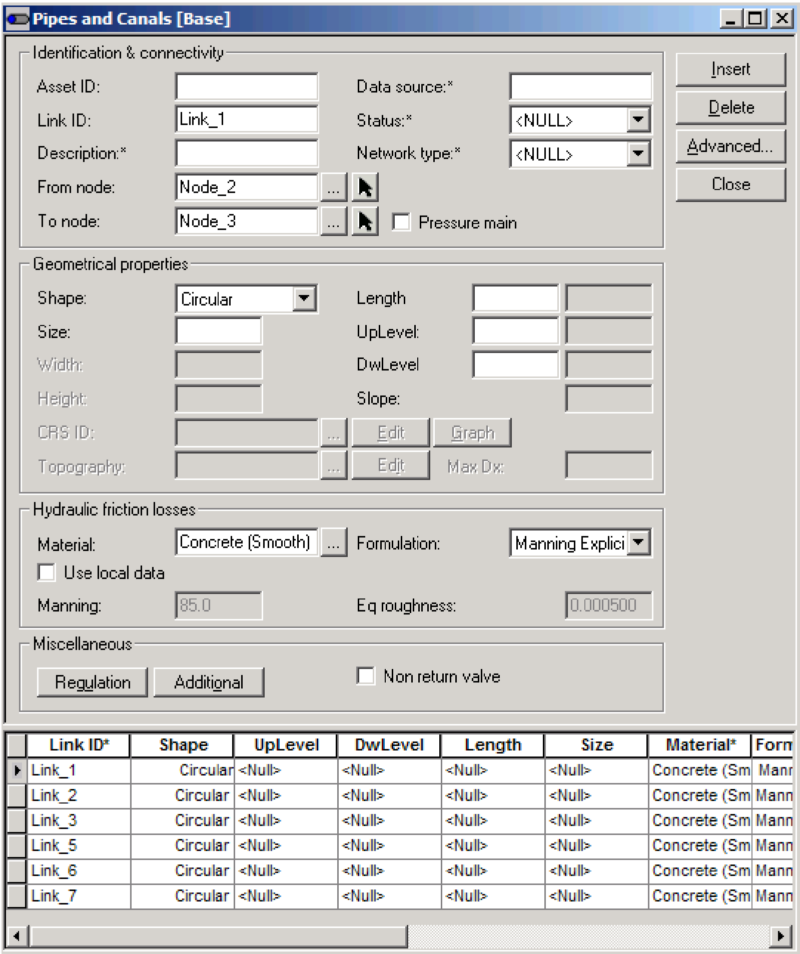
- Node type (Manhole) – automatically provided
- Diameter (1.0 m) – to be typed in.
- GL, BL (values taken from the Map seen in Figure 3.1)
- Cover Type (Normal) - default
- The Outlet head loss (MOUSE Classic - Engelund) – default



With regard to Links, you need to specify:

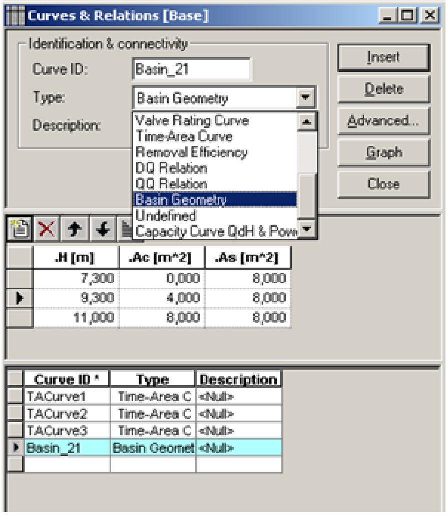
- Geometric properties such as shape (Circular) and size (0.3 m)
- Information about material (concrete), hydraulic friction losses (Manning's explicit)

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.

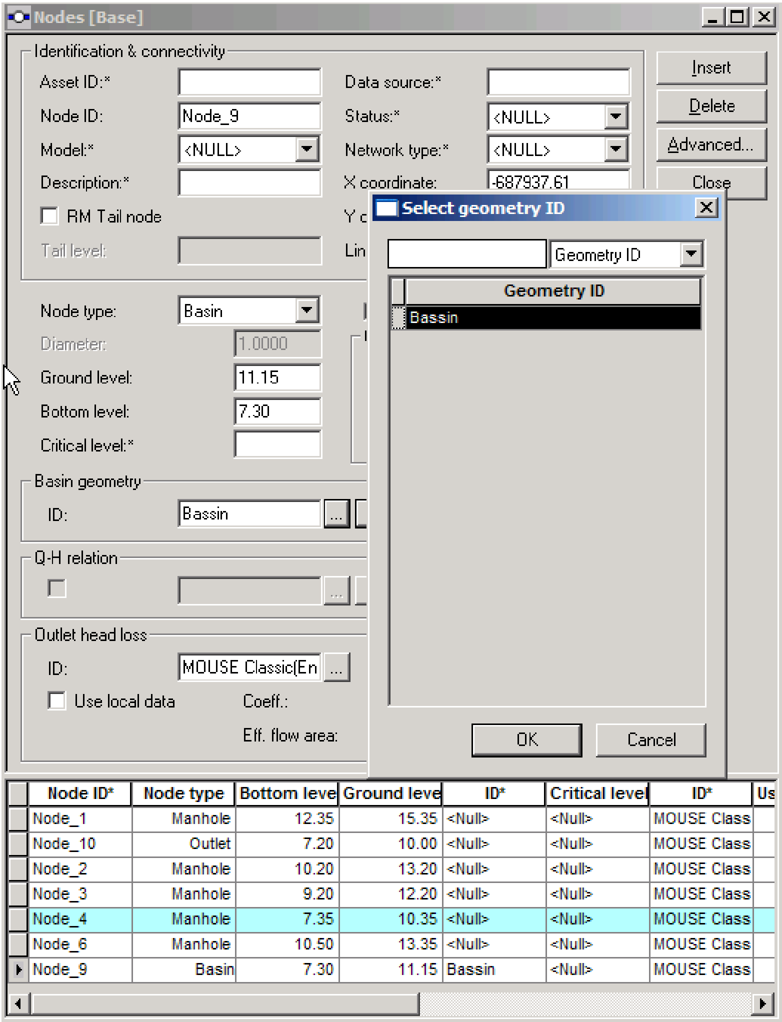


All our pipes are circular. The diameters are given in the illustration above.

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.



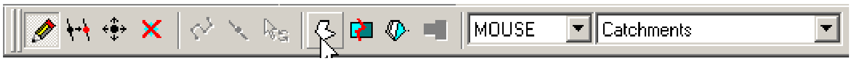
In MOUSE/'Nodes and Structures' select the created basin in the list and under 'Basin Geometry' choose the basin geometry.



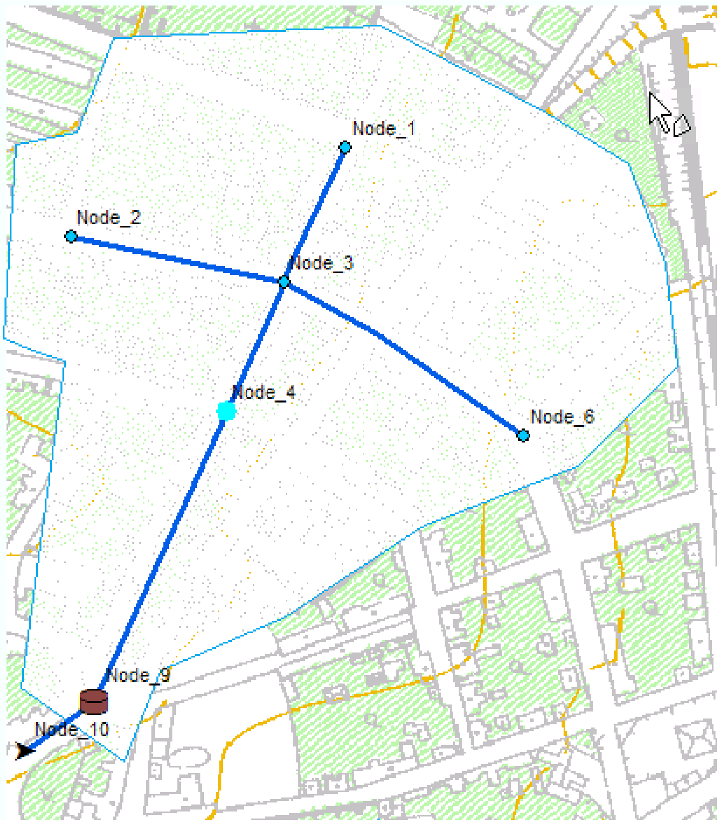
3.3 Definition of Catchments and the Hydrological Model

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 sub-catchments to be able to allocate runoff generated on the surfaces into the nodes in our network.

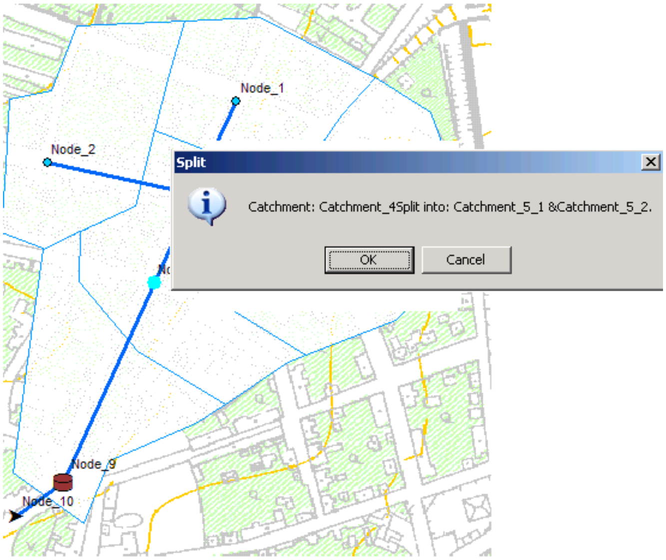
3.3.1 Step 1: Delineate the Catchments



- 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  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).



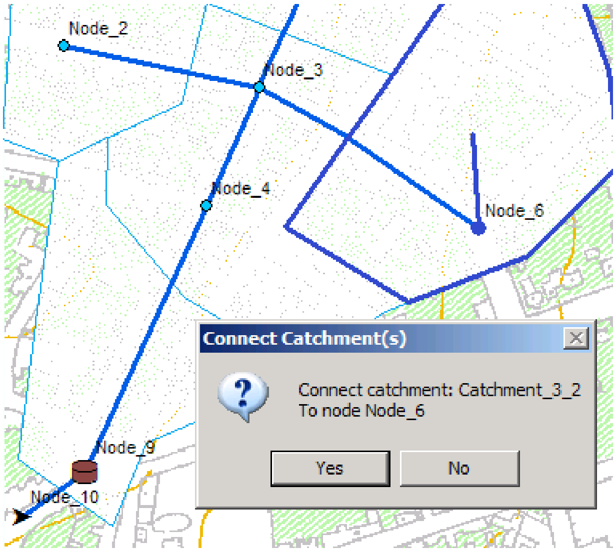
3.3.2 Step 2: Connecting Catchments to the Network

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.

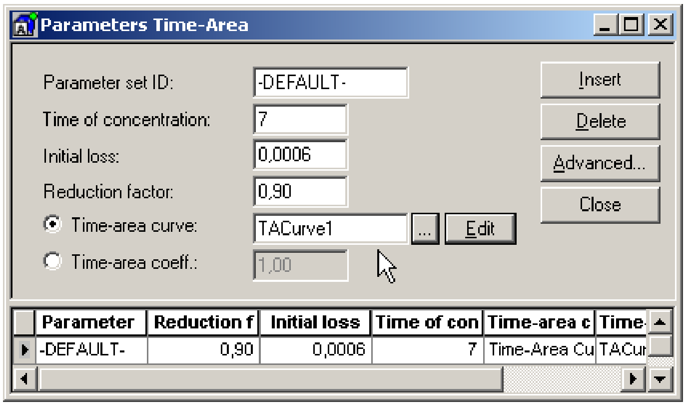
In this exercise we will use the Time Area Method (MOUSE model A) which is a simple surface runoff model with minimum data requirements.

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.

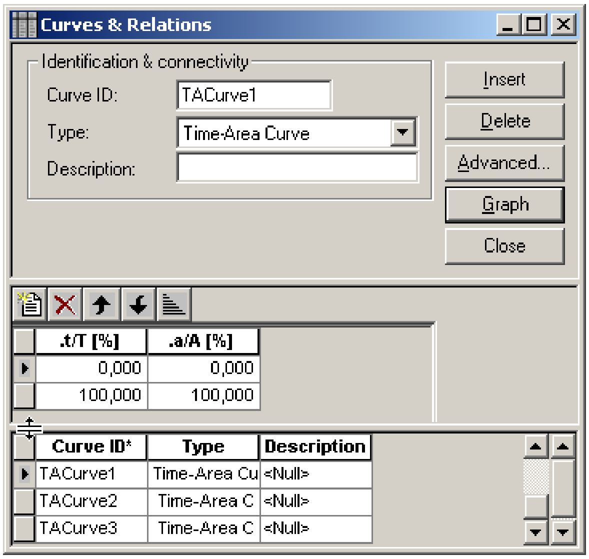
The missing parameters are:

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.



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



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

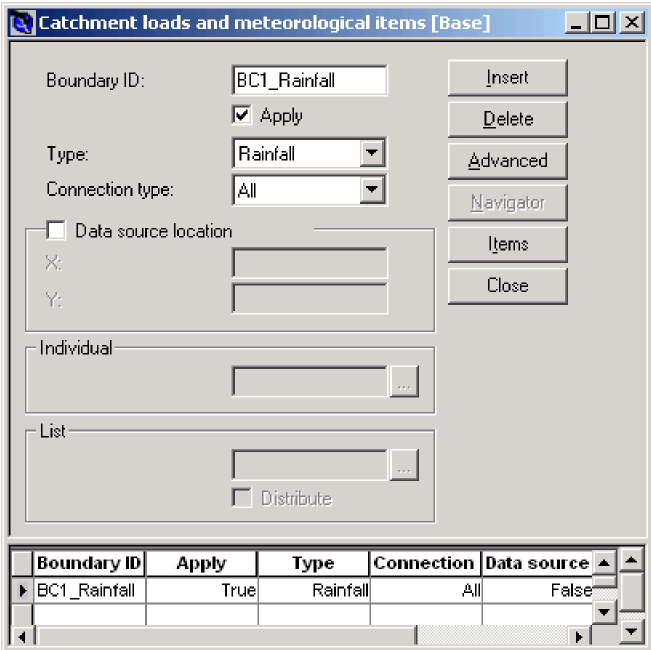
- External Water Levels

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

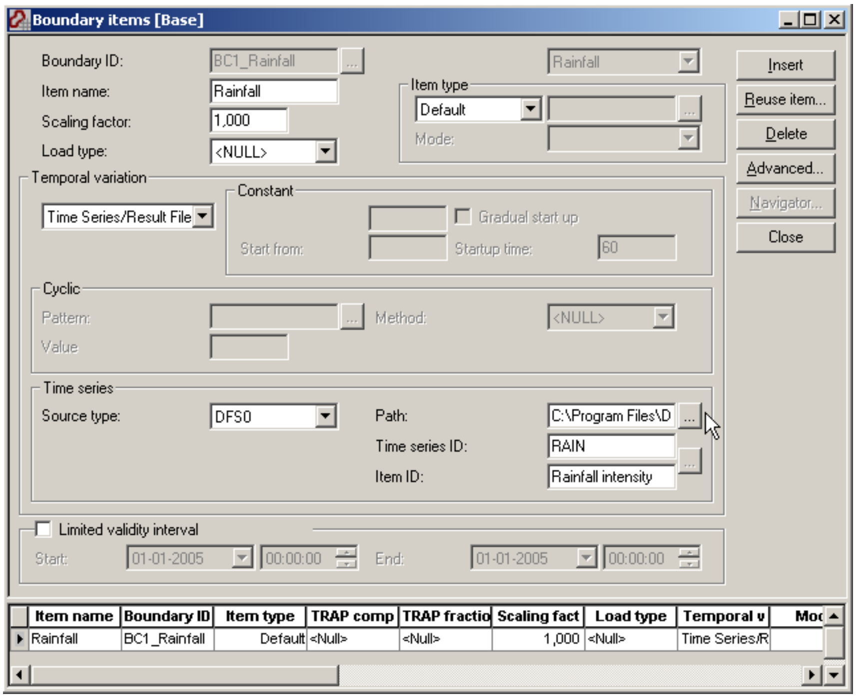


Having defined the boundary conditions, moving to the ‘Boundary items’ editor allows for a full definition of a boundary item in terms of type, temporal variation, quantity/quality and temporal validity.

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.

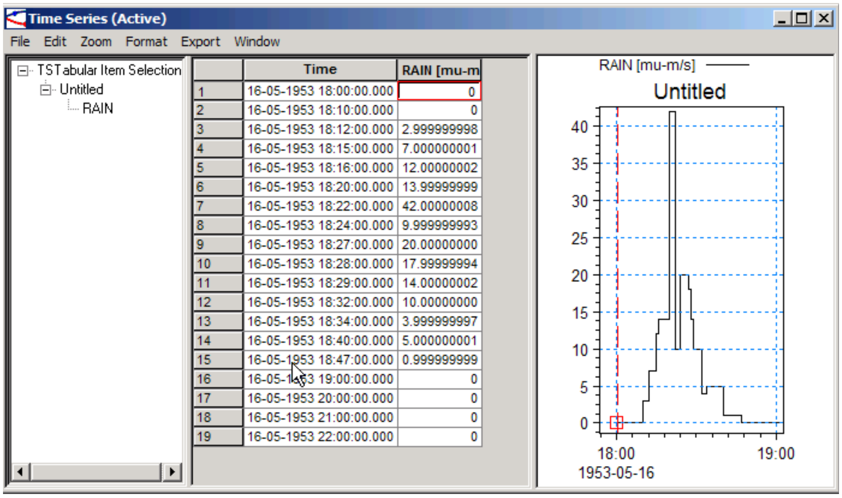
- Click on ‘Items’ in the ‘Catchment Load and…’ editor

- 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)



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.



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:

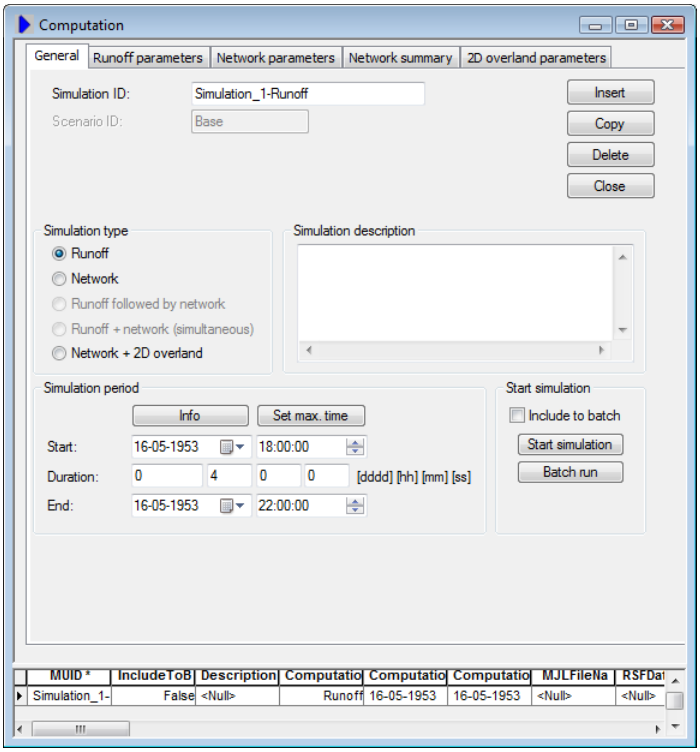
- General
- Runoff parameters
- Network Parameters
- Network Simulation Summary
- 2D overland parameters

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:

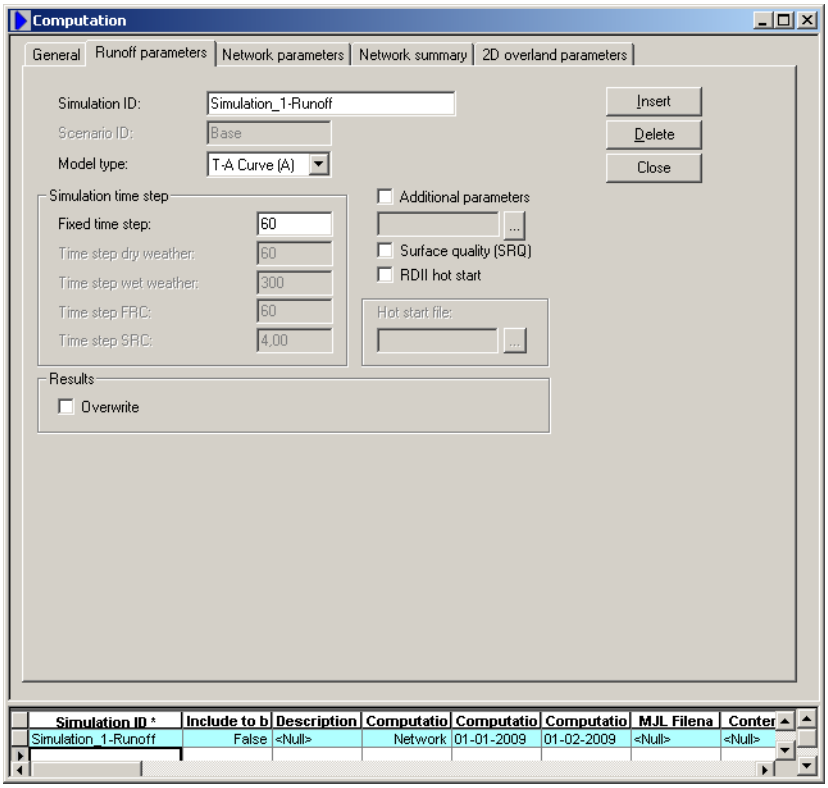
- The Name of the simulation (Simulation ID)
- The simulation period (xx xx xxxx – xx xx xxxx) – use 'Set max. time' tool
- The simulation type (Network, Runoff or 2D overland)



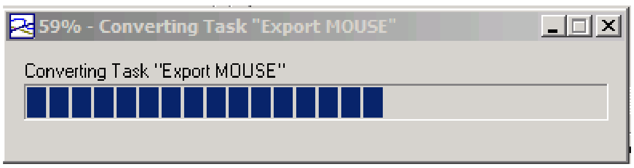
The Runoff Simulation

On the Runoff Parameters TAB page specify:

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

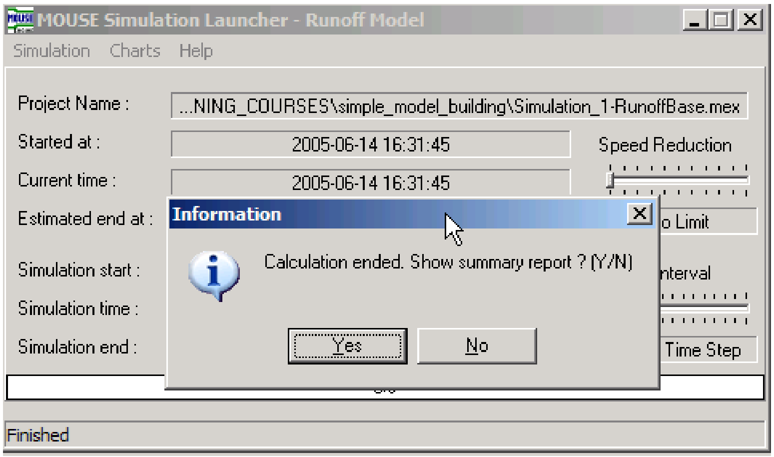


Return to the General TAB and press 'Start Simulation'. The system starts with export of model data to MOUSE.



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.



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.

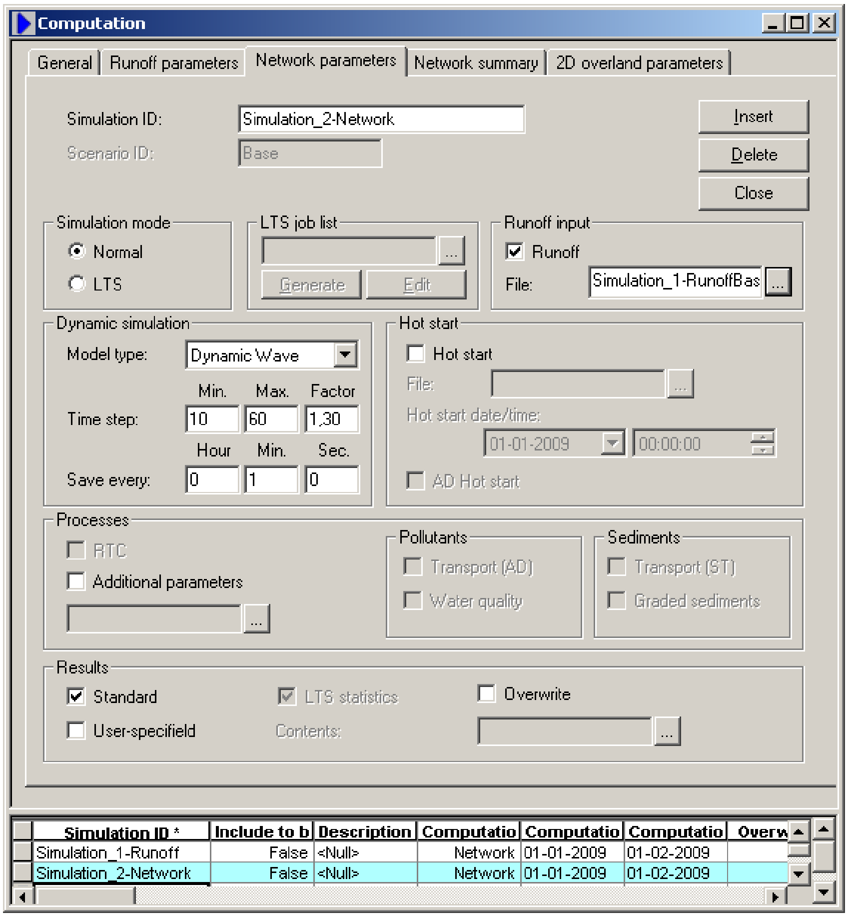
The result file from Runoff simulation gets the extension CRF (meaning: catchment result file). When the simulation is finished, the result file is automatically loaded into the memory and is ready for the presentation.

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)

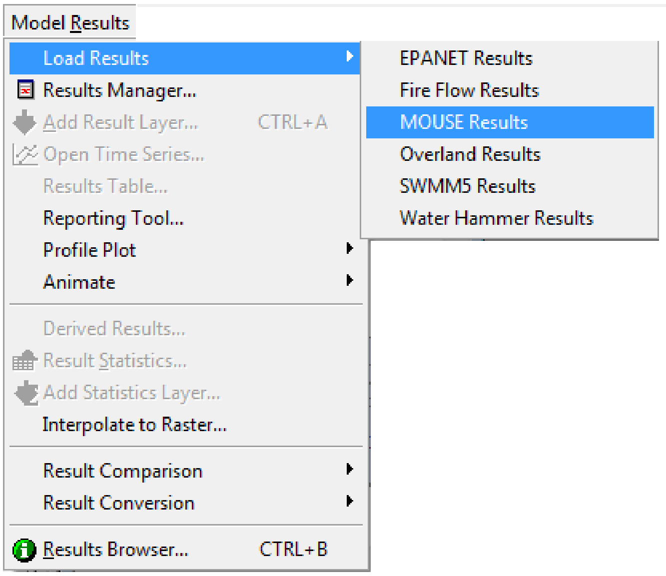


On the Network Summary tab page specify the type of results you would like to be available in the summary report presented after the network simulation.

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

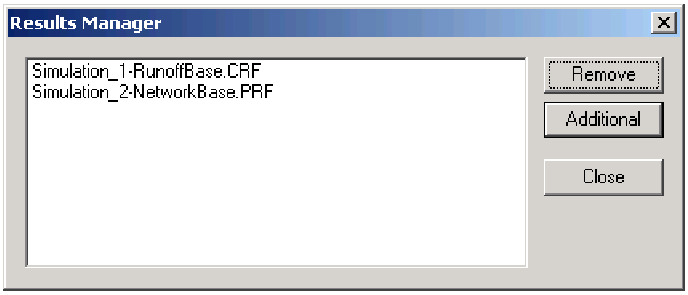


3.5.1 Step 1: Loading Results

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.

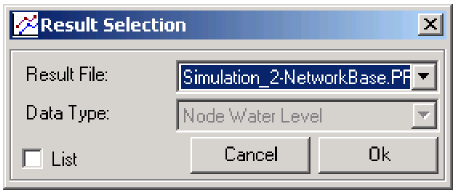


The result manager provides an overview of the files loaded and allows removing redundant files.

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.

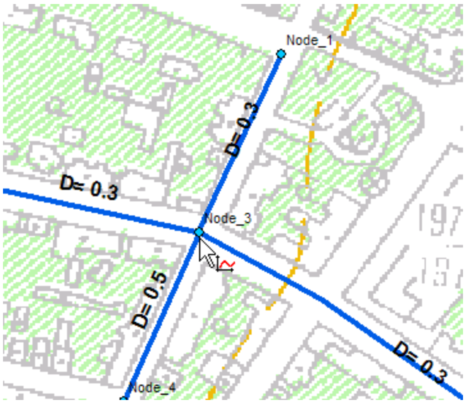
3.5.2 Step 2: Viewing Result Time Series

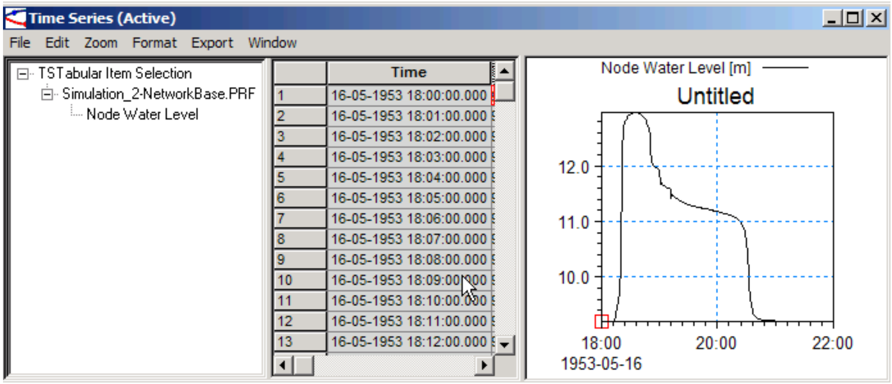
Go to Model Results / Open Time Series (or click on the Time Series button ) and choose the result file and data type from the result selection dialog.



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.



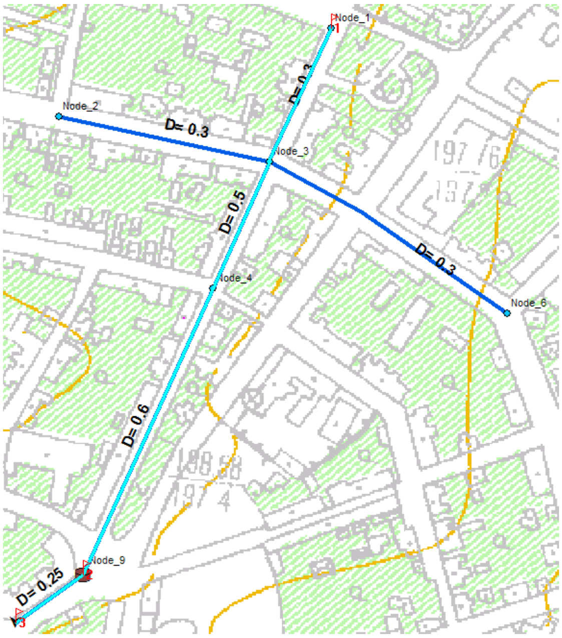


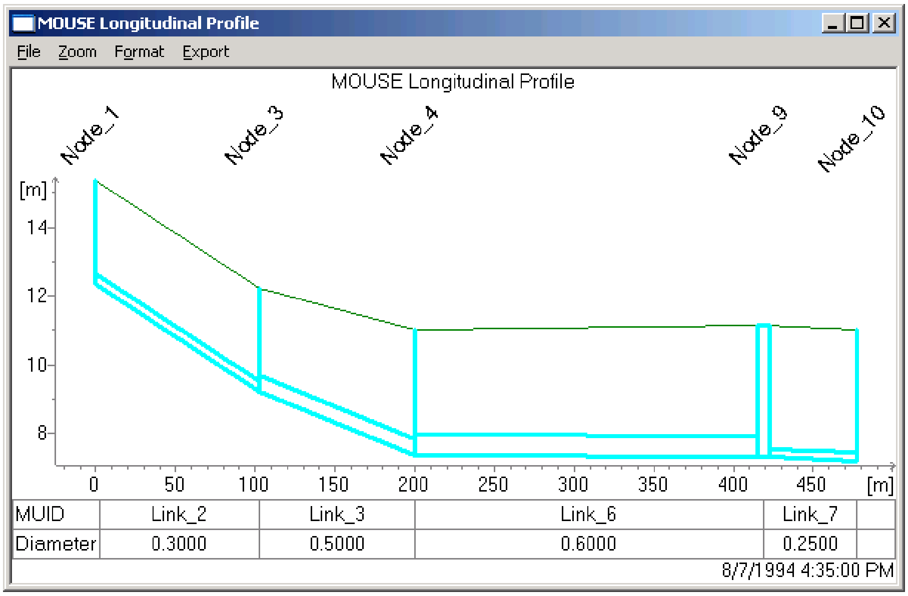
3.5.3 Step 3: Profile Plots and Animations

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

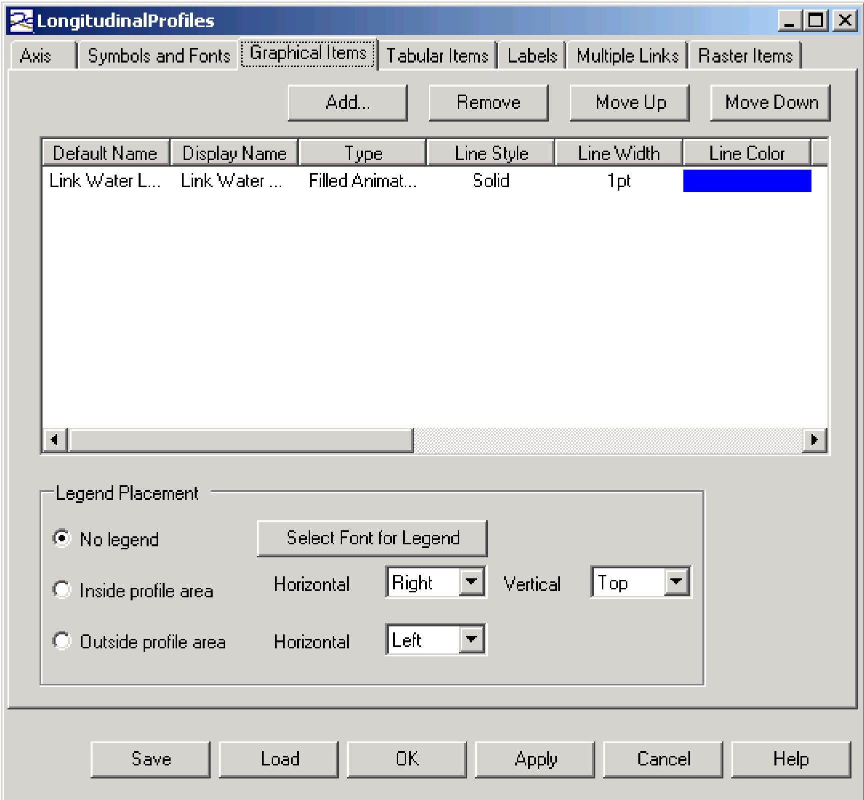
The profile plot is drawn between specified flags. Go to Model Results / Profile Plot and choose: 'Define Path Flags'

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'.

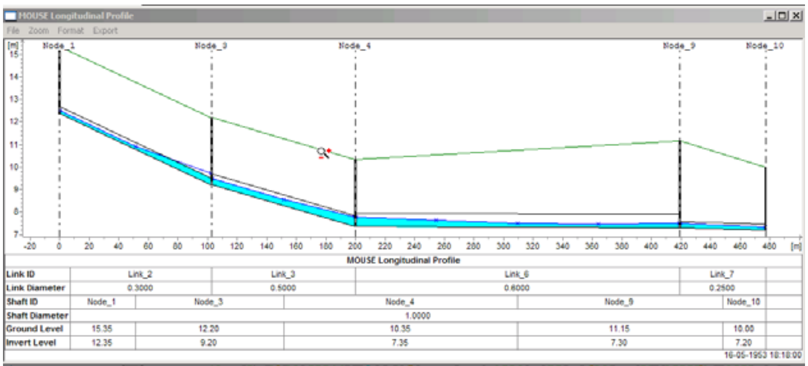




If results are loaded in, the longitudinal profile will be appearing with results added to the profile plot. For a MOUSE result file the water level will be added per default. If you wish to show other results or change the layout of the profile go to the profile plot properties dialog (right-click on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

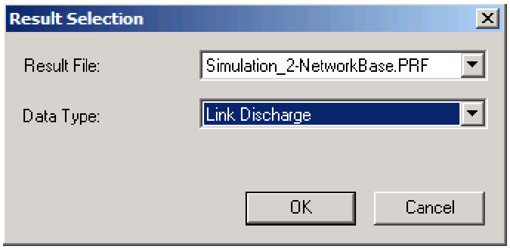


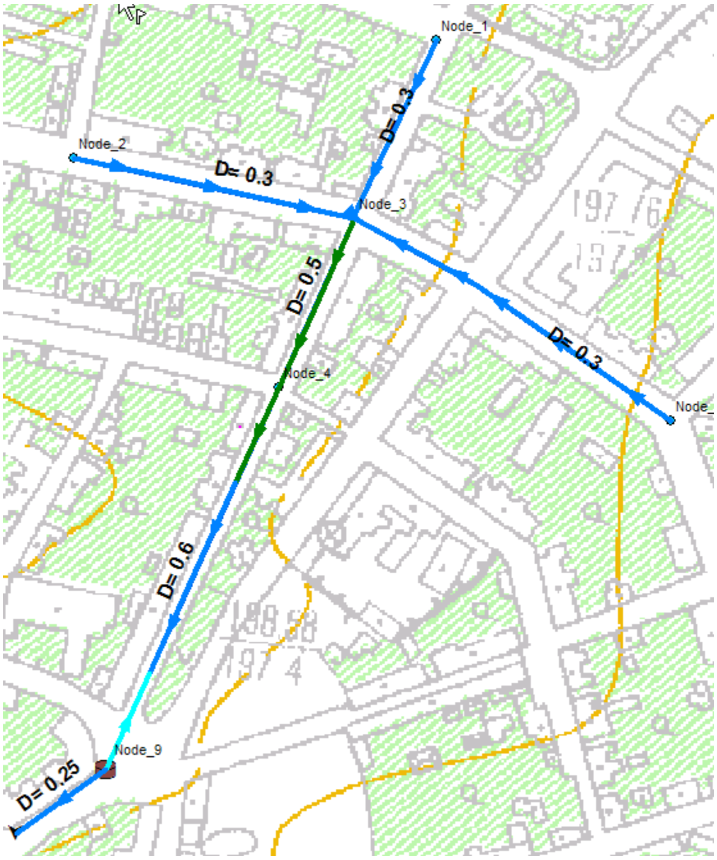
The profile plot may be animated. The actual time of the animation is seen in the lower right corner of the longitudinal profile.



3.5.4 Step 4: Adding Result Layers

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

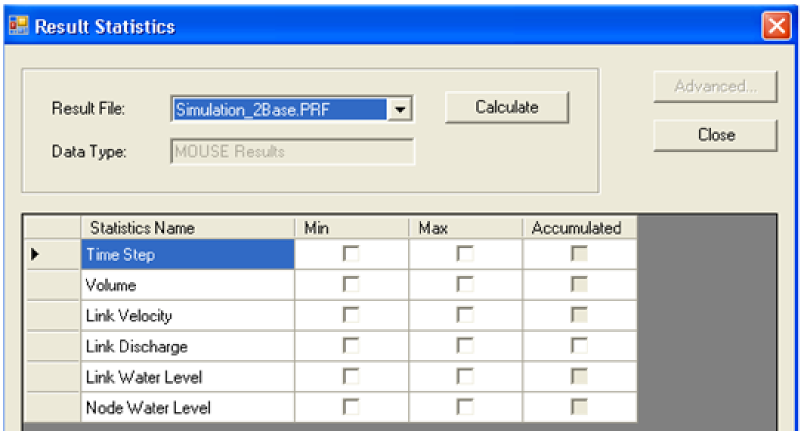




By appropriate symbology, the results may be displayed with various effects. The results in the map can be animated.

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.



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


Node_1
Node_2
Node_3
Node_4
Node_6
Node_9

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’:

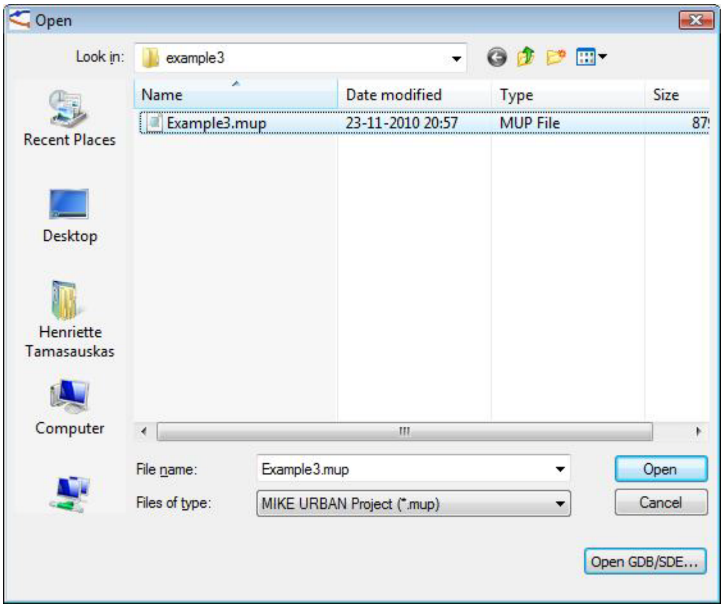


Figure 4.1 Opening the wastewater model

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.

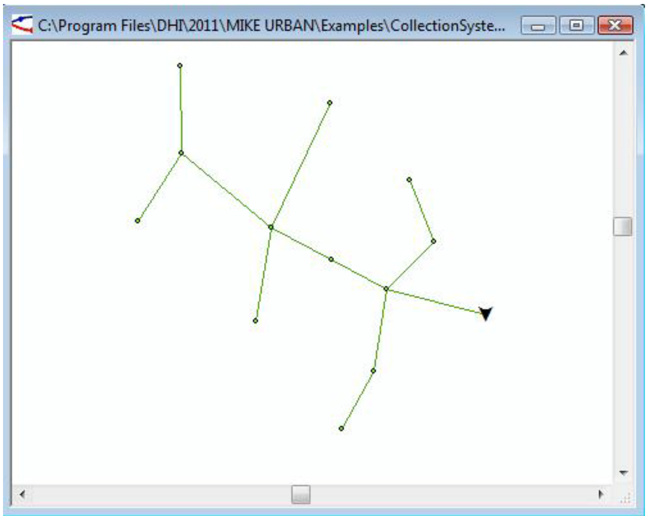


Figure 4.2 Wastewater model in MIKE URBAN

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.

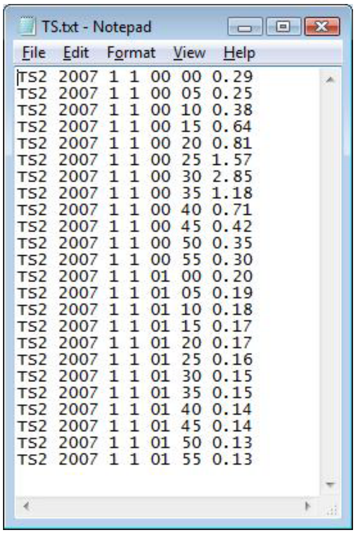

TS.txt - Notepad
File Edit Format View Help
```
TS2 2007 1 1 00 00 0.29
TS2 2007 1 1 00 05 0.25
TS2 2007 1 1 00 10 0.38
TS2 2007 1 1 00 15 0.64
TS2 2007 1 1 00 20 0.81
TS2 2007 1 1 00 25 1.57
TS2 2007 1 1 00 30 2.85
TS2 2007 1 1 00 35 1.18
TS2 2007 1 1 00 40 0.71
TS2 2007 1 1 00 45 0.42
TS2 2007 1 1 00 50 0.35
TS2 2007 1 1 00 55 0.30
TS2 2007 1 1 01 00 0.20
TS2 2007 1 1 01 05 0.19
TS2 2007 1 1 01 10 0.18
TS2 2007 1 1 01 15 0.17
TS2 2007 1 1 01 20 0.17
TS2 2007 1 1 01 25 0.16
TS2 2007 1 1 01 30 0.15
TS2 2007 1 1 01 35 0.15
TS2 2007 1 1 01 40 0.14
TS2 2007 1 1 01 45 0.14
TS2 2007 1 1 01 50 0.13
TS2 2007 1 1 01 55 0.13
```

Figure 4.3 The time series as it looks in Notepad

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

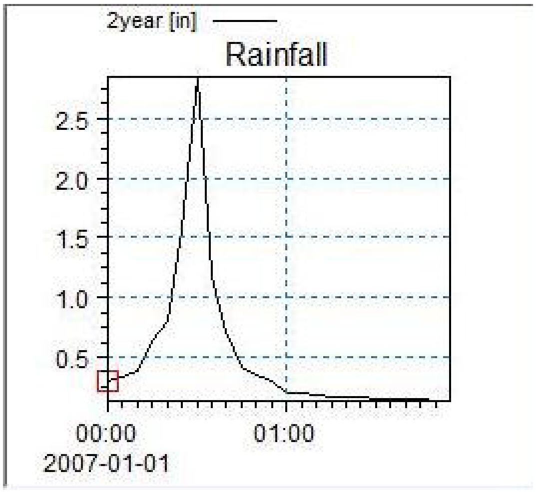


Figure 4.4 Graphical representation of the rainfall

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).

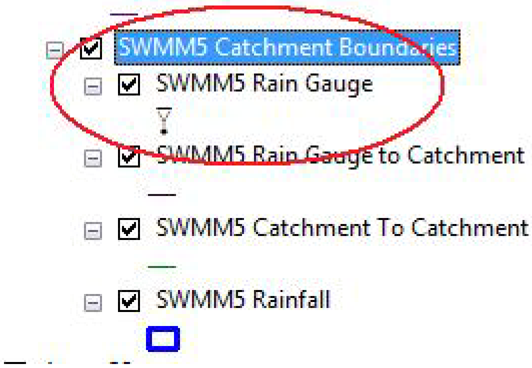


Figure 4.5 Toggling on the visibility of the rain gauge layer

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

Figure 4.6 Use the 'Create feature' to insert the raingauge

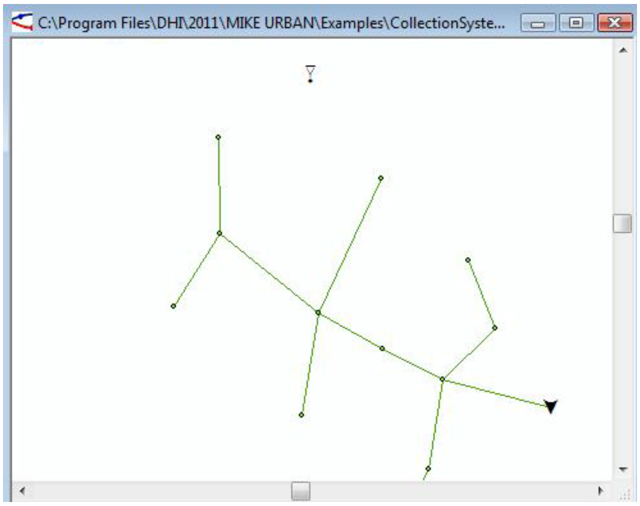


Figure 4.7 Raingauge is placed north of the system

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 4.1 Rain 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.

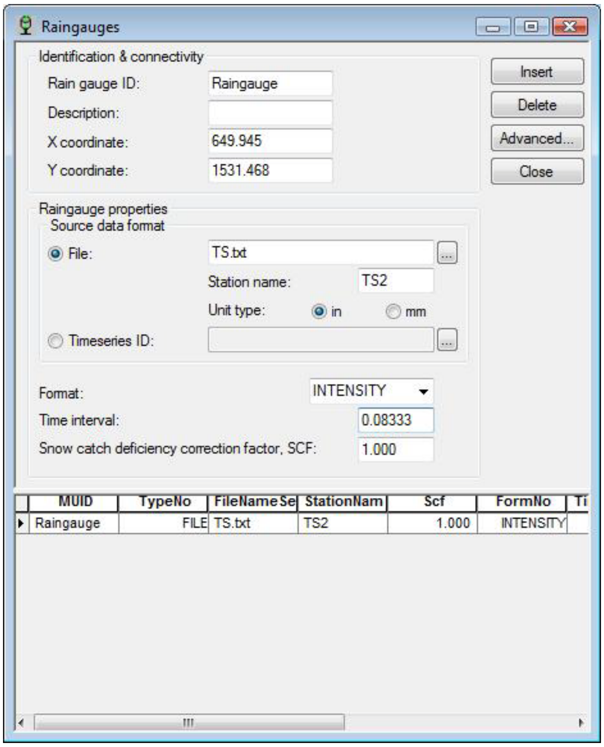


Figure 4.8 Rain gauge input

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.

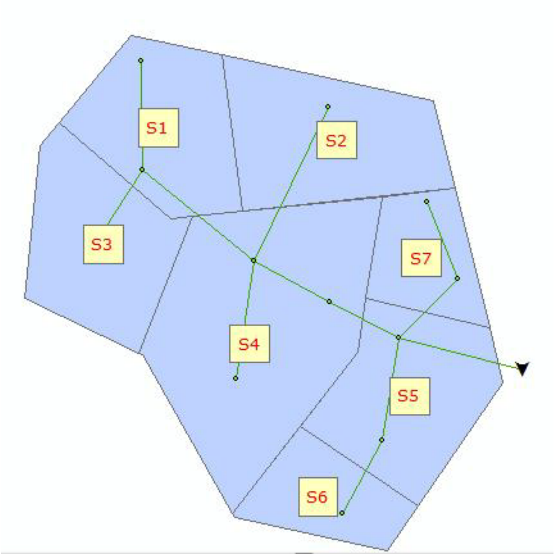


Figure 4.9 Catchment layout and ID's

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.

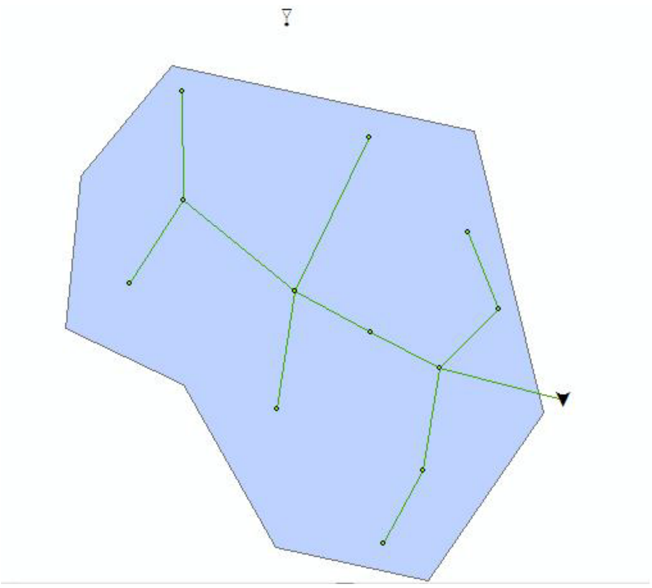


Figure 4.11 Start by drawing the contours of your total catchment area

- Switch to the ‘Split Polygon’ tool  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.

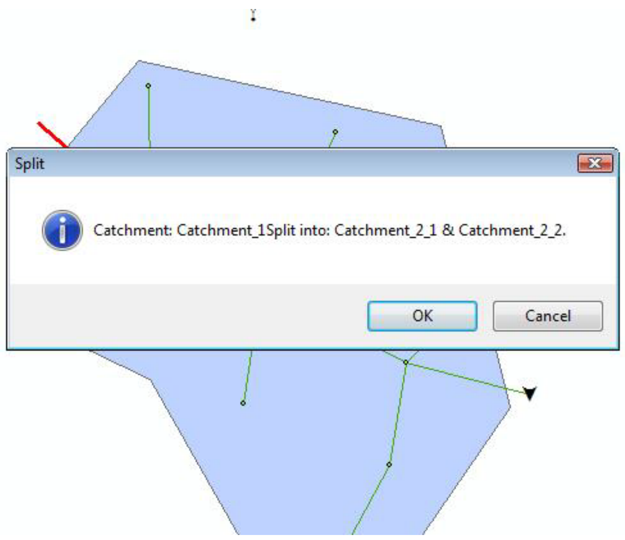


Figure 4.12 Using the Split Polygon tool

- 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.

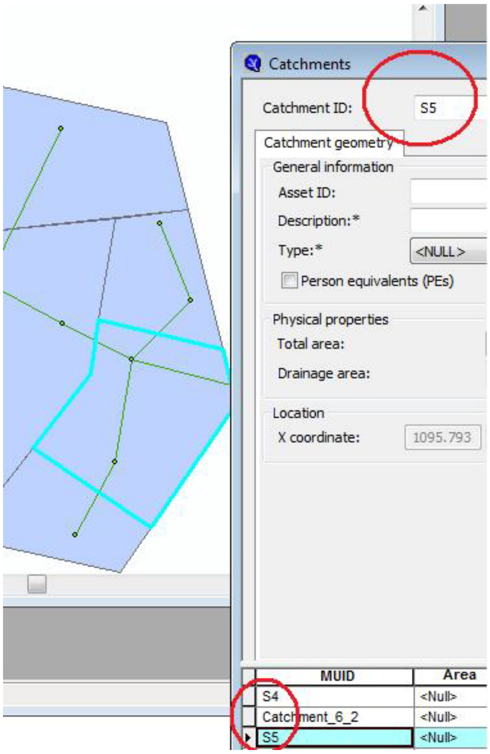


Figure 4.13 Renaming the catchments

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.

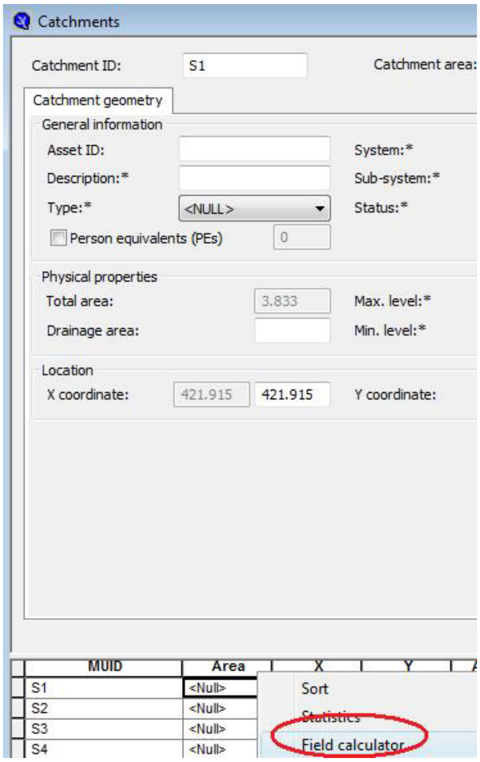


Figure 4.14 Activating the field calculator

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).

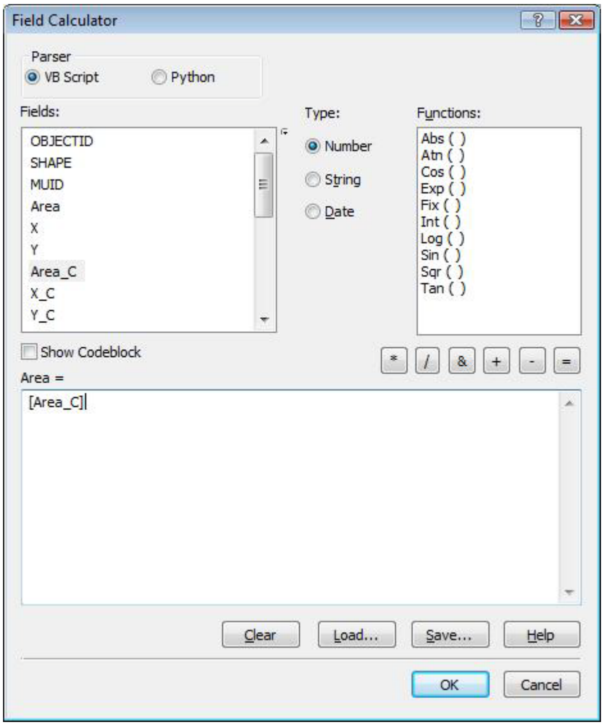


Figure 4.15 Setting the Drainage Area equal to the Total Area

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 Rain-gauge ID must be specified here.

All subcatchments:

- Load to Node
- Subarea Routing: Outlet

Table 4.2 Catchment routing data

Sub-catch-ment ID	Load Point ID	Width (ft)	Ground 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 impervious 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

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.

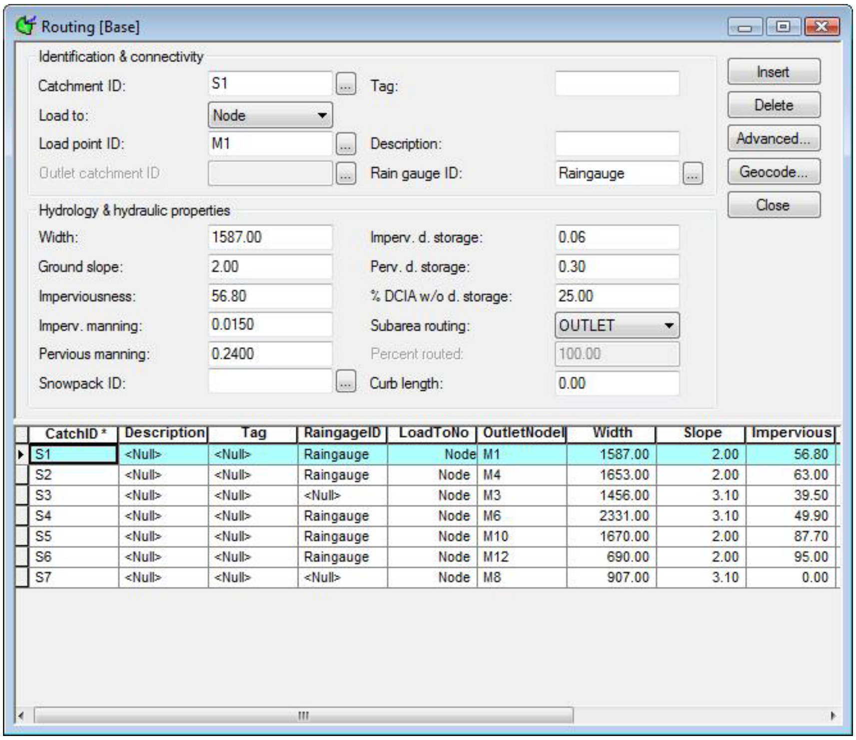


Figure 4.16 The Routing dialog after data has been entered

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.

Table 4.3 Catchment Infiltration Data

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

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

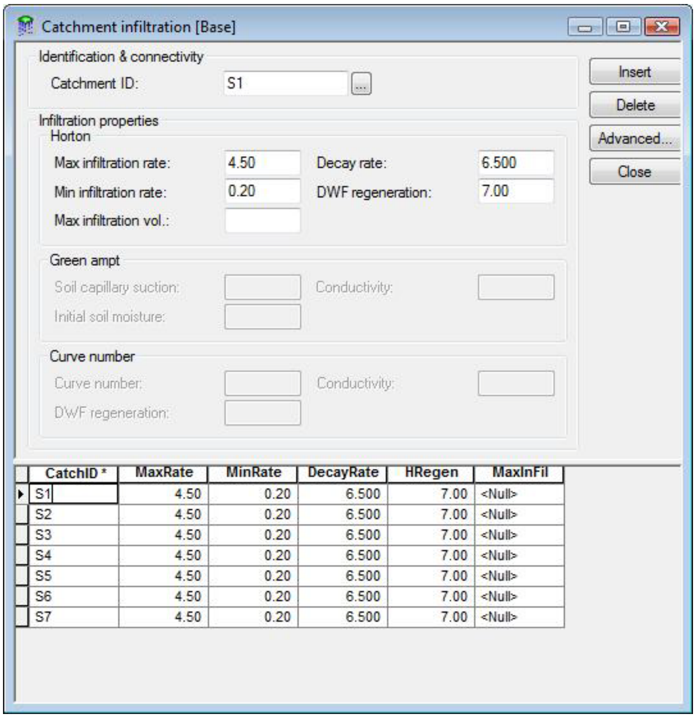


Figure 4.17 The Infiltration dialog after data has been entered

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.

Table 4.4 General settings

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

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

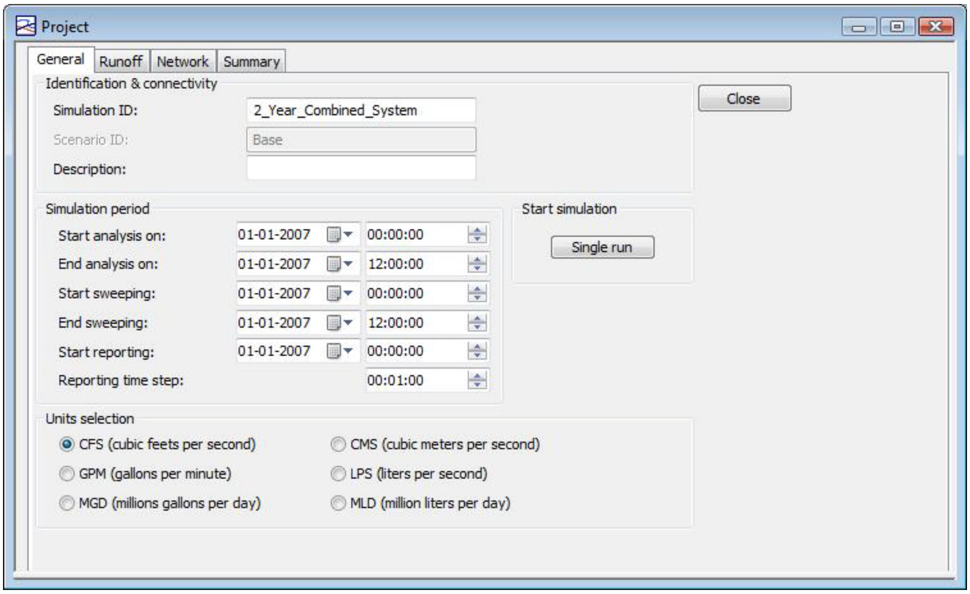


Figure 4.18 General settings for the simulation

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

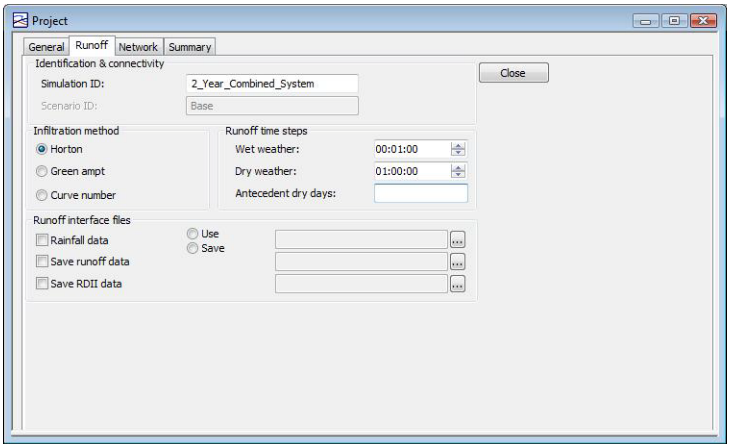


Figure 4.19 Runoff settings for the simulation

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

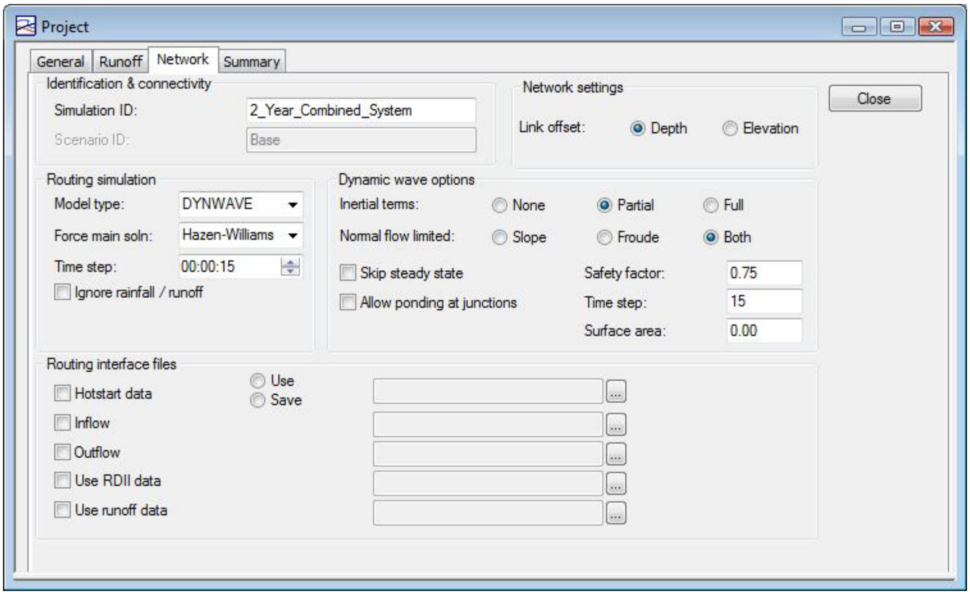


Figure 4.20 Network settings for the simulation

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.

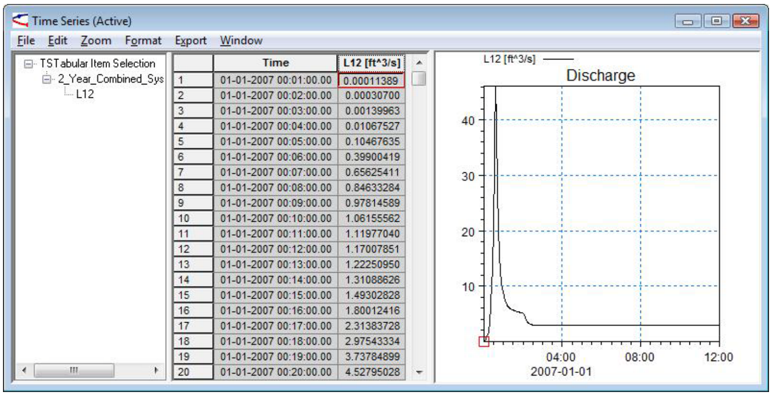


Figure 4.21 Discharge hydrograph

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

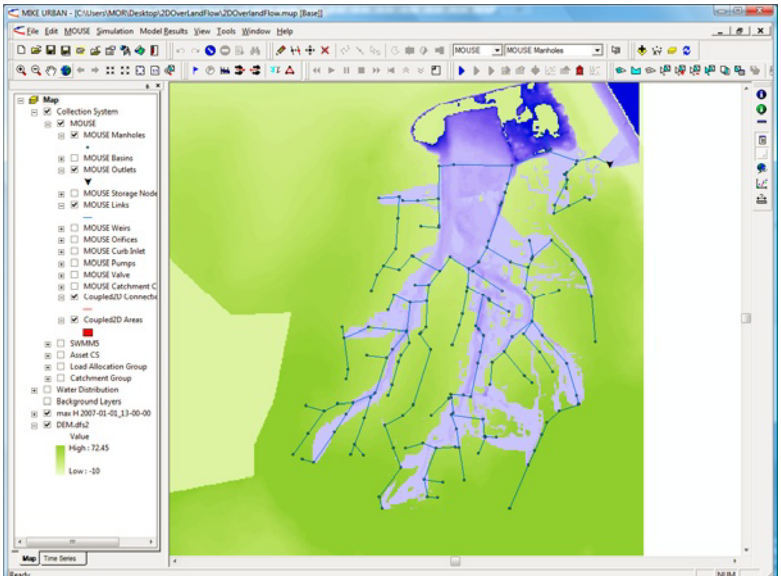


Figure 5.1 The model with DEM and flooding results

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.

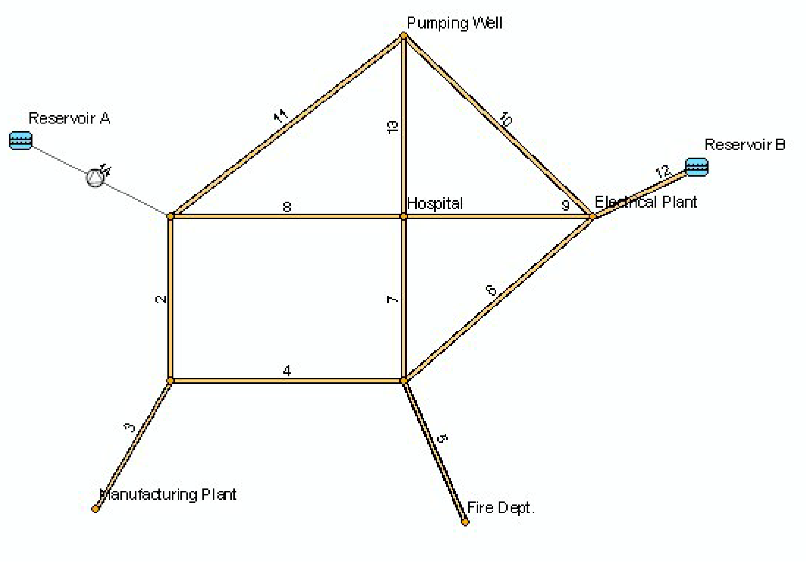


Figure 6.1 A schematic diagram of the pipe network system used in this lesson

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.

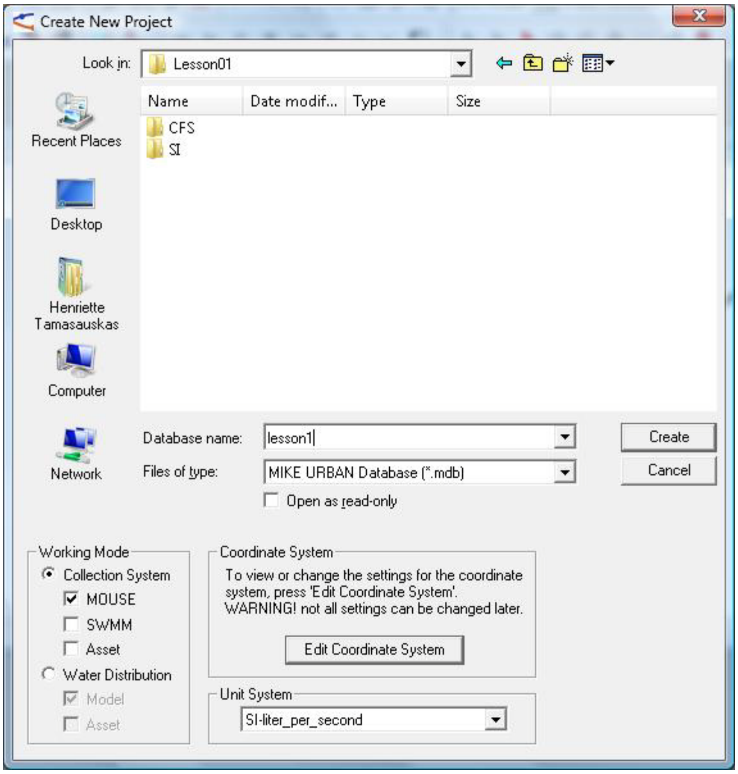


Figure 6.2 The Create New Project Dialog Box

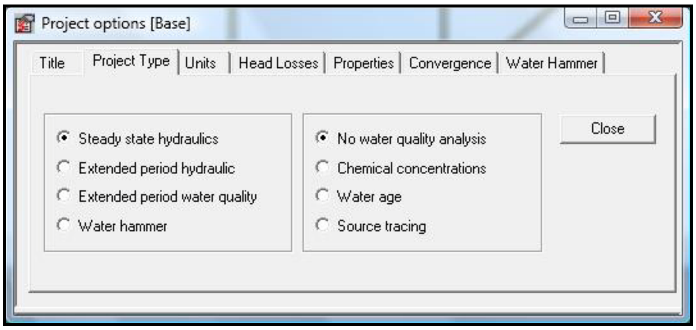


Figure 6.3 The Project Options Dialog Box

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.

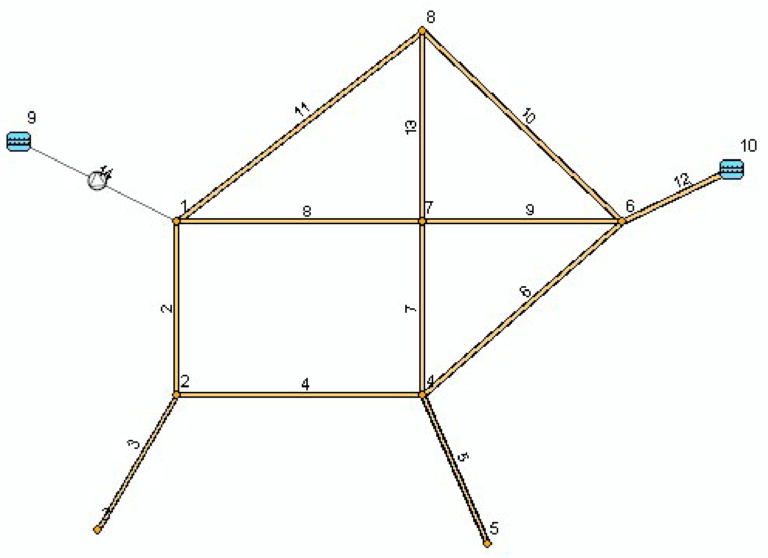


Figure 6.5 Node and pipe IDs

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.

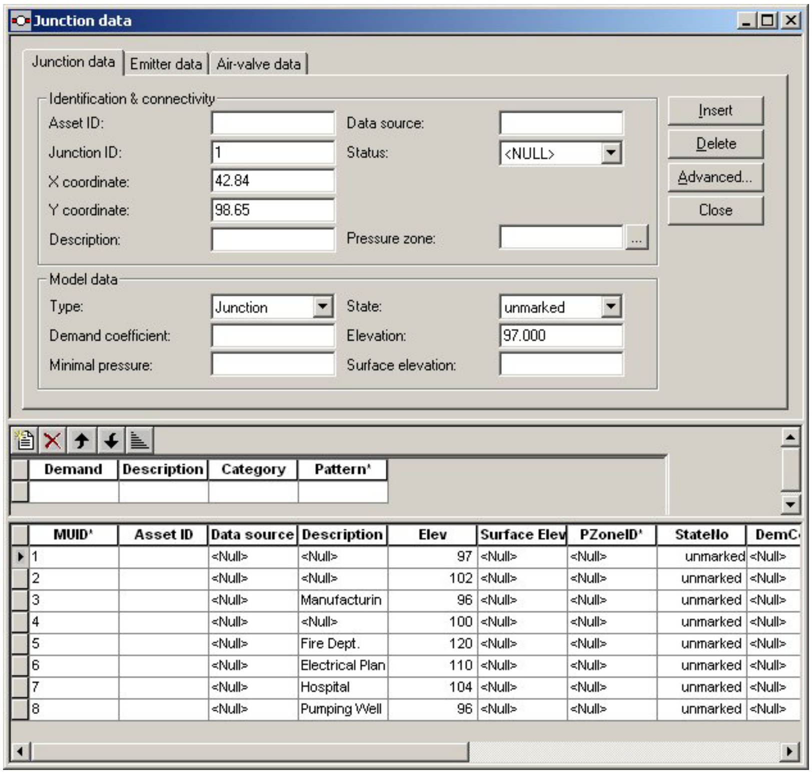


Figure 6.6 The Junction Editor dialog box

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.

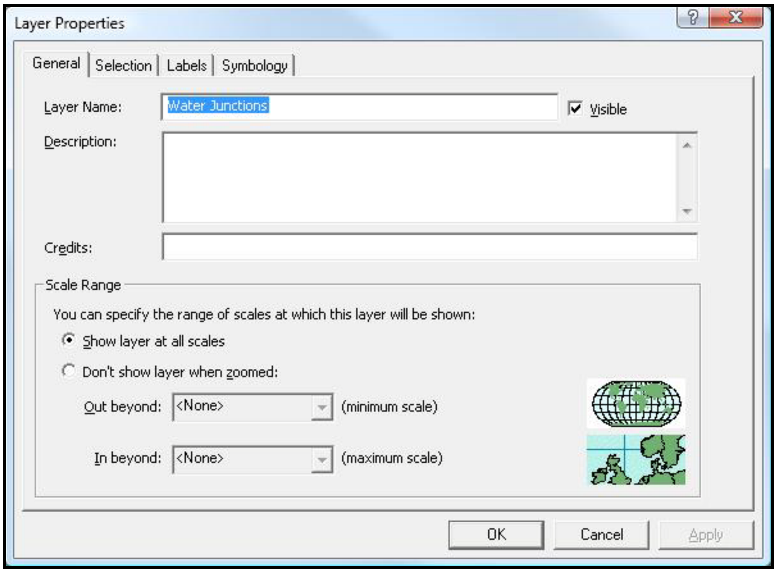


Figure 6.7 The Layer Properties dialog box is used to adjust the display settings for horizontal plan (map) window

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 pre-existing 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.

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

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

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–Y coordinates, 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 6.3 Pipe data for the pipe network system

Table 1:

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

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.

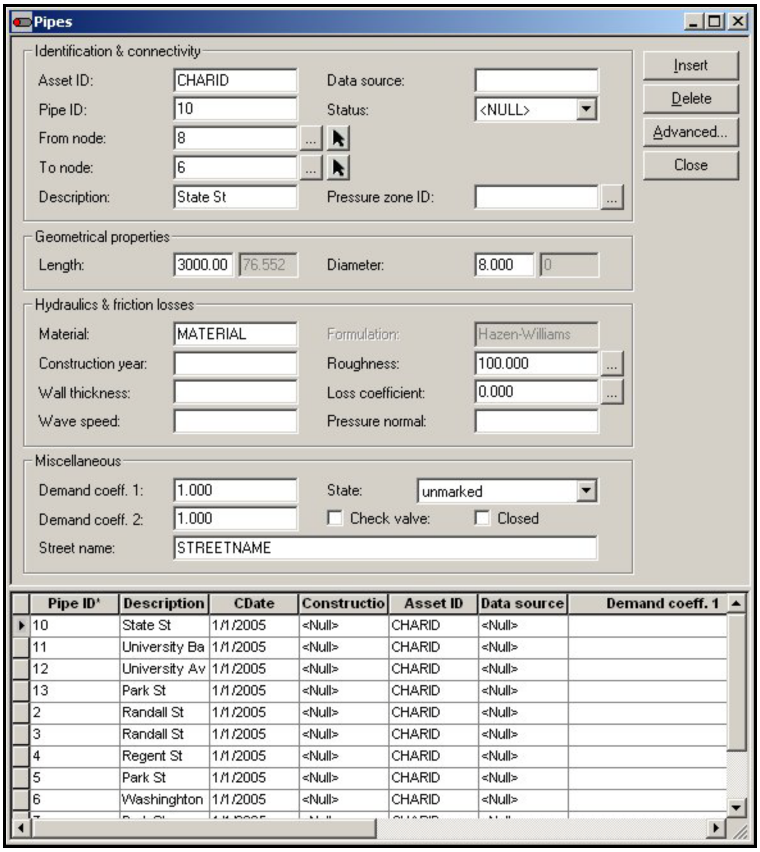


Figure 6.9 The Pipe Editor dialog box

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.

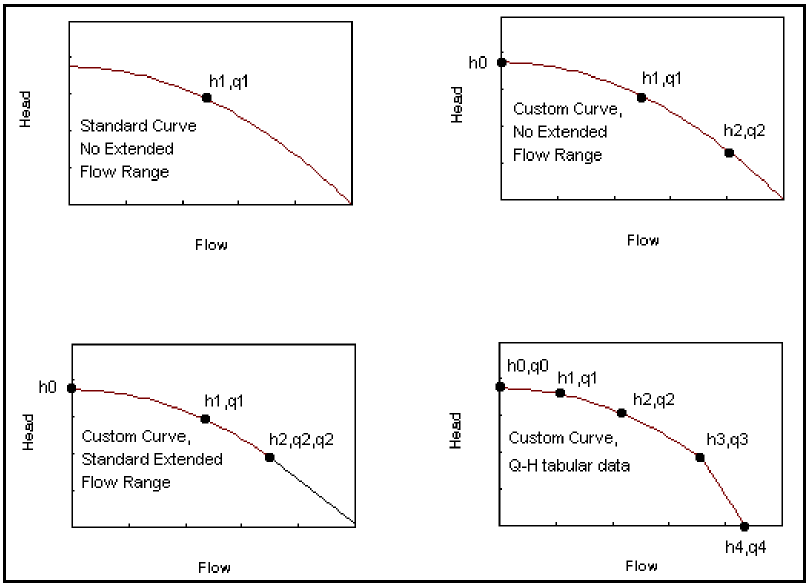


Figure 6.10 Input requirements for the various types of pump curves

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.

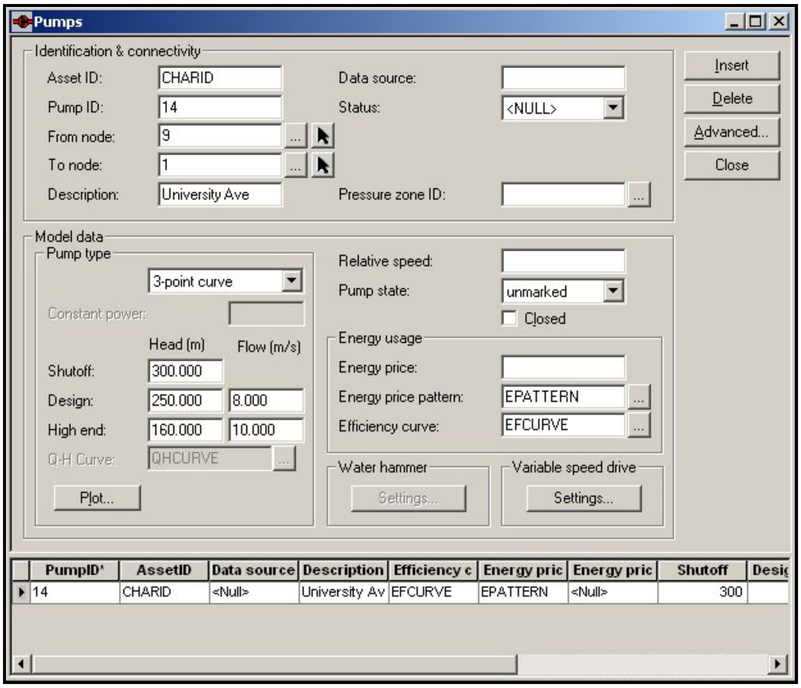


Figure 6.11 The Pump Editor dialog box

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.

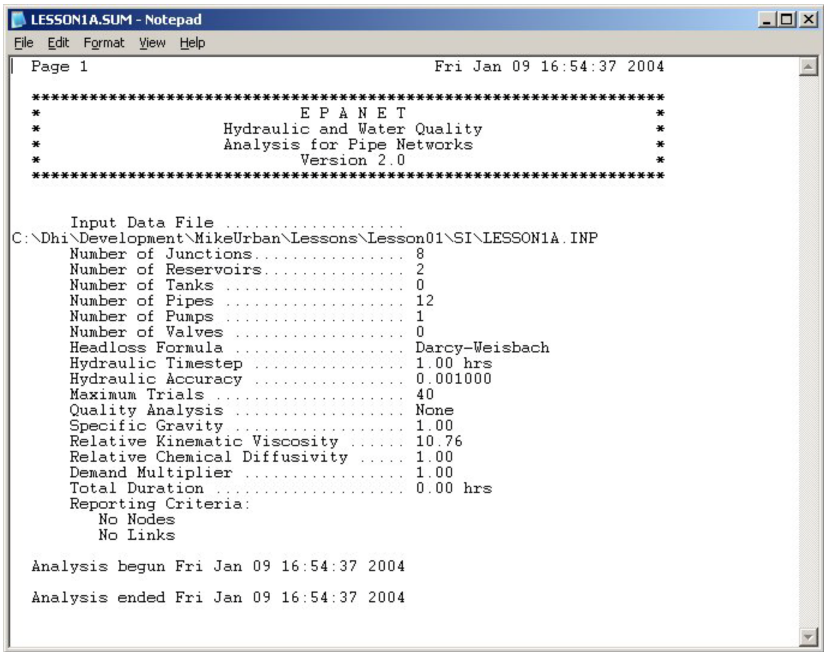
These files can be found in the LESSONS\LESSON01\CFS or LESSONS\LESSON01\SI subdirectory and can be used to perform the analysis and view the analysis results, without having to interactively enter the data for this lesson.

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 LESSON1.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 - Notepad
File  Edit  Format  View  Help
 Page 1                                        Fri Jan 09 16:54:37 2004

 ******************************************************************
 *                         E P A N E T                            *
 *                 Hydraulic and Water Quality                    *
 *                 Analysis for Pipe Networks                     *
 *                         Version 2.0                            *
 ******************************************************************

     Input Data File ..................
C:\Dhi\Development\MikeUrban\Lessons\Lesson01\SI\LESSON1A.INP
     Number of Junctions................ 8
     Number of Reservoirs............... 2
     Number of Tanks ................... 0
     Number of Pipes ................... 12
     Number of Pumps ................... 1
     Number of Valves .................. 0
     Headloss Formula .................. Darcy-Weisbach
     Hydraulic Timestep ................ 1.00 hrs
     Hydraulic Accuracy ................ 0.001000
     Maximum Trials .................... 40
     Quality Analysis .................. None
     Specific Gravity .................. 1.00
     Relative Kinematic Viscosity ...... 10.76
     Relative Chemical Diffusivity ..... 1.00
     Demand Multiplier ................. 1.00
     Total Duration .................... 0.00 hrs
     Reporting Criteria:
        No Nodes
        No Links

 Analysis begun Fri Jan 09 16:54:37 2004

 Analysis ended Fri Jan 09 16:54:37 2004
```

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.

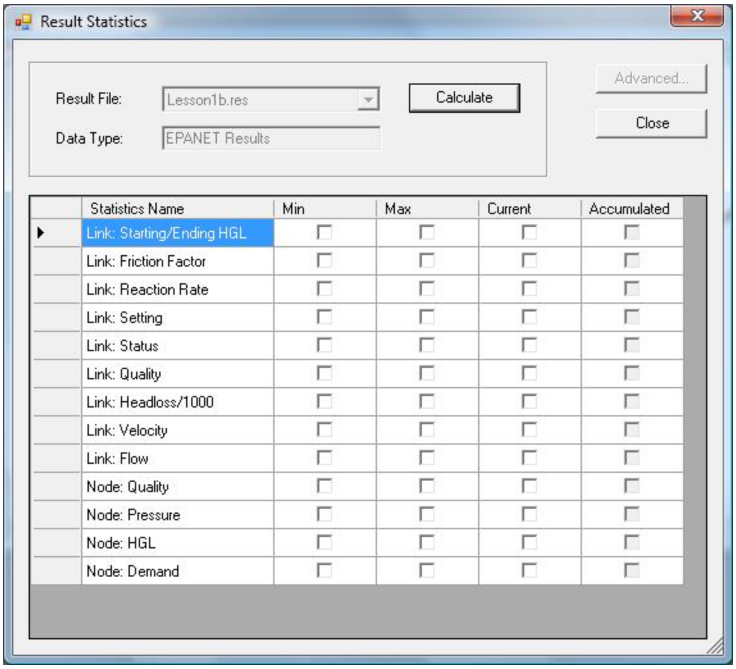


Figure 6.13 The Results Statistics Dialog box

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

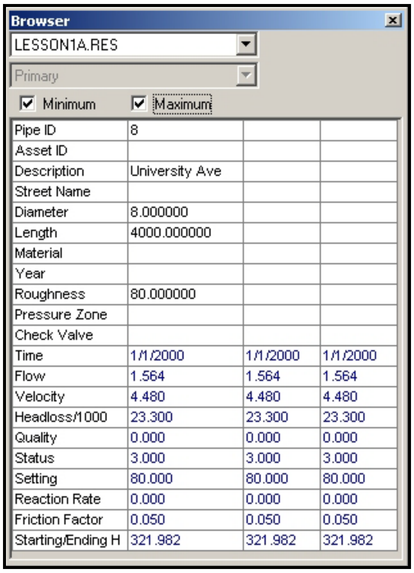
mw_RESJunction : Table

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	0
8	-35	-35	72.86816	72.86816	43.86815	43.86815
10	188.8012	188.8012	60	60	0	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, diameter, 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.



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.

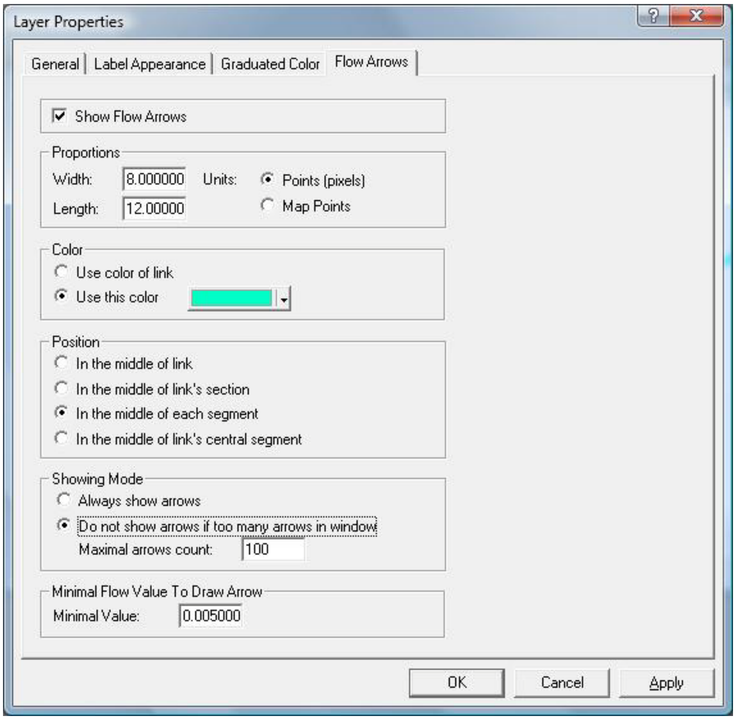


Figure 6.16 The Layer Properties dialog box allows you to customize the Horizontal Plan graphical plot

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.

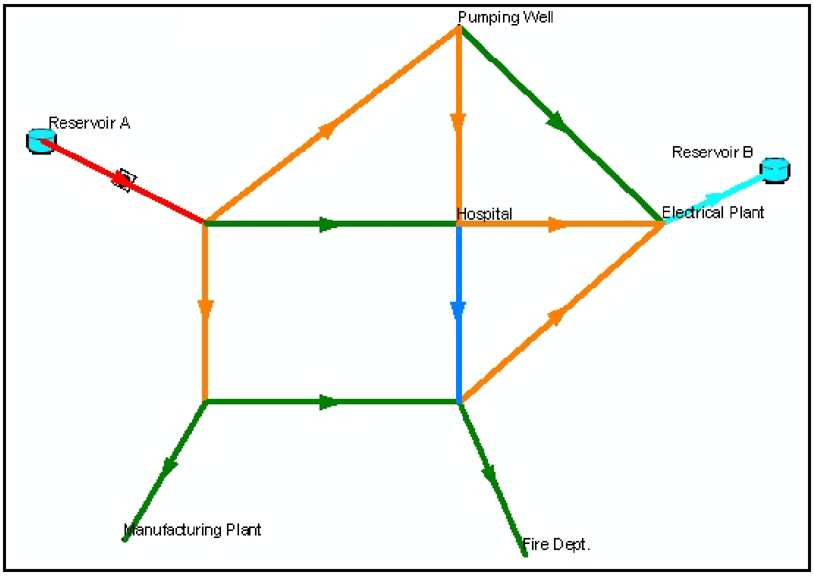


Figure 6.17 Computed flow direction arrows plotted on the pipe network in Horizontal Plan window

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.

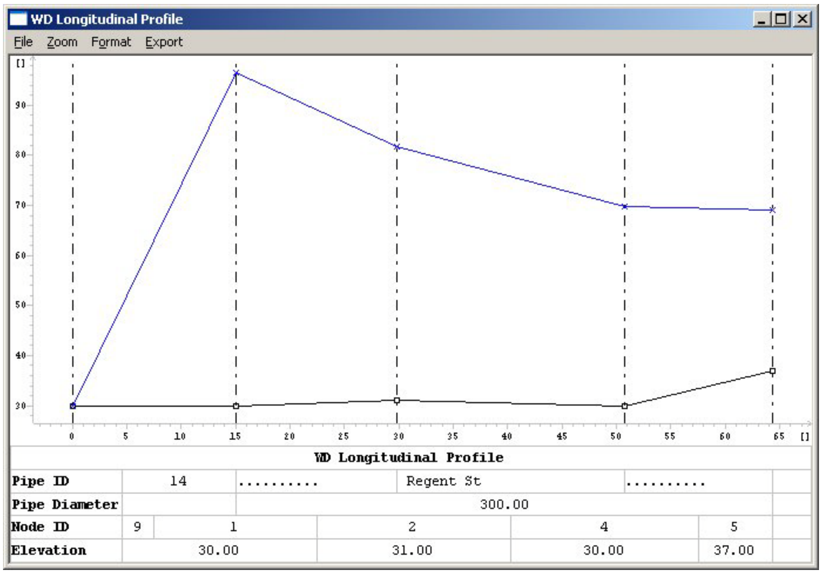


Figure 6.18 Profile plots allow to graphically plot the analysis results along any pipeline path

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.
2. 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.

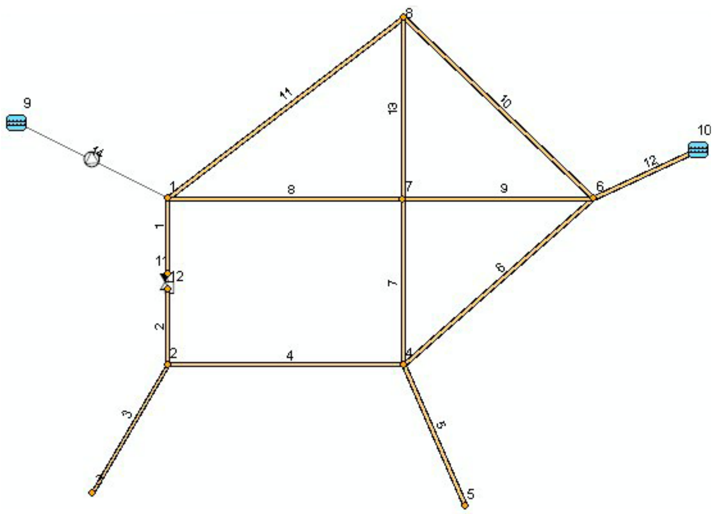


Figure 6.19 A schematic diagram of the pipe network system

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\LESSON02 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.

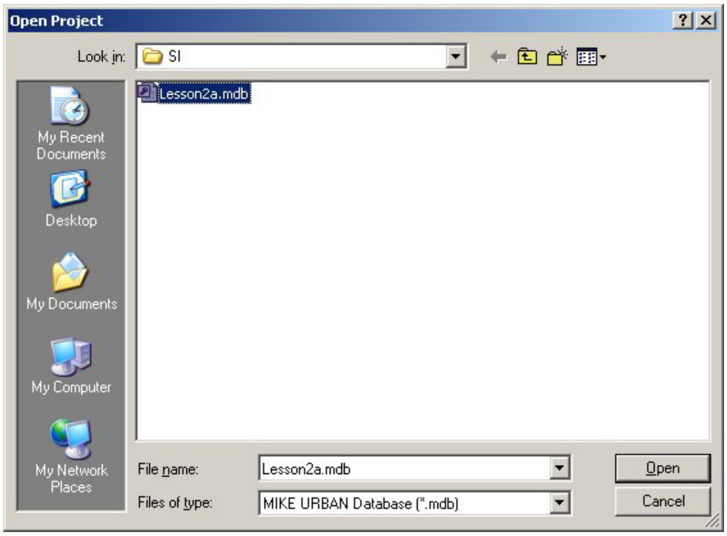


Figure 6.20 The Open dialog box

2. 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 <Del> key.
5. 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.

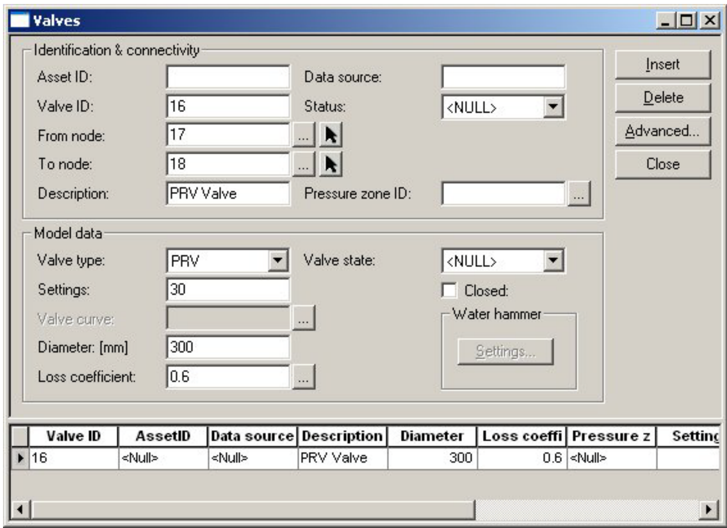


Figure 6.21 The Valve Editor dialog box

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.

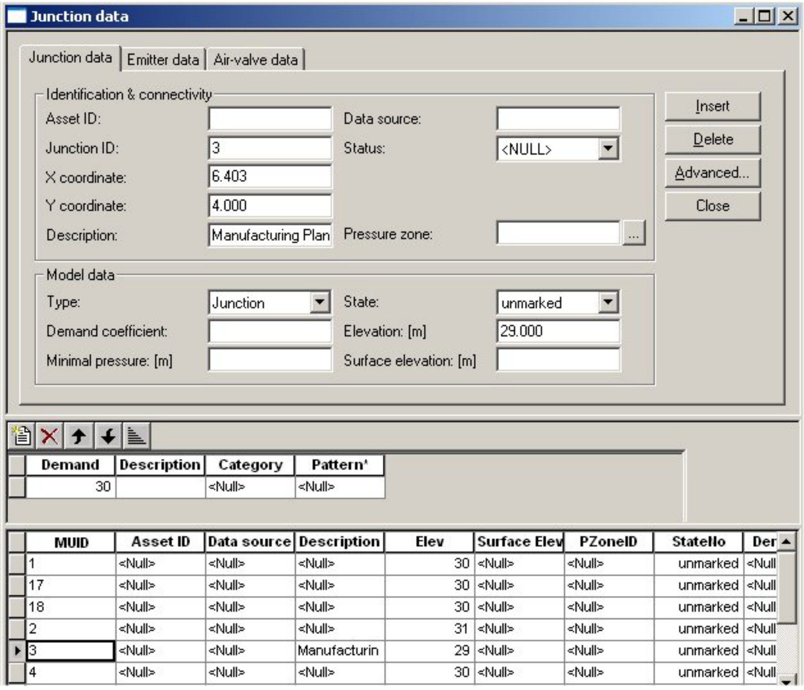


Figure 6.22 The Junction Editor dialog box

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 LESSON2B.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.
2. 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.

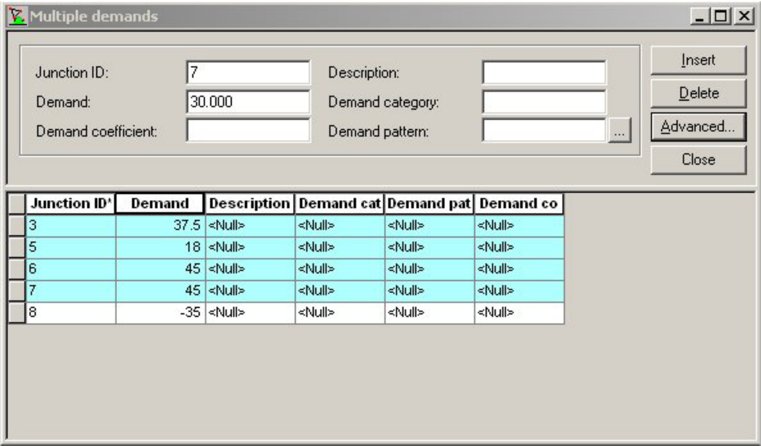


Figure 6.23 Select records, where you want to apply the global demand change

3. In the Field Calculator dialog, use the wizard to define the following statement: Demand = Demand * 1.5.
4. 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».

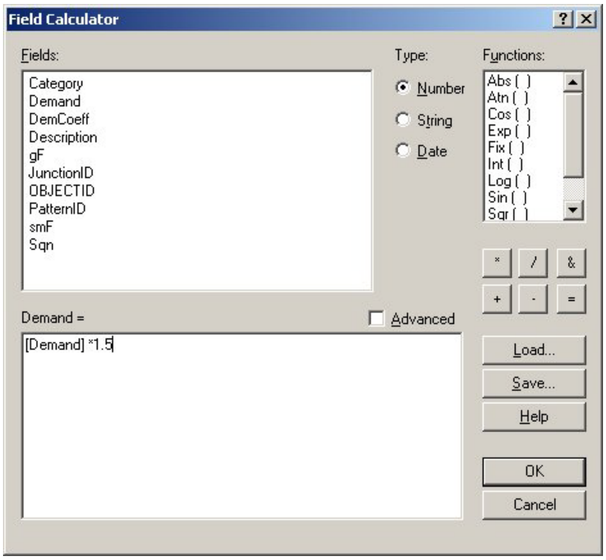


Figure 6.24 The Field Calculator dialog box

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.

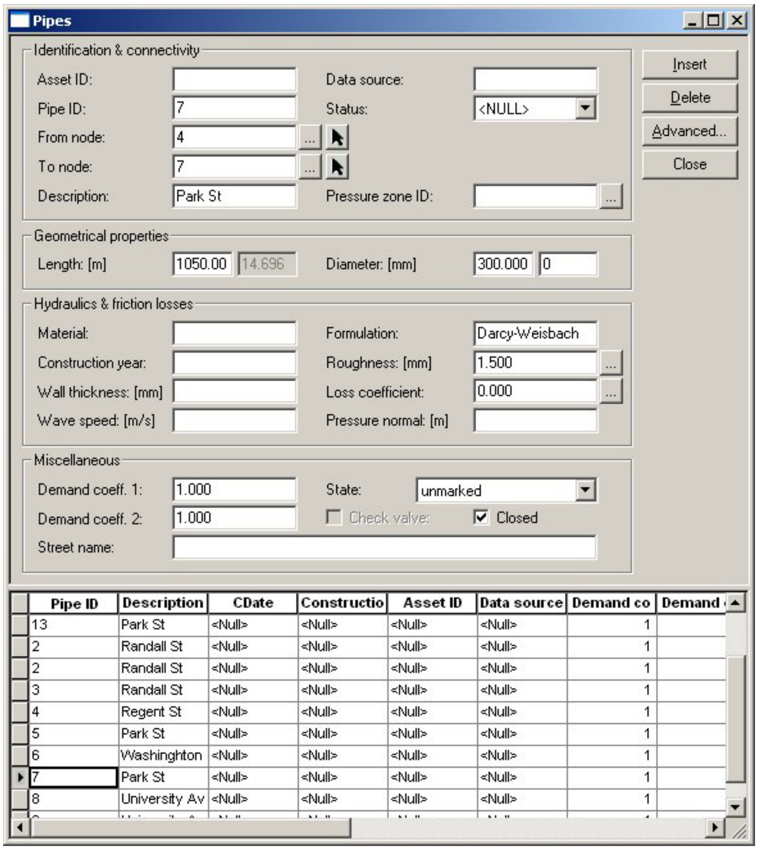


Figure 6.25 The Pipe Editor dialog box

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.

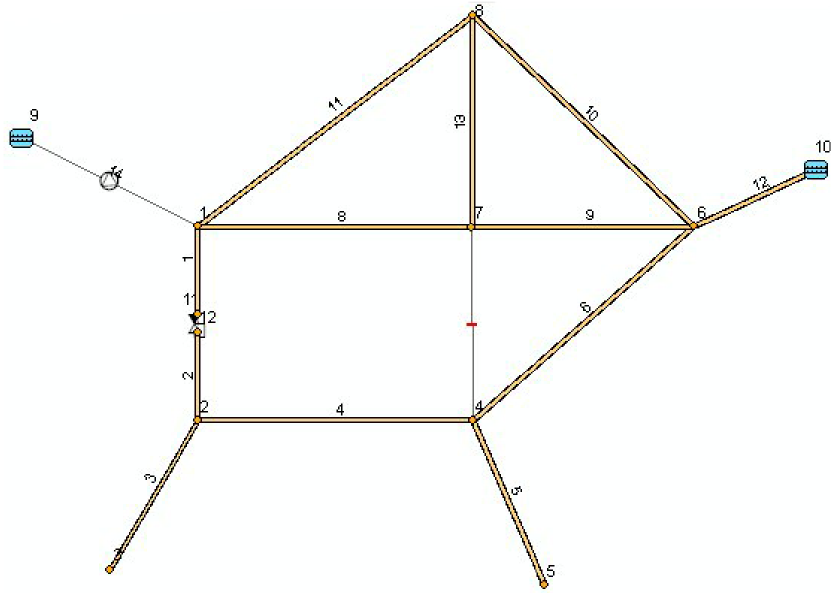


Figure 6.26 Schematic diagram with the closed pipe

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 (LESSON2B). 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 LESSON2C.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 LESSON2B.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 LESSONCB.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.

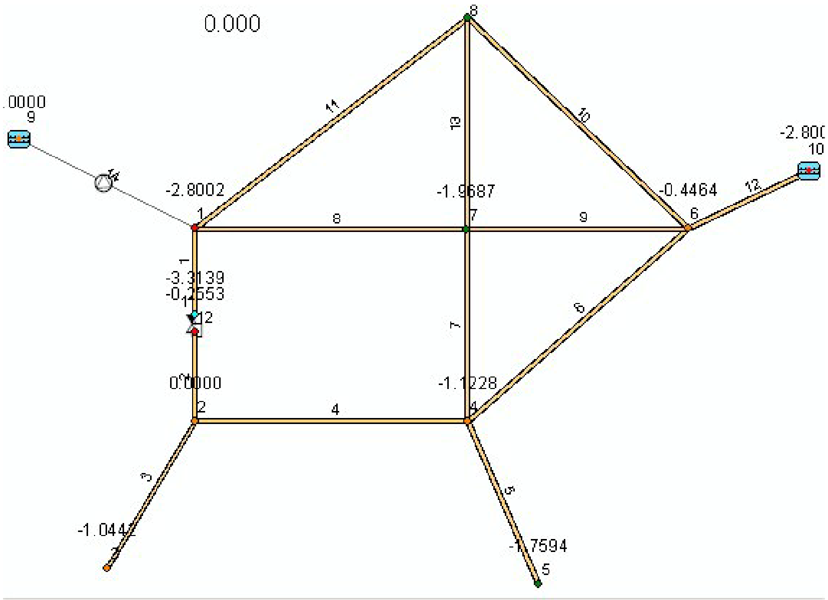


Figure 6.27 The comparison results (pressure differences in this case) are displayed in the Map window

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.

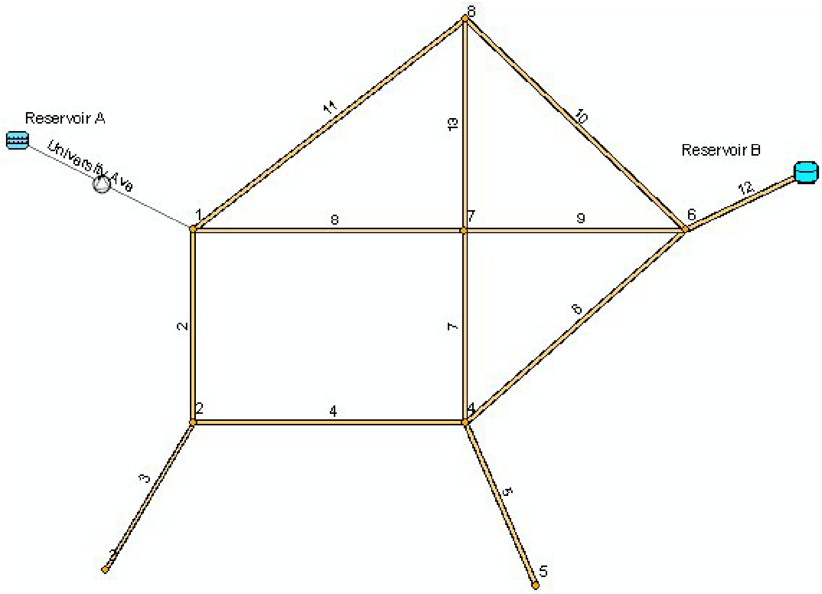


Figure 6.28 A schematic diagram of the pipe network system

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.

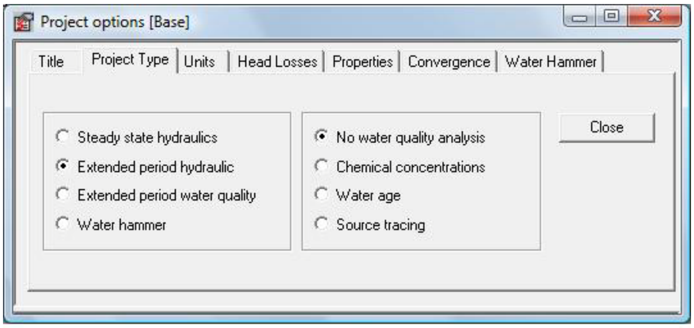


Figure 6.29 The Project Options dialog box

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.

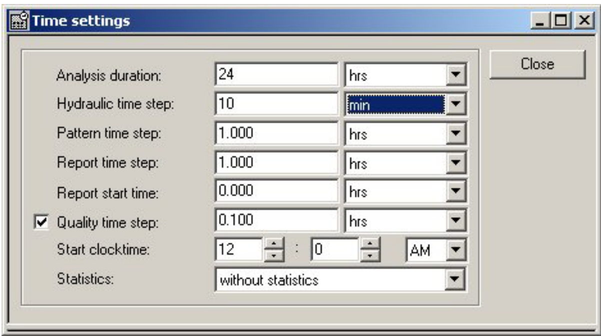


Figure 6.30 The Time Editor dialog box

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.

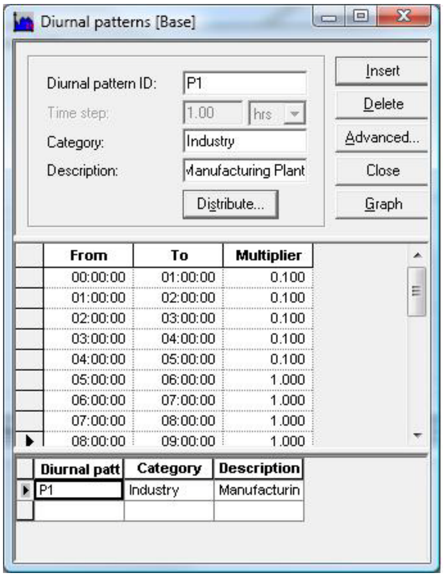


Figure 6.31 The Diurnal Profile dialog box

To define a demand pattern:

1. Select «Insert» and enter a Pattern ID of P1.
2. 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».

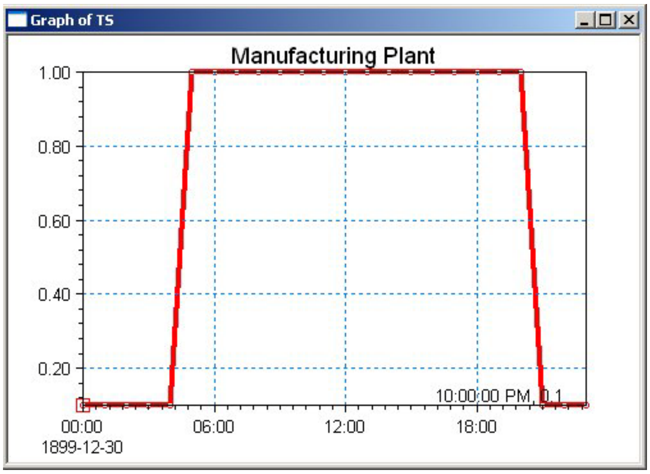


Figure 6.32 A demand diurnal pattern curve

4. 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).

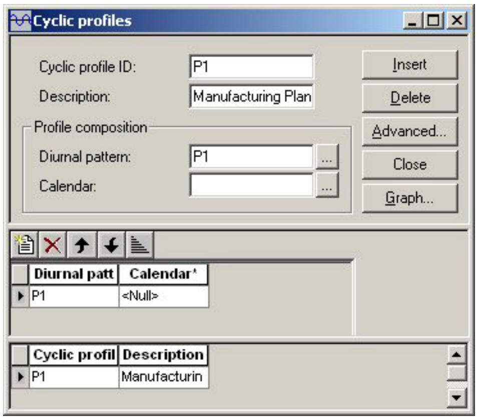


Figure 6.33 A demand diurnal pattern curve

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.

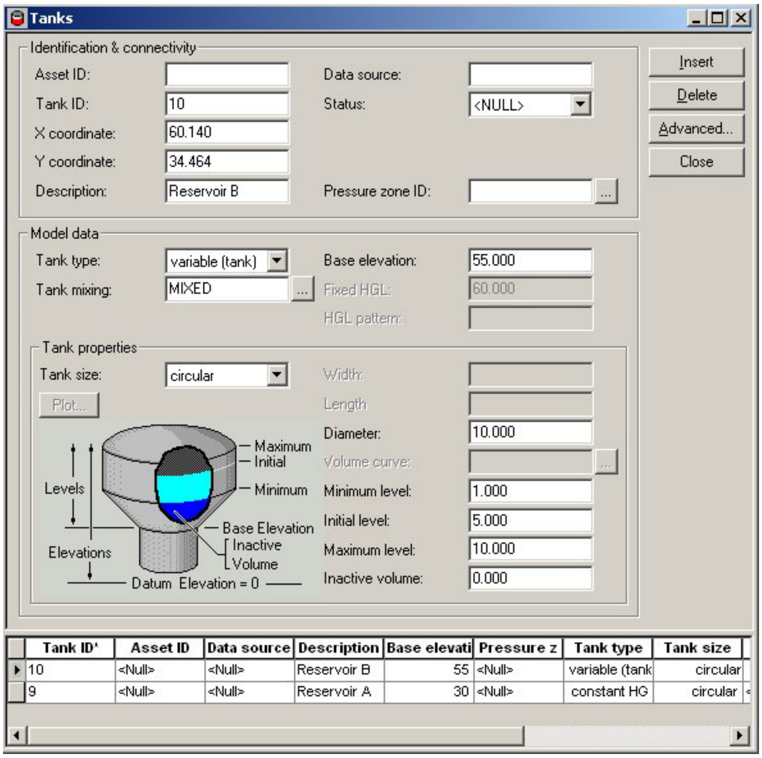


Figure 6.34 The Tank Editor dialog box

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.

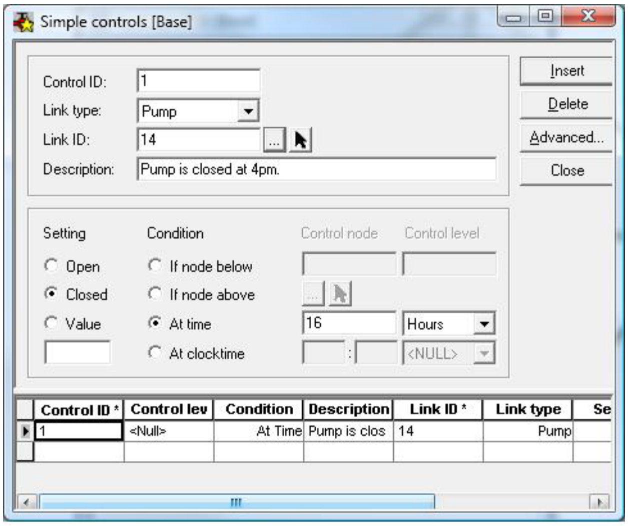


Figure 6.35 The Simple Control dialog box

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.

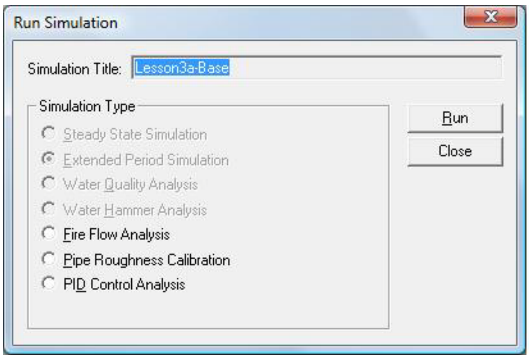


Figure 6.36 The Run Simulation dialog box

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.

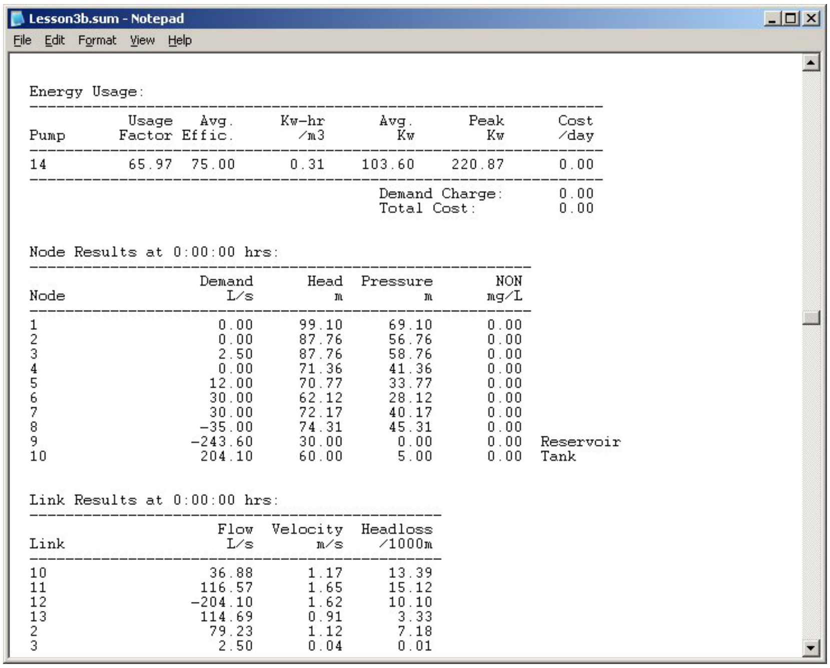
6.3.7 Viewing the Extended Period Analysis Results

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.



Lesson3b.sum - Notepad

File Edit Format View Help

Energy Usage:

Pump	Usage Factor	Avg. Effic.	Kw-hr /m3	Avg. Kw	Peak Kw	Cost /day
14	65.97	75.00	0.31	103.60	220.87	0.00

Demand Charge: 0.00
Total Cost: 0.00

Node Results at 0:00:00 hrs:

Node	Demand L/s	Head m	Pressure m	NON mg/L	
1	0.00	99.10	69.10	0.00	
2	0.00	87.76	56.76	0.00	
3	2.50	87.76	58.76	0.00	
4	0.00	71.36	41.36	0.00	
5	12.00	70.77	33.77	0.00	
6	30.00	62.12	28.12	0.00	
7	30.00	72.17	40.17	0.00	
8	-35.00	74.31	45.31	0.00	
9	-243.60	30.00	0.00	0.00	Reservoir
10	204.10	60.00	5.00	0.00	Tank

Link Results at 0:00:00 hrs:

Link	Flow L/s	Velocity m/s	Headloss /1000m
10	36.88	1.17	13.39
11	116.57	1.65	15.12
12	-204.10	1.62	10.10
13	114.69	0.91	3.33
2	79.23	1.12	7.18
3	2.50	0.04	0.01

Figure 6.37 Analysis Results Summary

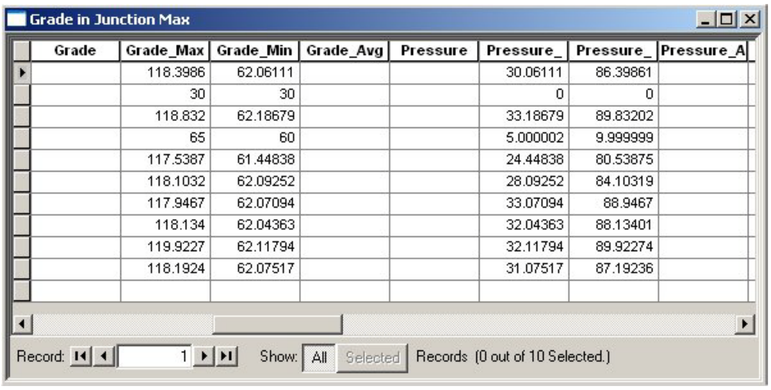
Further discussion on displaying analysis results is provided in the section titled Viewing the Analysis Results contained on -160.

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.



Grade	Grade_Max	Grade_Min	Grade_Avg	Pressure	Pressure_	Pressure_	Pressure_A
	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	
	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.

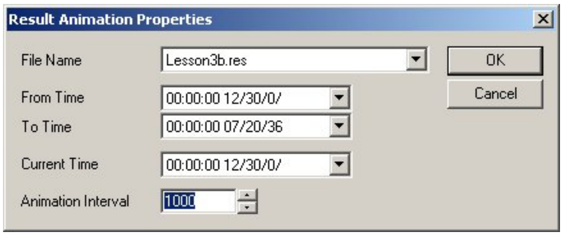


Figure 6.39 Animation - The Time Step dialog box allows you to select a different time step in which to display results

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.

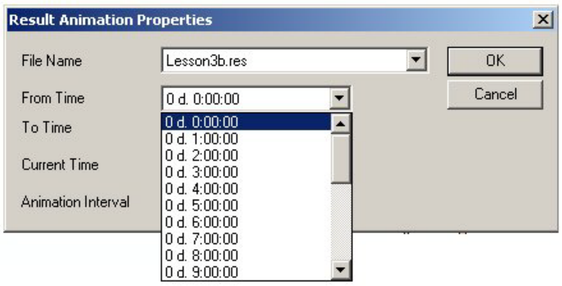


Figure 6.40 The select time level dialog box

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.
2. 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.

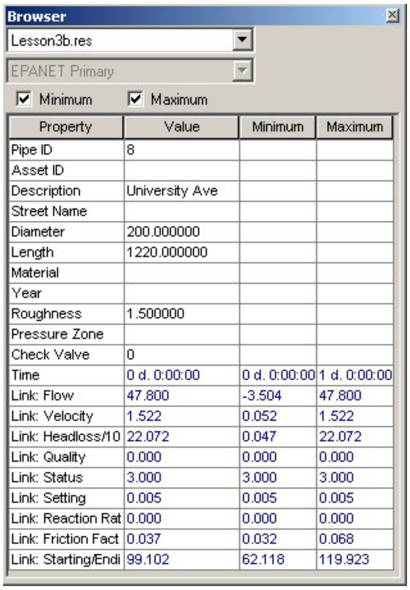


Figure 6.41 The Results Browser displays analysis results for the selected network component

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.

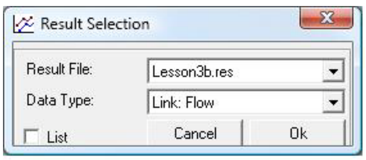


Figure 6.42 The Time Series Result item selection dialog box

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

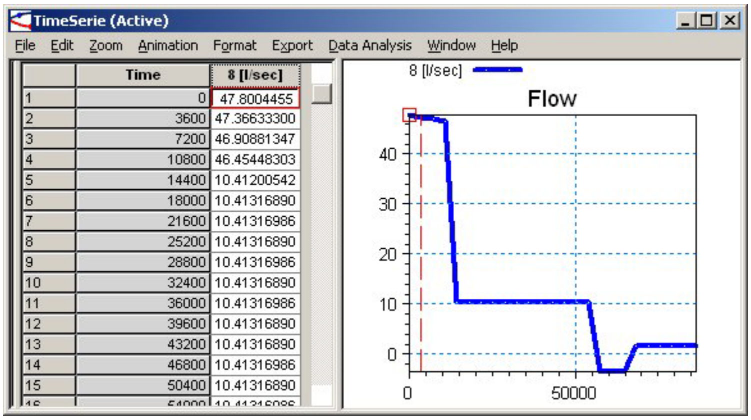


Figure 6.43 Time Series Plot for a flow in the link 8

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.
2. 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.
3. 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.

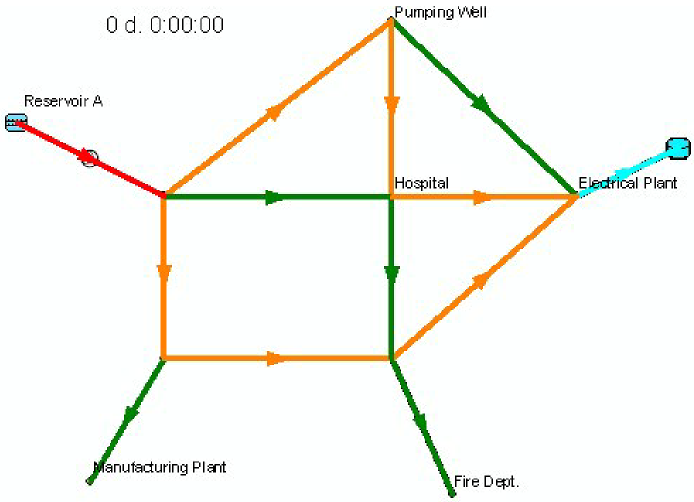


Figure 6.44 Flow arrows displayed for the result layer Link: Flow

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.

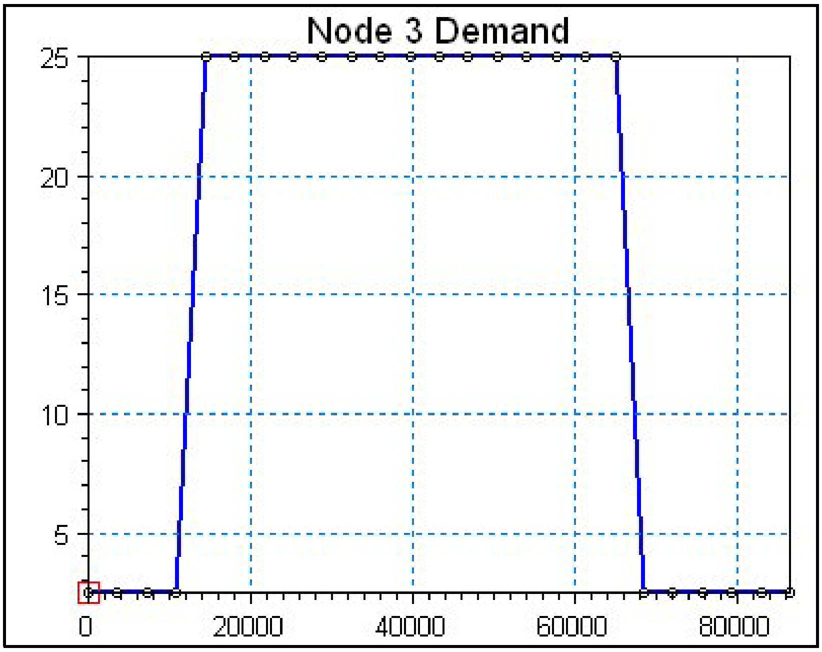


Figure 6.45 The calculated node demand for the node 3 (Pattern P1)

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.

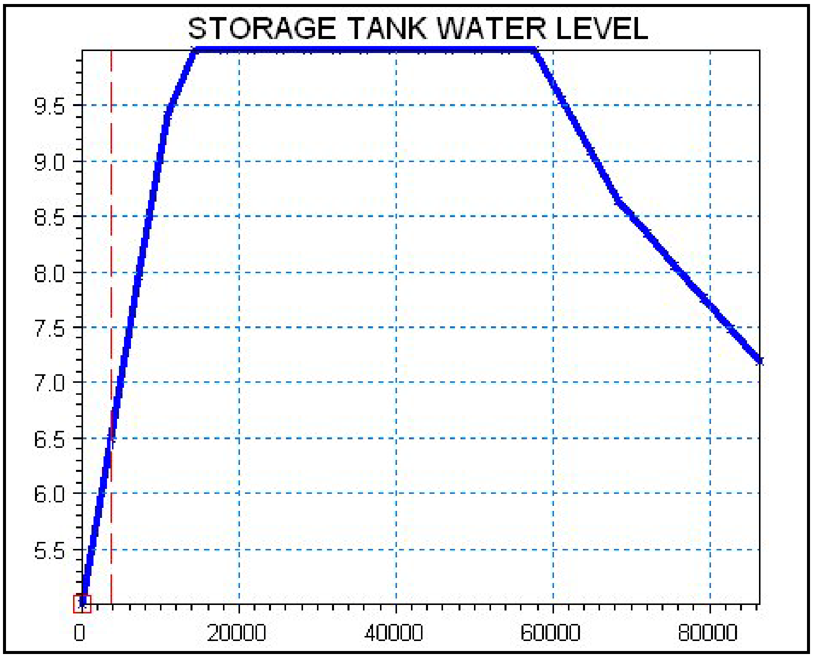


Figure 6.46 Extended period storage tank level results

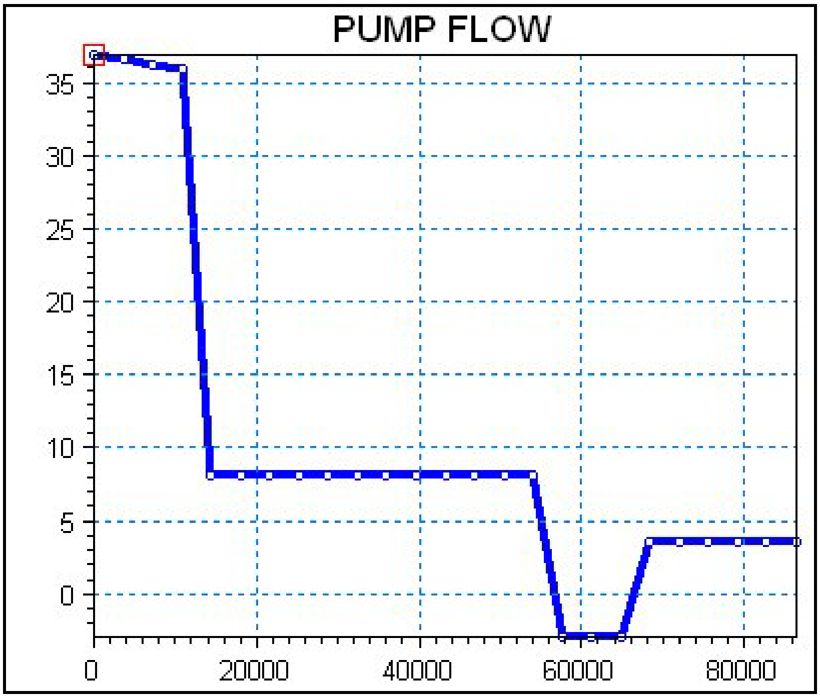


Figure 6.47 Extended period flow results for pump 14

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.

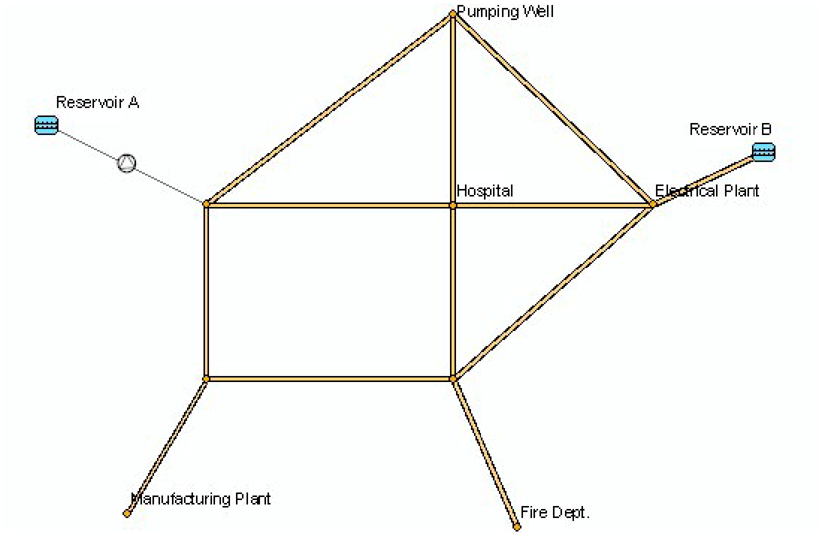


Figure 6.48 A schematic diagram of the pipe network system used to define a fire flow analysis

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.
5. Select Simulation | Run Simulation to display the Run Simulation dialog. Check on “Fire flow analysis“ and select «Run» to run the fire flow analysis.
6. 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.

7. 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.

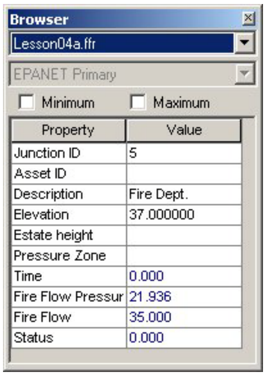


Figure 6.49 Results browser is used to view the simulated fire flow results

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:

1. 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).

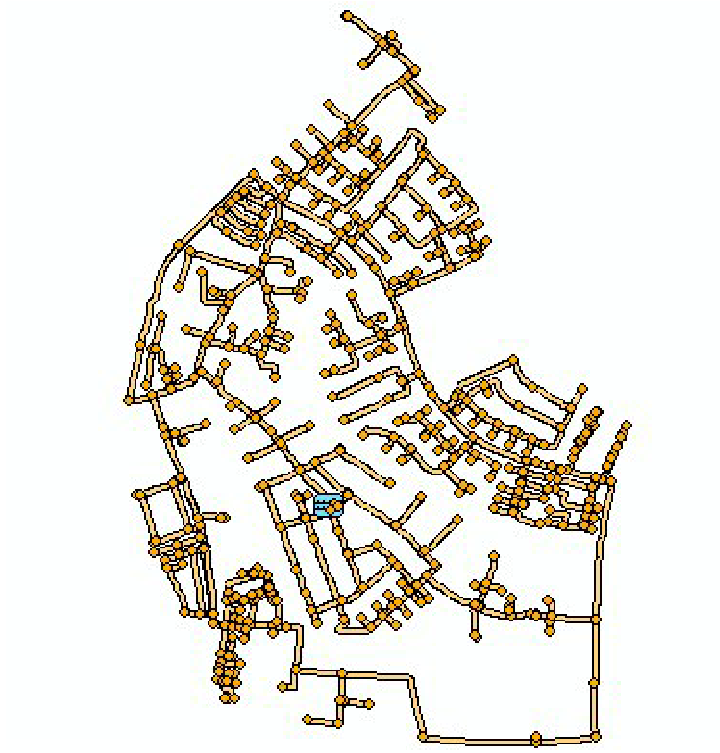


Figure 6.50 A schematic diagram of the pipe network system

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.

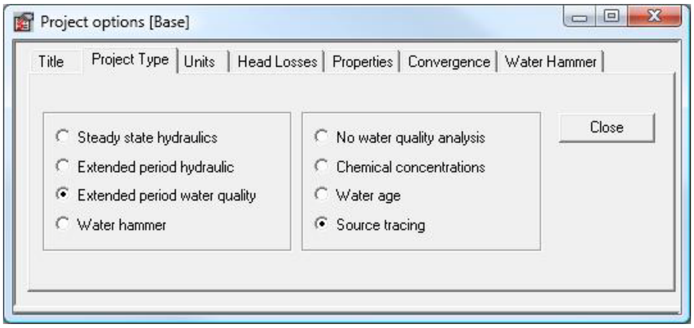


Figure 6.51 Project Options dialog box

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.

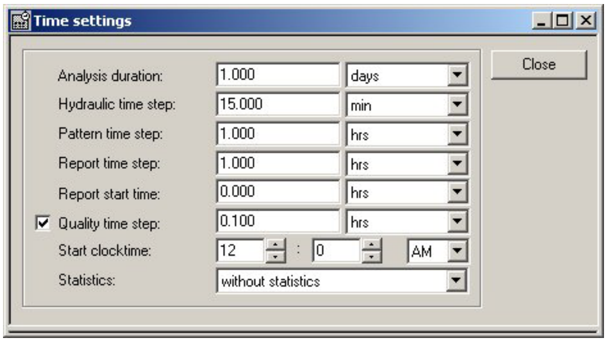


Figure 6.52 Time Settings dialog box

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.

Figure 6.53 Trace Node dialog box

2. 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.

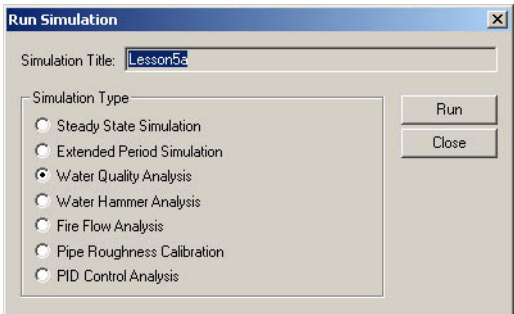


Figure 6.54 Run Simulation dialog box

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:

1. 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.

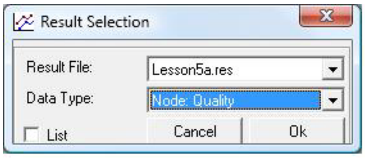


Figure 6.55 Results Selection dialog box

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.

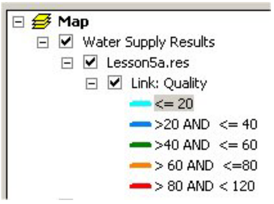


Figure 6.56 Result layer thematic mapping

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.

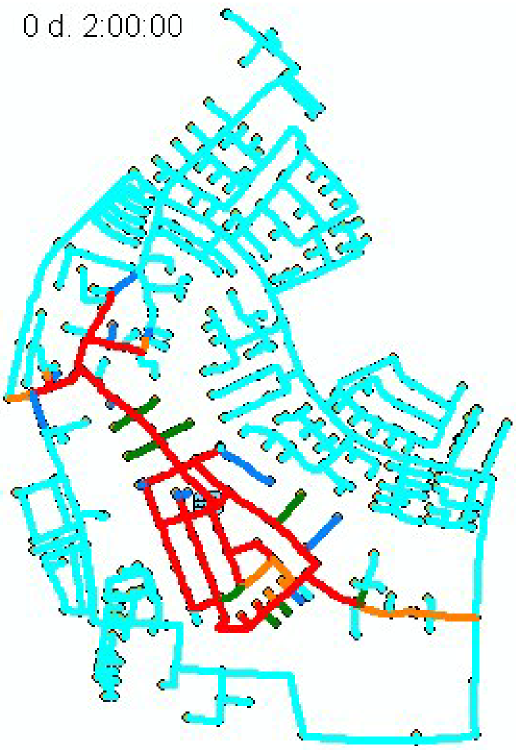


Figure 6.59 Analysis results on day 0 at 2:00 hours

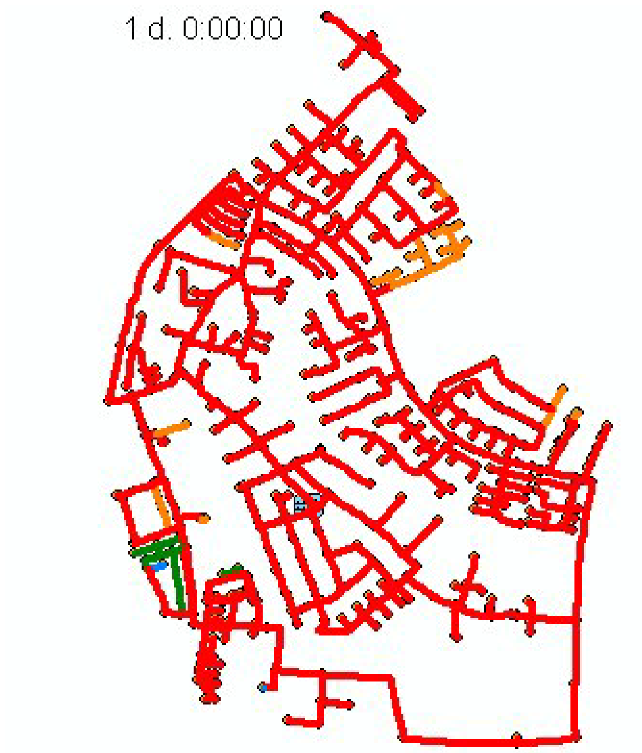


Figure 6.60 Analysis results on day 0 at 24:00 hours

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.

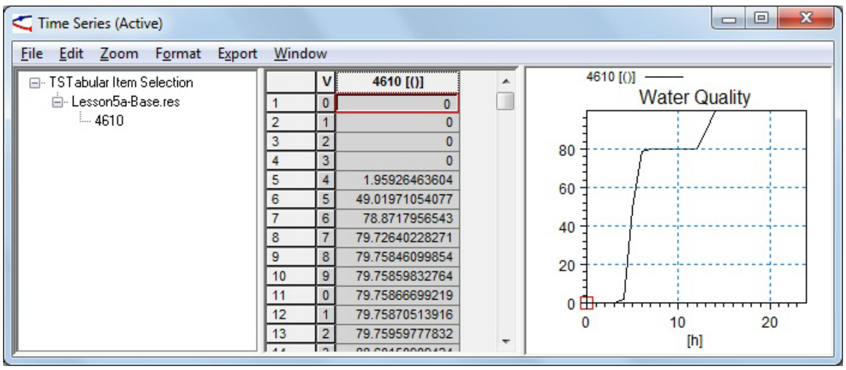


Figure 6.61 Source trace time series for the pipe 4610

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.

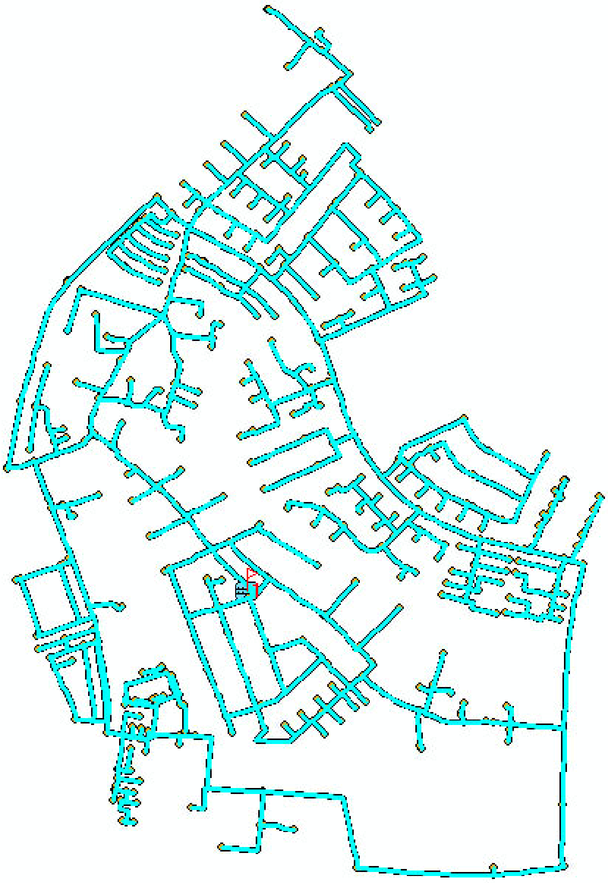


Figure 6.62 Forward tracing of water from reservoir 2000114

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.

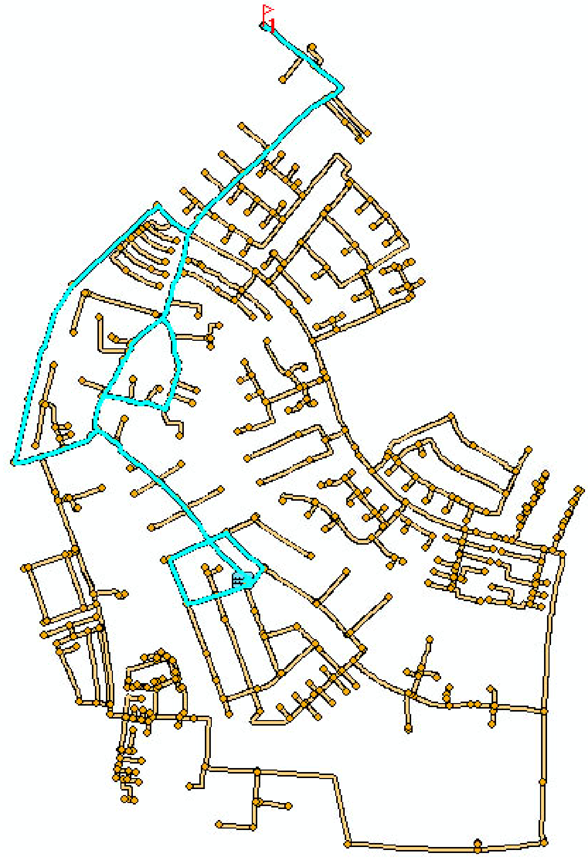


Figure 6.63 Backward tracing of water to junction node 11498

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).

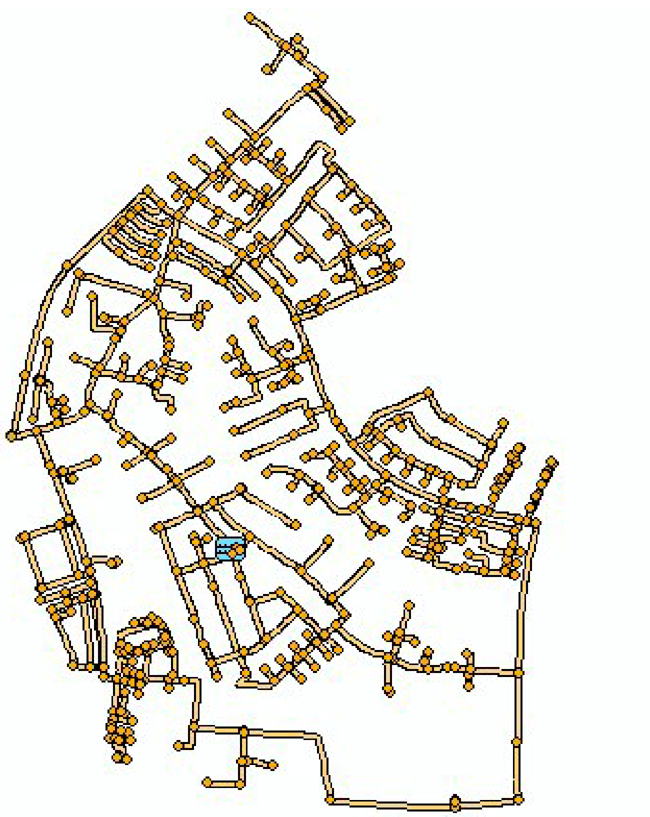


Figure 6.64 A schematic diagram of the pipe network system

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.

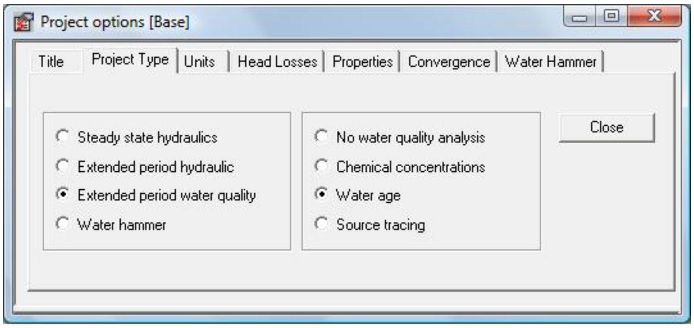


Figure 6.65 Project Options dialog box

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.

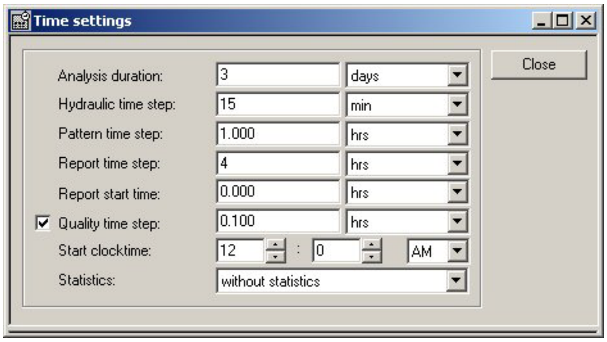


Figure 6.66 Time Settings dialog box

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.

For more details on viewing extended period analysis results, see the section titled Viewing the extended period Analysis Results in Lesson 3. In this lesson we will display the analysis results in the Horizontal Plan window.

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 LESSON6A.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.

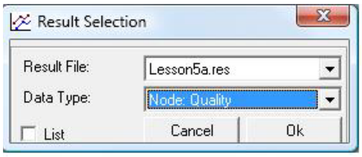


Figure 6.67 Results Selection dialog box

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.

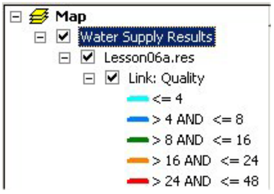


Figure 6.68 Result layer thematic mapping

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.

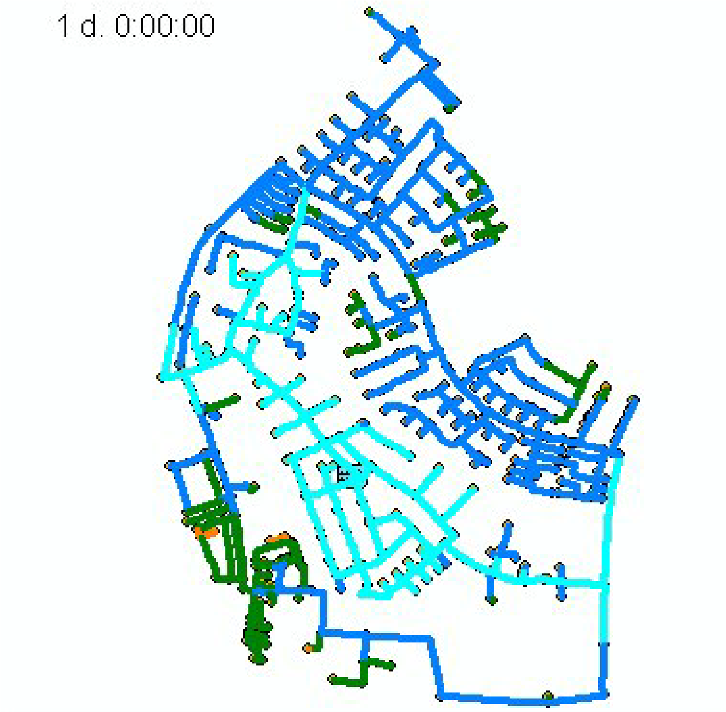


Figure 6.71 Analysis results on day 1 at 0:00 hours (24 hours)

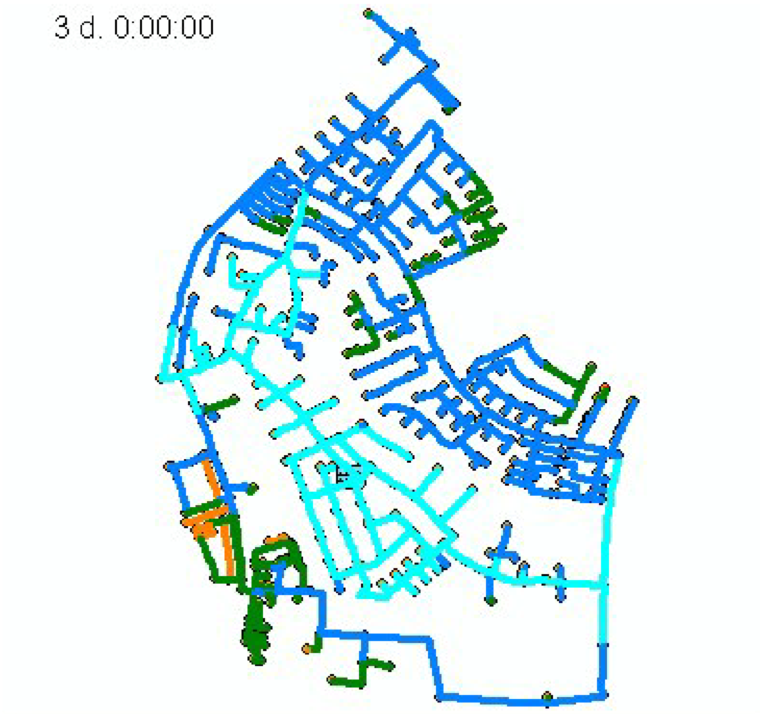
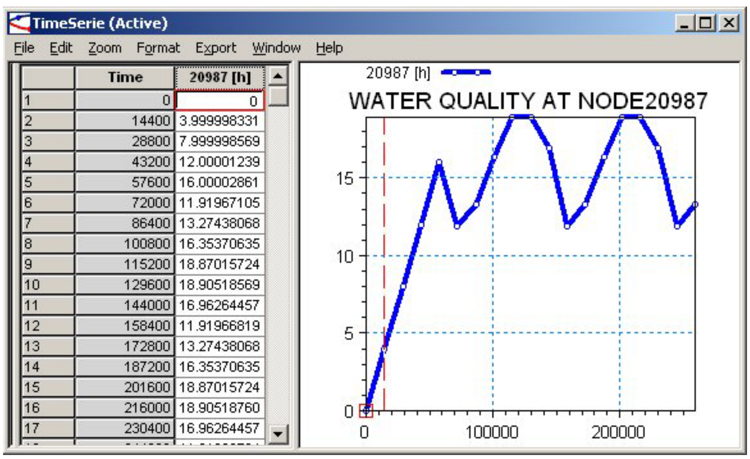


Figure 6.72 Analysis results on day 3 at 0:00 hours (72 hours)

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.



	Time	20987 [h]
1	0	0
2	14400	3.999998331
3	28800	7.999998569
4	43200	12.00001239
5	57600	16.00002861
6	72000	11.91967105
7	86400	13.27438068
8	100800	16.35370635
9	115200	18.87015724
10	129600	18.90518569
11	144000	16.96264457
12	158400	11.91966819
13	172800	13.27438068
14	187200	16.35370635
15	201600	18.87015724
16	216000	18.90518760
17	230400	16.96264457

Figure 6.73 Water age time series for the junction node 20987

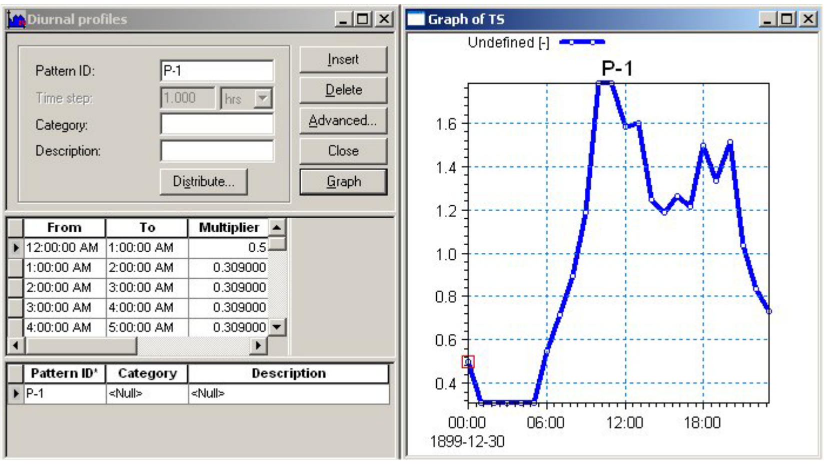


Figure 6.74 Diurnal Profile P-1 assigned to each junction node do model residential demand

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.

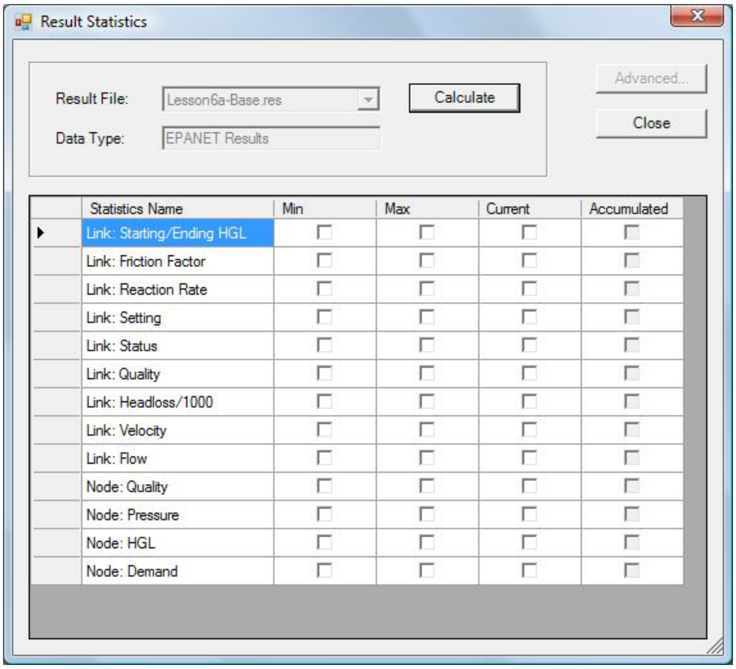


Figure 6.75 Results Statistics allows you to calculate minimum and maximum values for any selected result item

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.

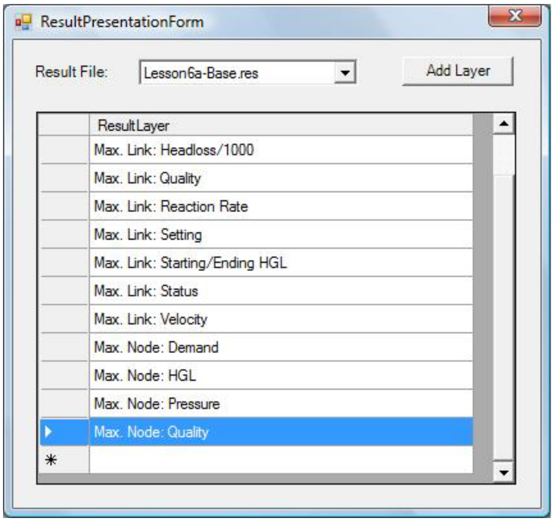


Figure 6.76 Add Statistic Layer

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).

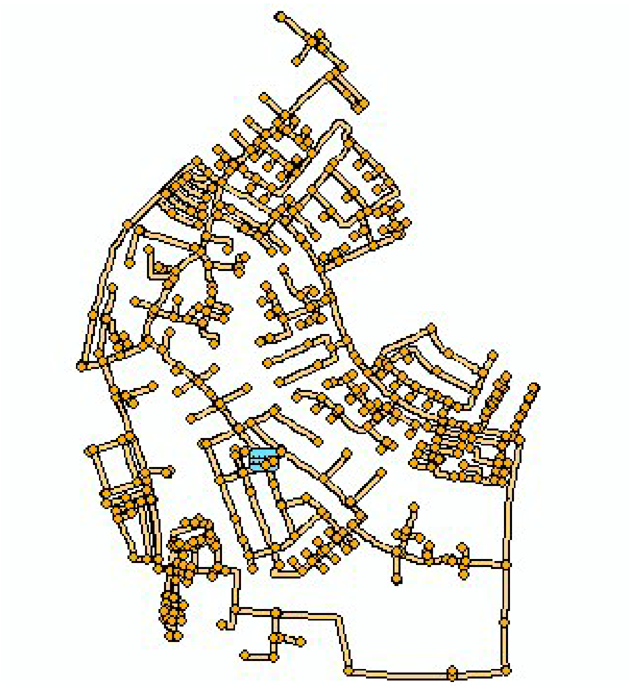


Figure 6.78 A schematic diagram of the pipe network system

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.

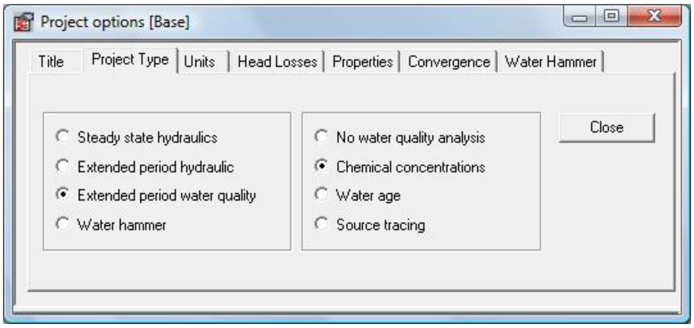


Figure 6.79 The Project Options dialog box

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.

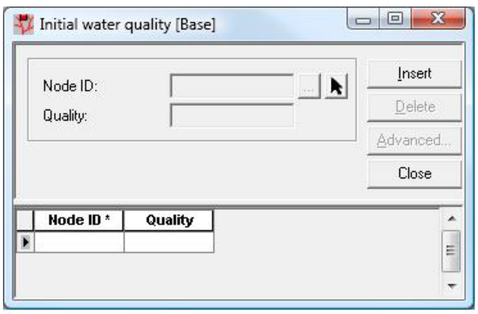


Figure 6.80 Initial Water Quality Editor dialog box

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.

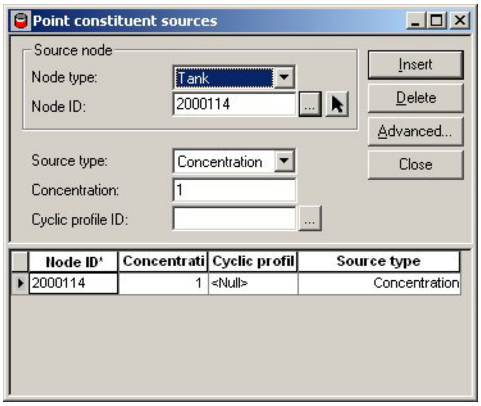


Figure 6.81 Point Constituent Source Editor dialog box

2. 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.

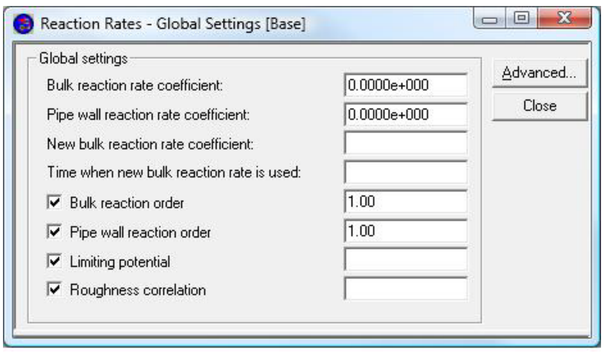


Figure 6.82 Reaction Rate Editor dialog box

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.
3. 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 LESSON7A.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.

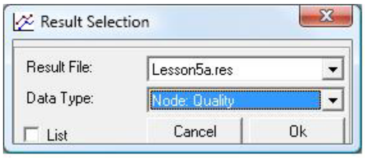


Figure 6.83 Results Selection dialog box

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 6.84 Result layer thematic mapping

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

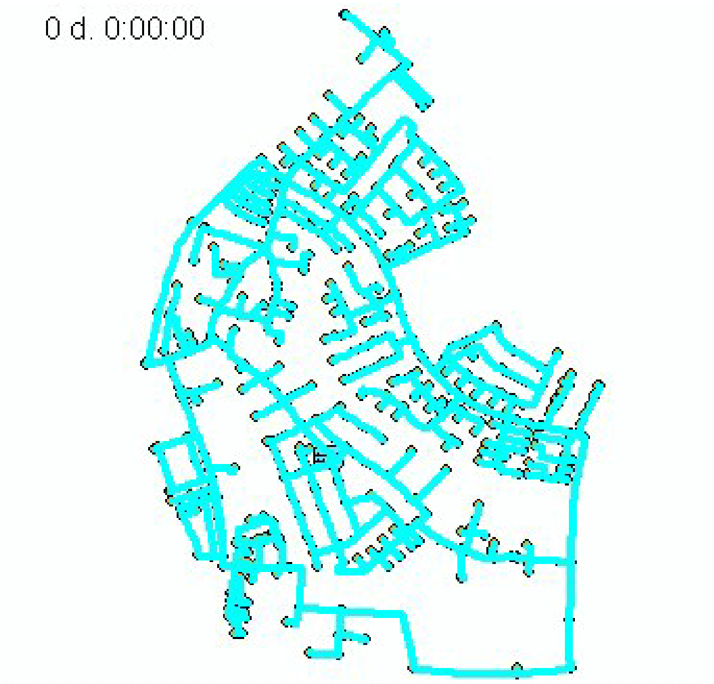


Figure 6.85 Analysis results on day 0 at 0:00 hours

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.

Figure 6.86 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 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.

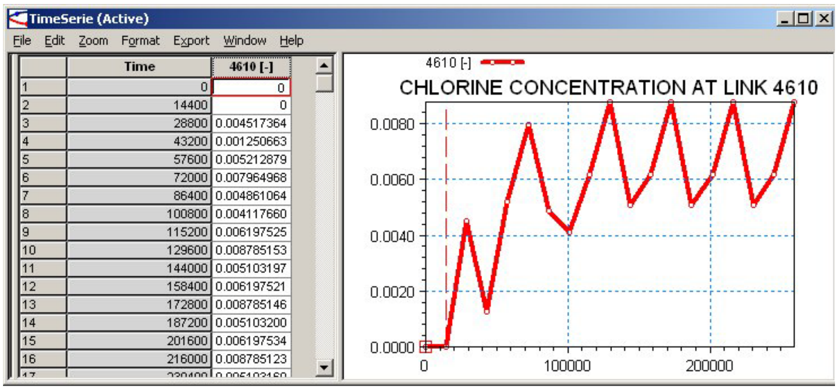


Figure 6.88 Water age time series for the pipe 4610

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.

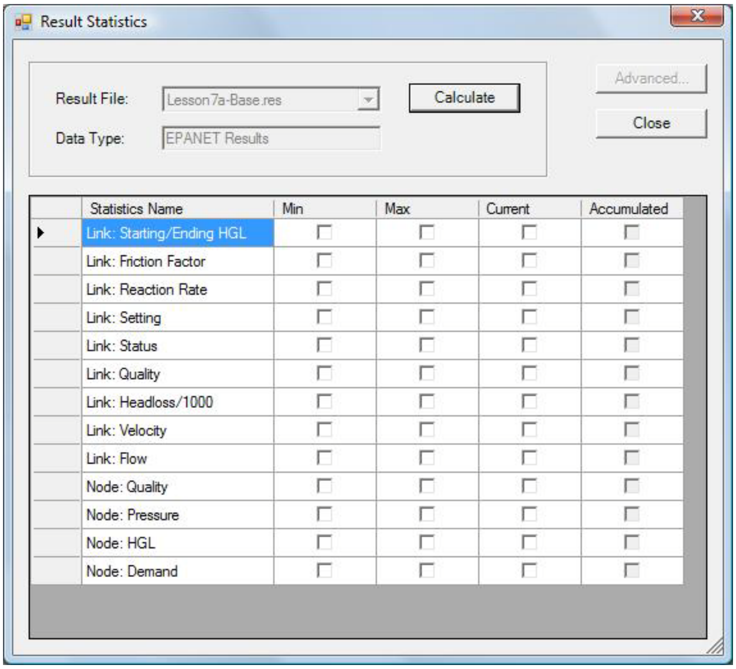


Figure 6.89 Results Statistics allows you to calculate minimum and maximum values for any selected result item

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.

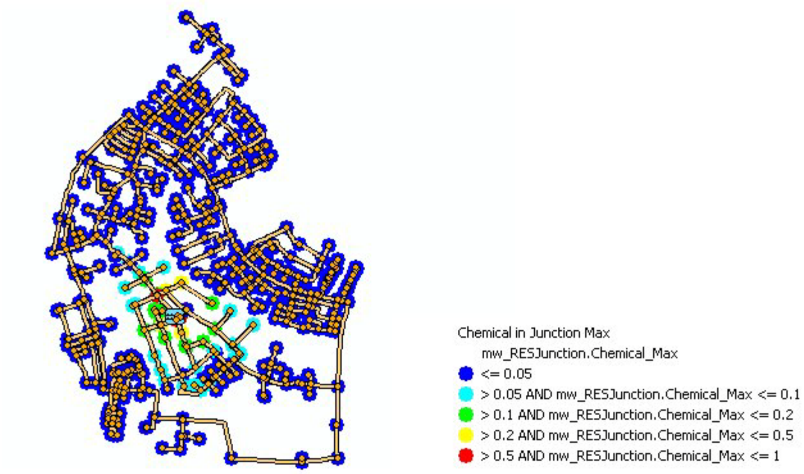


Figure 6.90 Maximum chlorine concentration at each junction node

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.

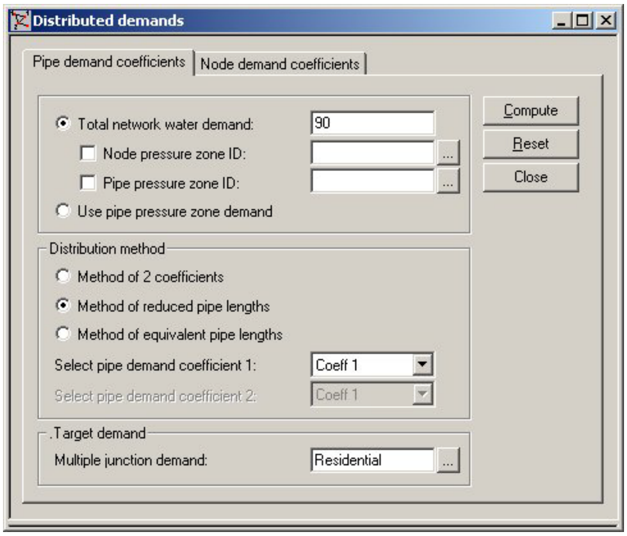


Figure 6.92 The Distributed Demands dialog box - Method of Reduced Pipe Lengths is used for distributing residential demand

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.

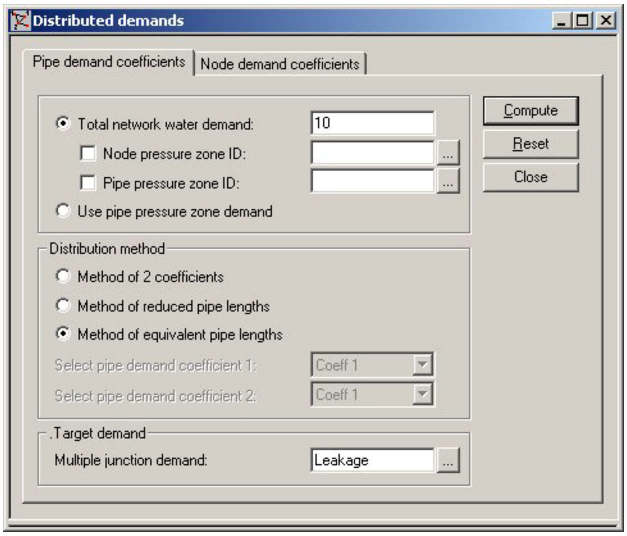


Figure 6.93 The Distributed Demands dialog box - Method of Equivalent Pipe Lengths is used for distributing demand corresponding to leakage

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.

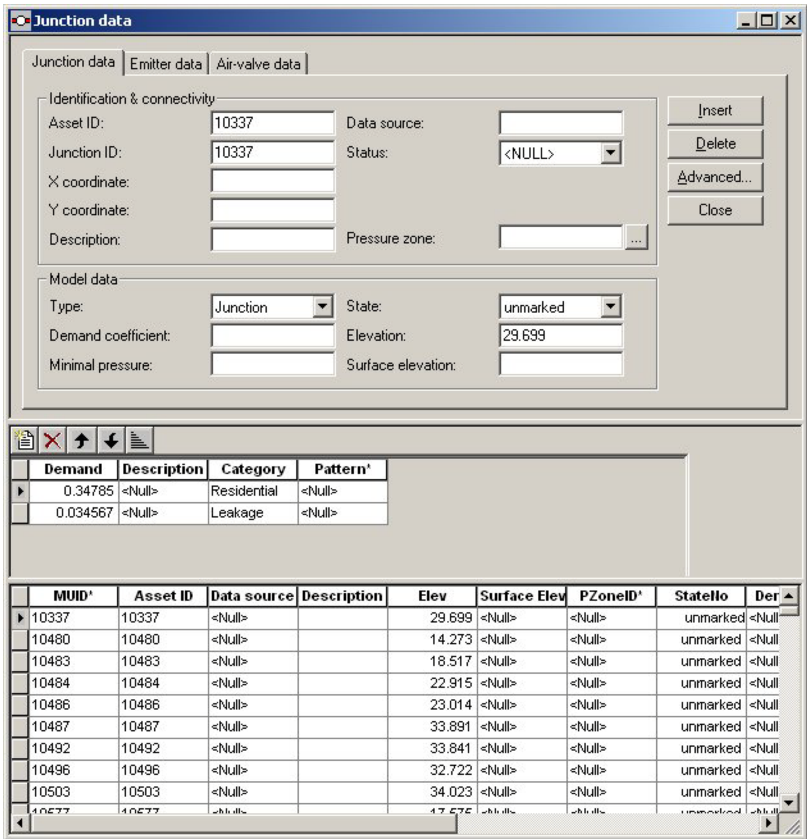
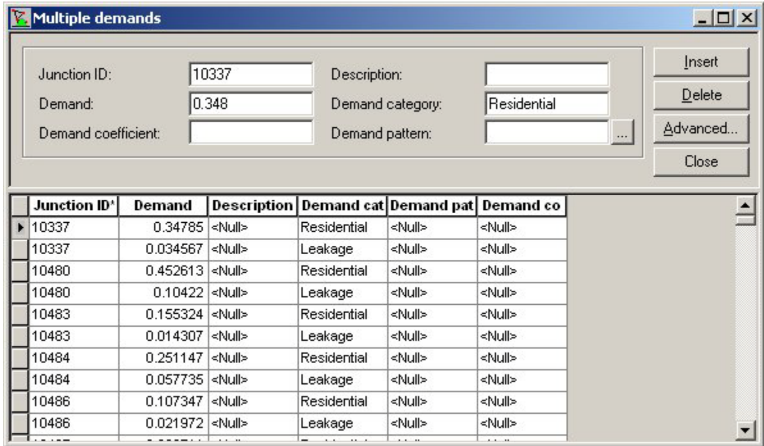


Figure 6.94 The Junction Editor - the computed demand values are shown in the demand field

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.

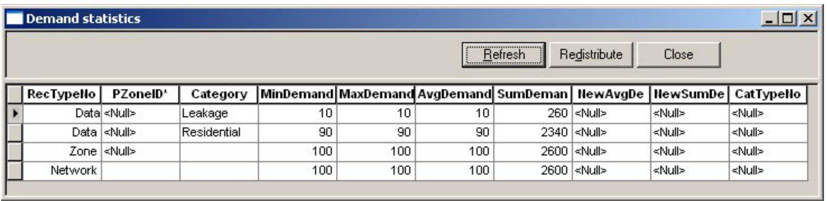


Junction ID	Demand	Description	Demand cat	Demand pat	Demand co
10337	0.34785	<Null>	Residential	<Null>	<Null>
10337	0.034567	<Null>	Leakage	<Null>	<Null>
10480	0.452613	<Null>	Residential	<Null>	<Null>
10480	0.10422	<Null>	Leakage	<Null>	<Null>
10483	0.155324	<Null>	Residential	<Null>	<Null>
10483	0.014307	<Null>	Leakage	<Null>	<Null>
10484	0.251147	<Null>	Residential	<Null>	<Null>
10484	0.057735	<Null>	Leakage	<Null>	<Null>
10486	0.107347	<Null>	Residential	<Null>	<Null>
10486	0.021972	<Null>	Leakage	<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.



RecTypeNo	PZoneID	Category	MinDemand	MaxDemand	AvgDemand	SumDeman	NewAvgDe	NewSumDe	CatTypeNo
Data	<Null>	Leakage	10	10	10	260	<Null>	<Null>	<Null>
Data	<Null>	Residential	90	90	90	2340	<Null>	<Null>	<Null>
Zone	<Null>		100	100	100	2600	<Null>	<Null>	<Null>
Network			100	100	100	2600	<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.

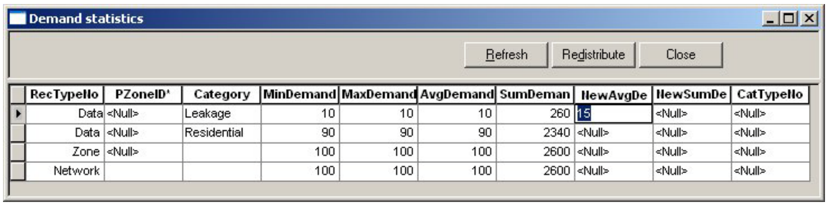


Figure 6.97 The zone or network demand can be redistributed from within Demand Statistics dialog box

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.

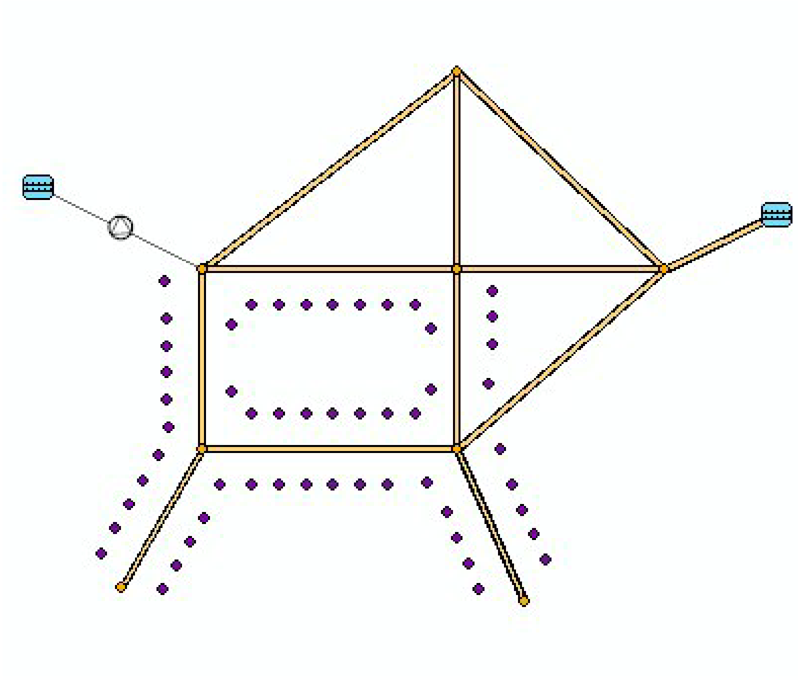


Figure 6.98 The pipe network system used in this lesson

Begin this lesson by loading LESSON10a.MDB from the LESSONS\LESSON10 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.

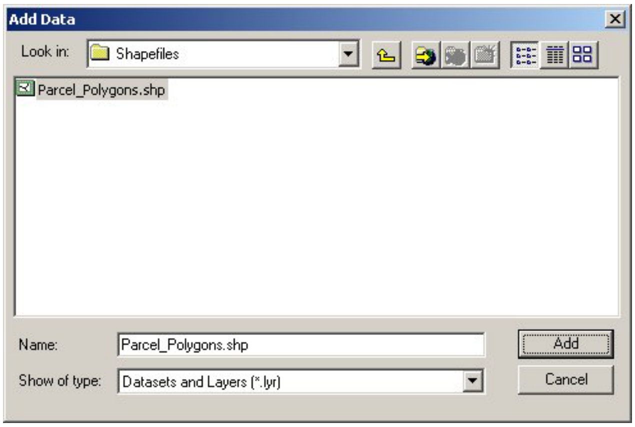


Figure 6.99 Use Add Data tool to load and display background shapefiles

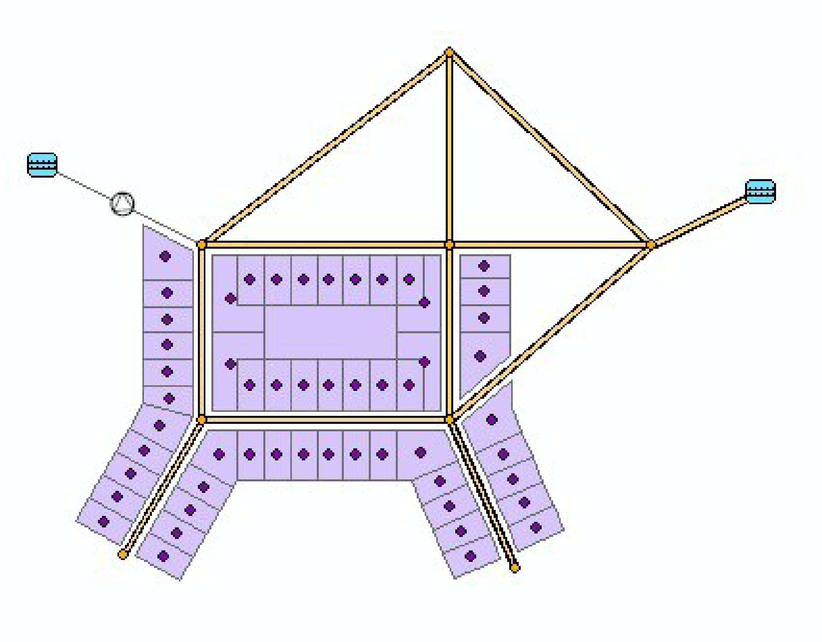


Figure 6.100 Demand points are displayed along with background Shapefiles in the Map window

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.

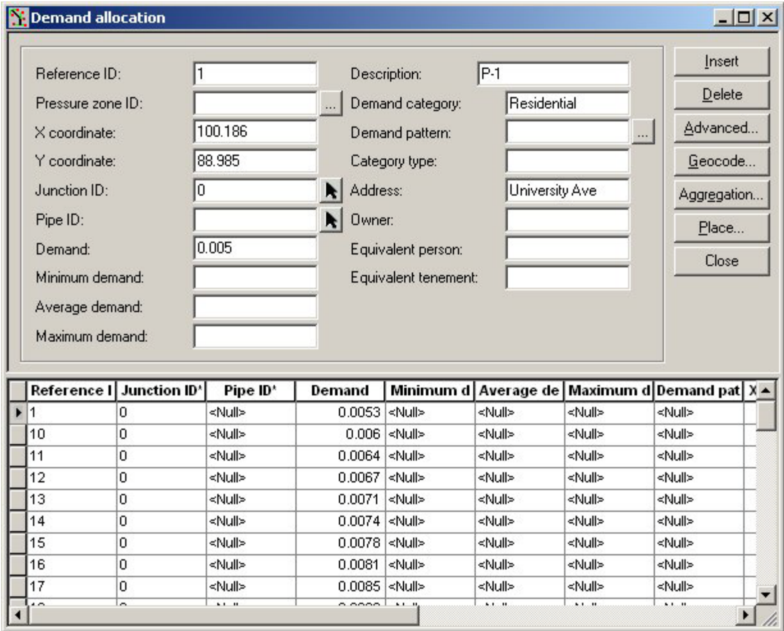


Figure 6.101 The Demand Allocation Editor

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.

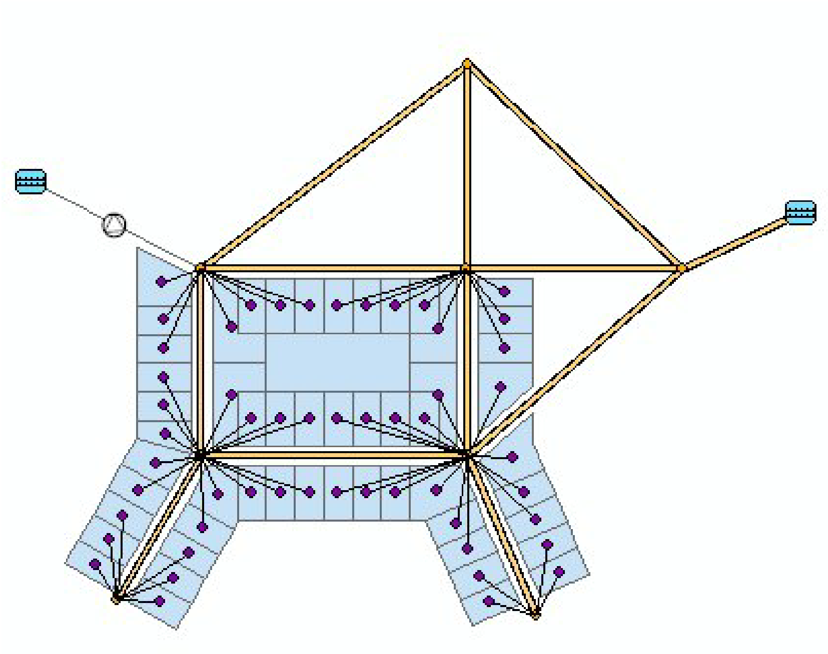


Figure 6.102 Demand points are geocoded to nodes

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.
2. 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.

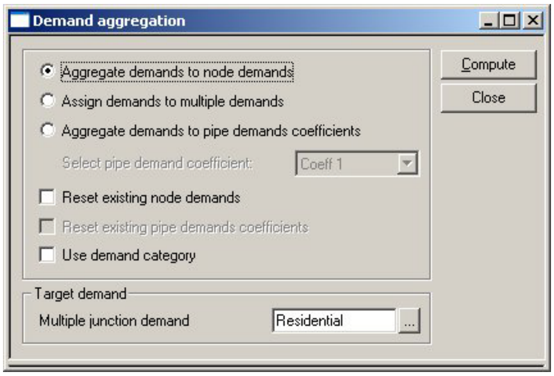


Figure 6.103 The Demand Aggregation dialog box

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 6.104 Pipeline overview

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.

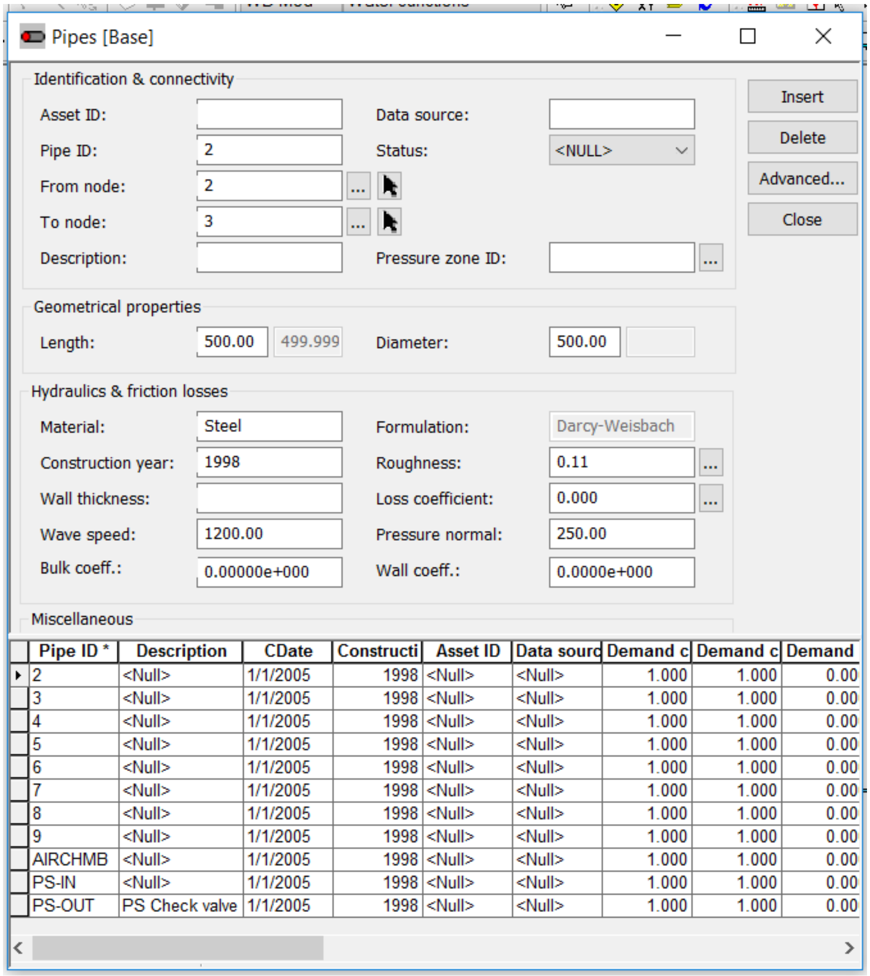


Figure 6.106 Pipe data

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.

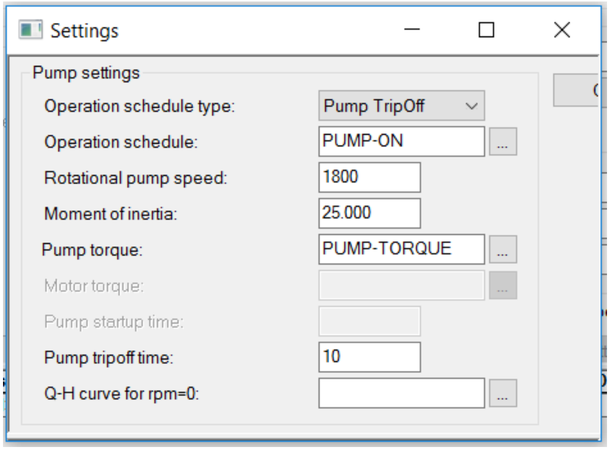


Figure 6.107 Pump data

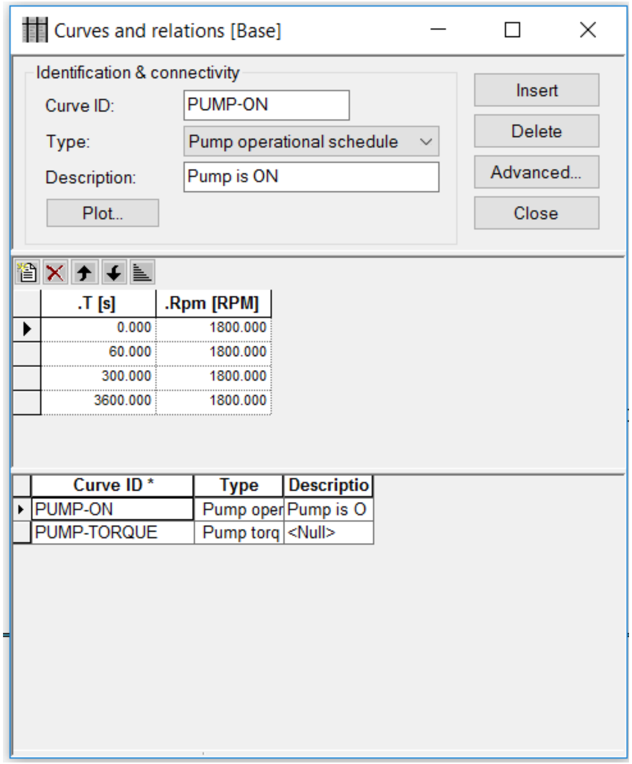


Figure 6.108 Curves data (pump schedule)

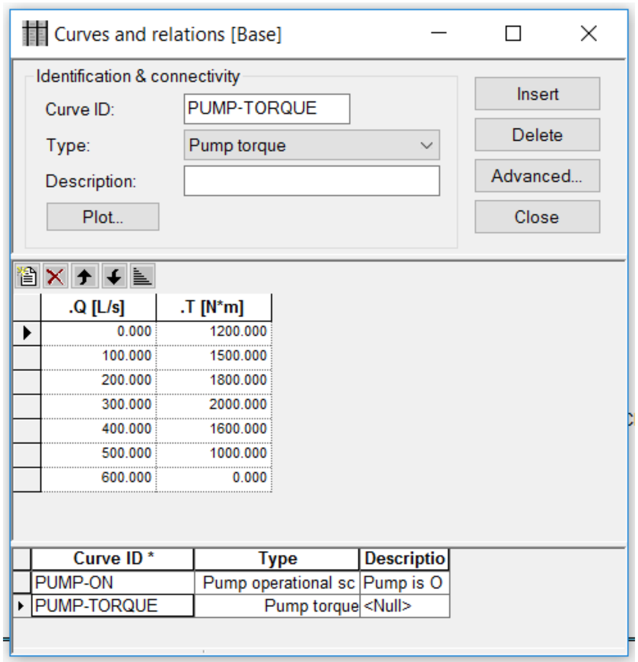


Figure 6.109 Curves data (pump torque)

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.

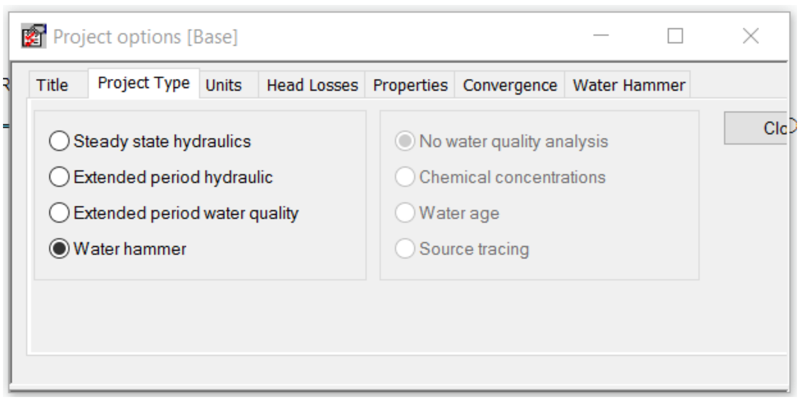


Figure 6.110 Project options

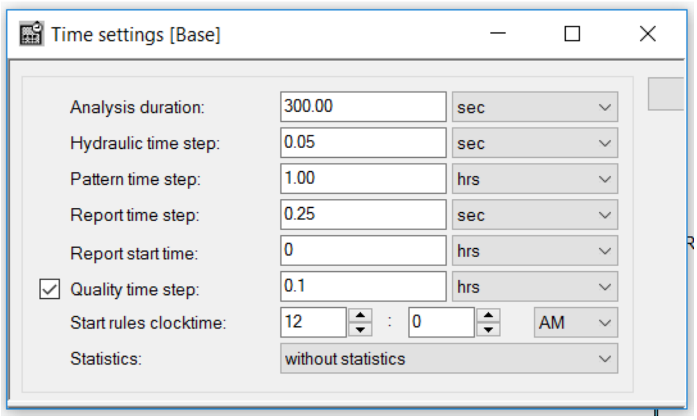


Figure 6.111 Time settings

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.

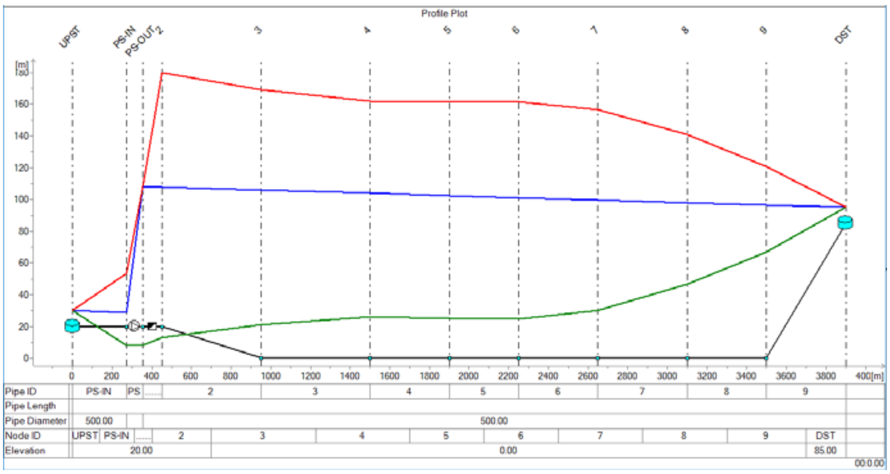


Figure 6.112 Profile plot with HGL levels

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 centre-line along the pipeline and cause water column separation.

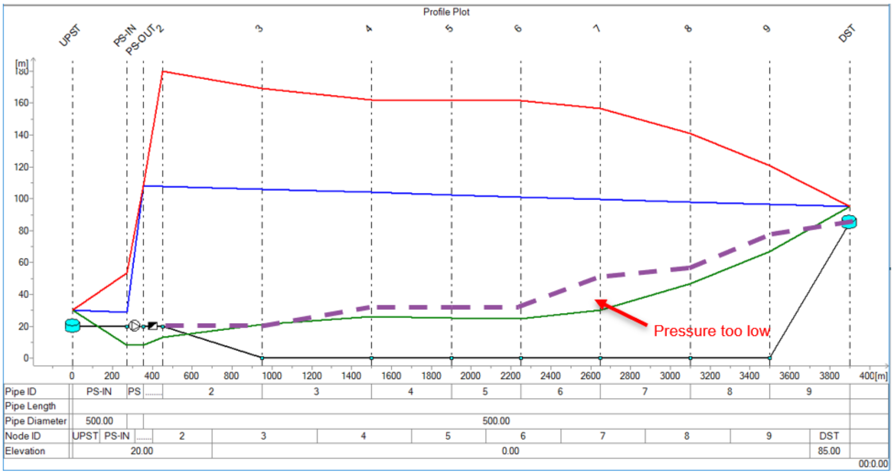


Figure 6.113 Profile plot with HGL levels and potential or future pipeline elevation

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, down-stream 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

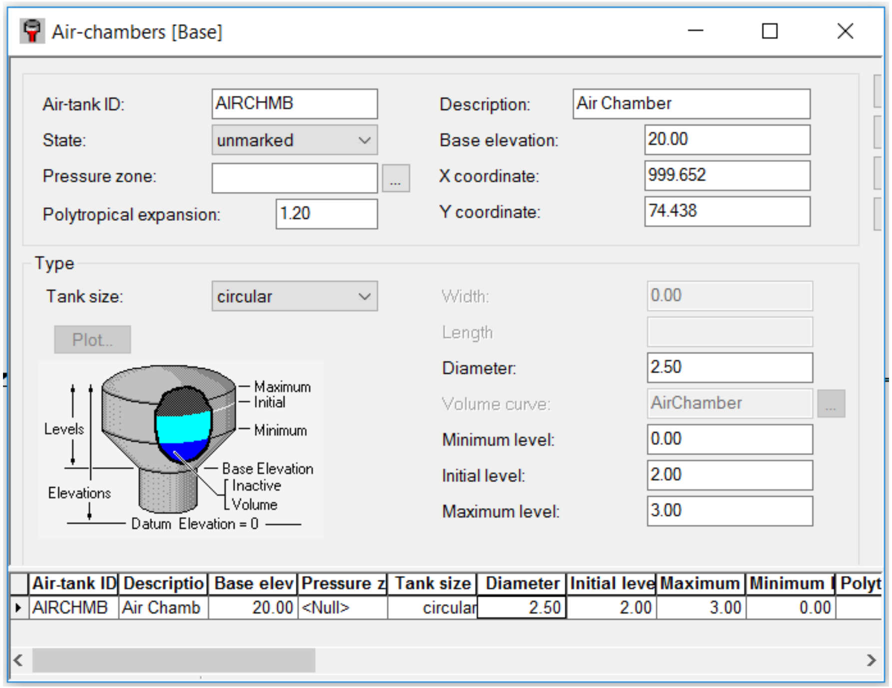


Figure 6.115 Air chamber editor

We will repeat the simulation and display the profile plot again.

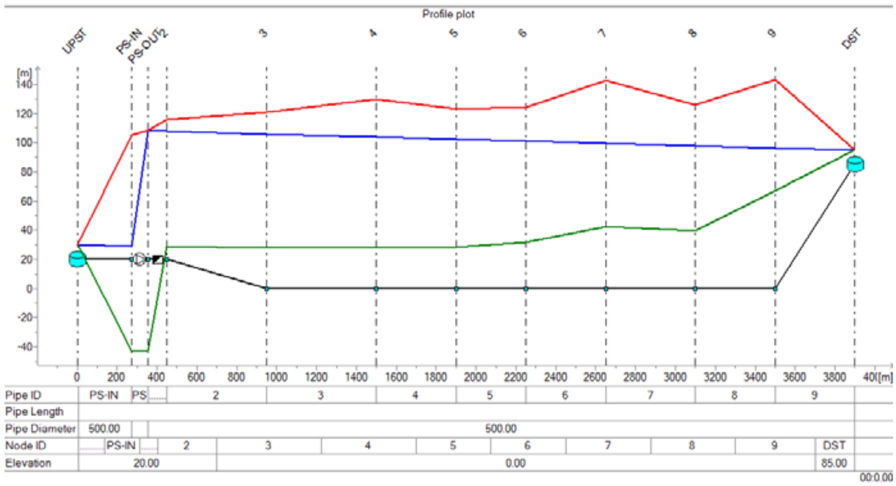


Figure 6.116 Profile plot levels

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.

INDEX