Sewer Networks

Version 11.0.0

USER GUIDE

www.technologismiki.com

TECHNO logismiki
Advanced Technical Software

5 Imitou str, 15561, Cholargos, Athens, Greece
tel: ++30 210 65 64 147 - fax: ++30 210 65 48 461
www.technologismiki.com - info@technologismiki.com

Sewer Networks

TechnoLogismiki

Sewer Networks

Publisher

TechnoLogismiki

Editors

Fotis Fotopoulos
Aristotelis Charalampakis

Technical Assistance

Antigoni Egglezou

Printed: September 2012 in Athens, Greece.

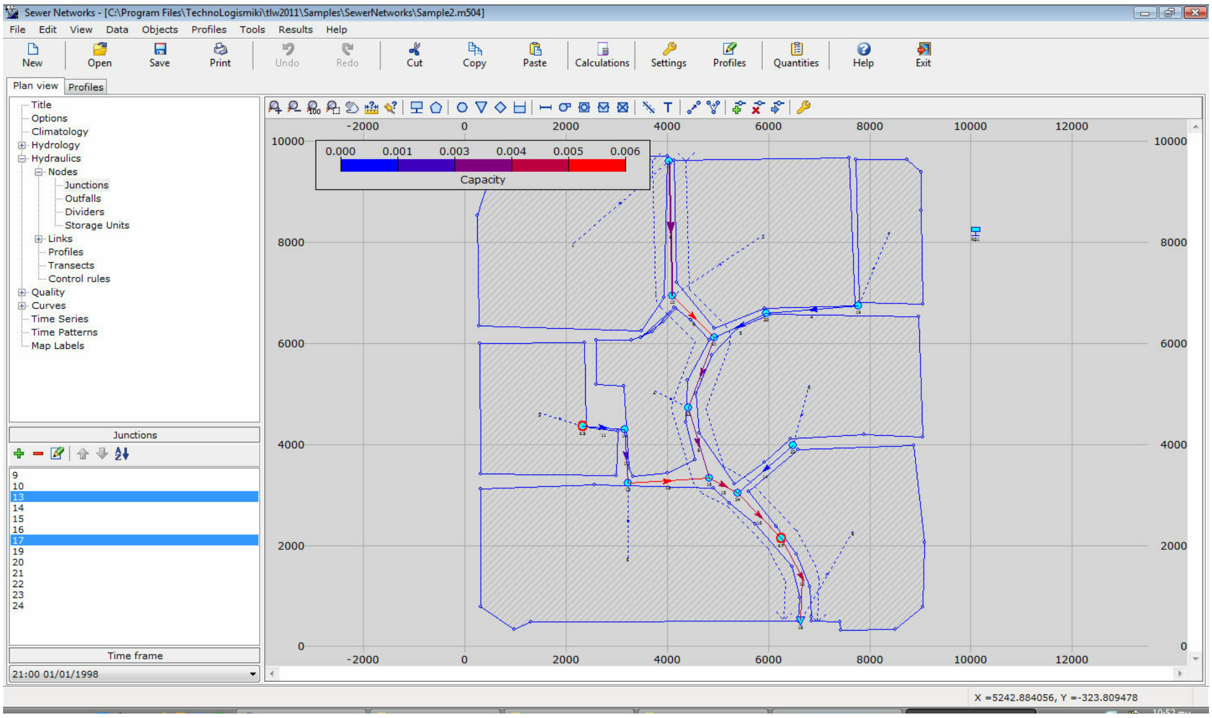
Table of Contents

Chapter I

1 About the program

1.1 What does the program do?

This program calculates sewer and storm networks in which flow is under pressure and/or with free surface. Apart from hydraulic calculations, the program can be used for water quality checks (for example, for combined networks). The program is based on and fully compatible with EPA SWMM 5. It offers unique design tools and extensive import/export capabilities through DXF/GTM/GIS files.



Some of the unique features of **Sewer Networks** are:

- Handles Sewer / Storm / Combined networks
- Easy data input in plan view and/or profile view with embedded spreadsheets.
- Generation of runoff by raingages, IDF curves, constant rain intensity or by direct input
- Fully compatible with American regulations (ASCE & WPCF) as well as Greek Regulations
- Easy data input from drawings and / or spreadsheets
- Automatic data input based on logical rules
- Active profile drawings with CAD capabilities
- Conduit / Manhole / Trench specifications
- One-click generation of professional reports

- One-click generation of plan view and profile drawings
- One-click generation of quantities report, including pipe lengths, excavations, backfill etc.

1.2 Minimum requirements

The minimum requirements for the usage of the programs are the following:

- Windows 2000/ XP/ 2003/ Vista/ 7 (for each case, the latest service packs, updates & patches must be installed)
- Pentium III 800 MHz
- 800x600 with 256 color palette
- 700 MB free disk space
- CD-Rom

If your system does not meet one or more of the above requirements, it is highly recommended that you upgrade it before installing the programs. The recommended system configuration is the following:

- Windows 2000/ XP/ 2003/ Vista/ 7 (for each case, the latest service packs, updates & patches must be installed)
- Pentium IV 2.0 GHz
- 1280x768 with 16-bit color palette
- 1.2 GB free disk space
- CD-Rom
- Internet connection

1.3 Technical support

Support through the Internet

TechnoLogismiki offers technical support 24 hours per day, 365 days per year, through the web site where you can get information on the latest programs and services.

Support by e-mail

Please use the dedicated e-mail addresses for better customer service:

- for questions regarding sales: sales@technologismiki.com
- for questions regarding the usage of programs: support@technologismiki.com
- for any other question or comment: info@technologismiki.com

The normal response time is within two business days. If your inquiry cannot be answered by e-mail, a customer service representative will contact you by telephone.

Interactive Support

Business days, 09:00 - 17:00 Eastern European Time:

- Telephone [3 lines]: ++30-210-656-4147
- Fax: ++30-210-654-8461
- Address: 5, Imittou street, Cholargos, 15561, Athens, Greece.

Chapter II

2 User Interface

2.1 Methodology

2.1.1 Computational Methods

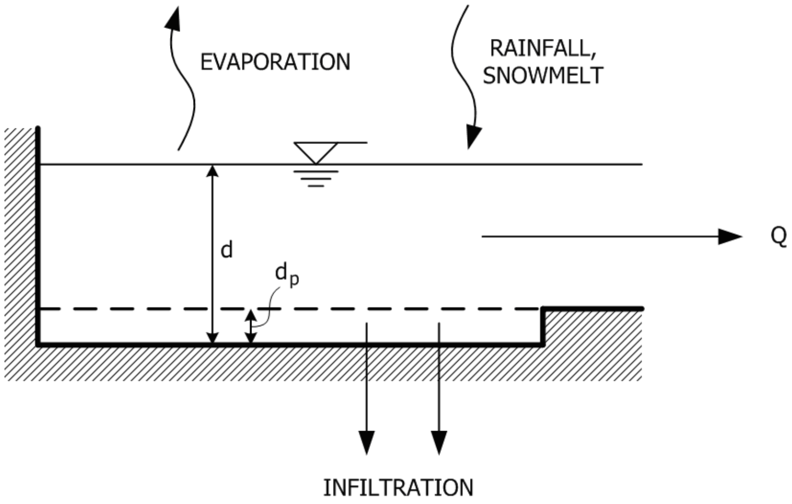
The program and this manual are based on EPA SWMM 5, which is a dynamic rainfall-runoff simulation model used for single event or long-term (continuous) simulation of runoff quantity and quality from primarily urban areas. The routing algorithm transports the runoff through a system of pipes, channels, storage/treatment devices, pumps, and regulators. The program tracks the quantity and quality of runoff generated within each subcatchment, and the flow rate, flow depth, and quality of water in each pipe and channel during a simulation period comprised of multiple time steps.

This section briefly describes the methodology employed by the program regarding the following topics:

- Surface Runoff
- Infiltration
- Groundwater
- Snowmelt
- Routing
- Surface Ponding
- Water Quality Ponding

2.1.2 Surface Runoff

The conceptual view of surface runoff used by Sewer Networks is illustrated in the following figure. Each subcatchment surface is treated as a nonlinear reservoir. Inflow comes from precipitation and any designated upstream subcatchments. There are several outflows, including infiltration, evaporation, and surface runoff. The capacity of this "reservoir" is the maximum depression storage, which is the maximum surface storage provided by ponding, surface wetting, and interception. Surface runoff per unit area, Q, occurs only when the depth of water in the "reservoir" exceeds the maximum depression storage, dp, in which case the outflow is given by Manning's equation. Depth of water over the subcatchment is continuously updated with time by solving numerically a water balance equation over the subcatchment.



2.1.3 Infiltration

Infiltration is the process of rainfall penetrating the ground surface into the unsaturated soil zone of pervious subcatchments areas. The program offers three choices for modeling infiltration:

Horton's equation

This method is based on empirical observations showing that infiltration decreases exponentially from an initial maximum rate to some minimum rate over the course of a long rainfall event. Input parameters required by this method include the maximum and minimum infiltration rates, a decay coefficient that describes how fast the rate decreases over time, and a time it takes a fully saturated soil to completely dry.

Green-Ampt Method

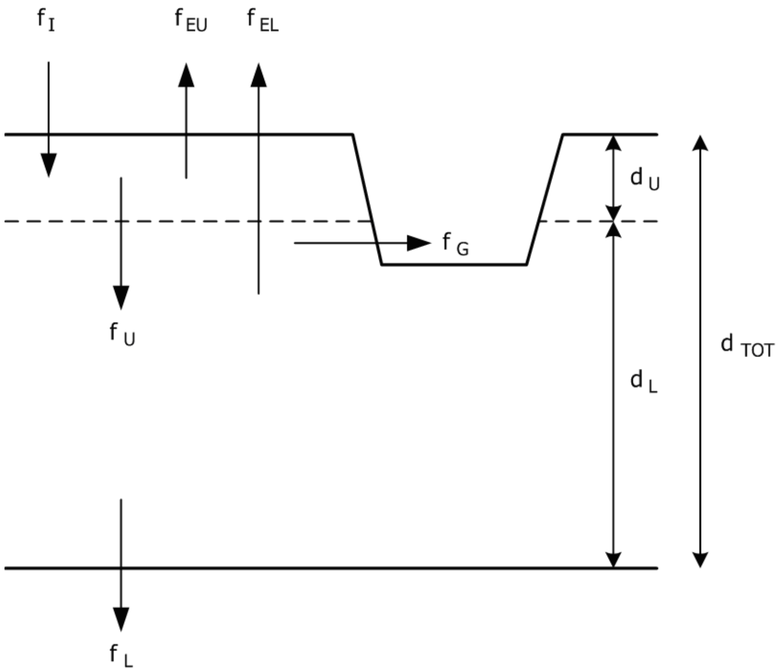
This method for modeling infiltration assumes that a sharp wetting front exists in the soil column, separating soil with some initial moisture content below from saturated soil above. The input parameters required are the initial moisture deficit of the soil, the soil's hydraulic conductivity, and the suction head at the wetting front.

Curve Number Method

This approach is adopted from the NRCS (SCS) Curve Number method for estimating runoff. It assumes that the total infiltration capacity of a soil can be found from the soil's tabulated Curve Number. During a rain event this capacity is depleted as a function of cumulative rainfall and remaining capacity. The input parameters for this method are the curve number, the soil's hydraulic conductivity (used to estimate a minimum separation time for distinct rain events), and a time it takes a fully saturated soil to completely dry.

2.1.4 Groundwater

The following figure is a definitional sketch of the two-zone groundwater model that is used by the program. The upper zone is unsaturated with a variable moisture content. The lower zone is fully saturated and therefore its moisture content is fixed at the soil porosity.



The fluxes shown in the figure, expressed as volume per unit area per unit time, consist of the following :

f_I infiltration from the surface
f_{EU} evapotranspiration from the upper zone which is a fixed fraction of the unused surface evaporation
f_U percolation from the upper to lower zone which depends on the upper zone moisture content θ and depth d_U
f_{EL} evapotranspiration from the lower zone, which is a function of the depth of the upper zone d_U
fL percolation from the lower zone to deep groundwater which depends on the lower zone depth d_L
fG lateral groundwater interflow to the drainage system, which depends on the lower zone depth d_L as well as the depth in the receiving channel or node.

After computing the water fluxes that exist during a given time step, a mass balance is written for the change in water volume stored in each zone so that a new water table depth and unsaturated zone moisture content can be computed for the next time step.

2.1.5 Snowmelt

The snowmelt routine is a part of the runoff modeling process. It updates the state of the snow packs associated with each subcatchment by accounting for snow accumulation, snow
redistribution by areal depletion and removal operations, and snow melt via heat budget accounting. Any snowmelt coming off the pack is treated as an additional rainfall input onto the subcatchment.

At each runoff time step the following computations are made:

1. Air temperature and melt coefficients are updated according to the calendar date.
2. Any precipitation that falls as snow is added to the snow pack.
3. Any excess snow depth on the plowable area of the pack is redistributed according to the removal parameters established for the pack.
4. Areal coverages of snow on the impervious and pervious areas of the pack are reduced according to the Areal Depletion Curves defined for the study area
5. The amount of snow in the pack that melts to liquid water is found using
   - a heat budget equation for periods with rainfall, where melt rate increases with increasing air temperature, wind speed, and rainfall intensity
   - a degree-day equation for periods with no rainfall, where melt rate equals the product of a melt coefficient and the difference between the air temperature and the pack's base melt temperature.
6. If no melting occurs, the pack temperature is adjusted up or down based on the product of the difference between current and past air temperatures and an adjusted melt coefficient. If melting occurs, the temperature of the pack is increased by the equivalent heat content of the melted snow, up to the base melt temperature. Any remaining melt liquid beyond this is available to runoff from the pack.
7. The available snowmelt is then reduced by the amount of free water holding capacity remaining in the pack. The remaining melt is treated the same as an additional rainfall input onto the subcatchment.

2.1.6 Routing

Flow routing within a conduit link is governed by the conservation of mass and momentum equations for gradually varied, unsteady flow (i.e., the Saint Venant flow equations). The program offers three levels of sophistication used to solve these equations:

- Steady Flow Routing
- Kinematic Wave Routing
- Dynamic Wave Routing

2.1.6.1 Steady Flow Routing

Steady Flow routing represents the simplest type of routing possible (actually no routing) by assuming that within each computational time step flow is uniform and steady. Thus it simply translates inflow hydrographs at the upstream end of the conduit to the downstream end, with no delay or change in shape. The normal flow equation is used to relate flow rate to flow area (or depth).

This type of routing cannot account for channel storage, backwater effects, entrance/exit losses, flow reversal or pressurized flow. It can only be used with dendritic conveyance networks, where each node has only a single outflow link (unless the node is a divider in which case two outflow links are required). This form of routing is insensitive to the time step employed and is really only appropriate for preliminary analysis using long-term continuous simulations.

NOTE: Older versions of the program use steady flow routing.

2.1.6.2 Kinematic Wave Routing

This routing method solves the continuity equation along with a simplified form of the momentum equation in each conduit. The latter requires that the slope of the water surface equal the slope of the conduit.

The maximum flow that can be conveyed through a conduit is the full normal flow value. Any flow in excess of this entering the inlet node is either lost from the system or can pond atop the inlet node and be re-introduced into the conduit as capacity becomes available.

Kinematic wave routing allows flow and area to vary both spatially and temporally within a conduit. This can result in attenuated and delayed outflow hydrographs as inflow is routed through the channel. However this form of routing cannot account for backwater effects, entrance/exit losses, flow reversal, or pressurized flow, and is also restricted to dendritic network layouts. It can usually maintain numerical stability with moderately large time steps, on the order of 5 to 15 minutes. If the aforementioned effects are not expected to be significant then this alternative can be an accurate and efficient routing method, especially for long-term simulations.

2.1.6.3 Dynamic Wave Routing

Dynamic Wave routing solves the complete one-dimensional Saint - Venant flow equations and therefore produces the most theoretically accurate results. These equations consist of the continuity and momentum equations for conduits and a volume continuity equation at nodes.

With this form of routing it is possible to represent pressurized flow when a closed conduit becomes full, such that flows can exceed the full normal flow value. Flooding occurs when the water depth at a node exceeds the maximum available depth, and the excess flow is either lost from the system or can pond atop the node and re-enter the drainage system.

Dynamic wave routing can account for channel storage, backwater, entrance/exit losses, flow reversal, and pressurized flow. Because it couples together the solution for both water levels at nodes and flow in conduits it can be applied to any general network layout, even those containing multiple downstream diversions and loops. It is the method of choice for systems subjected to significant backwater effects due to downstream flow restrictions and with flow regulation via weirs and orifices. This generality comes at a price of having to use much smaller time steps, on the order of a minute or less (the program will automatically reduce the user-defined maximum time step as needed to maintain numerical stability).

Each of these routing methods employs the Manning equation to relate flow rate to flow depth and bed (or friction) slope. The one exception is for circular Force Main shapes, where the Hazen-Williams equation is used instead.

2.1.7 Surface Ponding

Normally in flow routing, when the flow into a junction exceeds the capacity of the system to transport it further downstream, the excess volume overflows the system and is lost. An option exists to have instead the excess volume be stored atop the junction, in a ponded fashion, and be reintroduced into the system as capacity permits.

Under Steady and Kinematic Wave flow routing, the ponded water is stored simply as an excess volume. For Dynamic Wave routing, which is influenced by the water depths maintained at nodes, the excess volume is assumed to pond over the node with a constant surface area. This amount of surface area is an input parameter supplied for the junction.

Alternatively, the user may wish to represent the surface overflow system explicitly. In open channel systems this can include road overflows at bridges or culvert crossings as well as additional floodplain storage areas.

In closed conduit systems, surface overflows may be conveyed down streets, alleys, or other surface routes to the next available stormwater inlet or open channel. Overflows may also be impounded in surface depressions such as parking lots, back yards or other areas.

2.1.8 Water Quality Ponding

Water quality routing within conduit links assumes that the conduit behaves as a continuously stirred tank reactor (CSTR). Although a plug flow reactor assumption might be more realistic, the differences will be small if the travel time through the conduit is on the same order as the routing time step. The concentration of a constituent exiting the conduit at the end of a time step is found by integrating the conservation of mass equation, using average values for quantities that might change over the time step such as flow rate and conduit volume.

Water quality modeling within storage unit nodes follows the same approach used for conduits. For other types of nodes that have no volume, the quality of water exiting the node is simply the mixture concentration of all water entering the node.

2.2 Main Window (Plan View tab)

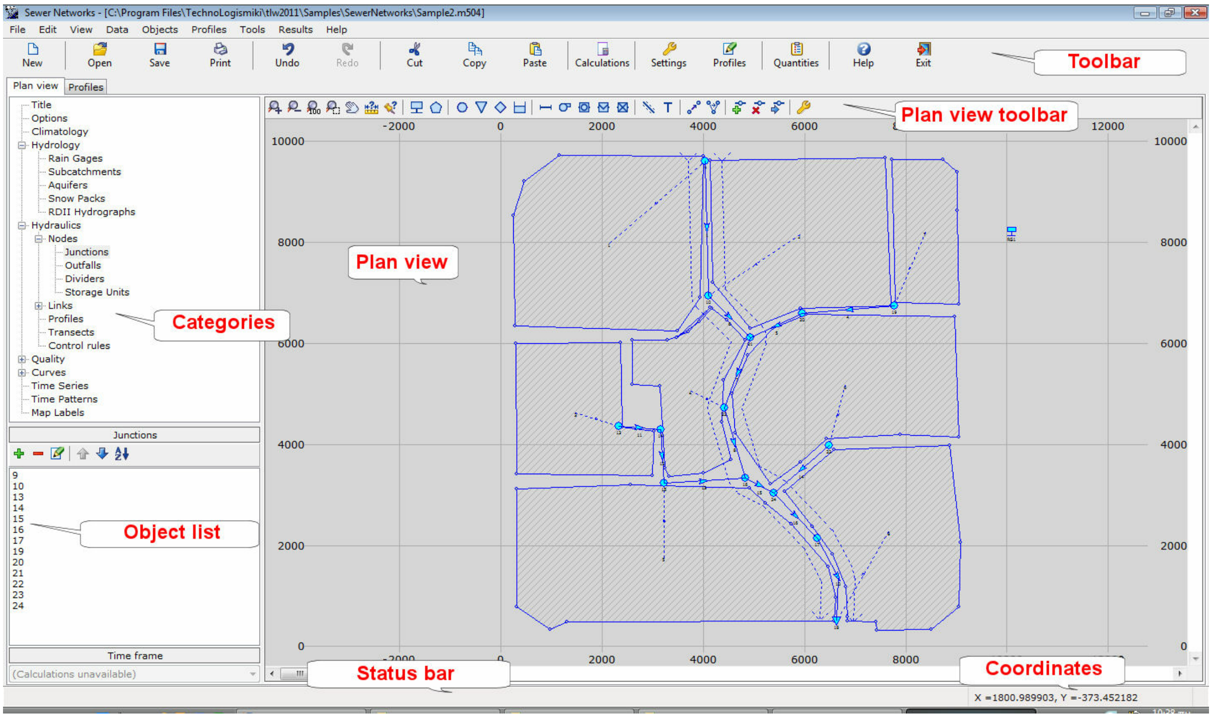
To select the Plan View mode, click on the corresponding tab on the top left corner of the main form. The main window consists of the following:

1. Menu: provides access to all program commands.
2. Toolbar: provides shortcut buttons to most common commands.
3. Categories: when plan view is selected, the categories tree view provides access to all project properties in a concise and comprehensive way.
4. Object List: enumerates all objects contained in the specified category. This list is equipped with a dedicated toolbar, containing the following:

- **Add**: adds an object in the specified category.
- **Delete**: deletes the selected object(s).
- **Edit**: edits the selected object(s).
- **Move Up:** moves the selected object one slot upwards.
- **Move Down**: moves the selected object one slot downwards.
- **Sort**: sorts all objects in the list alphabetically.

Not all options may be available at some instances.
5. Time Frame: when the calculations have been completed successfully, the time frame controls the time instant for which the results are displayed.
6. Plan View: the main plan view drawing. To select an object, click on it. Selected objects are drawn with a red line. Double-click on an object to display its properties.
7. Plan View Toolbar: a dedicated toolbar, containing shortcut buttons to most common commands regarding the plan view.
8. Status Bar: displays important messages regarding the state of the program.

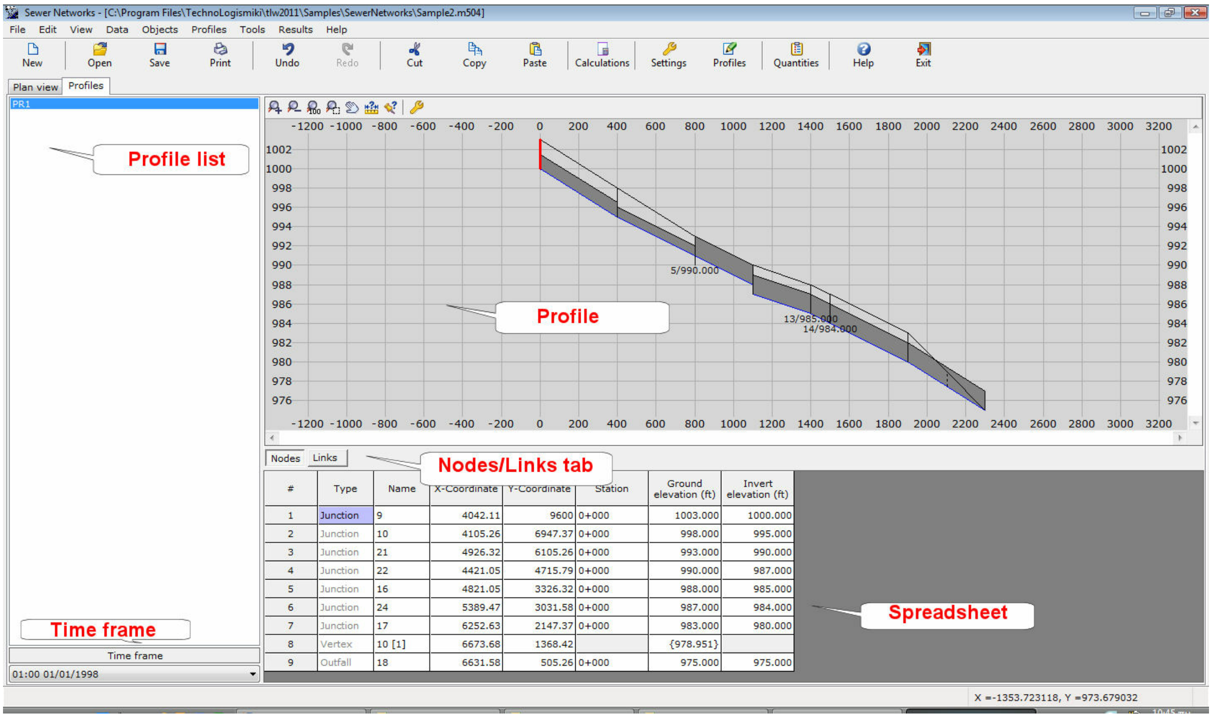
9. Coordinates: displays the current coordinates of the plan view or the profile.



2.3 Main Window (Profiles tab)

To select the Profiles mode, click on the corresponding tab on the top left corner of the main form. Apart from the controls already explained in the previous topic, the main window consists of the following:

1. Profile List: a list containing all profiles.
2. Profile: the profile drawing.
3. Spreadsheet: when in profile mode, the spreadsheet provides access to the properties of all objects consisting the specified profile.
4. Node/List tab: when in profile mode, select the appropriate tab to load the spreadsheet with the corresponding data.


Profile list
Profile
Nodes/Links tab
Spreadsheet
Time frame

Chapter III

3 File

3.1 File menu

With this menu, you can perform file operations and print reports. In the **File** menu you can select one of the following options:

- New project
- Open project
- Save project
- Save project as
- Import
  - Plan view
    - From DXF file
    - From ArcView Shapefile
    - From GTM
  - Data from SWMM
  - Data from LandXML
  - Background from DXF
  - Satellite image
- Export
  - Export selection
  - Plan view to DXF
  - Plan view to ArcView Shapefile
  - Plan view GTM
  - Plan view to BMP picture
  - Plan view to LandXML
  - Plan view to SWMM
- Print Setup
- Print
- Print to
  - Print to File
  - Print to Word
  - Print to Word (Formatted)
  - Print to Excel
- Exit

3.2 New project

With this option, a new project is started. All data, results, graphs, titles etc. of the previous project are erased.

To create a new project:

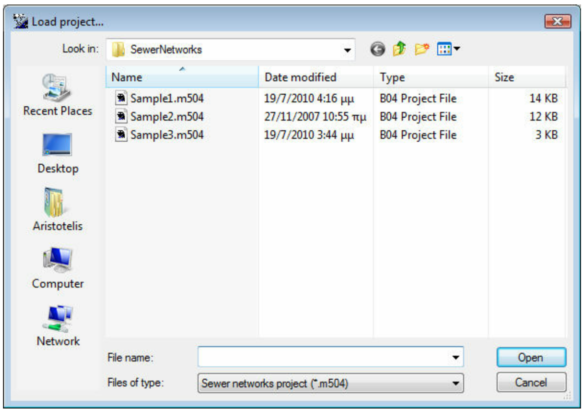
1. Select **New project** from the **File** menu.
2. If a project is already loaded and changes have been made, a warning message will appear that asks the user whether to save the changes or not.
3. The current project is erased and a new project is started.

3.3 Open project

With this option, an existing project is loaded. The project may be located locally, in a network or in an external media device such as a CD-Rom. If a project is already loaded and changes have been made, a warning message will appear that asks whether to save the changes or not. When a project is loaded, all data of the previous project are lost.

To open an existing project:

1. Select **Open project** from the **File** menu.
2. Select the path of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Sewer Networks project" with the extension .m504.
4. Select the file by clicking on it.
5. Select **Open** to open the selected file. Select **Cancel** to cancel the operation.



NOTE: You can find sample projects in the installation folder of the program:
C:\Program Files\TechnoLogismiki\TLW2013\Samples\SewerNetworks

Supported file types

- **M04** (Sewer networks project): Files created by versions 2012 and 2013 of Sewer Networks.
- **M504** (Sewer networks project): Files created by versions 2011, 2010, and 2008 of Sewer Networks.
- **M504** (Storm networks project v5.x - 2007): Files created by versions 2007 and 5.0 of Storm Networks.
- **M513** (Sewer networks project v5.x - 2007): Files created by versions 2007 and 5.0 of Sewer Networks.
- **MB4** (Storm networks project): Files created by versions 1.0 to 4.0 of Storm Networks.
- **MB13** (Sewer networks project): Files created by versions 1.0 to 4.0 of Sewer Networks.
- **BCK** (Backup files): If you have selected from program options the creation of backup copy when a file is loaded, then the file can be loaded by selecting Backup files (*.bck) from the Files of type drop-down list.
- ***.*** (All files): Displays all files in the current folder.

Backwards compatibility

This version implements full backwards compatibility; however, note that when a project is saved with the latest format, it cannot be used by previous versions.

NOTE: If a message "Could not load project. File may be corrupt or saved by an unknown or incompatible version of the program" then either you are trying to load a project that does not belong to this program or the file is used (and locked) by another process in your computer.

3.4 Save project

With this option, you can save all data of a project into a file. The file may be saved locally, in a network location or in an external media device such as a disk.

The filename and path will be asked only the first time you are saving a project. When the filename and path are set, all subsequent saves will be made to the same file.

When you want to rename a file or save it in a new location, use Save project as... from the **File** menu.

To save the current project:

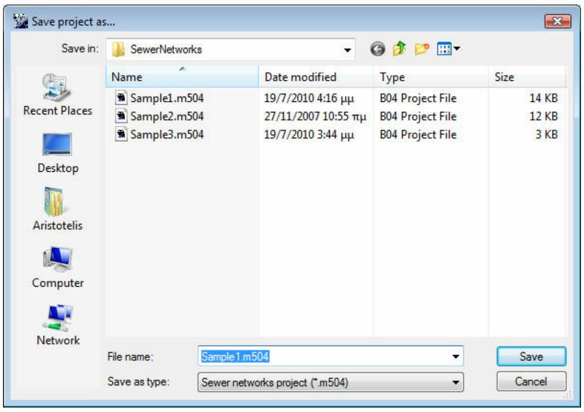
1. Select **Save project** from the **File** menu.
2. If the location of the file is already set, the project is saved to this file without any messages. If the filename is not set, a dialog box will appear that allows the selection of the filename and path.

3.5 Save project as

With this option, the current project is saved just as in the case of Save project, but with the difference that the name and/or location of the file can be changed. In this way, you can create backup files or move a project to another media device.

To save a project with another name and/or to another location:

1. Select **Save project as** from the **File** menu.
2. Select the path of the file.
3. Type the filename in the **File name** text box.
4. Select **Save** to save the project with the selected filename and path. Select **Cancel** to cancel the operation.



NOTE: If a file with the same name and in the same path already exists, a warning message will appear that asks whether to overwrite the file or not. If you answer Yes, then the existing file is erased and the new file takes its place. If you answer No, the existing file remains intact but NO changes of the current project are saved.

3.6 Import

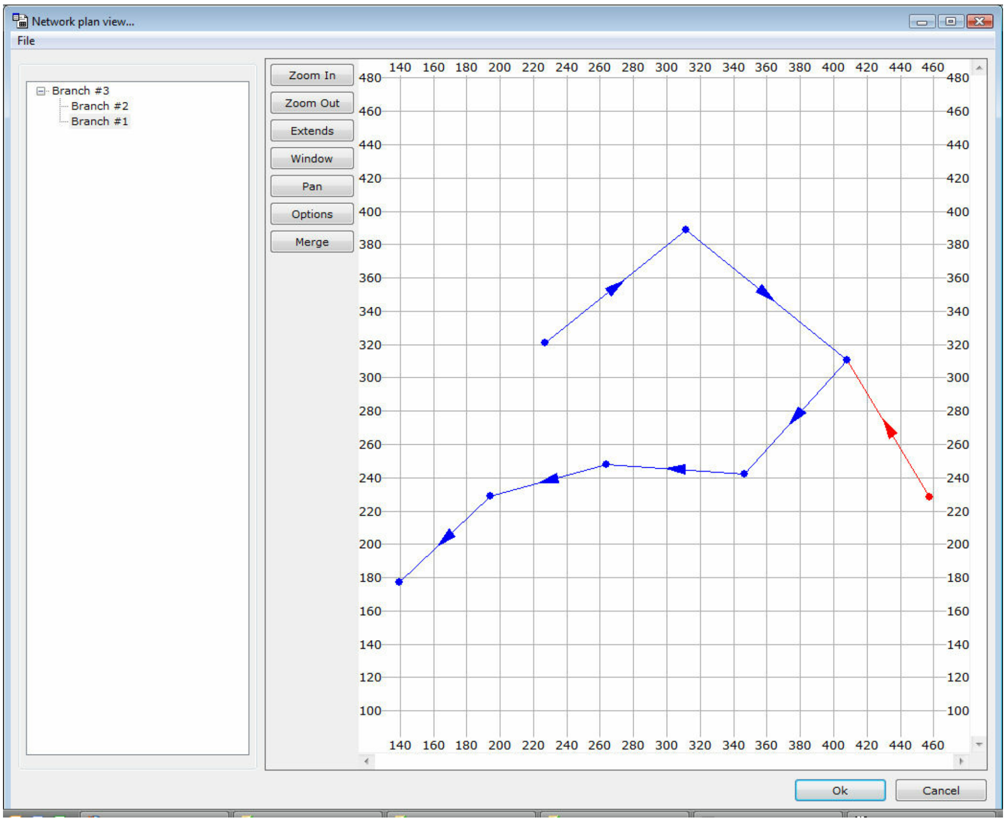
3.6.1 Plan view

With this option, you can import plan view data from an external file. All previous data will be erased.

To import plan view data from an external file:

1. Select **Import** from the **File** menu.
2. Select **Import plan view** from the **Import** menu. The following form appears:
3. From the **File** menu, select one of the following:
- **Import from DXF** to import plan view data from a DXF file.
- **Import from ArcView Shapefile** to import plan view data from an Arcview Shapefile.
- **Import from GTM** to import plan view data from a GTM GPS Trackmaker file.

4. Import the data following the specific instructions.
5. Click **Ok** to save the changes and close the form. Click **Cancel** to close the form without saving any changes.



To split an existing branch:

1. Right-click on the node you wish to split the branch.
2. Select **Split** from the drop-down menu. The branch is split into two branches.

To merge two branches:

1. Click the **Merge** button from the main form.
2. Click on the start node.
3. Click on the last node.

To reverse the flow of the selected branch:

1. Select the branch by clicking on it. The selected branch is drawn in red.
2. Right-click on the selected branch. Select **Reverse Flow** from the drop-down menu. The flow is reversed.

In order to manipulate the view of the drawing:

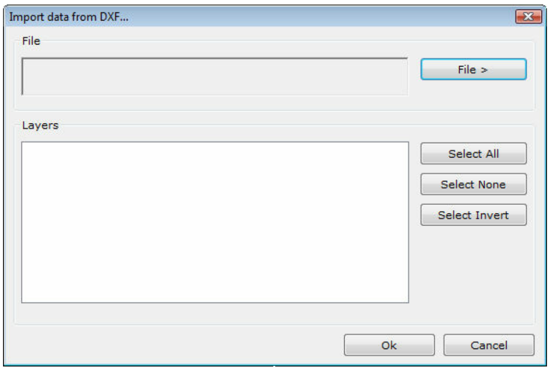
1. Click **Zoom In** from the main form and click anywhere on the drawing to zoom in.
2. Click **Zoom Out** from the main form and click anywhere on the drawing to zoom out.
3. Click **Extends** from the main form to view the whole drawing.
4. Click **Window** from the main form to zoom to a specified window.
5. Click **Options** from the main form to customize the appearance of the drawing.

NOTE: The number of collectors is equal to the number of flow exits.

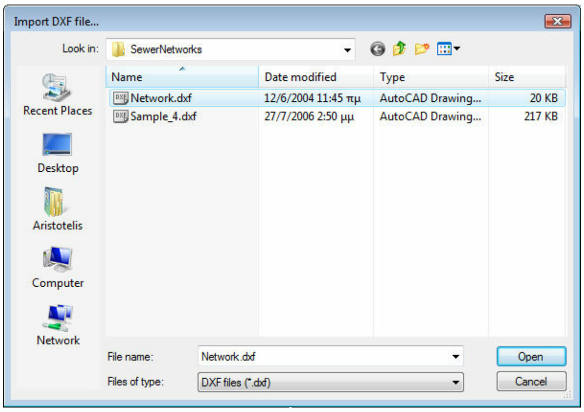
3.6.1.1 From DXF file

To import plan view data from a DXF file:

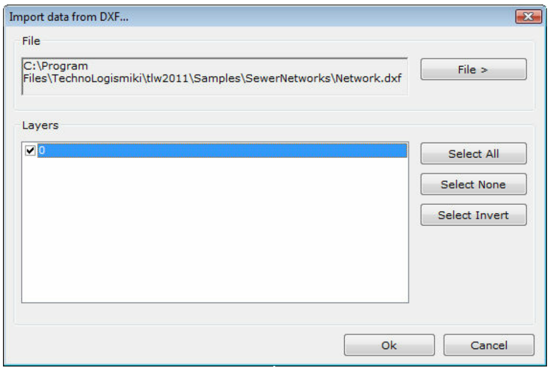
1. Select **Import From DXF file** from the **File** menu. The following form appears:



2. Click **File >** to select the DXF file. The file selection dialog box appears:



3. Select the path of the file.
4. Select the file type from the **Files of type** drop-down list. The default option is "DXF file" with the extension .dxf.
5. Select the file by clicking on it.
6. Select **Open** to open and analyze the file. The list in the **Layers** frame of step 1 is loaded with the layers contained in the DXF file:



7. Select one or more layers containing the data. The data should be defined in

polylines or a series of lines connected to each other. The connection must be exact, therefore you may need to use Snap or OSnap when using CAD software. The program will create stations at the nodes of the polylines and at the end points of the lines. Optionally, the coordinates of the stations can be used to calculate the distances between stations. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
8. Select **Ok** to import the data and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

NOTE: The following tips may be useful:

- Use lines and/or polylines in any combination. The nodes and start/end points of these objects define stations. You can use a different layer for each branch, although this is not necessary. You can use a single layer for the whole network.
- The names of the stations are filled automatically in descending order (upstream to downstream). The coordinates and distances between stations are also filled automatically.
- The DXF driver recognizes the following entities:
  - Lines
  - LWPolylines
  - Polylines
  - 3D Polylines

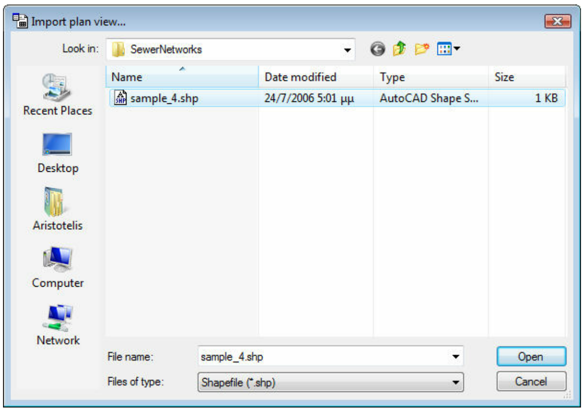
3.6.1.2 From ArcView Shapefile

With this option, you can import plan view data from an ArcView Shapefile. All previous data are erased.

NOTE: The ArcView Shapefile driver recognizes the following entities:
- Nullshapes
- Point/PointM/PointZ
- Multipoint/MultipointM/MultipointZ
- Polyline/PolylineM/PolylineZ

To import plan view data from an ArcView Shapefile:

1. Select **Import from Arcview Shapefile** from the **File** menu. The file selection dialog box appears:



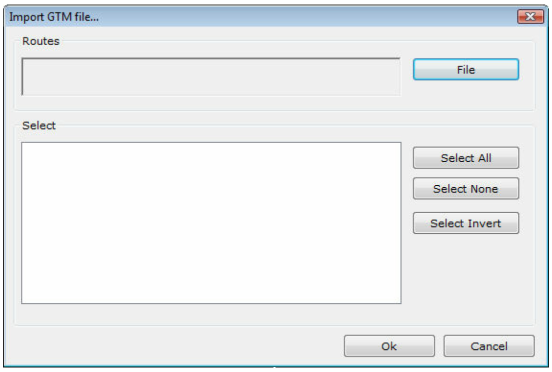
2. Select the path of the file.

3. Select the file type from the **Files of type** drop-down list. The default option is "ArcView shapefile" with the extension .shp.
4. Select the file by clicking on it.
5. Select **Open** to open and analyze the file. The current project data are erased. Select **Cancel** to cancel the operation.

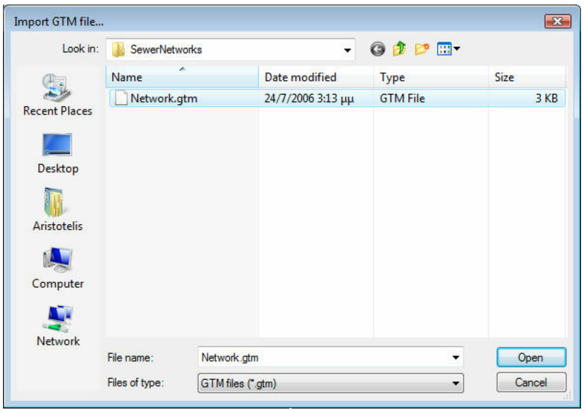
3.6.1.3 From GTM

To import plan view data from a GTM file:

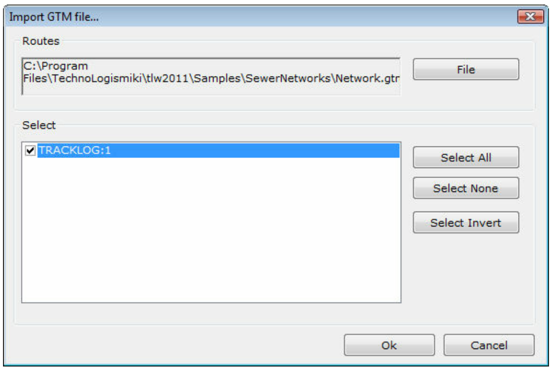
1. Select **Import from GTM file** from the **File** menu. The following form appears:



2. Click **File >** to select the GTM file. The file selection dialog box appears:



3. Select the path of the file.
4. Select the file type from the **Files of type** drop-down list. The default option is "GTM file" with the extension .gtm.
5. Select the file by clicking on it.
6. Select **Open** to open and analyze the file. The list in the **Selection** frame of step 1 is loaded with the layers contained in the GTM file:



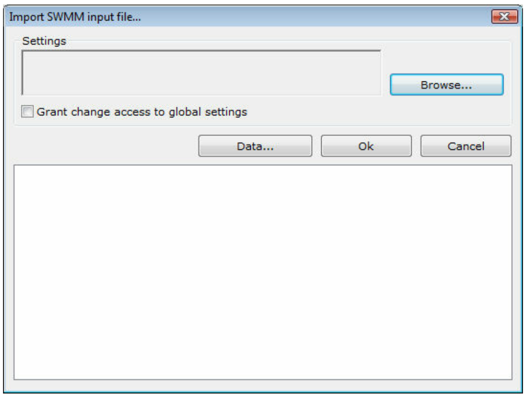
7. Select one or more routes or tracklog containing the data. Optionally, the coordinates of the stations can be used to calculate the distances between stations. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
8. Select **Ok** to import the data and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

3.6.2 Data from SWMM

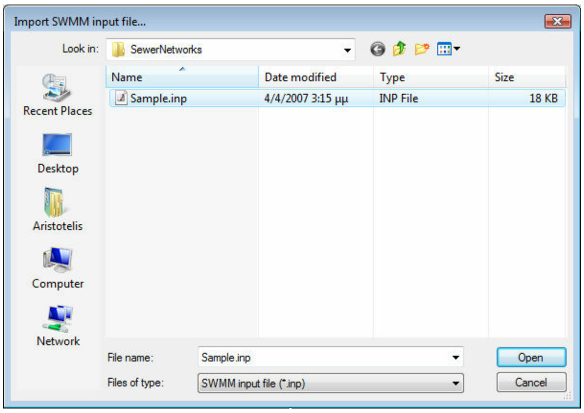
With this option, you can import data from files created by EPA's SWMM or Bentley's SewerGems. In most cases, import is complete there is no need for intervention. There are, however, some rare occasions in which you must correct some of the data.

To import data from EPA's SWMM or Bentley's SewerGems:

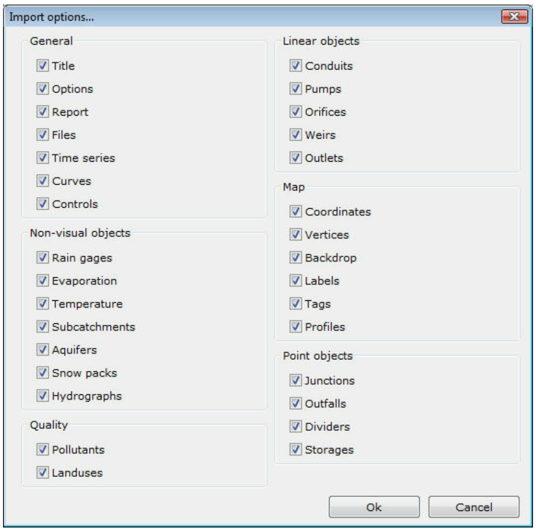
1. Select **Import** from the **File** menu.
2. Select **Data from SWMM** from the **Import** menu. The following form appears:



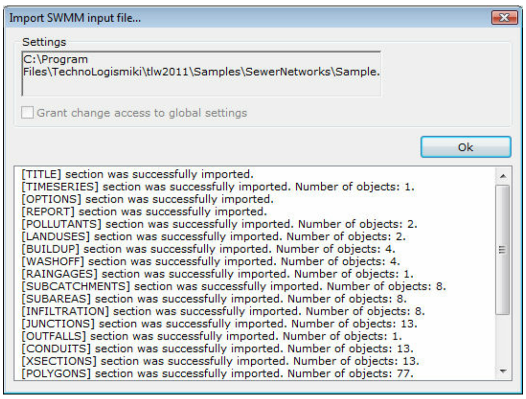
3. Click **Browse** to select the SWMM file. The file selection dialog box appears.
4. Select the path of the file.
5. Select the file type from the **Files of type** drop-down list. The default option is "SWMM Input file" with the extension .inp.
6. Select the file by clicking on it.
7. Select **Open** to select the file for import.



8. Select **Grant change access to global settings** to allow the file to change settings that are saved in per-project basis. Enabling this option is not recommended.
9. Optionally, click **Data...** to select which sections of the file will be imported.
9.1. Select the sections.
9.2. Click **Ok** to save changes. Click **Cancel** to ignore changes and close the form.



10. Click **Ok** to import the selected file. Click **Cancel** to cancel the import and close the form.
11. The results and potential error messages during import are loaded in the list.



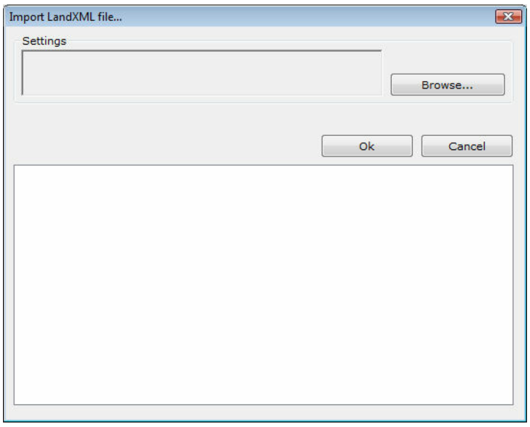
12. Click **Ok** to close the form.

3.6.3 Data from LandXML

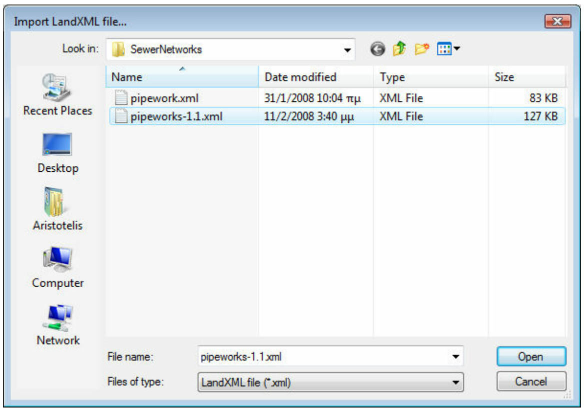
LandXML is an open source format specifically developed and tuned for exchanging design data used in the design / build process for land development and transportation projects. LandXML has evolved from previous efforts and is the cooperative work of representatives from all aspects of the engineering industry. For more information regarding programs supporting LandXML or for the LandXML scheme, please refer to LandXML forum.

To import LandXML data:

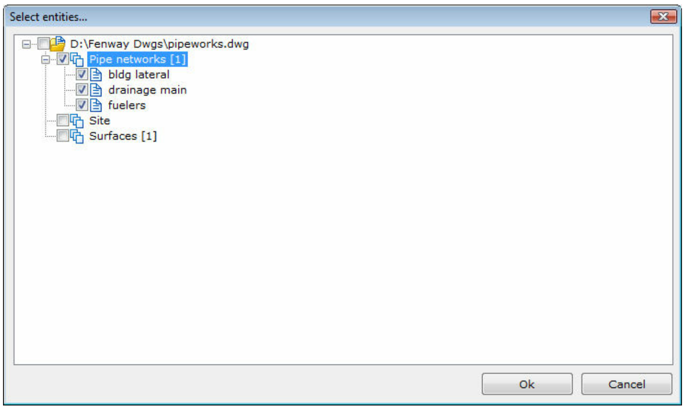
1. Select **Import** from the **File** menu.
2. Select **Data from LandXML** from the **Import** menu. The following form appears:



3. Click **Browse** to select the LandXML file. The file selection dialog box appears.



4. Select the path of the file.
5. Select the file type from the **Files of type** drop-down list. The default option is "LandXML file" with the extension .xml.
6. Select the file by clicking on it.
7. Select **Open** to open and analyze the file. If the file contains error, they are displayed on screen otherwise another form appears containing a treeview list of all entities that can be imported.



8. Select one or more entities that will be used as background or contain network information.
10. Press **Ok** to import the background (if available) or the pipe networks (if available) or **Cancel** to cancel the import process.

NOTE: In this version the following LandXML entities are supported:

- LandXML
- Units*
- CoordinateSystem*
- Project*
- Application*
- Alignments
- CgPoints
- Amendment*
- GradeModel
- Monuments
- Parcels
- PlanFeatures
- PipeNetworks
- Roadways
- Surfaces
- Survey
- FeatureDictionary*

* No visual representation.

NOTE: In this version the following LandXML schemes are supported:

- v1.2 beta Build32 03.12.2007
- v1.1
- v1.0
- v0.88

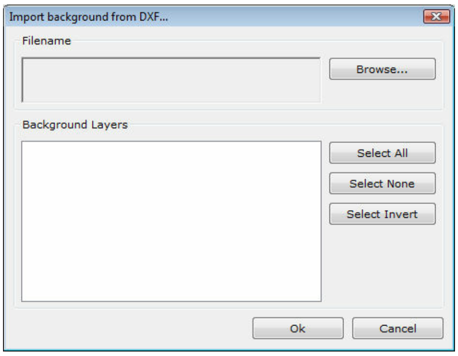
The program has been certified regarding its LandXML import and export functionality from LandXML forum. The certified versions are 1.0 and 1.1 and version 1.2 is pending until the scheme is finalized.

3.6.4 Background from DXF

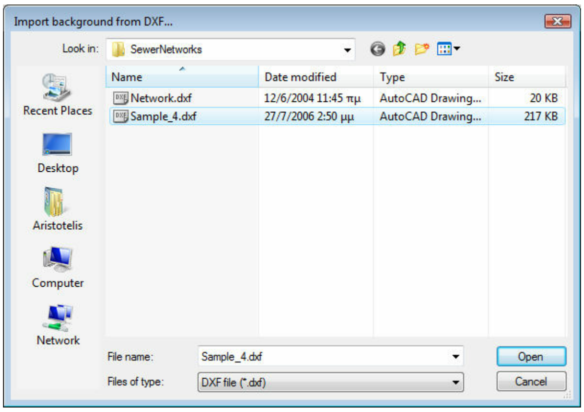
With this option, you can import background data from DXF files. The background is not active, but it is most helpful when designing a network. For performance reasons, only the most common entities of DXF files are imported, such as lines, points, polylines, arcs, circles, text etc.

To import background data from DXF:

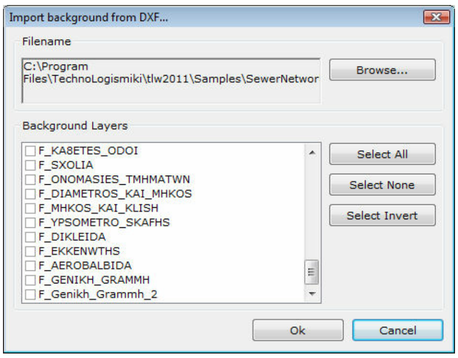
1. Select **Import** from the **File** menu.
2. Select **Background from DXF** from the **Import** menu. The following form appears:



3. Click **Browse** to select the DXF file. The file selection dialog box appears.



4. Select the path of the file.
5. Select the file type from the **Files of type** drop-down list. The default option is "DXF file" with the extension .dxf.
6. Select the file by clicking on it.
7. Select **Open** to open and analyze the file. The list is loaded with the layers contained in the DXF file.



8. Select one or more layers containing the data. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
9. Select **Ok** to import the data and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

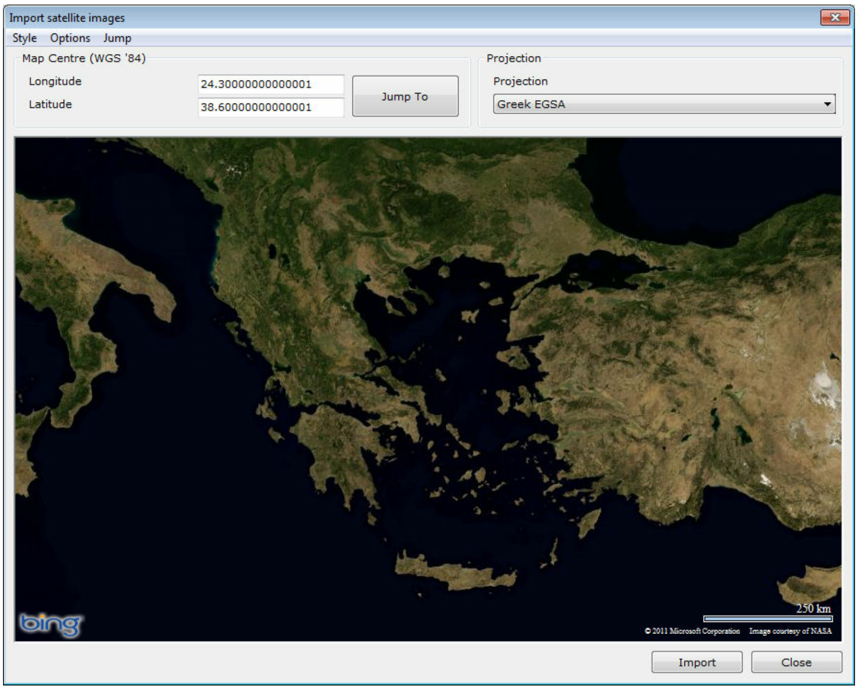
NOTE: To modify the background drawing, from the menu select **View > Plan View > Background**

3.6.5 Satellite image

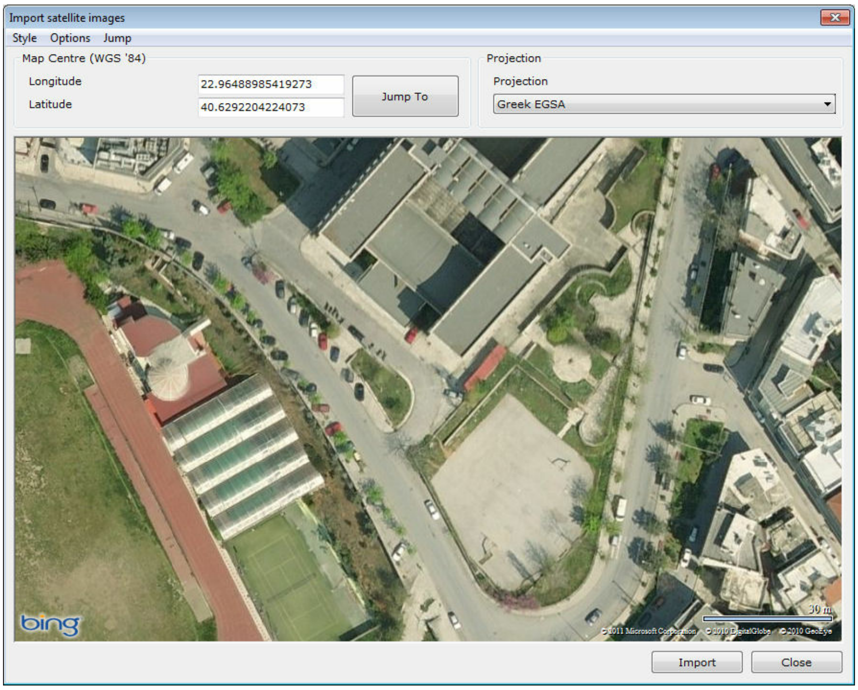
With this option you can insert a satellite image as a background image in plan view. The image is modified accordingly (translation, rotation, skewness) so that it is projected in the specified coordinate system.

To insert a satellite image as a background image in plan view:

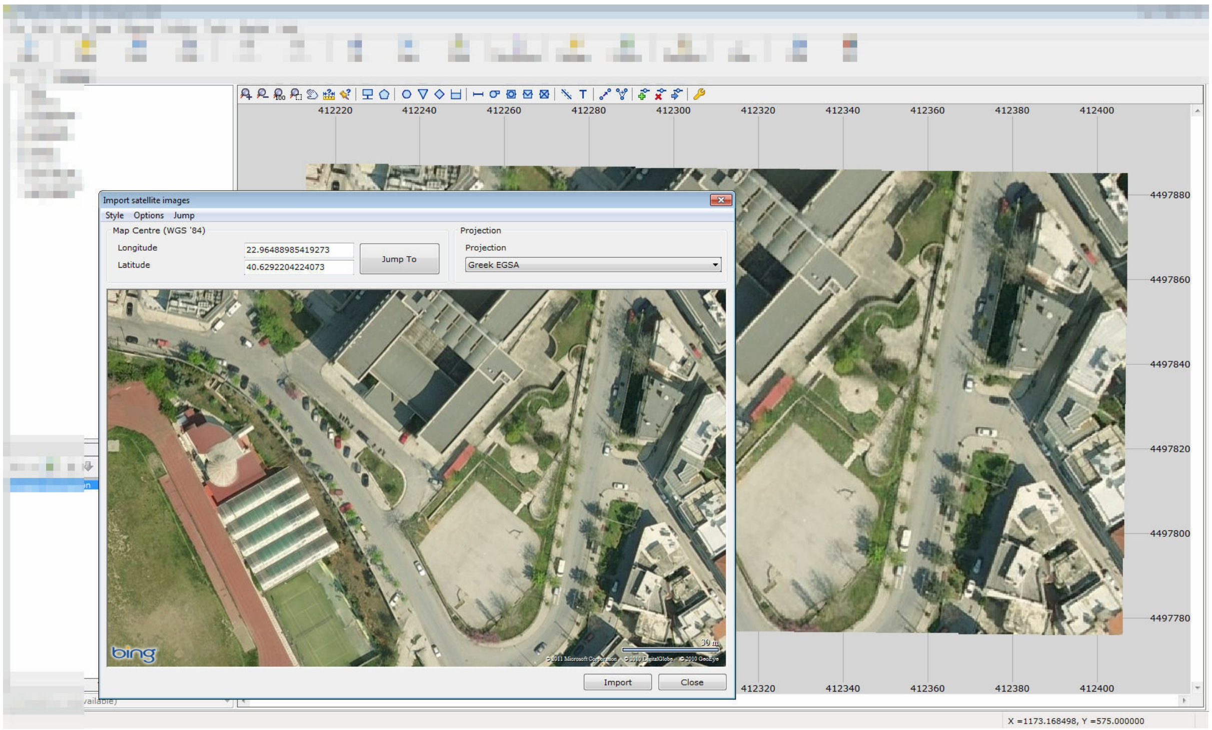
1. Select **Insert > Satellite image** from the **File** menu. The following form appears:



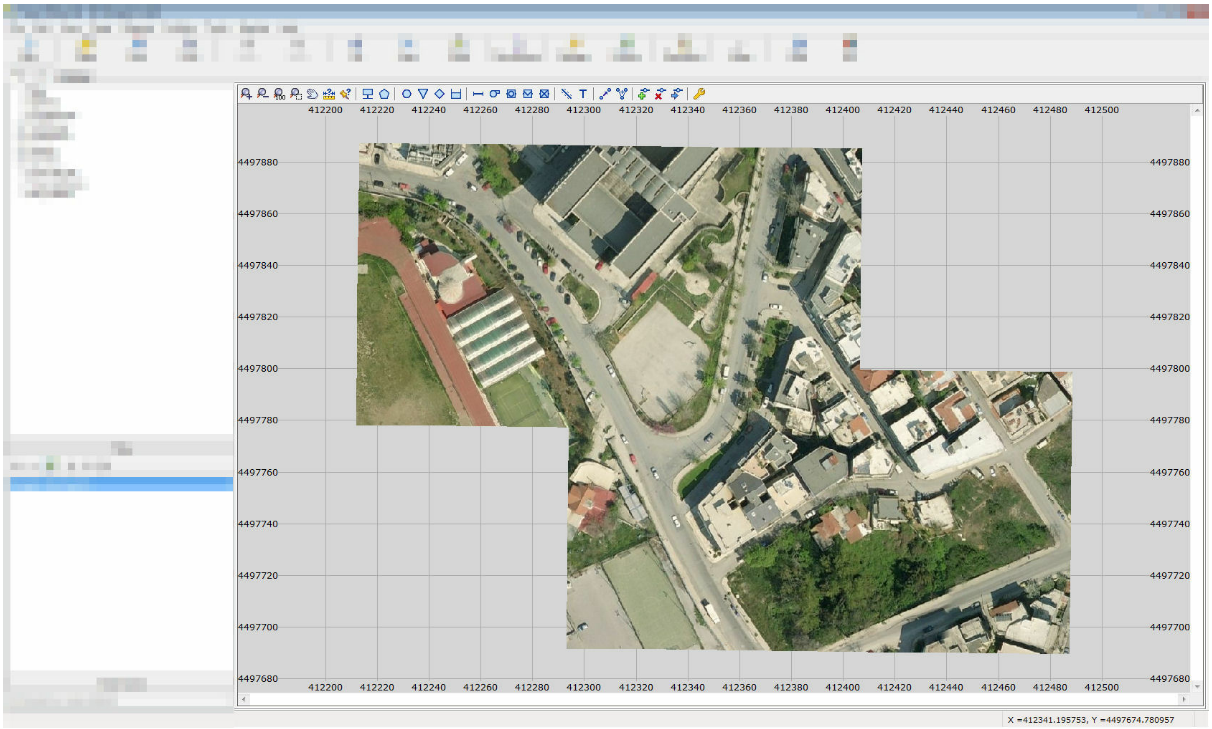
2. Navigate to the area of the project. You can pan the image by holding down the left mouse button. Using the roller you can change the resolution. Alternatively, enter the longitude and latitude in decimal degrees and press **Jump to:**



3. When you locate the area, select the appropriate resolution (the resolution varies, depending on the quality of the satellite image), and press Import. The current viewport is imported as a background image:



4. Without closing the window, pan the viewport and press Import again. A new image is imported, which may overlap with the previous one. When you cover the whole area of interest, press **Close**:



In the satellite image form, the following options are available:

- Style. Some options may not be available, or they may not have an effect, depending on the quality of the satellite image.
  - Roads
  - Shaded
  - Aerial
  - Hybrid
- Options
  - Show navigation tool.
  - Show locator tool.
  - Units
    - Metric
    - English
- Jump. These are quick selections for jumping to:
  - To Athens
  - To Greece
  - To Europe
  - To USA
  - To World

NOTE: The images are saved as TIFF files in the same path as the project. You can delete them selectively using the **View > Background images > Delete** menu.

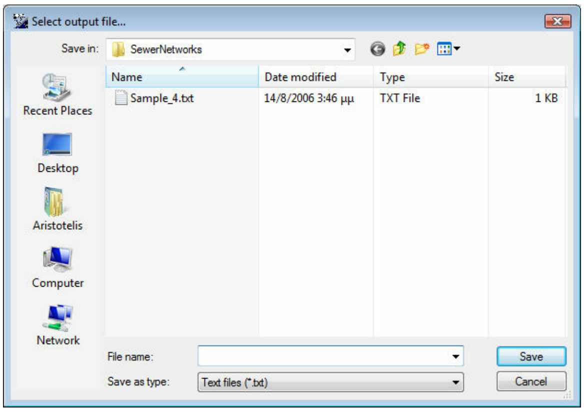
3.7 Export

3.7.1 Export selection

With this option, you can export the selected data in simple ASCII text files. The delimiter is customized by the user.

To export data to a file:

1. Select the cells containing the data.
2. Select the proper delimiter.
3. Select **Export** from the **File** menu.
4. Select **Export selection** from the **Export** menu. The following form appears:



5. Select the path of the file.
6. Type the filename in the **File name** text box.
7. Select **Save** to save the file with the selected filename and path. Select **Cancel** to cancel the operation.

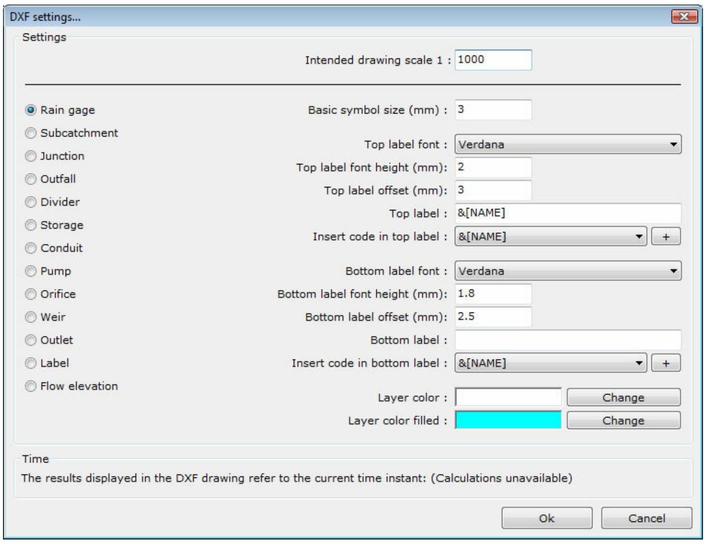
3.7.2 Plan view to DXF

With this option, a DXF file containing the plan view is created. The following data are included in the file:

- Station, station names
- Node data, pipe data
- Hydraulic calculations
- Contours (if visible)

To export the plan view to DXF file:

1. Select **Export** from the **File** menu.
2. Select **Plan view to DXF file** from the **Export** menu. The following dialog box appears:



3. Select the appropriate settings:

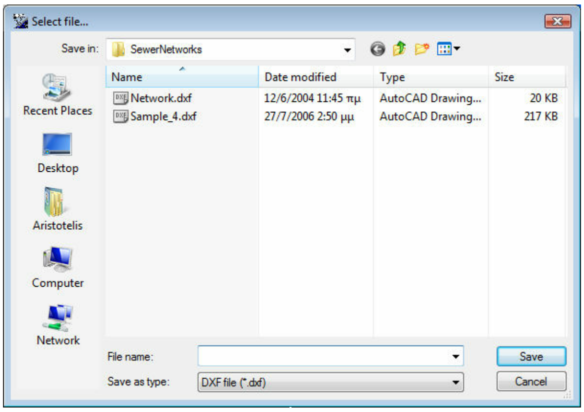
- **Intended drawing scale**: Select the intended drawing scale. The rest of the settings are measured in mm (millimeters) in the printed drawings. Based on the intended drawing scale, the sizes of all drawing elements are automatically derived.
- Select the object you wish to configure from the list on the left. Depending on the object, the following options may become available:
  - **Basic symbol size (mm)**: Enter the basic symbol size in mm (millimeters) in the printed drawing. For example, the basic symbol size for a junction refers to the diameter of the circle that represents it.
  - **Top label font**: select the font for the top label. All drawing elements are grouped into layers and font styles, so that they can be easily modified using CAD software.
  - **Top label font height (mm)**: Enter the top label font height in mm (millimeters) in the printed drawings. If the font height is selected to be 2mm and the scale is 1:1000, then the text height that will be used is equal to 2mm * 1000 = 2m. When printed, the desired font height will be obtained.
  - **Top label offset (mm) :** Enter the top label offset from the center of the object, in mm in the printed drawing.
  - **Top label** : Enter the text that will appear in the top label of the selected object. The text may contain **codes.** The codes have the prefix "&" and contain a special keyword within brackets. For example, if the top label is selected to be "&[NAME]", then the name of the object will be displayed. **Any combination of text and/or codes is allowed**. For example, if the model contains junctions named "J1", J2", etc and the manhole type is "T1", then the label "&[NAME] - &[MANHOLE_TYPE]" will create labels "J1 - T1", "J2 - T1" etc in the DXF drawing.
  - **Insert code in top label**: Depending on the selected object, select the code from the list and click on the " + " button to enter it in the top label. The insertion point is the current cursor point in the previous field entitled "Top label".
  - **Similar options may refer to the bottom label.**
  - **Layer color:** Select the layer color of the selected object.
  - **Layer color filled**: Select the layer color for the filled elements of the selected

object.

The available codes, depending on the type of the object, are (the names are self-explanatory):

- "&[NAME]"
- "&[STATION]"
- "&[DESCRIPTION]"
- "&[TAG]"
- "&[MANHOLE_TYPE]"
- "&[GROUND_ELEVATION]"
- "&[INVERT_ELEVATION]"
- "&[AREA]"
- "&[LENGTH]
- "&[SLOPE]"
- "&[SHAPE]"
- "&[DEPTH]"
- "&[DEPTH_RATIO]"
- "&[FLOW]"
- "&[VELOCITY]"

4. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.
5. The file selection dialog box appears:



6. Select the path of the file.
7. Type the filename in the **File name** text box.
8. Select **Save** to save the file with the selected filename and path. Select **Cancel** to cancel the operation.

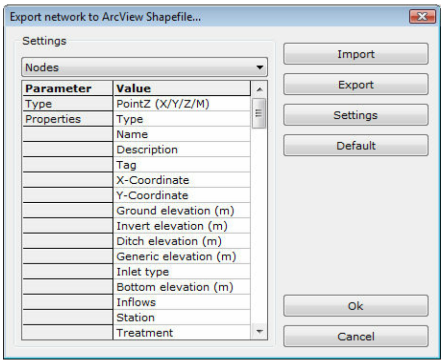
NOTE: If nothing is visible when viewing the newly created DXF file, select **Zoom Extents** to view the whole drawing.

3.7.3 Plan view to ArcView Shapefile

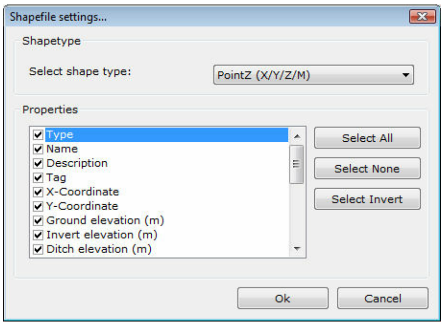
With this option, two ArcView Shapefiles containing the plan view data are created. The first file contains station data and the other contains pipe data. Note that apart from geometry, hydraulic calculation results are included in the Shapefiles.

To export the plan view to an ArcView Shapefile:

1. Select **Export** from the **File** menu.
2. Select **Plan view to ArcView Shapefile** from the **Export** menu.
3. The following dialog box appears:



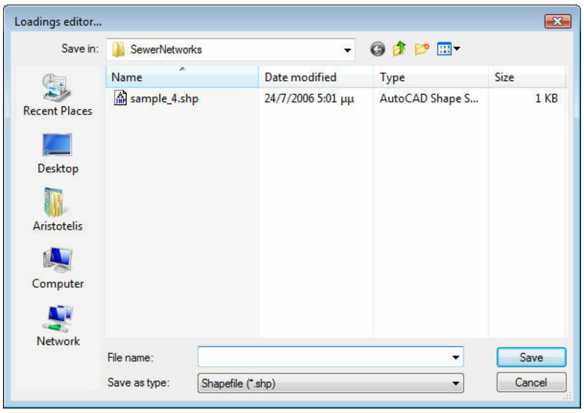
4. Select one of **Nodes**, **Pipes** from the drop-down list.
5. Select **Customization...** to customize the shape type and the properties that will be included:



6. Select the appropriate shape type.

NOTE: The GIS driver recognizes the following shape types:
- Nullshapes
- Point/PointM/PointZ
- Multipoint/MultipointM/MultipointZ
- Polyline/PolylineM/PolylineZ

7. Select the properties that you want to include in the file. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
8. Select **Ok** to proceed with the selection of the filename. Select **Cancel** to abort the operation and close the dialog box.



9. Select the path of the file.
10. Type the filename in the **File name** text box.
11. Select **Save** to create the file. Select **Cancel** to cancel the operation.

NOTE: If nothing is visible when viewing the newly created ArcView Shapefile, select **Zoom Extents** to view the whole drawing.

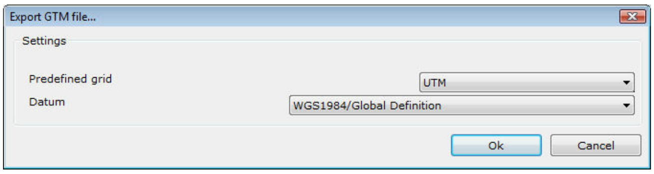
3.7.4 Plan view to GTM

With this option, a GPS Trackmaker (GTM) file containing the plan view is created. The following data are included in the plan view:

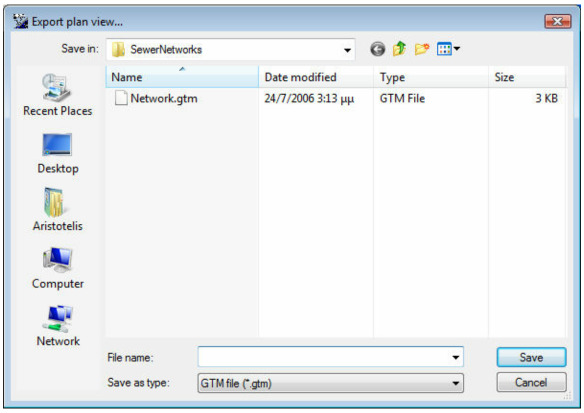
- Sections, Section names
- Pipes

To export the plan view to GTM file:

1. Select **Export** from the **File** menu.
2. Select **Plan view to GTM file** from the **Export** menu.
3. The following dialog box appears:



4. Select the predefined grid and the datum of the coordinate system. The default values are used in most cases.
5. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.
6. The file selection dialog box appears.



7. Select the path of the file.
8. Type the filename in the **File name** text box.
9. Select **Save** to save the file with the selected filename and path. Select **Cancel** to cancel the operation.

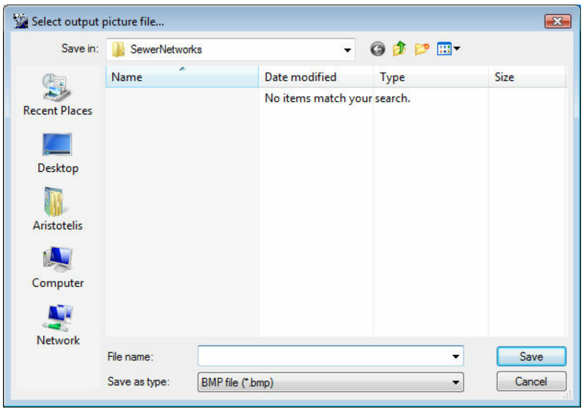
NOTE: If nothing is visible when viewing the newly created GTM file, select **Zoom Extents** to view the whole drawing.

3.7.5 Plan view to BMP picture

With this option, you can create a BMP file containing the plan view as it is currently displayed on screen.

To create a BMP file:

1. Select **Export** from the **File** menu..
2. Select **Plan view to BMP picture** from the **Export** menu. The following form will appear:



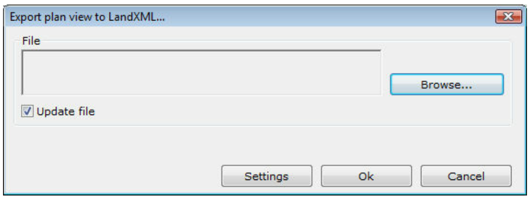
3. Select the path of the file.
4. Type the filename in the **File name** text box.
5. Select **Save** to save the file with the selected filename and path. Select **Cancel** to cancel the operation.

3.7.6 Plan view to LandXML

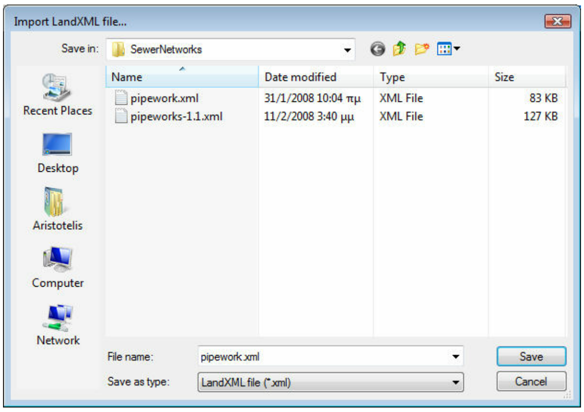
The plan view is exported (background, networks and calculations) to a LandXML file. For more information regarding programs supporting LandXML or for the LandXML scheme, please refer to LandXML forum.

To export plan view to LandXML:

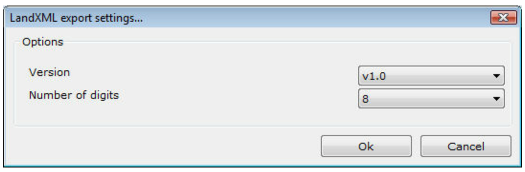
1. Select **Export** from the **File** menu..
2. Select **Plan view to LandXML** from the **Export** menu. The following form will appear:



3. Click **Browse** to select an existing or a new LandXML file. The file selection dialog box appears.

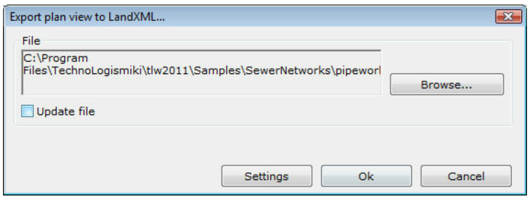


4. Select the path of the file.
5. Type the filename in the **File name** text box to select a new file or select an existing file.
6. Select **Save** to finalize your selection.
7. If you selected an existing LandXML file, then it is loaded and analyzed so that the program will be ready to update it. If not, then its name appears on the form and the update check box is disabled.
8. Optionally, click on **Settings** to change the driver's settings.



8.1. Select the LandXML scheme that will be used to create / update the file. It is recommended to select 1.0 for maximum compatibility or the scheme version

supported by the destination program (the program that will be used to read the exported file).
8.2. Select the number of decimal digits. It is recommended to select any number from 4 to 8 digits.
8.3. Click **Ok** to save changes. Click **Cancel** to ignore changes and close the form.



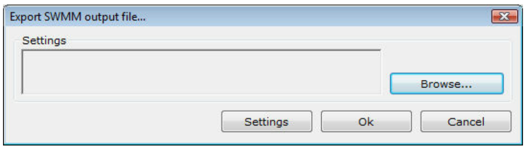
9. If an existing file has been selected, the **update file** check box is enabled. If this check box is selected then the target file will be updated with any changes made within the program.
10. Press **Ok** to close the form and finalize the export procedure or press **Cancel** to close the form and cancel the export procedure.

3.7.7 Plan view to SWMM

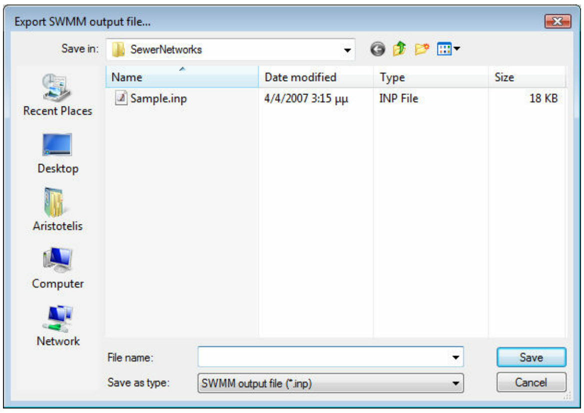
The plan view is exported (background and networks) to a SWMM file. This file can be read from other programs such as EPA's SWMM and Bentley's Sewergems.

To export the plan view to a SWMM file:

1. Select **Export** from the **File** menu..
2. Select **Plan view to SWMM** from the **Export** menu. The following form will appear:

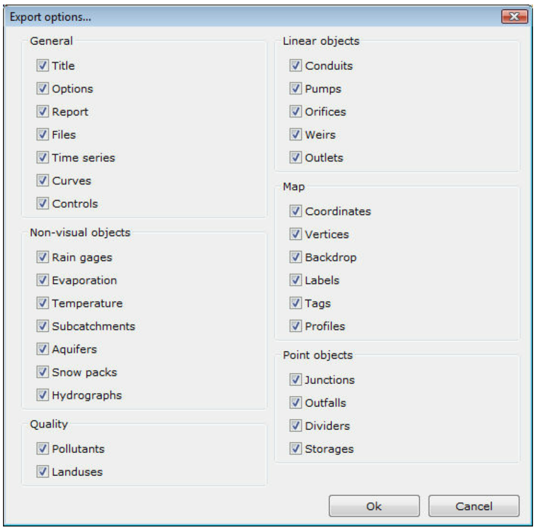


3. Click **Browse** to select a new SWMM file. The file selection dialog box appears.

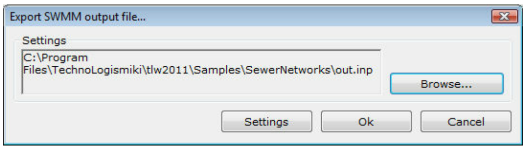


4. Select the path of the file.
5. Type the filename in the **File name** text box to select a new file.
6. Select **Save** to finalize your selection.
7. Optionally, click on **Settings** to select which sections will be exported to the SWMM

file.



7.1. Select the sections.
7.2. Click **Ok** to save changes. Click **Cancel** to ignore changes and close the form.



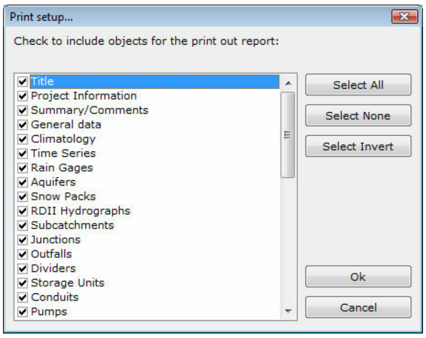
8. Press **Ok** to close the form and finalize the export procedure or press **Cancel** to close the form and cancel the export procedure.

3.8 Print Setup

With this option, you can select which parts of the project will be included in the printouts. When a new project is created, a full report is selected by default.

To modify the print setup:

1. Select **Print setup** from the **File** menu.
2. Select the **sections** (Title, Project information etc) that will be included in the reports.
3. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection of a list.

NOTE: The changes are saved with the project. The above preferences are used to all printouts, either to the printer or to other formats such as Word file, Excel file etc.

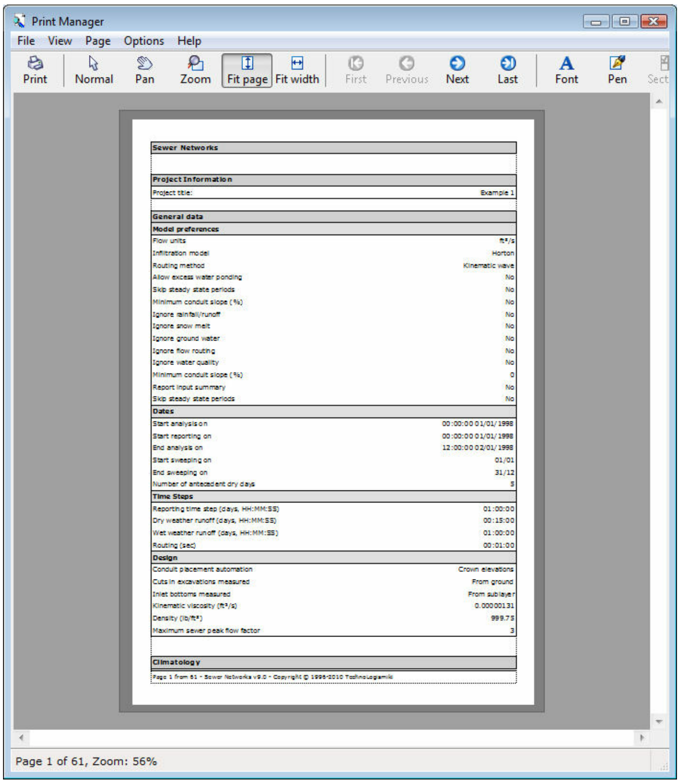
3.9 Print

With this option, you can prepare a report to be printed to a local, network or virtual printer such as Adobe PDF Writer. The parts of the project that will be included in the report are determined from print setup.

By selecting **Print**, the report is not printed directly; instead, a document is prepared and a preview of the printout is created by the **Print manager.** You can print the report by clicking the **Print** button of the toolbar of **Print manager.**

To create a report:

1. Select **Print** from the **File** menu.
2. A report is prepared and sent to **Print manager.** A preview of the document appears.
3. You can print the report by clicking the **Print** button of the toolbar.



NOTE: A complete user manual on the capabilities of **Print manager** can be found in the corresponding help file.

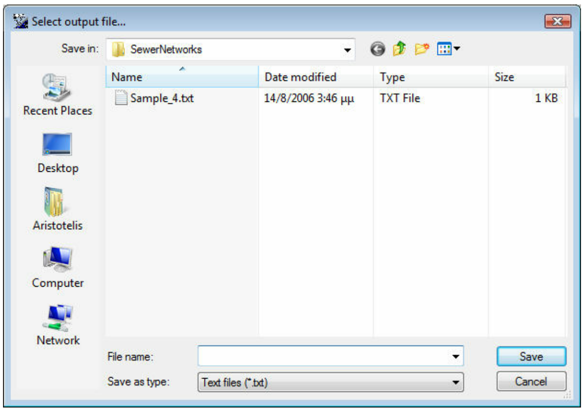
3.10 Print to

3.10.1 Print to File

With this option, you can create a simple text file containing a report of the project. This file is recognized and can be further modified by word processors such as Microsoft Word, OpenOffice Writer etc.

To print to a text file:

1. Select **Print to** from the **File** menu.
2. Select **Print to file** from the **Print to** menu.
3. Select the path of the file.
4. Type the filename in the **File name** text box.
5. Select **Save** to create the file.



The parts of the project that will be included in the report are determined from print setup.

NOTE: If a file with the same name and in the same path already exists, a warning message will appear that asks whether to overwrite the file or not. If you answer Yes, then the existing file is erased and the new file takes its place. If you answer No, the existing file remains intact but the report is NOT printed.

3.10.2 Print to Word

If Microsoft Word (version 97, 2000, XP, 2003 or later) has been installed in the system, then a Microsoft Word file containing the report can be created. Note that Microsoft Word is a separate program and it is not included in TechnoLogismiki's products. Moreover, no technical support is offered regarding the usage of Microsoft Word.

To print the report to a Microsoft Word file:

1. Select **Print to** from the **File** menu.
2. Select **Print to Word** from the **Print to** menu.

The parts of the project that will be included in the report are determined from print setup.

3.10.3 Print to Word (Formatted)

If Microsoft Word (version 97, 2000, XP, 2003 or later) has been installed in the system, then a Microsoft Word file containing the report can be created. Note that Microsoft Word is a separate program and it is not included in TechnoLogismiki's products. Moreover, no technical support is offered regarding the usage of Microsoft Word.

To print the report to a formatted Microsoft Word file:

1. Select **Print to** from the **File** menu.
2. Select **Print to Word (Formatted)** from the **Print to** menu.

The parts of the project that will be included in the report are determined from print setup. This operation is much slower than the regular print to word function. However, the final output requires minimal user intervention as it comes fully formatted with tables, alignment, font styles, etc.

NOTE: Do not use Copy (CTRL+C) on any of the programs running during this operation. If you do so, it will most likely affect the communication between Microsoft Word and the clipboard and as a result the final document will be corrupt.

3.10.4 Print to Excel

If Microsoft Excel (version 97, 2000, XP, 2003 or later) has been installed in the system, then a Microsoft Excel file containing the report can be created. Note that Microsoft Excel is a separate program and it is not included in TechnoLogismiki's products. Moreover, no technical support is offered regarding the usage of Microsoft Excel.

To print the report to a Microsoft Excel file:

1. Select **Print to** from the **File** menu.
2. Select **Print to Excel** from the **Print to** menu.

The parts of the project that will be included in the report are determined from print setup.

3.11 Exit

With this option, you can exit the program. If there are changes in the current project that have not been saved then the program will:

- either ask the user to save the changes
- or save the changes
- or ignore the changes

depending on what you have selected in General preferences.

To exit the program:

1. Select **Exit** from **File** menu.
2. If you are asked whether to save the changes or not, you can save changes or

ignore them.
3. The program is terminated.

Chapter IV

4 Edit

4.1 Edit menu

With this menu, you can perform basic operations regarding data. In the **Edit** menu you can select one of the following options:

- Undo
- Redo
- Copy
- Cut
- Paste
- Select all
- Clipboard delimiter
- Clipboard decimal separator
- Select objects
  - Create selection
  - Load selection
  - Save selection
  - Clear selection
- Locate objects

4.2 Undo

Undo cancels the last committed change in the project.

To cancel the last committed change:

1. Select **Undo** from the **Data** menu.
2. The last committed change is canceled.

To cancel an undo command, you may use the redo function which is described below. Redo becomes available once undo is used.

It is possible to undo more than one recent changes and to redo them, by following the step described above. The number of actions that are kept in memory and may be undone or redone is 20 by default. This means that the program is able to keep track of up to 20 successive changes and undo them. This number may change for all programs, using the option in the main menu. For more information, please consult main menu user guide.

NOTE: Some changes cannot be undone like the new project or the save project functions.

4.3 Redo

Redo cancels the latest undo command.

To redo the latest change that was undone:

1. Select **Redo** from the **Data** menu.
2. The latest undone change is redone.

To undo a redo, you may use the undo command.

It is possible to redo more than one changes that were previously undone by following the steps described above. The number of actions that are kept in memory and may be undone or redone is 20 by default. This means that the program is able to keep track of up to 20 successive changes that are undone and redo them. This number may change for all programs, using the option in the main menu. For more information, please consult main menu user guide.

4.4 Copy

With this option, you can copy the contents of the selected cells to the clipboard.

To copy the contents of the selected cells to the clipboard:

1. Select the cells from the data table.
2. Select **Copy** from the **Edit** menu. The contents of the selected cells are copied to the clipboard.

To copy data to be used with Microsoft Excel:

1. Select **TAB** as the delimiter.
2. Select **System** as the decimal separator.
3. Select the cells from the data table.
4. Select **Copy** from the **Edit** menu. The contents of the selected cells are copied to the clipboard.
5. Hit CTRL+V to paste the data when using Microsoft Excel.

4.5 Cut

With this option, you can copy the contents of the selected cells to the clipboard and clear the current selection.

To copy the contents of the selected cells to the clipboard and clear the current selection:

1. Select the cells from the data table.
2. Select **Cut** from the **Edit** menu. The contents of the selected cells are copied to the clipboard and the selection is cleared.

To cut data to be used with Microsoft Excel:

1. Select **TAB** as the delimiter.
2. Select **System** as the decimal separator.
3. Select the cells from the data table.
4. Select **Cut** from the **Edit** menu. The contents of the selected cells are copied to the clipboard and the selection is cleared.
5. Hit CTRL+V to paste the data when using Microsoft Excel.

4.6 Paste

With this option, you can paste data from the clipboard to the data table.

To paste data from the clipboard to the data table:

1. Select the top left cell.
2. Select **Paste** from the **Edit** menu. The data are copied from the clipboard to the data table.

To paste data from Microsoft Excel:

1. Select **TAB** as the delimiter.
2. Select **System** as the decimal separator.
3. Within Microsoft Excel, select all cells and hit CTRL+C to copy the data to the clipboard.
4. Within Sewer Networks, select the top left cell that corresponds to the data.
5. Select **Paste** from the **Edit** menu. The data are copied from the clipboard to the data table.

NOTE:
- You cannot paste data into the grayed cells.
- All data is transferred from consecutive columns, even if these are not visible.
- Other software may require other delimiter when using clipboard.

4.7 Select all

With this option, when in plan view mode all objects are selected. When in profiles mode, all cells of the data table are selected.

To select all:

1. Select **Select all** from the **Edit** menu.

4.8 Clipboard delimiter

With this option, you can select the delimiter that will be used that will be used when transferring data to and from the clipboard.

To select the delimiter:

1. Select **Clipboard delimiter** from the **Edit** menu.
2. Select one of **Tab**, **Comma**, **Space**.

NOTE: Although Sewer Networks can handle all three cases of delimiters, other software may have some restrictions. For example, to exchange data with Microsoft Excel, you should use **TAB** as delimiter.

4.9 Clipboard decimal separator

With this option, you can select the decimal separator that will be used when transferring data to and from the clipboard.

To select the decimal separator:

1. Select **Clipboard decimal separator** from the **Edit** menu.
2. Select one of **System**, **Period**.

NOTE: To exchange data with Microsoft Excel, you should use the **System** decimal separator (by default). It is possible to modify the settings in Microsoft Excel to accept period as decimal separator. Please refer to the manual of Microsoft Excel.

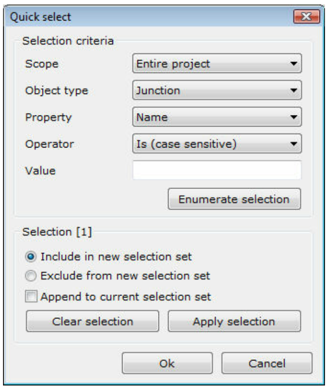
4.10 Select objects

4.10.1 Create selection

With this option, you can modify the set of selected objects. You can create a new selection set or add objects to the current selection set. The selection is based on various criteria. The successful completion of calculations is recommended before using this option.

To modify the set of selected objects:

1. Perform calculations. This step is optional, but if you do not follow it, the results for some objects may not be available and the selection may not function properly.
2. Select **Select Objects** from the **Edit** menu.
3. Select **Create selection** from the **Select Objects** menu. The following form appears:



4. Select the **scope** from the drop-down list. The scope can either be the entire project or the current selection.
5. Select the **object type** from the drop-down list.
6. Select the **object property** from the drop-down list. This list is depended on the object type.
7. Select the **operator** which will be used in the comparison.
8. Enter the **value** of the property.
9. Optionally click on **Enumerate Selection** to preview how many items will be selected if this query is applied.
10. Select either **Include in new selection set** to clear any existing selection and create a new one or **Exclude from new selection set** to remove matching items from the selection.
11. Enable **Append to current selection set** to preserve the current selection and expand it by adding items from current query.
12. Click on **Clear Selection** to clear the selected items.
13. Click on **Apply Selection** to perform the query according to the specified parameters.
14. Repeat steps 4 to 13 as necessary.

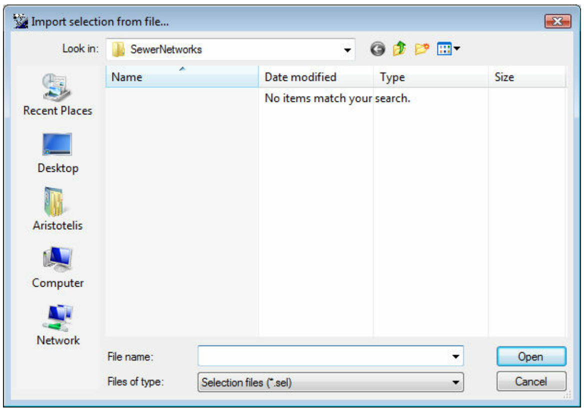
15. Select **Ok** to close the form and accept changes or **Cancel** to close the form and ignore any changed.

4.10.2 Load selection

With this option, you can load a selection set from an external file that was previously saved. This process will fail if the objects currently loaded do not correspond to the selection set.

To load a selection set from an external file:

1. Select **Select Objects** from the **Edit** menu.
2. Select **Load Selection** from the **Select Objects** menu. The following form appears:



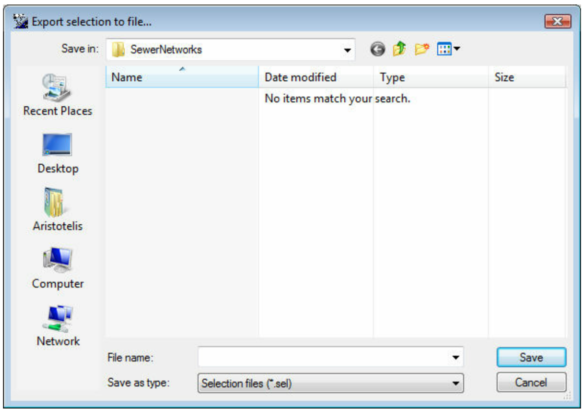
3. Select the path of the file.
4. Select the file type from the **Files of type** drop-down list. The default option is "Selection Files" with the extension .sel.
5. Select the file by clicking on it.
6. Select **Open** to open the selected file. Select **Cancel** to cancel the operation.

4.10.3 Save selection

With this option, you can save the current selection set to an external file. Note that no object properties are saved whatsoever. The file can be loaded and used at a later time, as long as the objects are not modified.

To save the current selection set to an external file:

1. Select **Selection** from the **Objects** menu.
2. Select **Save Selection** from the **Selection** menu. The following form appears:



3. Select the path of the file.
4. Type the filename in the **File name** text box.
5. Select **Save** to save the project with the selected filename and path. Select **Cancel** to cancel the operation.

4.10.4 Clear selection

With this option, you can deselect all items.

To deselect all items:

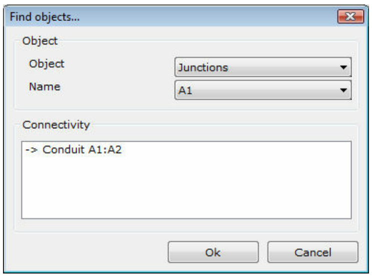
1. Select **Select objects** from the **Edit** menu.
2. Select **Clear Selection** from the **Select objects** menu. All objects are deselected.

4.11 Locate objects

With this option, the connectivity of a specified point, linear or surface object is revealed.

To locate objects:

1. Select **Locate objects** from the **Edit** menu. The following form appears:



2. Select the **Object type** from the drop-down list.
3. Select the **Name** of the object.
4. The connectivity of the specified object is loaded in the list.
5. Select **Ok** or **Cancel** to close the form.

Chapter V

5 View

5.1 View menu

With this menu, you can modify the appearance of the plan view and the profiles. In the **View** menu you can select one of the following options:

- Plan view
  - Visible objects
  - Background
- Background pictures
  - Add
  - Edit
  - Delete
  - Show
- Profile
  - Options

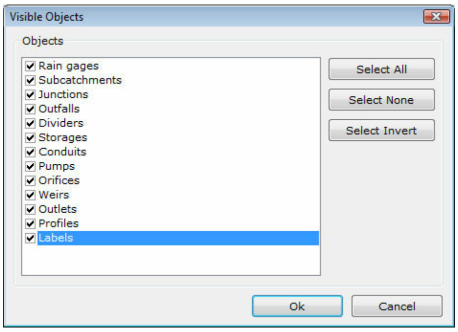
5.2 Plan view

5.2.1 Visible objects

With this option, you can select the object type(s) that will be visible in plan view. This option refers to the active objects of the plan view and not the background objects.

To select the object type(s) that will be visible in plan view:

1. Select **Plan view** from the **View** menu.
2. Select **Visible objects** from the **Plan view** menu. The following form appears:



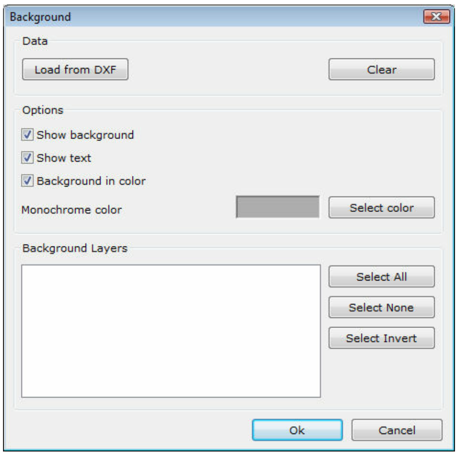
3. Select the object type(s) that you wish to be visible. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
4. Select **Ok** to accept changes and close the form. Select **Cancel** to close the form without any changes.

5.2.2 Background

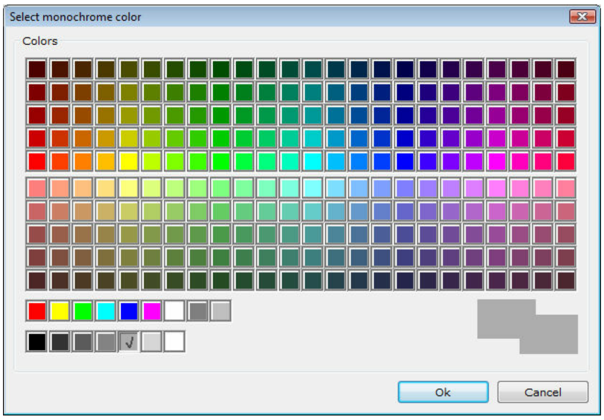
With this option, you have full access to the background data from DXF files. This includes the option to import data from a DXF file , under the **File > Import** menu.

To modify the background:

1. Select **Plan view** from the **View** menu.
2. Select **Background** from the **Plan view** menu. The following form appears:



3. To load data from a DXF file, select **Load from DXF.** This process is described in the **File > Import > Background from DXF** section.
4. Select **Clear** to clear the existing background. A confirmation message will be displayed.
5. Select **Show background** to toggle the visibility of the background.
6. Select **Show text** to toggle the visibility of TEXT objects of the background.
7. Select **Background in color** if you wish to view the background in color.
8. If the option of step 7 is deselected, a single color is used for all background objects. This color can be modified by selecting **Select color.**
8.1. The color selection dialog box appears.



8.2. Select the **color** from the 256 available colors. The currently selected color is marked with a tick. On top of the **Cancel** button, the old and the new color are displayed.
8.3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the

dialog box without saving the changes.

NOTE: The color palette follows standard CAD color palettes.

9. Select the **Background layers** that you wish to be active (visible). The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
10. Select **Ok** to accept changes and close the form. Select **Cancel** to close the form without any changes.

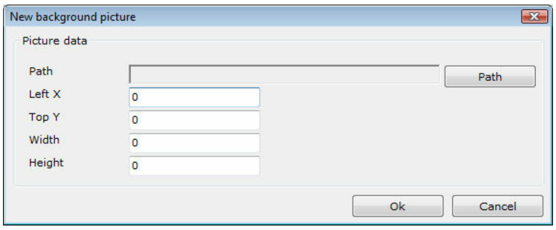
5.3 Background pictures

5.3.1 Add

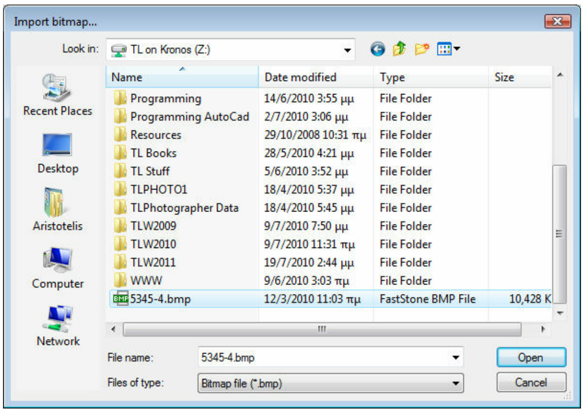
With this option, you can add a picture to the background.

To add a picture to the background:

1. Select **Background Pictures** from the **View** menu.
2. Select **Add** from the **Background Pictures** menu. The following form will appear:



3. Select **Path.** The file selection dialog box will appear:



4. Select the path of the file.
5. Select the file type from the **Files of type** drop-down list. The default option is "Bitmap file" with the extension .bmp.
6. Select the file by clicking on it.
7. Select **Open** to open the selected file. Select **Cancel** to cancel the operation.
8. Enter the **Left X, Top Y, Width** and **Height** of the picture in drawing units. If you provide only the height or the width, the ratio of the source picture will be used to calculate the missing data.

9. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

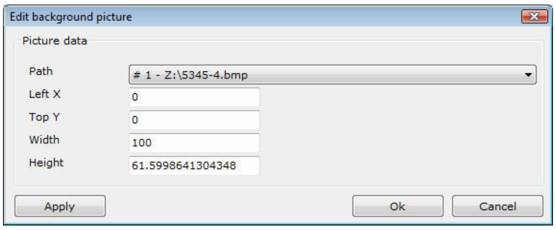
NOTE: The following image file types are supported: bitmaps (.bmp) and JPEG (.jpg).

5.3.2 Edit

With this option, you can modify the position and dimensions of an existing background picture.

To modify the position and dimensions of an existing background picture:

1. Select **Background Pictures** from the **View** menu.
2. Select **Edit** from the **Background Pictures** menu. The following form will appear:



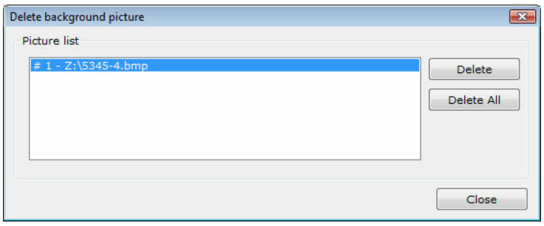
3. Select the picture from the drop-down list.
4. Make the appropriate changes. If you provide only the height or the width, the ratio of the source picture will be used to calculate the missing data.
5. Select **Apply** to apply the changes without closing the form. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

5.3.3 Delete

With this option, you can delete one or more existing background pictures.

To delete one or more existing background pictures:

1. Select **Background Pictures** from the **Map** menu.
2. Select **Delete** from the **Background Pictures** menu. The following form will appear:



3. Select the picture from the list.
4. Select **Delete** to delete the selected picture.
5. Select **Delete all** to delete all pictures.
6. Select **Close** to close the dialog box.

5.3.4 Show

With this option, you can show or hide all pictures in the background.

To show or hide all pictures in the background:

1. Select **Background pictures** from the **View** menu.
2. Select **Show** from the **Background pictures** menu.
3. If the background pictures were visible, they become invisible and vice-versa. A checkbox in the menu indicates if this option is enabled.

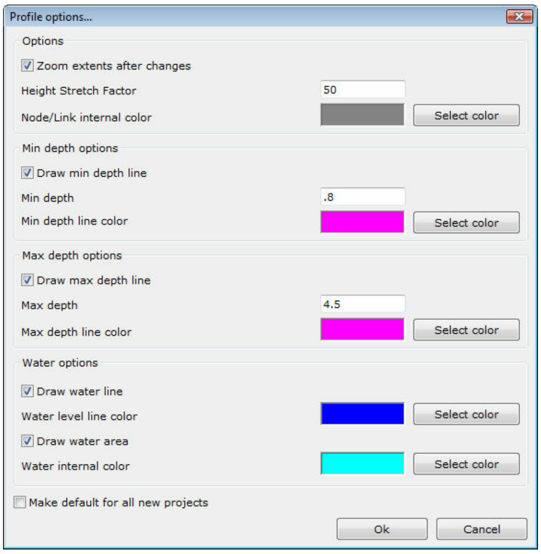
5.4 Profile

5.4.1 Options

With this option, you can modify the appearance of the profile sketch.

To modify the appearance of the profile sketch:

1. Select **Profile** from the **View** menu.
2. Select **Options** from the **Profile** menu. The following form appears:



3. Select **Zoom extents after changes** if you wish the sketch to zoom automatically to the extents of the drawing whenever you make a change.
4. Select an appropriate **Height stretch factor**. This factor magnifies distances in the Y direction so that small elevation differences are clear.
5. Select **Draw min depth line** to draw a line parallel to the ground at a specified depth. This line is a design tool that shows the minimum depth that a conduit should be placed (for example, to ensure a minimum backfill depth)
6. Type the depth of the min depth line in the **Min depth** field and select its color by pressing the corresponding **Select color** button.
7. Select **Draw max depth line** to draw a line parallel to the ground at a specified

depth. This line is a design tool that shows the maximum depth that a conduit should be placed (for example, to show the maximum depth that an excavator can reach)
8. Type the depth of the max depth line in the **Max depth** field and select its color by pressing the corresponding **Select color** button
9. Select **Make default for all new projects** to use these options for all new projects.
10. Select **Ok** to accept changes and close the form. Select **Cancel** to close the form without any changes.

Chapter VI

6 Data

6.1 Data menu

With this menu, you can add and modify data. In the **Data** menu you can select one of the following options:

- Project info
- Project summary
- General data
  - General
  - Dates
  - Time steps
  - Dynamic wave
  - Interface files
  - Design
  - Checks
  - Excavation options
- Climatology: Temperature, Evaporation, Wind speed, Snow melt, Areal depletion, IDF curve
- Hydrology: Raingages, Subcatchments, Aquifers, Snow packs, RDII Hydrographs
- Quality: Pollutants, Land uses, Quality data
- Curves: Management, Add, Delete, Edit, Move, Sort
- Time series: Management, Add, Delete, Edit, Move, Sort
- Time patterns: Management, Add, Delete, Edit, Move, Sort
- Runoff areas: Management, Add, Delete, Edit
- Sewer flow (population): Management, Add, Delete, Edit
- Sewer flow (area): Management, Add, Delete, Edit
- Conduit shapes: Management, Add, Delete, Edit, Import, Export
- Manhole specifications: Management, Add, Delete, Edit, Import, Export
- Trench specifications: Management, Add, Delete, Edit, Import, Export
- Network consistency
- Options
  - General preferences
  - Sketch
  - Grid editing
  - Customize toolbar
  - Default values
  - Algorithm

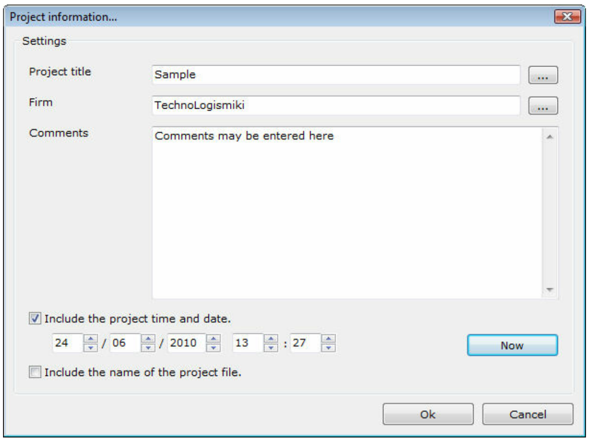
6.2 Project info

With this option, you can add project information that include title, firm title and comments. If you want, this information can be included in the reports. The empty fields are ignored.

To add or modify the project information:

1. Select **Project info** from the **Data** menu.
2. Type the **project title**, **firm** title and comments.

3. Check **Include project time and date** if you want to include the time and date in the project. In this case, type the day, month, year, hours and seconds in the corresponding text boxes. Alternatively, press **Today** to insert the current values automatically.
4. Check **Include the name of the project file** if you want the full path and filename of the project to be included in the report.
5. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



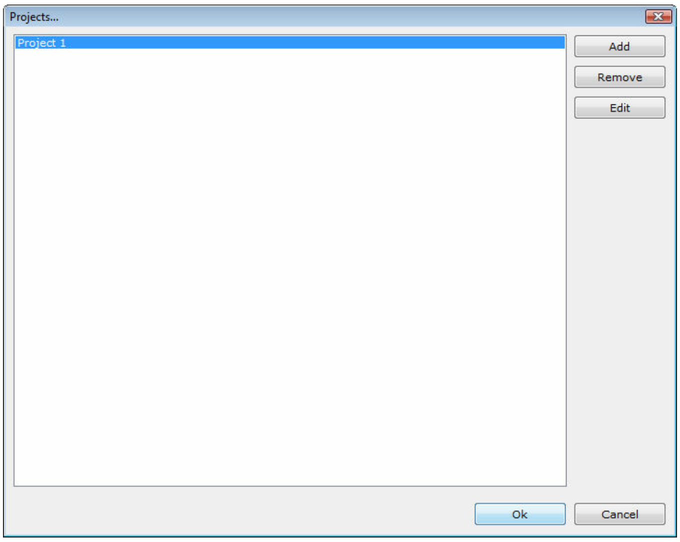
By selecting the buttons with the ellipses (...) next to the **project title** and **firm**, you can access the corresponding databases.

Project title database

For the completion of a project, more than one programs may be needed. For convenience, you can add the project title to the database and retrieve it from all programs.

To use the project title database:

1. Select the button with the ellipses (...) next to the project title text box. The project title database appears.
2. Select **Add** to add a new title to the database.
3. Select **Remove** to remove the selected entry from the database. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
4. Select **Edit** to modify the selected entry.
5. Select **Ok** to use the currently selected project title and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

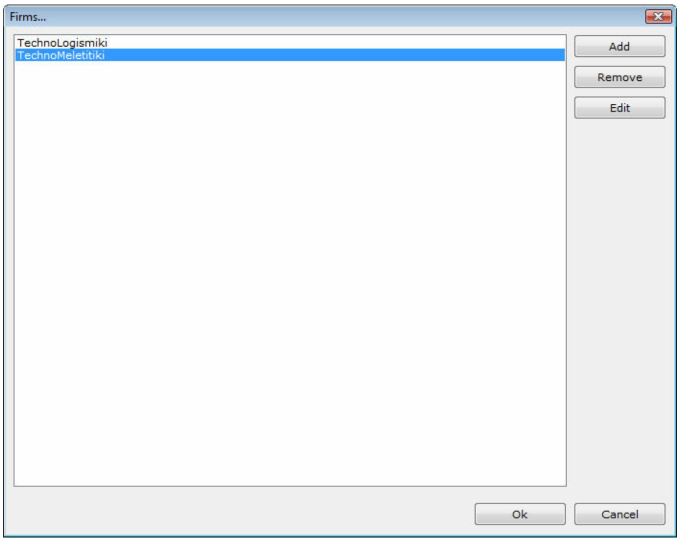


Firm database

An engineer or firm may be involved in multiple projects. For convenience, you can add the title to the database and retrieve it from all programs.

To use the firm database:

1. Select the button with the ellipses (...) next to the firm text box. The firm database appears.
2. Select **Add** to add a new firm/author to the database.
3. Select **Remove** to remove the selected entry from the database. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
4. Select **Edit** to modify the selected entry.
5. Select **Ok** to use the currently selected firm and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

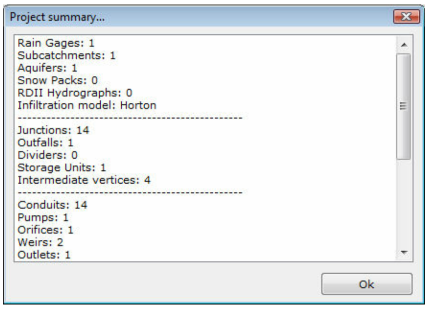


6.3 Project summary

With this option, a summary of the project is displayed. This includes the number of objects and the most important simulation and calculation options.

To display the project summary:

1. Select **Project summary** from the **Data** menu.
2. The project summary is displayed:



3. Press **Ok** to close the form.

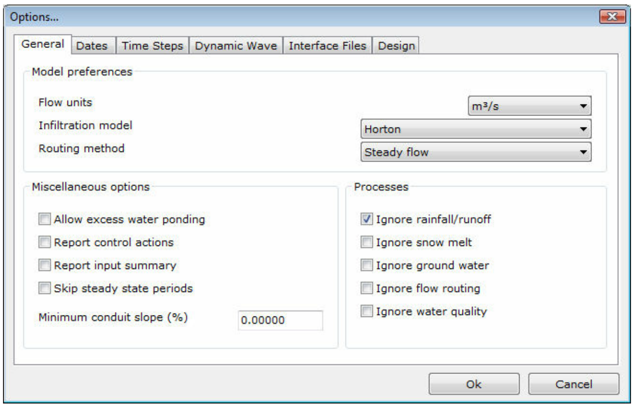
6.4 General data

6.4.1 General

With this option, you can input general data.

To input general data:

1. Select **General data > General** from the **Data** menu. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Model preferences

Flow units: select the appropriate flow units. There are six combination available, three of which belong to the metric system (m^3/s, L/s, ML/day) and three (ft^3/s, g/m, Mg/day) to the English systems. This affects all data input/output.

NOTE: If you change this setting, existing data will **not** be converted.

The units that are used in both the metric and English system are enumerated in the Appendix.

Infiltration model: select the appropriate infiltration model, which can be one of Horton, Green-Ampt or SCS Curve Number. Depending on the model, different data are required for each subcatchment.

Routing method: select the appropriate routing method, which can be one of: None, Kinematic wave, Dynamic wave and Steady flow. For compatibility with previous versions of the program, select steady flow.

Miscellaneous options:

Allow excess water ponding: Checking this option will allow excess water to collect atop nodes and be re-introduced into the system as conditions permit. In order for ponding to actually occur at a particular node, a nonzero value for its Ponded Area attribute must be used.

Report control actions: This option exists for compatibility with EPA's SWMM. It is always enabled in Sewer Networks.

Report input summary: This option exists for compatibility with EPA's SWMM. It is always enabled in Sewer Networks.

Skip steady state periods: Checking this option will make the simulation use the most recently computed conveyance system flows during a steady state period instead of computing a new flow routing solution. A time step is considered to be in steady state if the change in external inflow at each node is below 0.5 ft^3/s and the relative difference between total system inflow and outflow is below 5%.

Ignore rainfall / runoff, snow melt, ground water, flow routing and water quality: Check this option to ignore the respective models and computations. If rainfall / runoff calculations are disabled, then only user-specified direct inflow time series and dry weather inflows will be considered.

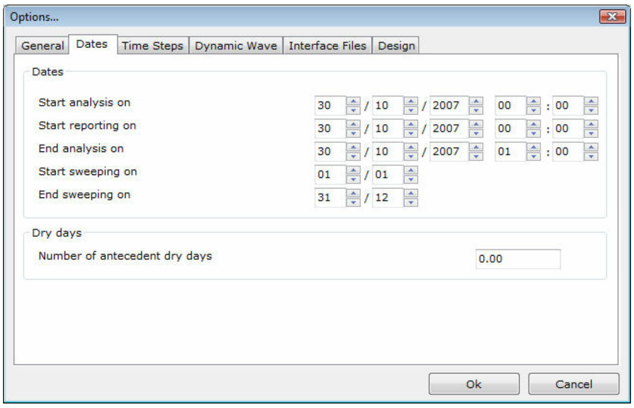
Minimum Conduit Slope: The minimum value allowed for a conduit's slope (%). If zero (the default) then no minimum is imposed (although the program uses a lower limit on elevation drop of 0.001 ft (0.00035 m) when computing a conduit slope).

6.4.2 Dates

With this option, you can input data regarding dates, simulation duration, report frequency etc.

To input data regarding dates:

1. Select **General data > Dates** from the **Data** menu. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Dates:

Start analysis on: select the day, month, year, hour and minute of the beginning of the simulation.

Start reporting on: select the day, month, year, hour and minute when reporting of simulation results is to begin. This must be on or after the simulation starting date and time.

End analysis on: Enter the date and time when the simulation is to end. This must be after the simulation starting date and time.

Start sweeping on: Enter the day of the year (month/day) when street sweeping operations begin. The default is January 1.

End sweeping on: Enter the day of the year (month/day) when street sweeping operations end. The default is December 31.

Dry days:

Number of antecedent dry days: Enter the number of days with no rainfall prior to the start of the simulation. This value is used to compute an initial buildup of pollutant load on the surface of subcatchments.

NOTE: If you are interested in steady state analysis then:

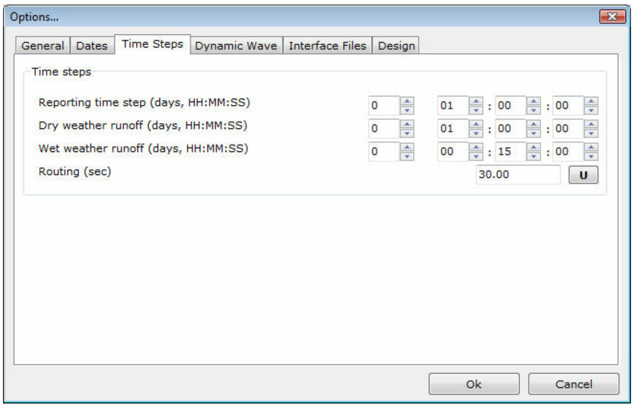
1. The start of the reporting is set equal to the start of the analysis.
2. The end of the analysis is set equal to the start of the analysis plus 15 minutes.
3. You should not change the default values of the times steps.

6.4.3 Time steps

With this option, you can input data regarding time steps.

To input data regarding time steps:

1. Select **General data > Time steps** from the **Data** menu. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Time steps:

Reporting time step: Enter the time interval for reporting of computed results. You should select an appropriate time step so that the number of results is about 100 per project.

Dry weather runoff: Enter the time step length used for runoff computations (consisting essentially of pollutant buildup) during periods when there is no rainfall and no ponded water. This must be greater or equal to the wet weather time step.

Wet weather runoff: Enter the time step length used to compute runoff from subcatchments during periods of rainfall or when ponded water still remains on the surface..

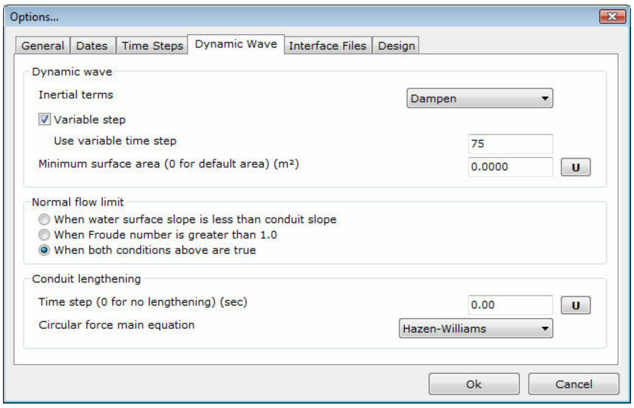
Routing (sec): Enter the time step length in decimal seconds used for routing flows and water quality constituents through the conveyance system. Values around 600 seconds are generally sufficient. However, note that Dynamic wave routing requires a much smaller time step than the other methods of flow routing

6.4.4 Dynamic wave

With this option, you can input data regarding the dynamic wave solver. These settings are ignored in case other solvers are employed.

To input data regarding the dynamic wave solver:

1. Select **General data > Dynamic wave** from the **Data** menu. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Dynamic wave:

Inertial terms: Indicates how the inertial terms in the St. Venant momentum equation will be handled. Select **keep** to maintain these terms at their full value under all conditions. Select **Dampen** to reduce the terms as flow comes closer to being critical and ignores them when flow is supercritical. Select **Ignore** to drop the terms altogether from the momentum equation, producing what is essentially a Diffusion Wave solution.

Variable step: Check the box if an internally computed variable time step should be used at each routing time period and select an adjustment (or safety) factor to apply to this time step. The variable time step is computed so as to satisfy the Courant condition within each conduit and yet prevent an excessive change in water depth at each node. A typical adjustment factor would be 75% to provide some margin of conservatism. The computed variable time step will not be less than 0.5 seconds nor be greater than the fixed time step specified on the Time Steps page of the dialog. If the latter was set lower than 0.5 seconds then the variable time step option is ignored.

Minimum surface area: This is a minimum surface area used at nodes when computing changes in water depth. If 0 is entered, then the default value of 12.566 ft2 (1.167 m2) is used. This is the area of a 4-ft diameter manhole.

Normal flow limit

Selects the basis used to determine when supercritical flow occurs in a conduit. The choices are (a) water surface slope only (i.e., water surface slope > conduit slope) (b) Froude number only (i.e., Froude number > 1.0) and (c) both water surface slope and Froude number. The last option is recommended.

Conduit lengthening:

Time step (0 for no lengthening) (sec): This is a time step, in seconds, used to artificially lengthen conduits so that they meet the Courant stability criterion under full-flow conditions (i.e., the travel time of a wave will not be smaller than the specified conduit lengthening time step). As this value is decreased, fewer conduits will require lengthening. A value of 0 means that no conduits will be lengthened.

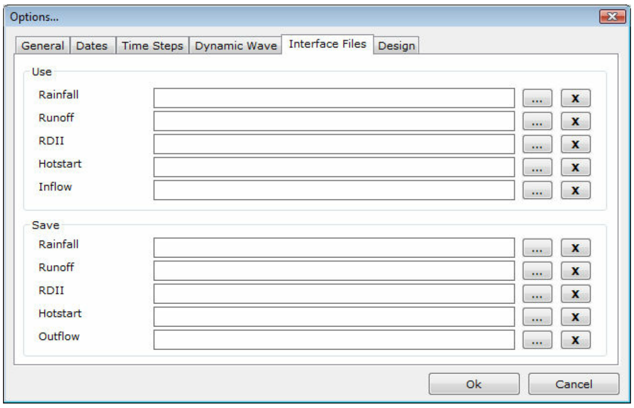
Circular force main equation: select one of Hazen-Williams or Darcy-Weisbach. This equation will be used to compute friction losses during pressurized flow for conduits that have been assigned a Circular Force Main cross-section.

6.4.5 Interface files

With this option, you can input data regarding interface files.

To input data regarding interface files:

1. Select **General data > Interface files** from the **Data** menu. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Use:

The external file references that are defined serve as source for data input. Press the corresponding button with the ellipses "..." to define a reference to an external file. Press the "X" button to remove the reference. Source files can be defined for the following cases:

- Rainfall
- Runoff
- RDII
- Hotstart
- Inflow

Save:

The external file references that are defined serve as data output. Press the corresponding button with the ellipses "..." to define a reference to an external file. Press the "X" button to remove the reference. Target files can be defined for the following cases:

- Rainfall

- Runoff
- RDII
- Hotstart
- Outflow

General information on the interface files

SWMM can use several different kinds of interface files that contain either externally imposed inputs (e.g., rainfall or infiltration/inflow hydrographs) or the results of previously run analyses (e.g., runoff or routing results). These files can help speed up simulations, simplify comparisons of different loading scenarios, and allow large study areas to be broken up into smaller areas that can be analyzed individually.

The **rainfall** and **runoff** interface files are binary files created internally by the program. These can be saved and reused from one analysis to the next.

If the same rainfall data are being used with many different analyses, saving the rainfall interface file after the first run and then reusing this file in subsequent runs can save computation time.

The rainfall interface file should not be confused with a rainfall data file. The latter is an external text file that provides rainfall time series data for a single rain gage. The former is a binary file created internally that processes all of the rainfall data files used by a project.

The runoff interface file can be used to save the runoff results generated from a simulation run. If runoff is not affected in future runs, the user can request that the program use this interface file to supply runoff results without having to repeat the runoff calculations again.

Hot start files are binary files created by the program that contain hydraulic and water quality variables for the drainage system at the end of a run. These data consist of the water depth and concentration of each pollutant at each node of the system as well as the flow rate and concentration of each pollutant in each link. The hot start file saved after a run can be used to define the initial conditions for a subsequent run.

Hot start files can be used to avoid the initial numerical instabilities that sometimes occur under Dynamic Wave routing. For this purpose they are typically generated by imposing a constant set of base flows (for a natural channel network) or set of dry weather sanitary flows (for a sewer network) over some startup period of time. The resulting hot start file from this run is then used to initialize a subsequent run where the inflows of real interest are imposed

It is also possible to both use and save a hot start file in a single run, starting off the run with one file and saving the ending results either to the same or to another file. The resulting file can then serve as the initial conditions for a subsequent run if need be. This technique can be used to divide up extremely long continuous simulations into more manageable pieces

The **RDII** interface file is a text file that contains a time series of rainfall-derived infiltration/inflow flows for a specified set of drainage system nodes. This file can be generated from a previous run when Unit Hydrographs and nodal RDII inflow data have been defined for the project, or it can be created outside of the program using some other source of RDII data (e.g., through measurements or output from a different computer program). The format of the file is the same as that of the routing interface file discussed below, where Flow is the only variable contained in the file.

A **routing** interface file stores a time series of flows and pollutant concentrations that are discharged from the outfall nodes of drainage system model. This file can serve as the source of inflow to another drainage system model that is connected at the outfalls of the first system. This allows very large systems to be broken into smaller sub-systems that can be analyzed separately and linked together through the routing interface file.

Interface file format (RDII/routing)

RDII interface files and routing interface files have the same format. They are simple text files. Their structure is as follows:

1. the first line contains the keyword "SWMM5" (without the quotes).
2. a line of text that describes the file (can be blank).
3. the time step used for all inflow records (integer seconds).
4. the number of variables stored in the file, where the first variable must always be flow rate.
5. the name and units of each variable (one per line), where flow rate is the first variable listed and is always named FLOW.
6. the number of nodes with recorded inflow data
7. the name of each node (one per line).
8. a line of text that provides column headings for the data to follow (can be blank).
9. for each node at each time step, a line with:
- the name of the node
- the date (year, month, and day separated by spaces)
- the time of day (hours, minutes, and seconds separated by spaces)
- the flow rate followed by the concentration of each quality constituent

Time periods with no values at any node can be skipped. An excerpt from an RDII / routing interface file is shown below:

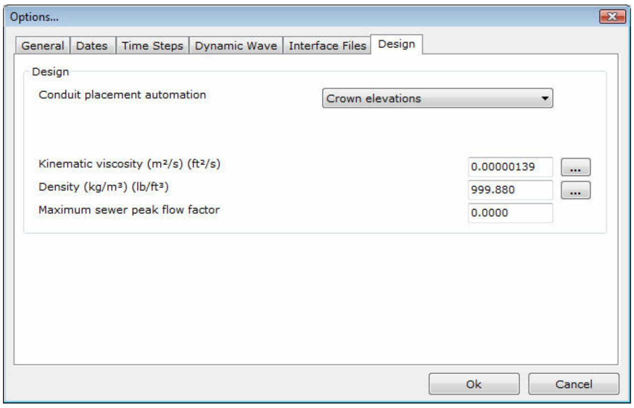
```
SWMM5
Example File
300
1
FLOW CFS
2
N1
N2
Node Year Mon Day Hr Min Sec Flow
N1   2002 04  01  00 20  00  0.000000
N2   2002 04  01  00 20  00  0.002549
N1   2002 04  01  00 25  00  0.000000
N2   2002 04  01  00 25  00  0.002549
```

6.4.6 Design

With this option, you can input data regarding design.

To input data regarding design:

1. Select **General data > Design** from the **Data** menu. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Design:

Conduit placement automation: select one of **crown elevations**, **invert elevations**, **80% of conduit height**, **axes** or **none**. Whatever this setting may be, the user can always define the position of a conduit manually.

Cuts in excavations measured: select one of **from ground** or **from trench**. In the second case, trench data are required.

Inlet bottoms measured: select one **from bottom slab** or **from sublayer**.

Kinematic viscosity (m^2/s): select the kinematic viscosity of the fluid. Press the button with the ellipses "..." to invoke the corresponding database. This value is needed when quality checks are performed and when the friction is calculated based on the Darcy - Weisbach method.

Density (kg/m^3): select the density of the fluid. Press the button with the ellipses "..." to invoke the corresponding database. This value is needed when quality checks are performed and when the friction is calculated based on the Darcy - Weisbach method.

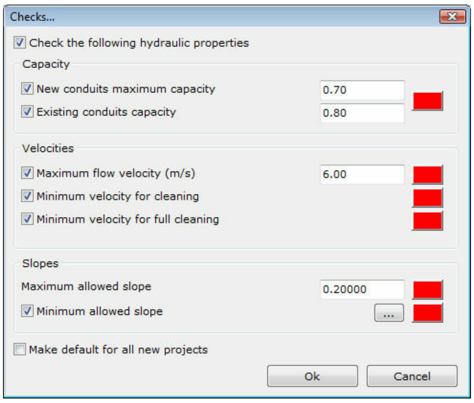
Maximum sewer peak flow factor: this is the maximum acceptable value of peak flow factor. According to Greek Regulations, this value is equal to 3.

6.4.7 Checks

With this option, the maximum and minimum values of various parameters can be set. If these values are breached, the corresponding field is typed in different (but customizable) color. The default color option is red.

To set the maximum and minimum values of various parameters:

1. Select **General data > Checks** from the **Data** menu. The following form appears:



2. Select **Check the following hydraulic properties** to enable or disable all checks.
3. Make the appropriate selections as described below.
4. Select **Make default for all new projects** if you wish to use these settings for all new projects.
5. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Capacity

- Check **New conduits maximum capacity** to enable this check. Set the value in the corresponding field.
- Check **Existing conduits maximum capacity** to enable this check. Set the value in the corresponding field. This check does not necessarily refer to old conduits, but to existing conduits as set in conduit properties.
- Press the corresponding colored button to change the font color of the results that fail these checks.

NOTE: This check is not restrictive for the network. A conduit is functional even when its capacity is 80%. However, the desired maximum capacity may be only 50%.

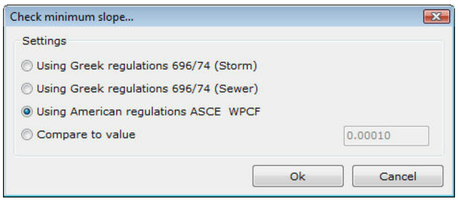
Velocities

- Check **Maximum flow velocity** to enable this check. Set the value in the corresponding field.
- Check **Minimum velocity for cleaning** to enable this check.
- Check **Minimum velocity for full cleaning** to enable this check.
- Press the corresponding colored button to change the font color of the results that fail these checks.

NOTE: The minimum velocity for cleaning and for full cleaning can be determined only when appropriate quality data are provided.

Slopes

- Set the **Maximum allowed slope** in the corresponding field.
- Check **Minimum allowed slope** to enable this check. Press the button with the ellipses to select the calculation method. The following form appears:



Select the method of calculating the minimum slope. This can be one of the following:
- Greek Regulations 696/74 for storm networks
- Greek Regulations 696/74 for sewer or combined networks
- American regulations ASCE & WPCF
- Compare to value (in this case, you must provide the threshold value in the corresponding field)

Press **Ok** to accept the changes. Press **Cancel** to close the form without any changes.

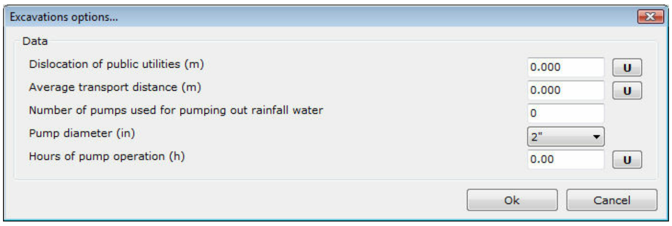
- Press the corresponding colored buttons to change the font color of the results that fail these checks.

6.4.8 Excavation options

With this option, you can set several excavation options.

To set excavation options:

1. Select **General data > Checks** from the **Data** menu. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Data

- Enter the **expected dislocation of public utilities** in ft or m.
- Enter the **average transport distance** of excavations in ft or m.
- Enter the **number of pumps used for pumping out rainfall water**.
- Select the **pump diameter**.
- Enter the **pump operation hours**.

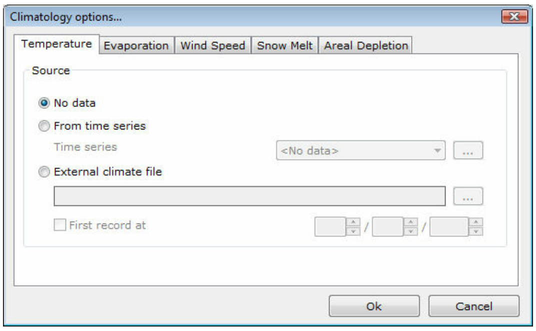
6.5 Climatology

6.5.1 Temperature

With this option, you can input temperature data.

To input temperature data:

1. Select **Climatology > Temperature** from the **Data** menu. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Source

- Select **No data** if you do not want to input temperature data. Make this selection if you do not want to simulate snow melt.
- Select **From time series** if you have already entered a temperature time series and you wish to use it. Otherwise, press the corresponding button with the ellipses "..." to define a time series.
- Select **External climate file** if you wish to use an external file containing data. Press the corresponding button with the ellipses "..." to define a reference to an external file. You can optionally select the first record to be taken into account by setting its day, month and time.

Climate files

The program can use an external climate file that contains daily air temperature, evaporation, and wind speed data. The program recognizes the following formats:

- **DSI-3200/DSI-3210** file available from the National Climatic Data Center (USA).
- **Canadian climate** files available from Environment Canada.
- A **user-prepared climate file** where each line contains a recording station name, the year, month, day, maximum temperature, minimum temperature, and optionally, evaporation rate, and wind speed. If no data are available for any of these items on a given date, then an asterisk should be entered as its value.

When a climate file has days with missing values, the program will use the value from the most recent previous day with a recorded value. For a user-prepared climate file, the data must be in the same units as the project being analyzed. For US units, temperature is in degrees F, evaporation is in inches/day, and wind speed is in miles/hour. For metric units, temperature is in degrees C, evaporation is in mm/day, and wind speed is in km/hour:

```
ST001   2004   12   01   10.0   28.0   145.0   1.9
ST002   2004   12   01   11.1   27.6   155.3   2.0
ST001   2004   12   01   9.9    26.5   144.0   1.9
```

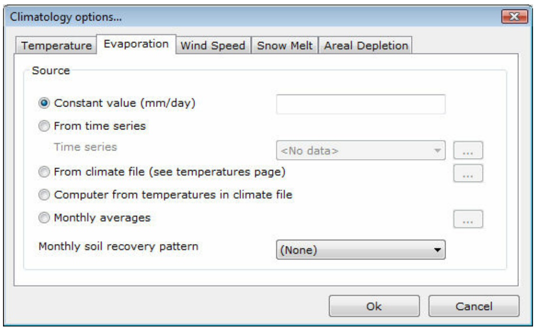
```
ST002  2004  12  01  10.2  26.2  150.1  1.8
```

6.5.2 Evaporation

With this option, you can input data regarding evaporation rates. Evaporation occurs for standing water on subcatchment surfaces, for subsurface water in groundwater aquifers, and for water held in storage units.

To input data regarding evaporation rates:

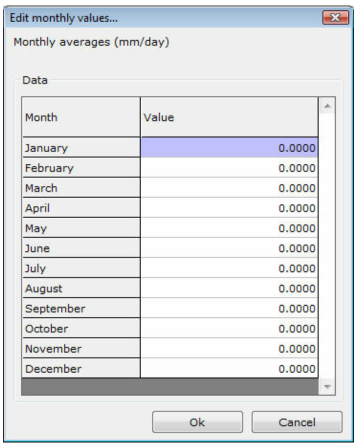
1. Select **Climatology > Evaporation** from the **Data** menu. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Source

- Select **Constant value** if evaporation remains constant over time. Provide the evaporation rate in the corresponding field.
- Select **From time series** if you have already entered a evaporation time series and you wish to use it. If you have not defined a time series, press the corresponding button with the ellipses "..." to define one.
- Select **From climate file** if you wish to use the external climate file defined in the temperature page. The file must contain evaporation data.
- Select **Monthly averages** if you wish to use mean annual monthly values. Press the corresponding button with the ellipses "...". The following form appears:



The user can specify an optional **Monthly Soil Recovery Pattern**. This is a time pattern whose factors adjust the rate at which infiltration capacity is recovered during periods with no precipitation. It applies to all subcatchments for any choice of infiltration method. For example, if the normal infiltration recovery rate was 1% during a specific time period and a pattern factor of 0.8 applied to this period, then the actual recovery rate would be 0.8%. The Soil Recovery Pattern allows one to account for seasonal soil drying rates. In principle, the variation in pattern factors should mirror the variation in evaporation rates but might be influenced by other factors such as seasonal groundwater levels.

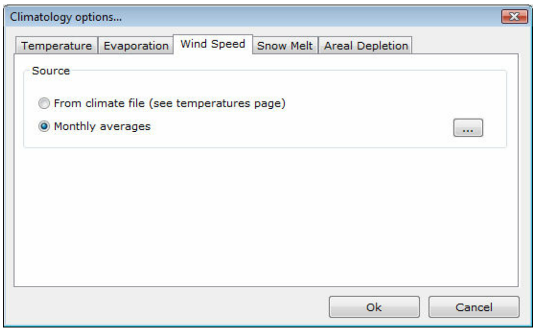
Type the values directly onto the table. Press **Ok** to close the form saving all changes. Press **Cancel** to close the form without any changes.

6.5.3 Wind speed

With this option, you can input wind speed data. Wind speed is an optional climatic variable that is only used for snowmelt calculations.

To input wind speed data:

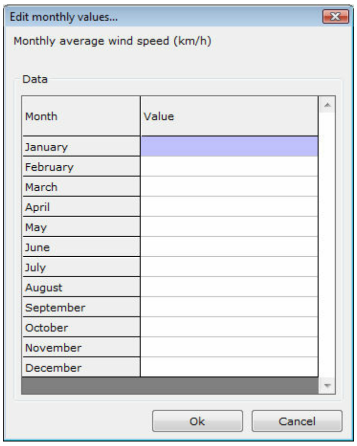
1. Select **Climatology > Wind speed** from the **Data** menu. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Source

- Select **From climate file** if you wish to use the external climate file defined in the temperature page. The file must contain wind speed data.
- Select **Monthly averages** if you wish to use mean annual monthly values. Press the corresponding button with the ellipses "...". The following form appears:



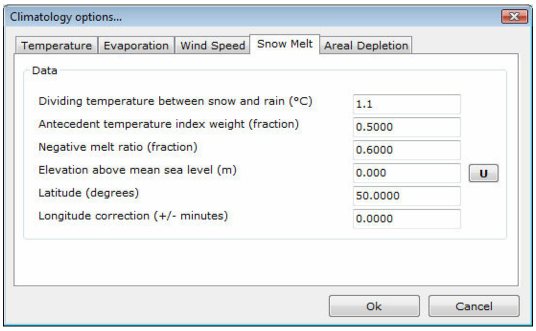
Type the values directly onto the table. Press **Ok** to close the form saving all changes. Press **Cancel** to close the form without any changes.

6.5.4 Snow melt

With this option, you can input snow melt data.

To input snow melt data:

1. Select **Climatology > Snow melt** from the **Data** menu. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

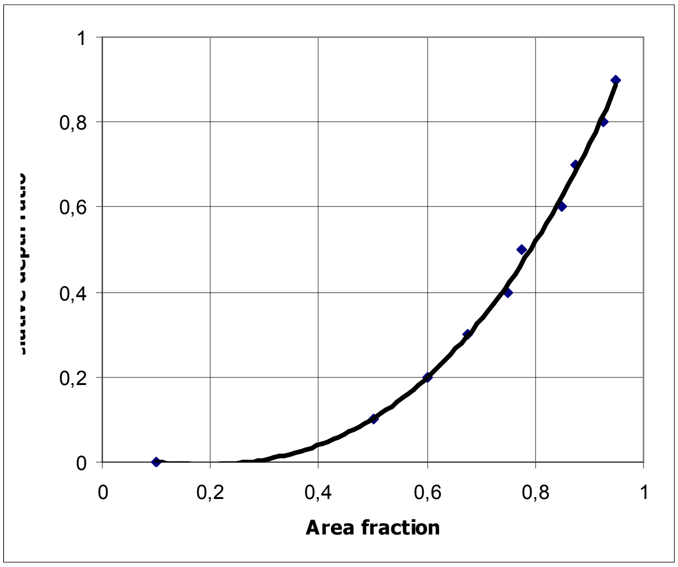
Data

- **Dividing temperature between snow and rain:** If the temperature is below this value then the precipitation falls as snow instead of rain.
- **Antecedent Temperature Index weight:** This parameter reflects the degree to which heat transfer within a snow pack during non-melt periods is affected by prior air temperatures. Smaller values reflect a thicker surface layer of snow which results in reduced rates of heat transfer. Values must be between 0 and 1, and the default is 0.5.
- **Negative melt ratio:** This is the ratio of the heat transfer coefficient of a snow pack during non-melt conditions to the coefficient during melt conditions. It must be a number between 0 and 1. The default value is 0.6.

- **Elevation above mean sea level:** Enter the average elevation above mean sea level for the study area, in feet or meters. This value is used to provide a more accurate estimate of atmospheric pressure. The default is 0.0, which results in a pressure of 29.9 inches Hg. The effect of wind on snow melt rates during rainfall periods is greater at higher pressures, which occur at lower elevations.
- **Latitude:** Enter the latitude of the study area in degrees North. This number is used when computing the hours of sunrise and sunset, which in turn are used to extend min/max daily temperatures into continuous values. The default is 50 degrees North
- **Longitude correction (+/- minutes):** This is a correction, in minutes of time, between true solar time and the standard clock time. It depends on a location's longitude (θ) and the standard meridian of its time zone (SM) through the expression 4(θ-SM). This correction is used to adjust the hours of sunrise and sunset when extending daily min/max temperatures into continuous values. The default value is 0.

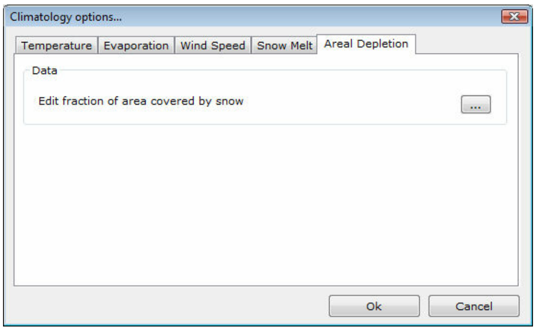
6.5.5 Areal depletion

With this option, you can input areal depletion data. These are used to specify points on the Areal Depletion Curves for both impervious and pervious surfaces within a project's study area. These curves define the relation between the area that remains snow covered and snow pack depth. Each curve is defined by 10 equal increments of relative depth ratio between 0 and 0.9. (Relative depth ratio is the ratio of an area's current snow depth to the depth at which there is 100% areal coverage). A typical graph is as follows:



To input area depletion data:

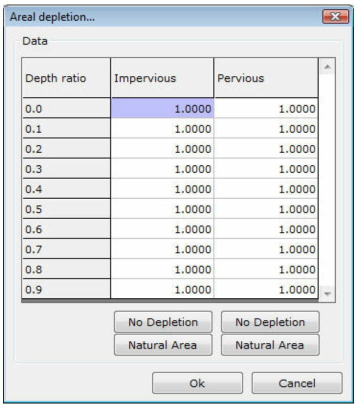
1. Select **Climatology > Areal depletion** from the **Data** menu. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Data

- Select the button with the ellipses to **edit the fraction of area covered by snow.** The following form appears:



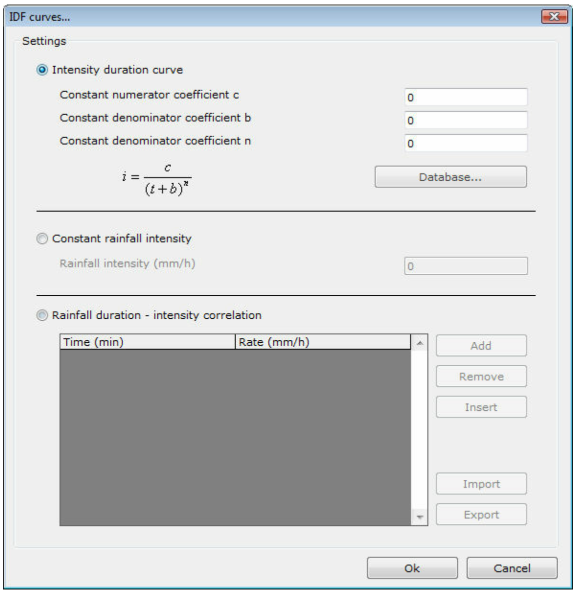
Enter values in the data grid provided for the fraction of each area that remains snow covered at each specified relative depth ratio. Valid numbers must be between 0 and 1, and be increasing with increasing depth ratio.
Press the **Natural Area** button to fill the grid with values that are typical of natural areas.
Press the **No Depletion** button to fill the grid with all 1's, indicating that no areal depletion occurs.
Press **Ok** to save the changes and close the dialog box. Press **Cancel** to close the dialog box without saving any changes.

6.5.6 IDF curve

With this option, you can select the IDF curve that will be used for the calculation of the design storm. The rain intensity can be either constant or calculated from an IDF curve or by using linear interpolation in a data matrix that correlates the rainfall duration and intensity.

To select the IDF curve:

1. Select **Climatology > IDF curve** from the **Data** menu. The following form appears:



2. Make the appropriate changes.
3. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

To use an IDF curve:

1. Click the **Intensity duration curve** option button.
2. Enter the values of the coefficients **c**, **b** and **n**, in such a way that when time **t** is entered in hours, the intensity will be given in mm/h.
3. Optionally, select **Database...** to invoke the IDF database.

NOTE: The selection of IDF curve is not required in case the design flow rate is known or raingages are to be used. IDF curves are used in conjunction with runoff areas.

To select a constant rainfall intensity:

1. Click the **Constant rainfall intensity** option button.
2. Enter the **rainfall intensity** in mm/h. The design flow rate is calculated based on this value; it is not based on the initial concentration time.

To use linear interpolation in a data matrix that correlates the rainfall duration and intensity:

1. Click the **Rainfall duration - intensity correlation** option button.
2. Enter a curve that describes the correlation between rainfall duration (in min) and intensity (in mm/h). The program will use the exact values, if found, or will estimate a value using linear interpolation. The curve can be entered manually or imported from a RCV (TechnoLogismiki) file or a HYDRO IDF file.
3. Select **Import** to import a curve from a RCV (TechnoLogismiki) file or a HYDRO IDF file.
4. Select **Export** to export the current curve from a RCV (TechnoLogismiki) file or a HYDRO IDF file.

5. Select **Add, Remove** and **Insert** to add, remove and insert a record, respectively.

6.6 Hydrology

6.6.1 Raingages

This option is described in detail in Objects > Properties > Raingages.

6.6.2 Subcatchments

This option is described in detail in Objects > Properties > Subcatchments.

6.6.3 Aquifers

Aquifers are sub-surface groundwater areas used to model the vertical movement of water infiltrating from the subcatchments that lie above them. They also permit the infiltration of groundwater into the drainage system, or exfiltration of surface water from the drainage system, depending on the hydraulic gradient that exists.

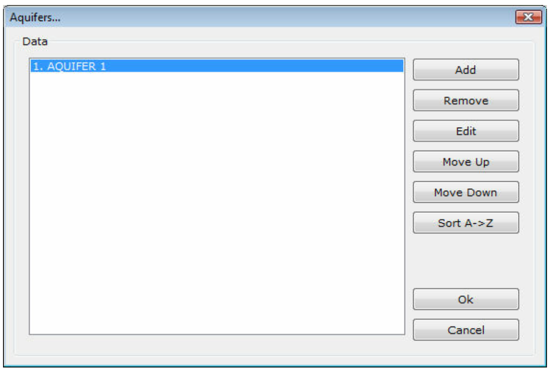
The same aquifer object can be shared by several subcatchments. Aquifers are only required in models that need to explicitly account for the exchange of groundwater with the drainage system or to establish baseflow and recession curves in natural channels and non-urban systems.

Aquifers are represented using two zones – an un-saturated zone and a saturated zone. Their behavior is characterized using such parameters as soil porosity, hydraulic conductivity, evapotranspiration depth, bottom elevation, and loss rate to deep groundwater. In addition, the initial water table elevation and initial moisture content of the unsaturated zone must be supplied.

Aquifers are connected to subcatchments and to drainage system nodes.

To manage aquifers:

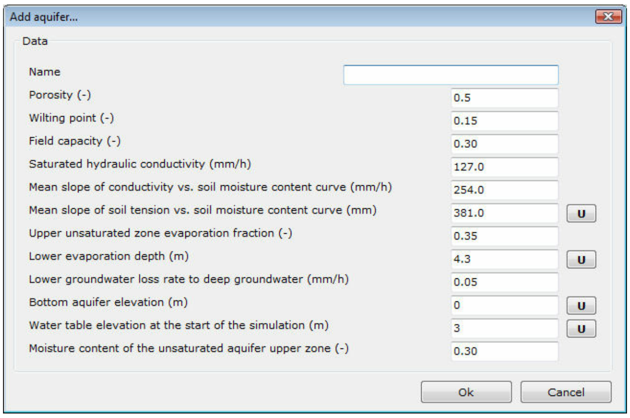
1. Select **Hydrology > Aquifers** from the **Data** menu. The following form appears:



2. Make the necessary modifications.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To add a new aquifer:

1. Press **Add**. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To edit an existing aquifer:

1. Select the aquifer from the list on the left.
2. Press **Add**. The data form appears.
3. Make the appropriate selections as described below.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To delete an existing aquifer:

1. Select the aquifer from the list on the left.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The aquifer is deleted from the list.

To move an existing aquifer upwards in the list:

1. Select the aquifer from the list on the left.
2. Press **Move Up**.
3. The aquifer is moved one place upwards.

To move an existing aquifer downwards in the list:

1. Select the aquifer from the list on the left.
2. Press **Move Down**.
3. The aquifer is moved one place downwards.

To sort the aquifer list:

1. Press **Sort A->Z**.
2. The list is sorted alphabetically.

Aquifer properties

- **Name**: enter a user-assigned aquifer name. This name cannot be null or used for another aquifer in the project.

- **Porosity (volumetric fraction)**: enter the ration of volume of voids over the total soil volume
- **Wilting point (volumetric fraction)**: enter the soil moisture content at which plants cannot survive.
- **Field capacity (volumetric fraction):** enter the soil moisture content after all free water has drained off.
- **Saturated hydraulic conductivity (in/hr or mm/hr)**: enter the soil's saturated hydraulic conductivity.
- **Mean slope of conductivity vs soil moisture content curve (in/hr or mm/hr)**: enter the average slope of conductivity versus soil moisture content curve.
- **Mean slope of soil tension vs soil moisture content curve (inches or mm):** enter the average slope of soil tension versus soil moisture content curve.
- **Upper unsaturated zone evaporation fraction :** enter the fraction of total evaporation available for evapotranspiration in the upper unsaturated zone.
- **Lower evaporation depth (ft or m):** enter the maximum depth into the lower saturated zone over which evapotranspiration can occur.
- **Lower groundwater loss rate to deep groundwater (in/hr or mm/hr) :** enter the rate of percolation from saturated zone to deep groundwater.
- **Bottom aquifer elevation (ft or m):** enter the elevation of the bottom of the aquifer.
- **Water table elevation at the start of the simulation (ft or m):** enter the elevation of the water table in the aquifer at the start of the simulation.
- **Moisture content of the unsaturated aquifer upper zone (volumetric fraction):** enter the moisture content of the unsaturated upper zone of the aquifer at the start of the simulation (it cannot exceed soil porosity).

6.6.4 Snow packs

Snow Pack objects contain parameters that characterize the buildup, removal, and melting of snow over three types of sub-areas within a subcatchment:

- The **Plowable** snow pack area consists of a user-defined fraction of the total impervious area. It is meant to represent such areas as streets and parking lots where plowing and snow removal can be done.
- The **Impervious** snow pack area covers the remaining impervious area of a subcatchment.
- The **Pervious** snow pack area encompasses the entire pervious area of a subcatchment.

Each of these three areas is characterized by the following parameters:

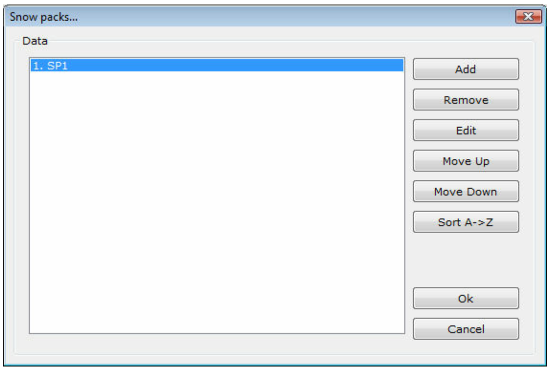
- Minimum and maximum snow melt coefficients
- minimum air temperature for snow melt to occur
- snow depth above which 100% areal coverage occurs
- initial snow depth
- initial and maximum free water content in the pack

In addition, a set of snow removal parameters can be assigned to the Plowable area. These parameters consist of the depth at which snow removal begins and the fractions of snow moved onto various other areas.

A single snow pack object can be applied to any number of subcatchments. Assigning a snow pack to a subcatchment simply establishes the melt parameters and initial snow conditions for that subcatchment. Internally, the program creates a "physical" snow pack for each subcatchment, which tracks snow accumulation and melting for that particular subcatchment based on its snow pack parameters, its amount of pervious and impervious area, and the precipitation history it sees.

To manage snow packs:

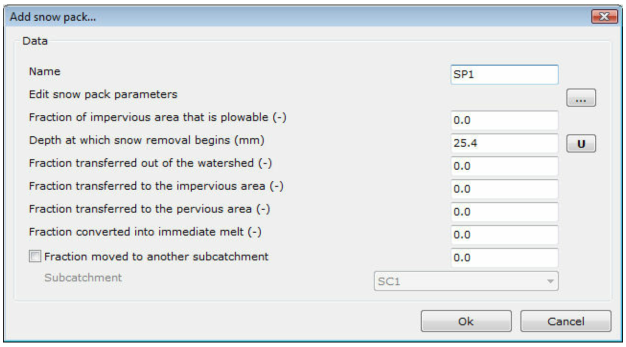
1. Select **Hydrology > Snow packs** from the **Data** menu. The following form appears:



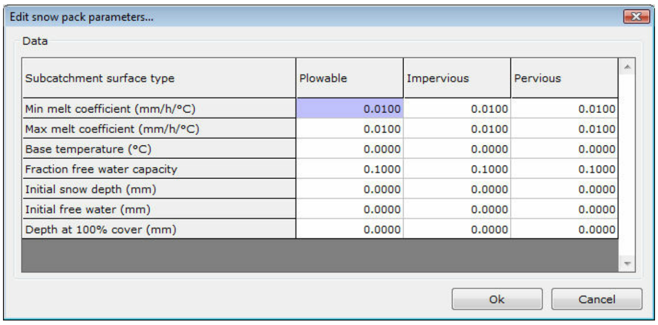
2. Make the necessary modifications.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To add a new snow pack:

1. Press **Add**. The following form appears:



2. Make the appropriate selections as described below.
3. Press the button with the ellipses "..." to edit the snow pack parameters. The following form appears:



Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To edit an existing snow pack:

1. Select the snow pack from the list on the left.
2. Press **Add**. The data form appears.
3. Make the appropriate selections as described below.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To delete an existing snow pack:

1. Select the snow pack from the list on the left.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The snow pack is deleted from the list.

To move an existing snow pack upwards in the list:

1. Select the snow pack from the list on the left.
2. Press **Move Up**.
3. The snow pack is moved one place upwards.

To move an existing snow pack downwards in the list:

1. Select the snow pack from the list on the left.
2. Press **Move Down**.
3. The snow pack is moved one place downwards.

To sort the snow pack list:

1. Press **Sort A->Z**.
2. The list is sorted alphabetically.

Snow pack properties

- **Name**: enter a user-assigned snow pack name. This name cannot be null or used for another snow pack in the project.
- **Fraction of impervious area that is plowable**: enter the fraction of the impervious area that is plowable and therefore is not subject to areal depletion.

- **Depth at which snow removal begins (in or mm)**: no removal occurs at depths below this and the fractions specified below are applied to the snow depths in excess of this number.
- **Fraction transferred out of the watershed**: enter the fraction of excess snow depth that is removed from the system (and does not become runoff).
- **Fraction transferred to the impervious area**: enter the fraction of excess snow depth that is added to snow accumulation on the pack's impervious area.
- **Fraction transferred to the pervious area**: enter the fraction of excess snow depth that is added to snow accumulation on the pack's pervious area.
- **Fraction converted into immediate melt**: enter the fraction of excess snow depth that becomes liquid water which runs onto any subcatchment associated with the snow pack.
- **Fraction moved to another subcatchment**: enter the fraction of excess snow depth which is added to the snow accumulation on some other subcatchment. The name of the subcatchment must also be provided.

Snow pack parameters

- **Min melt coefficient (in/h/deg.F or mm/h/deg.C)**: enter the degree-day snow melt coefficient that occurs on December 21.
- **Max melt coefficient (in/h/deg.F or mm/h/deg.C)**: enter the degree-day snow melt coefficient that occurs on June 21. For a short term simulation of less than a week or so it is acceptable to use a single value for both the minimum and maximum melt coefficients. The minimum and maximum snow melt coefficients are used to estimate a melt coefficient that varies by day of the year. The latter is used in the following degree-day equation to compute the melt rate for any particular day:

  Melt Rate = (Melt Coefficient) * (Air Temperature – Base Temperature).
- **Base temperature (deg. F or deg. C):** enter the temperature at which snow begins to melt.
- **Fraction free water capacity:** enter the volume of a snow pack's pore space which must fill with melted snow before liquid runoff from the pack begins, expressed as a fraction of snow pack depth.
- **Initial snow depth (water equivalent in or mm):** enter the depth of snow at the start of the simulation.
- **Initial free water (in or mm):** enter the depth of melted water held within the pack at the start of the simulation. This number should be at or below the product of the initial snow depth and the fraction free water capacity.
- **Depth at 100% cover (in or mm):** enter the depth of snow beyond which the entire area remains completely covered and is not subject to any areal depletion effect.

6.6.5 RDII hydrographs

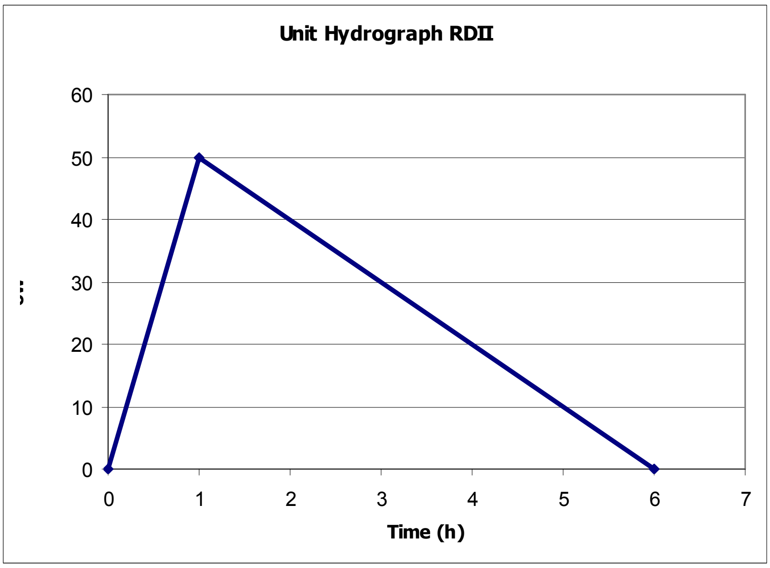
Unit Hydrographs (UHs) estimate rainfall-derived infiltration/inflow (RDII) into a sewer system. A UH set contains up to three such hydrographs, one for a short-term response, one for an intermediate-term response, and one for a long-term response. A UH group can have up to 12 UH sets, one for each month of the year. Each UH group is considered as a separate object by the program, and is assigned its own unique name along with the name of the rain gage that supplies rainfall data to it.

Each unit hydrograph is defined by three parameters:

- **R**: the fraction of rainfall volume that enters the sewer system
- **T**: the time from the onset of rainfall to the peak of the UH (h)
- **K**: the ratio of time to recession of the UH to the time to peak

A UH group can also have a set of Initial Abstraction (IA) parameters associated with it. These determine how much rainfall is lost to interception and depression storage before any excess rainfall is generated and transformed into RDII flow by a unit hydrograph. The IA parameters consist of:

- a **maximum possible depth** of IA (inches or mm).
- a **recovery rate** (inches/day or mm/day) at which stored IA is depleted during dry periods.
- an **initial depth** of stored IA (inches or mm).

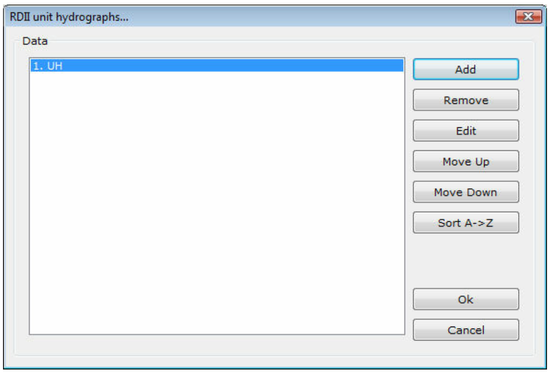


In the above example, the time to the peak of the UH is 1 hour, the base time is Tb=6 hours and hence K=(Tb-T)/T=5. The peak flow is 50 units.

An alternative to using unit hydrographs to define RDII flow is to create an external RDII interface file, which contains RDII time series data.

To manage UH RDIIs:

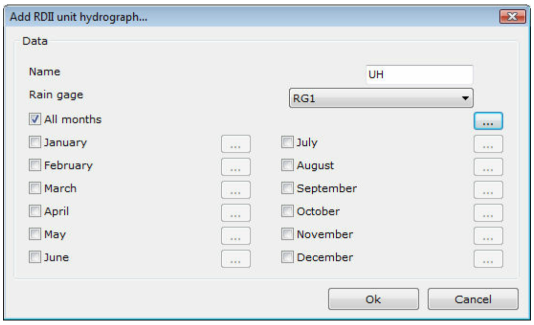
1. Select **Hydrology > RDII hydrographs** from the **Data** menu. The following form appears:



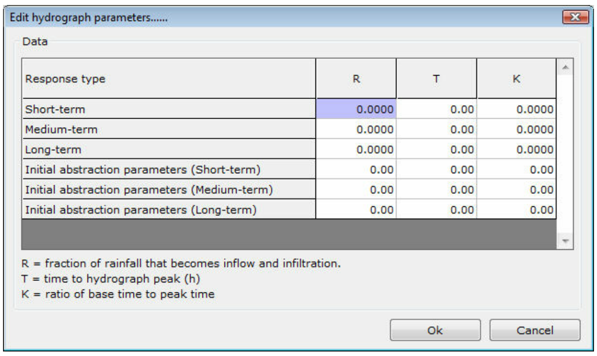
2. Make the necessary modifications.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To add a new UH RDII:

1. Press **Add**. The following form appears:



2. Make the appropriate selections as described below.
3. Select **All months** if you wish to use common values for all months. If you wish to use separate values for each month, check the corresponding months. If there is no data for a specified month, the common values are used.
4. Press the corresponding button with the ellipses "..." to enter data for a specified month (or common values). The following form appears:



Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.
5. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To edit an existing UH RDII:

1. Select the UH RDII from the list on the left.
2. Press **Add**. The data form appears.
3. Make the appropriate selections as described below.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To delete an existing UH RDII:

1. Select the UH RDII from the list on the left.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The UH RDII is deleted from the list.

To move an existing UH RDII upwards in the list:

1. Select the UH RDII from the list on the left.
2. Press **Move Up**.
3. The UH RDII is moved one place upwards.

To move an existing UH RDII downwards in the list:

1. Select the UH RDII from the list on the left.
2. Press **Move Down**.
3. The UH RDII is moved one place downwards.

To sort the UH RDII list:

1. Press **Sort A->Z**.
2. The list is sorted alphabetically.

UH properties

- **Name**: enter a user-assigned UH name. This name cannot be null or used for another UH in the project.
- **Raingage**: select the raingage that corresponds to the specified UH from the drop-down list.

UH parameters

- **R**: the fraction of rainfall volume that enters the sewer system
- **T**: the time from the onset of rainfall to the peak of the UH (h)
- **K**: the ratio of time to recession of the UH to the time to peak
- **maximum possible depth** of IA (inches or mm).
- **recovery rate** (inches/day or mm/day) at which stored IA is depleted during dry periods.
- **initial depth** of stored IA (inches or mm).

6.7 Quality

6.7.1 Pollutants

The program can simulate the generation, inflow and transport of any number of user-defined pollutants. Required information for each pollutant includes:

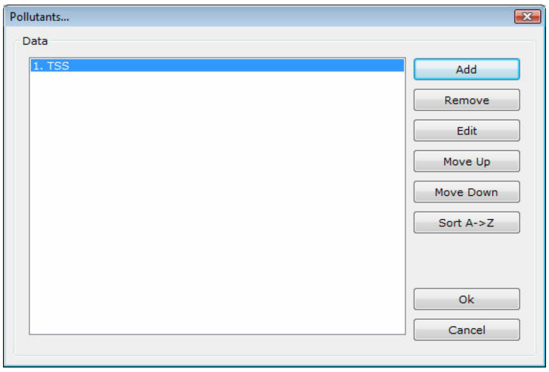
- pollutant name
- concentration units (i.e., milligrams/liter, micrograms/liter, or counts/liter)
- concentration in rainfall
- concentration in groundwater
- concentration in direct infiltration/inflow
- first-order decay coefficient

Co-pollutants can also be defined. For example, pollutant X can have a co-pollutant Y, meaning that the runoff concentration of X will have some fixed fraction of the runoff concentration of Y added to it.

Pollutant buildup and washoff from subcatchment areas are determined by the land uses assigned to those areas. Input loadings of pollutants to the drainage system can also originate from external time series inflows as well as from dry weather inflows.

To manage aquifers:

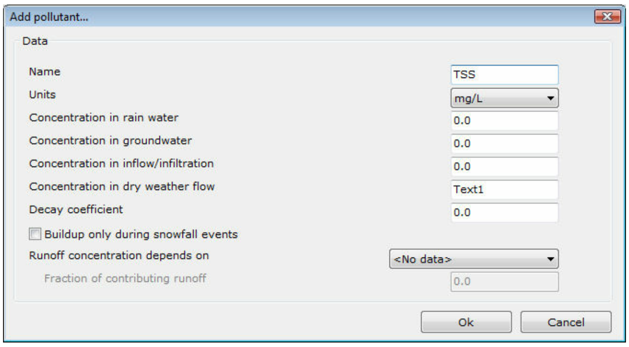
1. Select **Hydrology > Pollutants** from the **Data** menu. The following form appears:



2. Make the necessary modifications.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To add a new pollutant:

1. Press **Add**. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To edit an existing pollutant:

1. Select the pollutant from the list on the left.
2. Press **Add**. The data form appears.
3. Make the appropriate selections as described below.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To delete an existing pollutant:

1. Select the pollutant from the list on the left.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The pollutant is deleted from the list.

To move an existing pollutant upwards in the list:

1. Select the pollutant from the list on the left.
2. Press **Move Up**.
3. The pollutant is moved one place upwards.

To move an existing pollutant downwards in the list:

1. Select the pollutant from the list on the left.
2. Press **Move Down**.
3. The pollutant is moved one place downwards.

To sort the pollutant list:

1. Press **Sort A->Z**.
2. The list is sorted alphabetically.

pollutant properties

- **Name**: enter a user-assigned pollutant name. This name cannot be null or used for another pollutant in the project.
- **Units:** enter the concentration units (mg/L, ug/L, or #/L (counts/L)) in which the pollutant concentration is expressed.

- **Concentration in rain water:** enter the concentration of the pollutant in rain water.
- **Concentration in groundwater:** enter the concentration of the pollutant in ground water.
- **Concentration in inflow/infiltration:** enter the concentration of the pollutant in any Infiltration/Inflow.
- **Decay coefficient (1/days):** enter the first-order decay coefficient of the pollutant.
- **Buildup only during snowfall events**: check this option if pollutant buildup occurs only when snowfall occurs.
- **Runoff concentration depends on**: enter (optionally) the name of another pollutant whose runoff concentration contributes to the runoff concentration of the current pollutant.
- **Fraction of contributing runoff**: enter the fraction of the co-pollutant's runoff concentration that contributes to the runoff concentration of the current pollutant. An example of a co-pollutant relationship would be where the runoff concentration of a particular heavy metal is some fixed fraction of the runoff concentration of suspended solids. In this case suspended solids would be declared as the co-pollutant for the heavy metal.

6.7.2 Land uses

Land Uses are categories of development activities or land surface characteristics assigned to subcatchments. Examples of land use activities are residential, commercial, industrial, and undeveloped. Land surface characteristics might include rooftops, lawns, paved roads, undisturbed soils, etc. Land uses are used solely to account for spatial variation in pollutant buildup and washoff rates within subcatchments.

The program has many options for defining land uses and assigning them to subcatchment areas. One approach is to assign a mix of land uses for each subcatchment, which results in all land uses within the subcatchment having the same pervious and impervious characteristics. Another approach is to create subcatchments that have a single land use classification along with a distinct set of pervious and impervious characteristics that reflects the classification.

The following processes can be defined for each land use category:

- pollutant buildup
- pollutant washoff
- street cleaning

Pollutant buildup

Pollutant buildup that accumulates within a land use category is described (or "normalized") by either a mass per unit of subcatchment area or per unit of curb length. Mass is expressed in pounds for US units and kilograms for metric units. The amount of buildup is a function of the number of preceding dry weather days and can be computed using one of the following functions:

Power Function: Pollutant buildup (B) accumulates proportionally to time (t) raised to some power, until a maximum limit is achieved:

$$B = C_2 t^{C_3} \leq C_1$$

where C_1 is the maximum buildup possible (mass per unit of area or curb length), C_2 is the buildup rate constant and C_3 is the time exponent.

Exponential Function: Buildup follows an exponential growth curve that approaches a maximum limit asymptotically:

$$B = C_1 \cdot \left(1 - e^{-C_2 t}\right)$$

where C_1 is the maximum buildup possible (mass per unit of area or curb length) and C_2 is the buildup rate constant (1/days).

Saturation Function: Buildup begins at a linear rate that continuously declines with time until a saturation value is reached:

$$B = \frac{C_1 t}{C_2 + t}$$

where C_1 is the maximum buildup possible (mass per unit of area or curb length) and C_2 is the half-saturation constant (days to reach half of the maximum buildup).

Pollutant Washoff

Pollutant washoff from a given land use category occurs during wet weather periods and can be described in one of the following ways:

Exponential Washoff: The washoff load (W) in units of mass per hour is proportional to the product of runoff raised to some power and to the amount of buildup remaining:

$$W = C_1 q^{C_2} B$$

where C_1 is the washoff coefficient, C_2 is the washoff exponent, q is the runoff rate per unit area (inches/hour or mm/hour), and B is the pollutant buildup in mass (lbs or kg) per unit area or curb length. Washoff mass units are the same as used to express the pollutant's concentration (milligrams, micrograms, or counts).

Rating Curve Washoff: The rate of washoff W in mass per second is proportional to the runoff rate raised to some power:

$$W = C_1 Q^{C_2}$$

where C_1 is the washoff coefficient, C_2 is the washoff exponent, and Q is the runoff rate in user-defined flow units.

Event Mean Concentration: This is a special case of Rating Curve Washoff where the exponent C_2 is 1.0 and the coefficient C_1 represents the washoff pollutant concentration in mass per liter (Note:the conversion between user-defined flow units used for runoff and liters is handled internally by the program).

Note that in each case buildup is continuously depleted as washoff proceeds, and washoff ceases when there is no more buildup available.

Washoff loads for a given pollutant and land use category can be reduced by a fixed percentage by specifying a BMP Removal Efficiency that reflects the effectiveness of any BMP controls associated with the land use. It is also possible to use the Event Mean Concentration option by itself, without having to model any pollutant buildup at all.

Street Sweeping

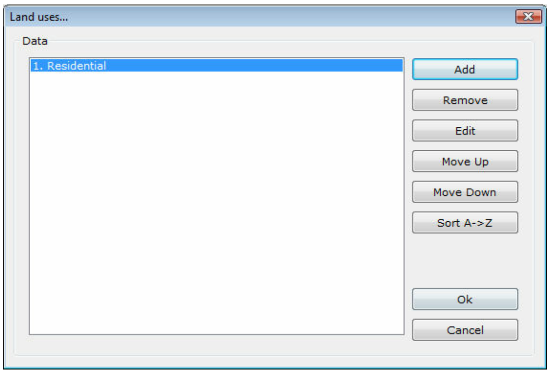
Street sweeping can be used on each land use category to periodically reduce the accumulated buildup of specific pollutants. The parameters that describe street sweeping include:

- days between sweeping
- days since the last sweeping at the start of the simulation
- the fraction of buildup of all pollutants that is available for removal by sweeping
- the fraction of available buildup for each pollutant removed by sweeping

Note that these parameters can be different for each land use, and the last parameter can vary also with pollutant.

To manage land uses:

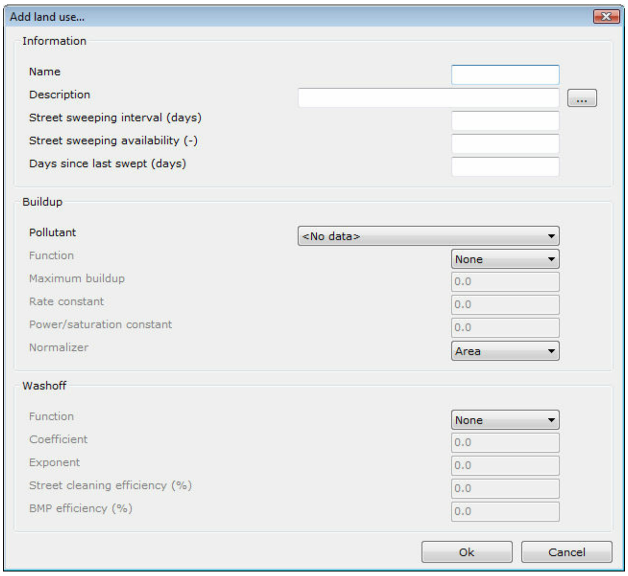
1. Select **Hydrology > Land uses** from the **Data** menu. The following form appears:



2. Make the necessary modifications.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To add a new land use:

1. Press **Add**. The following form appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To edit an existing land use:

1. Select the land use from the list on the left.
2. Press **Add**. The data form appears.
3. Make the appropriate selections as described below.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To delete an existing land use:

1. Select the land use from the list on the left.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The land use is deleted from the list.

To move an existing land use upwards in the list:

1. Select the land use from the list on the left.
2. Press **Move Up**.
3. The land use is moved one place upwards.

To move an existing land use downwards in the list:

1. Select the land use from the list on the left.
2. Press **Move Down**.
3. The land use is moved one place downwards.

To sort the land use list:

1. Press **Sort A->Z**.
2. The list is sorted alphabetically.

Information

- **Name**: enter a user-assigned land use name. This name cannot be null or used for another land use in the project.
- **Description**: enter an optional comment or description of the land use. Press the button with the ellipses "..." to edit multi-line text.
- **Street sweeping interval (days):** Days between street sweeping within the land use.
- **Street sweeping availability:** Fraction of the buildup of all pollutants that is available for removal by sweeping.
- **Days since last swept (days):** Number of days since last swept at the start of the simulation.

Buildup

- **Pollutant:** select the pollutant whose buildup properties are being edited.
- **Function:** select the type of buildup function to use for the pollutant. The choices are **None, Power, Exponential** and **Saturation**.
- **Maximum buildup (lbs or kg):** The maximum buildup of the pollutant that can occur per unit of the normalizer variable. This is the same as the C_1 coefficient used in the buildup formulas.
- **Rate constant:** The time constant that governs the rate of pollutant buildup. This is the C_2 coefficient in the Power and Exponential buildup formulas. For Power buildup its units are mass / days raised to a power, while for Exponential buildup its units are 1/days.
- **Power/saturation constant:** The exponent C_3 used in the Power buildup formula, or the half-saturation constant C_2 used in the Saturation buildup formula. For the latter case, its units are days
- **Normalizer:** The variable to which buildup is normalized on a per unit basis. The choices are either land area (in acres or hectares) or curb length. Any units of measure can be used for curb length,as long as they remain the same for all subcatchments in the project.

Washoff

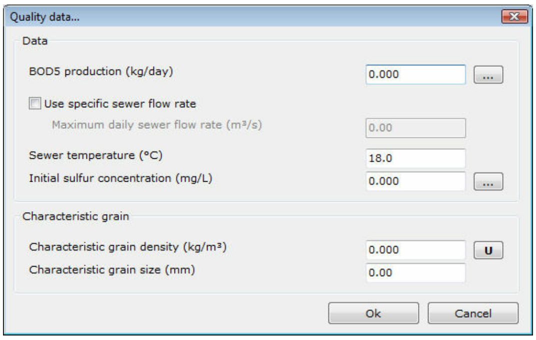
- **Function:** select the choice of washoff function to use for the pollutant. The choices are: **None, Exponential, Rating curve** and **Event mean concentration.**
- **Coefficient:** enter the value of C_1 in the exponential and rating curve formulas, or the event-mean concentration.
- **Exponent:** enter the exponent used in the exponential and rating curve washoff formulas.
- **Street cleaning efficiency (%):** The street cleaning removal efficiency (percent) for the pollutant. It represents the fraction of the amount that is available for removal on the land use as a whole which is actually removed.
- **BMP efficiency:** Removal efficiency (percent) associated with any Best Management Practice that might have been implemented. The washoff load computed at each time step is simply reduced by this amount.

6.7.3 Quality data

With this option, you can enter water quality data. These are optional. The water quality data are the same throughout the network.

To enter water quality data:

1. Select **Quality data** from the **Data** menu. The following form will appear:



2. Enter the data as follows:

- **BOD_5 production:** enter the BOD_5 production in kg/day. Click the button with the ellipses (...) on the right of the text box to invoke the corresponding database .
- Check **Use specific sewer flow rate** if the sewer flow rate is known. This is usually valid in mixed-type networks, in which case the flow rate is entered in the corresponding text box.
- **Sewer temperature (deg. F or deg. C)**: enter the mean sewer temperature in degrees Celsius.
- **Initial sulfur concentration (mg/L)**: enter the initial sulfur concentration in mg/L. Click the button with the ellipses (...) on the right of the text box to invoke the corresponding database.
- **Characteristic grain density (lb/ft^3 or kg/m^3):** enter the characteristic grain density in kg/m^3. This is used for the calculation of the cleaning velocity and the full cleaning velocity.
- **Characteristic grain size (in or mm)**: enter the characteristic grain size in mm. This is used for the calculation of the cleaning velocity and the full cleaning velocity.

3. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.8 Curves

6.8.1 Management

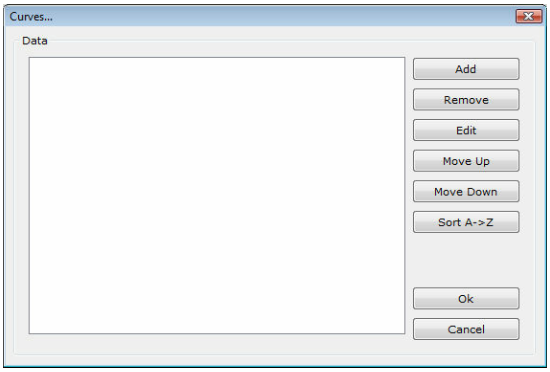
Curves are objects that describe the relation between two quantities. The following curve types are available:

- Storage curves
- Diversion curves

- Tidal curves
- Shape curves
- Pump curves
- Rating curves
- Control curves

To manage curves:

1. Select **Curves** from the **Data** menu.
2. Select the appropriate curve type from the submenu. The following form appears:

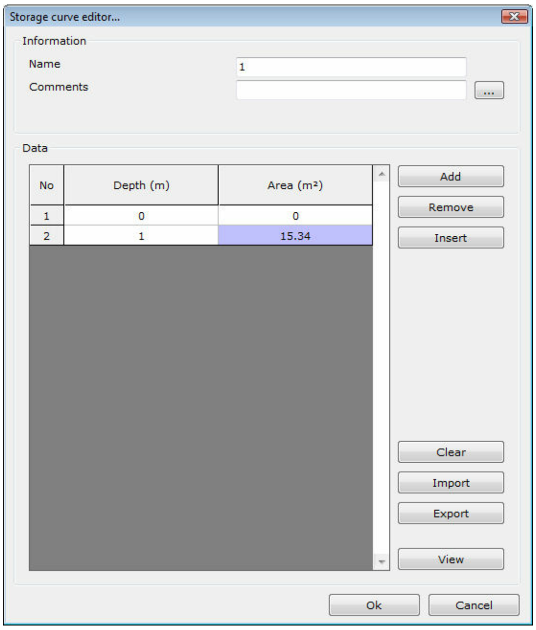


3. Make the necessary modifications.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.8.2 Add

To add a new curve:

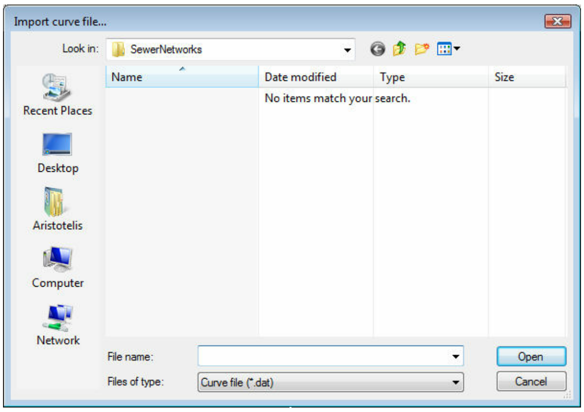
1. Press **Add**. The data form that corresponds to the specified curve type appears.



2. Enter a user-assigned **name**. This name cannot be null or used for another curve in the project.
3. For pump curves, select the pump curve type from the drop-down list.
4. Optionally, add some **comments** to the curve. Press the ellipsis button to edit multiline text.
5. Press **add** to add a line at the end of the data list.
6. Press **remove** to remove the current line of the data list
7. Press **insert** to insert a line above the current line of the data list.
8. Type the data in the data list.
9. If you wish to clear the data list, press **clear**.
10. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To import a curve from an external file:

1. Press **Import**. The file selection dialog box appears:

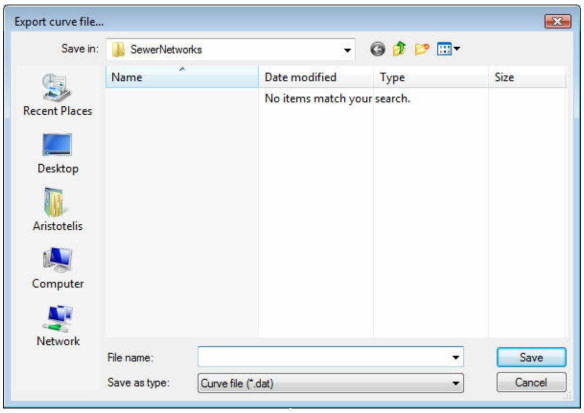


2. Select the path of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Curve file" with the extension .dat.
4. Select the file by clicking on it.
5. Select **Open** to open and analyze the file.

NOTE: A warning message will be displayed if the curve type of the imported file does not match the current curve type.

To export a curve to an external file:

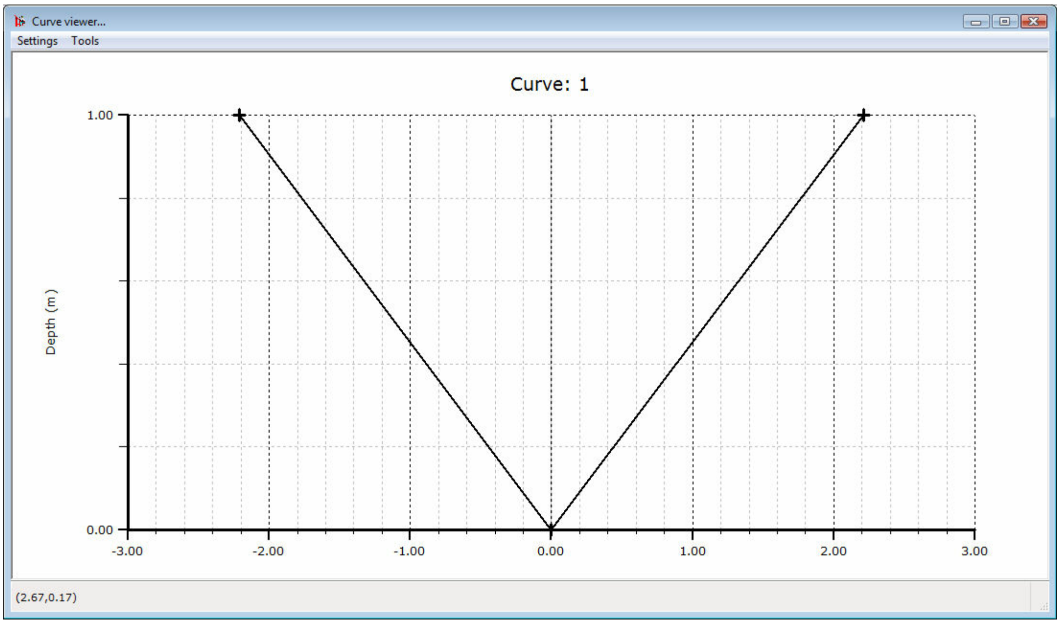
1. Press **Export.** The file selection dialog box appears:



6. Select the path of the file.
7. Type the filename in the **File name** text box.
8. Select **Save** to save the file with the selected filename and path. Select **Cancel** to cancel the operation.

To view a curve:

1. Press **View.** The curve is drawn in a separate window:



6.8.3 Delete

To delete an existing curve:

1. Select the curve from the list on the left.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The curve is deleted from the list.

6.8.4 Edit

To edit an existing curve:

1. Select the curve from the list on the left.
2. Press **Edit**. The data form appears.
3. Make the appropriate selections as described in the add curve topic.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.8.5 Move

To move an existing curve upwards in the list:

1. Select the curve from the list on the left.
2. Press **Move Up**.
3. The curve is moved one place upwards.

To move an existing curve downwards in the list:

1. Select the curve from the list on the left.
2. Press **Move Down**.
3. The curve is moved one place downwards.

6.8.6 Sort

To sort the curve list:

1. Press **Sort A->Z**.
2. The list is sorted alphabetically.

6.9 Time series

6.9.1 Management

Time Series objects are used to describe how certain object properties vary with time. Time series can be used to describe:

- temperature data
- evaporation data
- rainfall data
- water stage at outfall nodes
- external inflow hydrographs at drainage system nodes
- external inflow pollutographs at drainage system nodes
- control settings for pumps and flow regulators

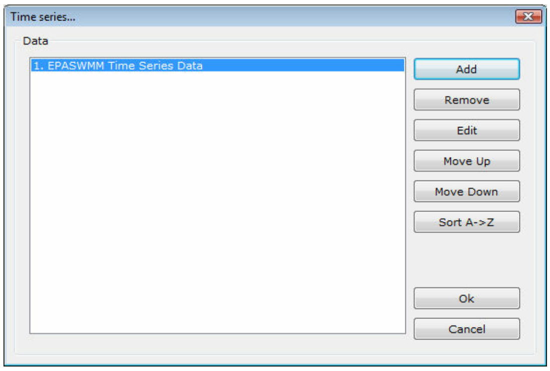
Each time series must be given a unique name and can be assigned any number of time-value data pairs. Time can be specified either as hours from the start of a simulation or as an absolute date and time-of-day.

For rainfall time series, it is only necessary to enter periods with non-zero rainfall amounts. The program interprets the rainfall value as a constant value lasting over the recording interval specified for the rain gage that utilizes the time series. For all other types of time series, the program uses interpolation to estimate values at times that fall in between the recorded values.

For times that fall outside the range of the time series, the program will use a value of 0 for rainfall and external inflow time series, and either the first or last series value for temperature, evaporation, and water stage time series.

To manage time series:

1. Select **Time series** from the **Data** menu. The following form appears:



2. Make the necessary modifications.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Time series files

Time series files are external text files that contain data for time series objects. Examples of time series data include rainfall, evaporation, inflows to nodes of the drainage system, and water stage at outfall boundary nodes. Normally these data are entered and edited through the time series editor. However there is an option to import data from an external file into the editor. Creating and editing this file can be done outside of the program, using text editors or spreadsheet programs.

The format of a time series file consists of two lines of descriptive text followed by the actual time series data, with one time series value per line. Typically, the first text line identifies the time series and the second line includes a detailed description of the time series. Time series values can either be in date / time / value format or in time / value format, where each entry is separated by one or more spaces or tab characters. For the date / time / value format, dates are entered as month/day/year (e.g., 7/21/2004) and times as 24-hour military time (e.g., 8:30 pm is 20:30). After the first date, additional dates need only be entered whenever a new day occurs. For the time / value format, time can either be decimal hours or military time since the start of a simulation (e.g., 2 days, 4 hours and 20 minutes can be entered as either 52.333 or 52:20). An example of a time series file is shown below:

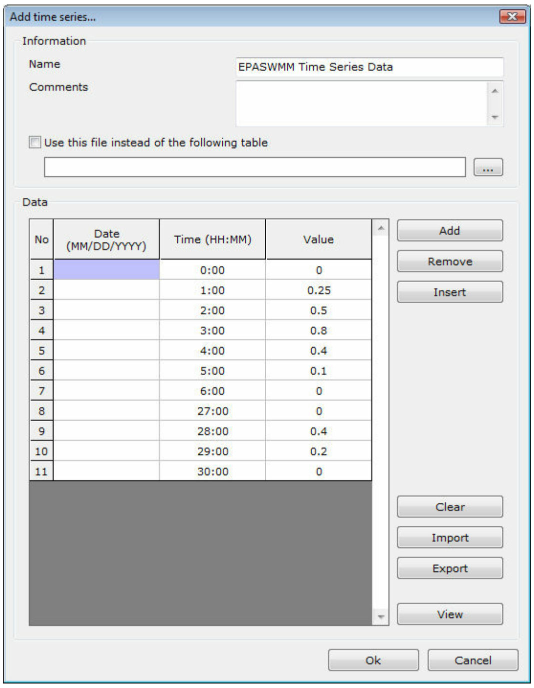
```
EPASWMM Time Series Data
<optional description goes here>
12/31/2005  00:00   0.00
            06:00   3.22
            12:00   5.78
```

```
              18:00   2.11
01/01/2006    00:00   1.08
              06:00   0.03
              12:00   0.00
```

6.9.2 Add

To add a new time series:

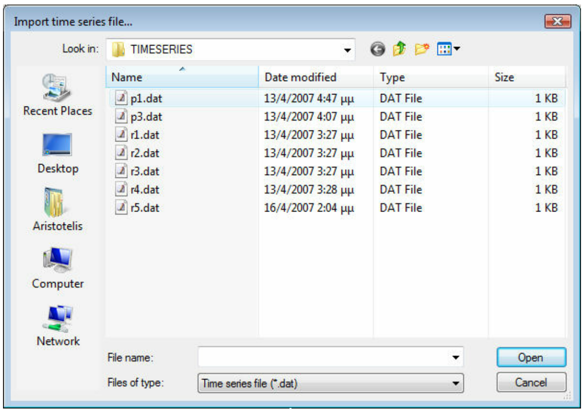
1. Press **Add**. The data form appears:



2. Enter a user-assigned **name**. This name cannot be null or used for another time series in the project.
3. Optionally, add some **comments** to the curve. Press the ellipsis button to edit multiline text.
4. If the data are located in an external file, check the **use this file instead of the following table**. Then click the ellipsis button to select the external time-series file.
5. Press **add** to add a line at the end of the data list.
6. Press **remove** to remove the current line of the data list
7. Press **insert** to insert a line above the current line of the data list.
8. Type the data in the data list.
9. If you wish to clear the data list, press **clear**.
10. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To import a time series from an external file:

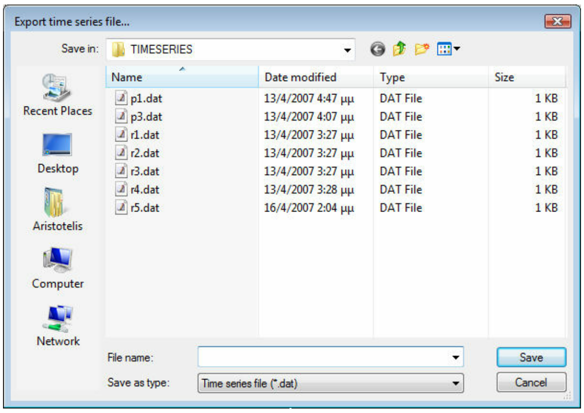
1. Press **Import**. The file selection dialog box appears:



2. Select the path of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Time series file" with the extension .dat.
4. Select the file by clicking on it.
5. Select **Open** to open and analyze the file.

To export a time series to an external file:

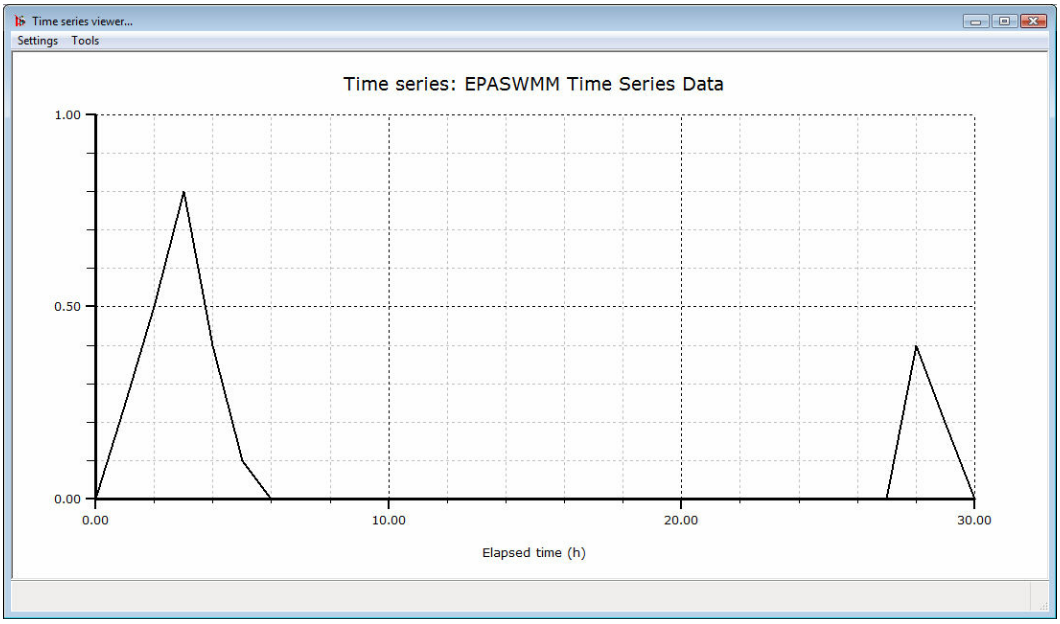
1. Press **Export.** The file selection dialog box appears:



6. Select the path of the file.
7. Type the filename in the **File name** text box.
8. Select **Save** to save the file with the selected filename and path. Select **Cancel** to cancel the operation.

To view a time series:

1. Press **View.** The time series is drawn in a separate window:



6.9.3 Delete

To delete an existing time series:

1. Select the time series from the list on the left.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The time series is deleted from the list.

6.9.4 Edit

To edit an existing time series:

1. Select the time series from the list on the left.
2. Press **Edit**. The data form appears.
3. Make the appropriate selections as described in the add time series topic.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.9.5 Move

To move an existing time series upwards in the list:

1. Select the time series from the list on the left.
2. Press **Move Up**.
3. The time series is moved one place upwards.

To move an existing time series downwards in the list:

1. Select the time series from the list on the left.
2. Press **Move Down**.
3. The time series is moved one place downwards.

6.9.6 Sort

To sort the time series list:

1. Press **Sort A->Z**.
2. The list is sorted alphabetically.

6.10 Time patterns

6.10.1 Management

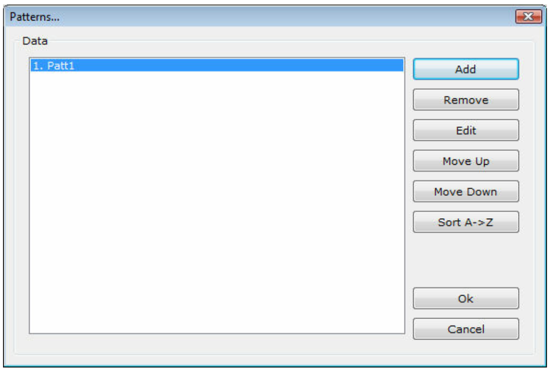
Time Patterns allow external Dry Weather Flow (DWF) to vary in a periodic fashion. They consist of a set of adjustment factors applied as multipliers to a baseline DWF flow rate or pollutant concentration. The different types of time patterns include:

- **monthly**: one multiplier for each month of the year
- **daily**: one multiplier for each day of the week
- **hourly**: one multiplier for each hour from 12 AM to 11 PM
- **weekend**: hourly multipliers for weekend days

Each Time Pattern must have a unique name and there is no limit on the number of patterns that can be created. Each dry weather inflow (either flow or quality) can have up to four patterns associated with it, one for each type listed above.

To manage time patterns:

1. Select **Time patterns** from the **Data** menu. The following form appears:

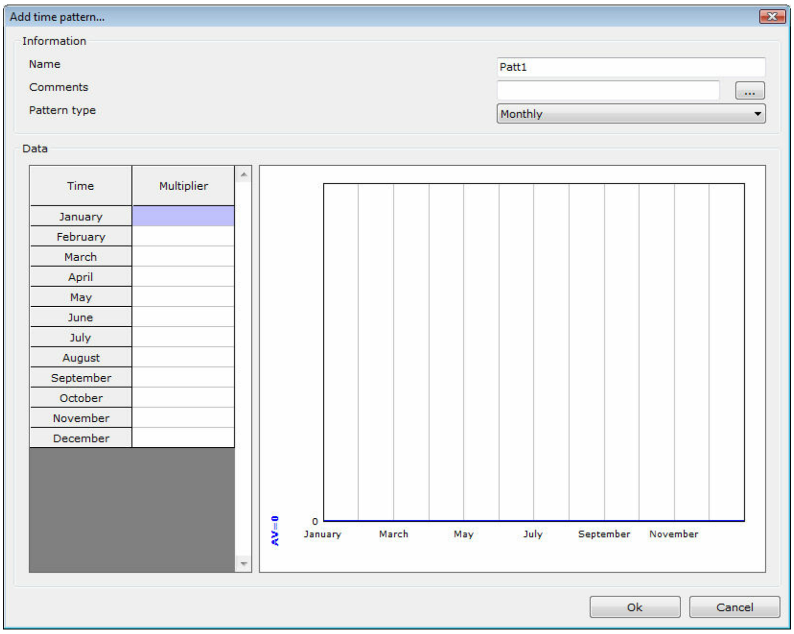


2. Make the necessary modifications.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.10.2 Add

To add a new time pattern:

1. Press **Add**. The data form appears:



2. Enter a user-assigned **name**. This name cannot be null or used for another time series in the project.
3. Optionally, add some **comments** to the curve. Press the ellipsis button to edit multiline text.
4. Select the **pattern type** from the drop-down list.
5. Type the data in the data list. The time pattern is drawn in the picture box in real time.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

NOTE: In order to maintain an average dry weather flow or pollutant concentration at its specified value, the multipliers for a pattern should average to 1.0. The average value is displayed graphically as a horizontal blue line. It is also displayed analytically in the sketch as "AV".

6.10.3 Delete

To delete an existing time pattern:

1. Select the time pattern from the list on the left.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The time pattern is deleted from the list.

6.10.4 Edit

To edit an existing time pattern:

1. Select the time pattern from the list on the left.
2. Press **Edit**. The data form appears.
3. Make the appropriate selections as described in the add time pattern topic.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the

dialog box without saving any changes.

6.10.5 Move

To move an existing time pattern upwards in the list:

1. Select the time pattern from the list on the left.
2. Press **Move Up**.
3. The time pattern is moved one place upwards.

To move an existing time pattern downwards in the list:

1. Select the time pattern from the list on the left.
2. Press **Move Down**.
3. The time pattern is moved one place downwards.

6.10.6 Sort

To sort the time pattern list:

1. Press **Sort A->Z**.
2. The list is sorted alphabetically.

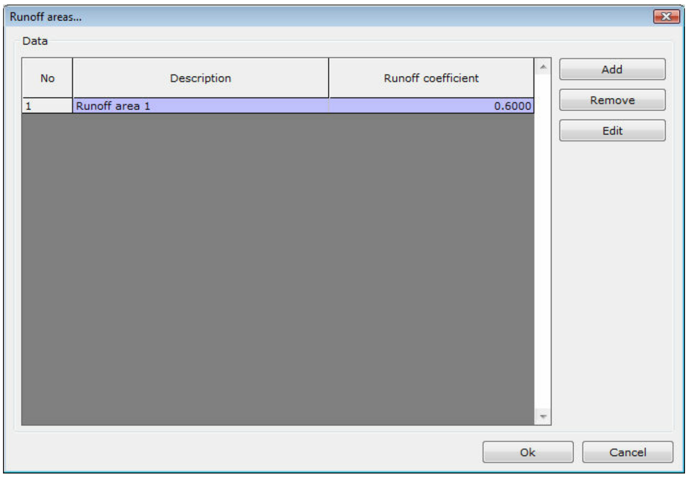
6.11 Runoff areas

6.11.1 Management

With this option, you can enter data regarding runoff areas. Each area can have different runoff coefficient. For example, a storm network can include inflow from road surface and an external basin.

To enter data regarding runoff areas:

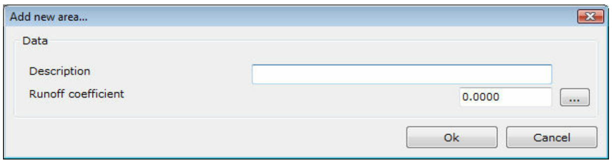
1. Select **Runoff areas** from the **Data** menu.
2. Enter the data as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.



6.11.2 Add

To add a new area:

1. Select **Add**. The following form appears:



2. Enter a unique name for the area.
3. Enter the **runoff coefficient**. Optionally, select the button with the ellipses (...) to invoke the corresponding database.
4. Select **Ok** to add the new area to the list and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.11.3 Delete

To delete an existing runoff area:

1. Select the runoff area from the list.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The runoff area is deleted from the list.

NOTE: If an existing area is deleted, all corresponding entries in the inflow forms will also be deleted.

6.11.4 Edit

To edit an existing area:

1. Select the area from the list.
2. Select **Edit**. The data form appears.
3. Make the appropriate changes as described in the add area topic.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.12 Sewer flow (population)

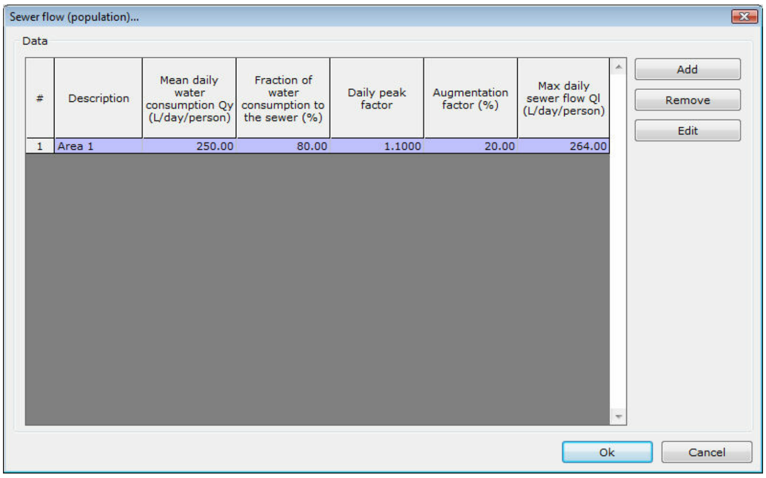
6.12.1 Management

For sewer networks, the program can compute the inflows using the actual number of people being serviced at each conduit. For example, a small village has a population of 800. Suppose that 50 people live in a block and only one pipe services their apartments. The program can compute the flow rate of sewage into that pipe, as the percentage (50/800) of the total flow rate. You can specify different water consumptions that lead to different sewer design flow rates, by adding areas corresponding to residents, tourists, factories or special consumers.

To enter data regarding sewer flow calculated using population data:

1. Select **Sewer Flow (Population)** from the **Data** menu.

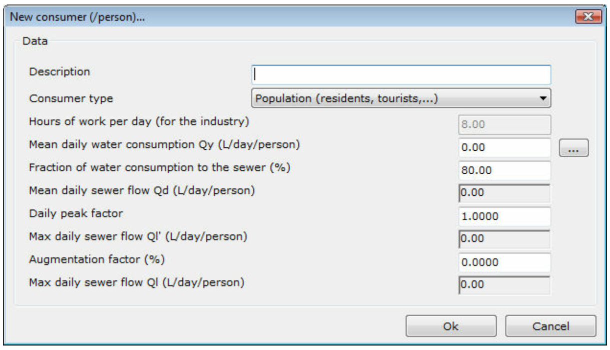
2. Enter the data as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.



6.12.2 Add

To add a new sewer flow (population) area:

1. Select **Add**. The following form appears:



2. Enter a unique name for the consumer.
3. Select the consumer type from the drop down list. Depending on your choice, different data is required.
4. If you selected **industry**, then the number of hours that the factory is in operation is required.
5. Enter the **mean daily water consumption**. Click the button with the ellipses (...) on the right of the text box to invoke the corresponding database.
6. Enter the **fraction of water consumption** that ends in the sewer. Usually this amount is around 80%.
7. Enter the **daily peak factor** which will be used to convert the mean daily sewer flow to maximum daily sewer flow. If you have selected industry, then the peak factor is equal to 24/(hours of operation), while for special consumers the peak factor is always equal to unity.
8. Optionally you can enter an **augmentation factor** to increase the maximum daily sewer flow.

9. Select **Ok** to add the new area to the list and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.12.3 Delete

To delete an existing sewer flow (population) data:

1. Select the data from the list.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The runoff area is deleted from the list.

NOTE: If an existing area is deleted, all corresponding entries in the inflow forms will also be deleted.

6.12.4 Edit

To edit existing sewer flow (population) data:

1. Select the data from the list.
2. Select **Edit**. The data form appears.
3. Make the appropriate changes as described in the add sewer flow (population) topic.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

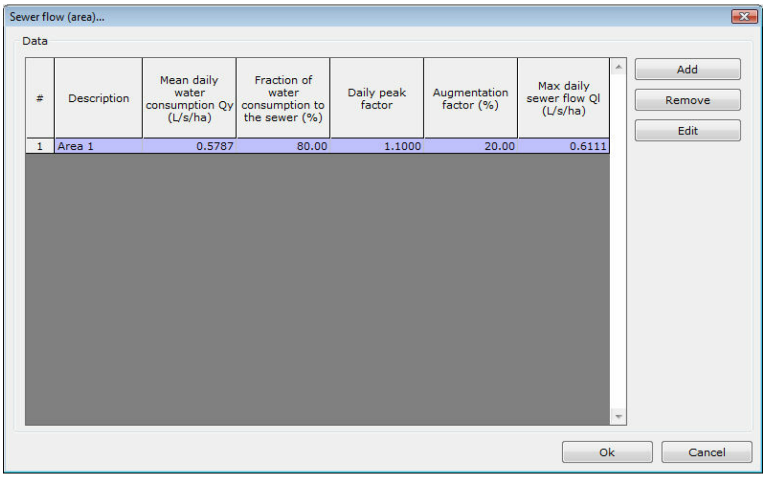
6.13 Sewer flow (area)

6.13.1 Management

For sewer networks, the program can compute the inflows using the actual size of the areas being serviced at each conduit. For example, a small village has a total area of 50 hectares. Suppose that one block has an area of 2 hectares and only one pipe services the apartments built in this block. The program can compute the flow rate of sewage into that pipe, as the percentage (2/50) of the total flow rate. You can specify different water consumptions that lead to different sewer design flow rates, by adding areas corresponding with varying characteristics.

To enter data regarding sewer flow calculated using areas:

1. Select **Sewer Flow (Area)** from the **Data** menu.
2. Enter the data as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

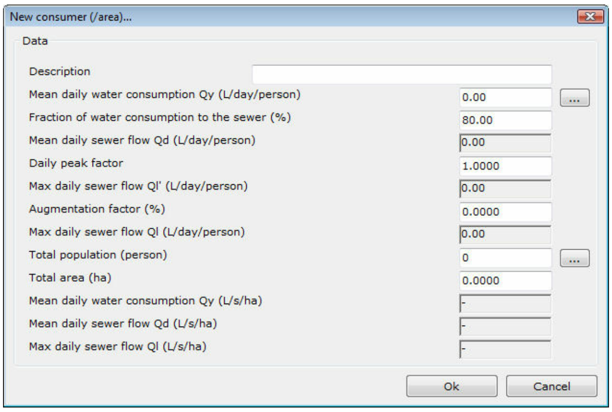


6.13.2 Add

With this option, you can estimate the maximum daily flow rate per each serviced area. The total area of each entity can be broken down to subareas, which are added to different nodes.

To add a new area entity:

1. Click on Add.
2. Enter the data as follows:



- Enter the description of the area.
- Enter the **mean daily water consumption**. Click the button with the ellipses (...) on the right of the text box to invoke the corresponding database.
- Enter the **fraction of water consumption** that ends in the sewer. Usually this amount is around 80%.
- Enter the **daily peak factor**, which is used to convert mean daily flow rates to maximum daily flow rates.
- Optionally you can enter an **augmentation factor** to increase the maximum daily sewer flow.
- Enter the total **population.** If this is unknown, you can use the estimation tool by selecting the corresponding ellipsis button.

- Enter the **total area** in units of area.

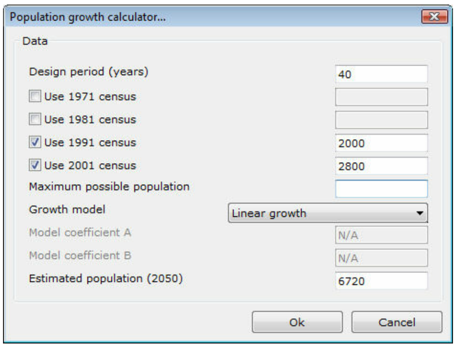
2. Select **Ok** to add this area to the serviced areas list and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Estimating the total population

If the total population is now known, you can estimate the design population using two or more census entries.

To estimate the design population:

1. Enter the data as follows:



- Enter the **design period** in years. When added to the current simulation year it will mark the target year for population growth calculations.
- Enter two or more **census** entries. Check the entries that will be taken into account.
- Enter the **maximum possible population** of the area. This value is required for some growth models. It is based on criteria regarding the area, the location, the usage of land etc.
- Select one of **Linear**, **Geometric**, **Decaying**, **S curve** as the method of growth. Some of the methods may not be available, subject to the number of available census entries. Some methods are calibrated by various coefficients. Although the program will provide the optimum values of these coefficients, you can modify them.

2. Select **Ok** to use the estimated value of the population and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.13.3 Delete

To delete an existing sewer flow (area) data:

1. Select the data from the list.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The runoff area is deleted from the list.

NOTE: If an existing area is deleted, all corresponding entries in the inflow forms will also be deleted.

6.13.4 Edit

To edit existing sewer flow (population) data:

1. Select the data from the list.
2. Select **Edit**. The data form appears.
3. Make the appropriate changes as described in the add sewer flow (area) topic.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

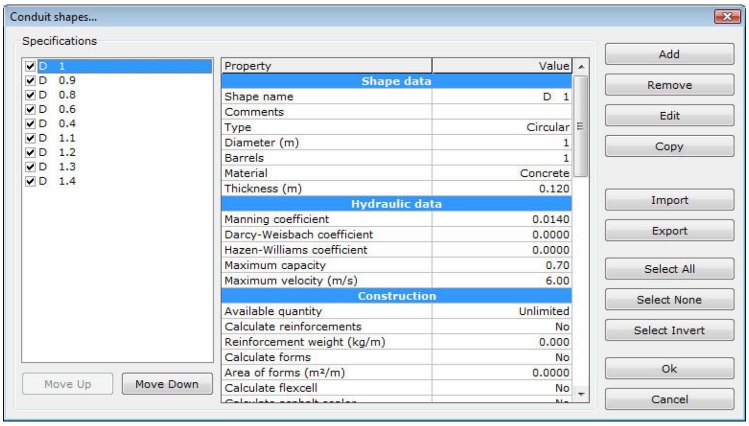
6.14 Conduit shapes

6.14.1 Management

With this option, you can add or modify conduit shape specifications. These are recommended in case you to calculate quantities.

To add or modify conduit shape specifications:

1. Select **Conduit shapes** from the **Data** menu. The following form will appear:



2. Make the necessary modifications.
3. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To change the order of pipe specifications:

1. Select an entry from the list.
2. Select **Up** to move the selected entry up by one place. This option is not available for the first entry in the list.
3. Select **Down** to move the selected entry down by one place. This option is not available for the last entry in the list.
4. Repeat the above steps as necessary.

NOTE: You can activate or deactivate a specific conduit shape specification using the corresponding check box.

The quick keys perform the following actions:

- **Select all**: activates all conduit shape specifications in the list.

- **Select none**: deactivates all conduit shape specifications in the list.
- **Invert**: inverts the status of all conduit shape specifications.

Conduit shape specifications can be imported from a file or exported to a file, as described later in this chapter.

The next sections describe the process of adding, modifying or deleting a conduit shape specification.

6.14.2 Add

With this option, you can add a conduit shape specification. There are two ways of adding a new specification. The first way is to add a new (empty) entry; the second is to create a copy of an existing entry and modify only some of its properties.

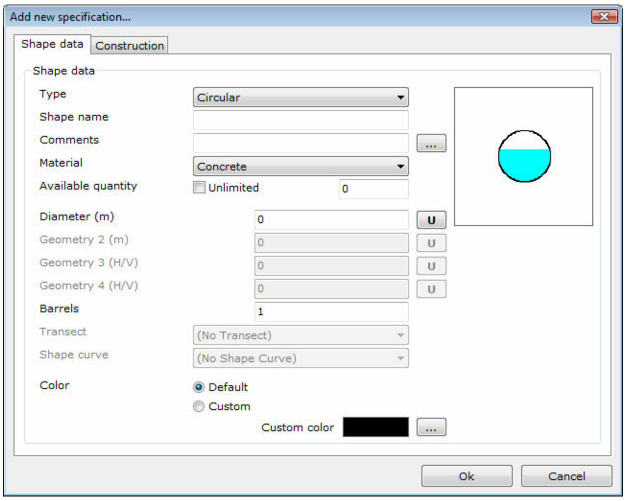
To add a new (empty) entry:

1. Select **Add**. A new entry is added to the list.
2. Enter the appropriate data as described below.
3. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To add a new entry by copying an existing entry:

1. Select the existing (base) entry from the list.
2. Select **Copy**. A copy of the existing entry is created.
3. Select **Edit** to modify the properties of the clone entry, as described below.
4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

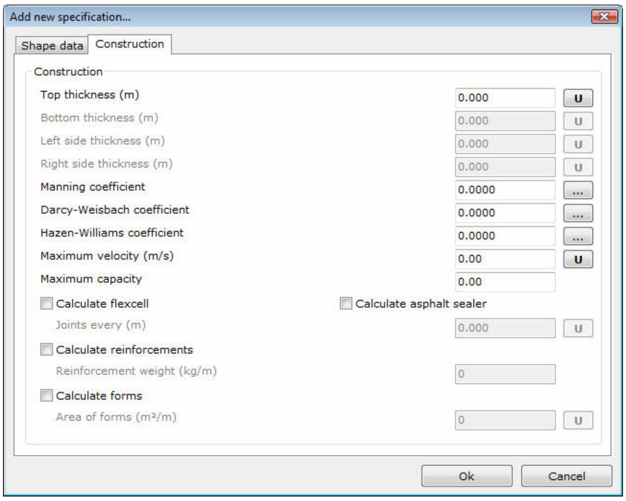
Shape Data Tab



- Select the **type** of the shape from the drop-down list.
- Enter a unique **shape name.**
- Optionally, you can enter some **comments.** Use the ellipsis button to edit multiline text.
- Select the **material** of the conduit from the drop-down list.

- Select whether the specified shape is available in **unlimited quantity.** If not, enter the available quantity in ft or m.
- Enter the **dimensions** of the shape. In case of circular shape, only the internal diameter is required.
- Enter the number of **barrels**. In case of twin conduits, this equals 2. Normally, the number of barrels is equal to 1.
- If the shape is **irregular**, select the corresponding transect from the drop-down list.
- If the shape is **custom,** select the corresponding curve from the drop-down list.
- Select the **color** that will be used to draw the conduit on the plane view. It can be either the default color or a custom color. This color will also be used when the plane view DXF drawing is created.

Construction Tab



- Enter the wall **thicknesses** of the conduit.
- Enter the Manning, Darcy - Weisbach, and Hazen - Williams friction coefficients. If these are not known, press the corresponding ellipsis button to invoke the corresponding database. The Darcy - Weisbach and Hazen - Williams friction coefficients are used only in circular shapes under pressure and only if the appropriate settings are made in the dynamic wave settings form.
- Enter the **maximum velocity** in ft/s or m/s of the conduit shape. This is not a restriction but a calculation check. If violated, the velocity is printed in red. Additional checks can be defined. However, the checks in the specifications are dominant.
- Enter the **maximum capacity** of the conduit shape. This is not a restriction but a calculation check. If violated, the capacity of the conduit is printed in red. Additional checks can be defined. However, the checks in the specifications are dominant.
- Check **Calculate flexcell, Calculate asphalt sealer** if these materials are used at the joints. In this case, enter the characteristic length that will define the frequency of the application of the aforementioned materials.
- Check **Calculate reinforcements** if you want to calculate the weight of the reinforcement using a mean weight per unit length value, which needs to be specified in the corresponding field.
- Check **Calculate forms** if you want to calculate the area of the forms using a mean area per unit length value, which needs to be specified in the corresponding field.

6.14.3 Delete

To delete an existing specification:

1. Select the specification from the list on the left.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The specification is deleted from the list.

6.14.4 Edit

To edit an existing specification:

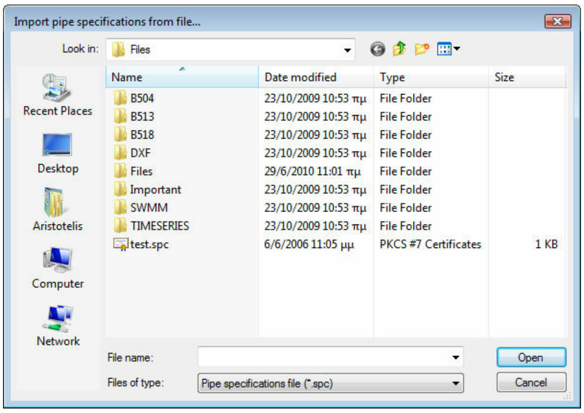
1. Select the specification from the list on the left.
2. Press **Edit**. The data form appears.
3. Make the appropriate selections as described in the add specification topic.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.14.5 Import

With this option, you can import specifications from an external file. Existing specifications are erased.

To import specifications from an external file:

1. Select **Import.** The file selection dialog box will appear:



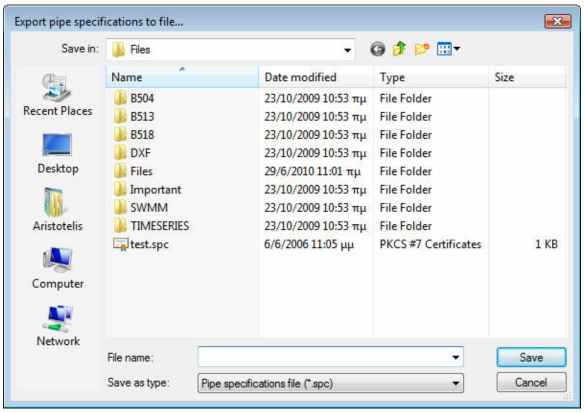
2. Select the path of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Pipe specification file" with the extension .spc.
4. Select the file by clicking on it.
5. Select **Open** to import the specifications.

6.14.6 Export

With this option, you can export specifications to an external file. These can be imported at a later time.

To export pipe specifications to an external file:

1. Select **Export**. The following form will appear:



2. Select the path of the file.
3. Type the filename in the **File name** text box.
4. Select **Save** to export pipe specifications with the selected filename and path. Select **Cancel** to cancel the operation.

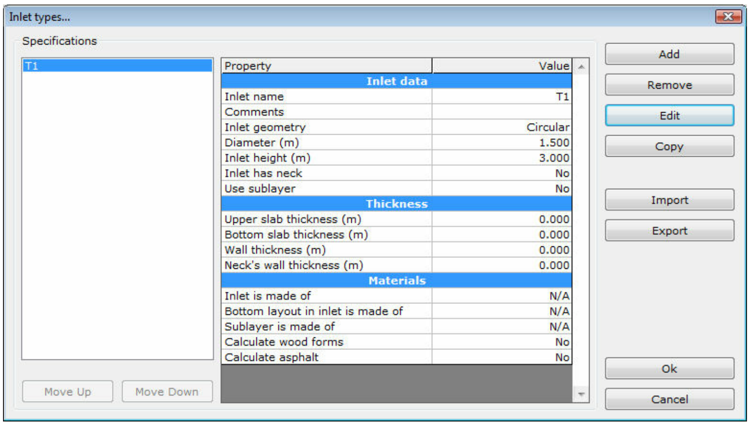
6.15 Manhole specifications

6.15.1 Management

With this option, you can add or modify inlet specifications. After defining inlet specifications, you can use them by selecting inlets from the **Profile** menu.

To add or modify inlet specifications:

1. Select **Inlet specifications** from the **Data** menu. The following form will appear:



2. Enter the inlet specifications.
3. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Inlet specifications can be imported from a file or exported to a file, as described later in this chapter.

The next sections describe the process of adding, modifying or deleting an inlet specification.

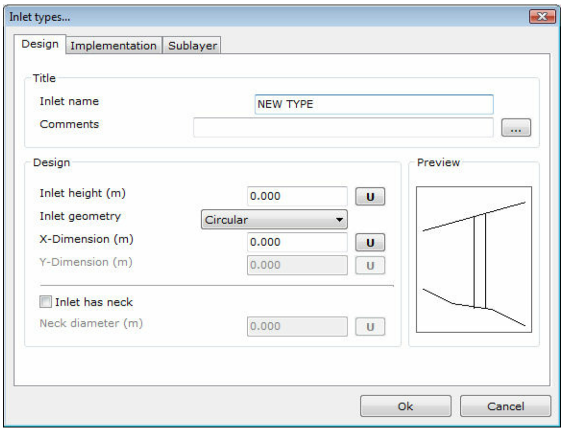
6.15.2 Add

With this option, you can add an inlet specification.

To add an inlet specification:

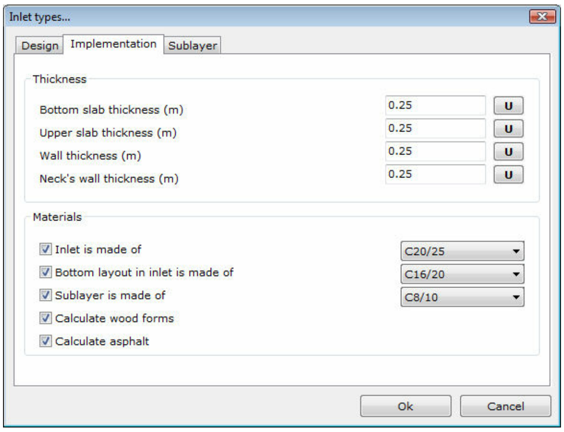
1. Select **Add**. A new entry is added to the list.
2. Enter the appropriate data as described below.
3. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Design Tab



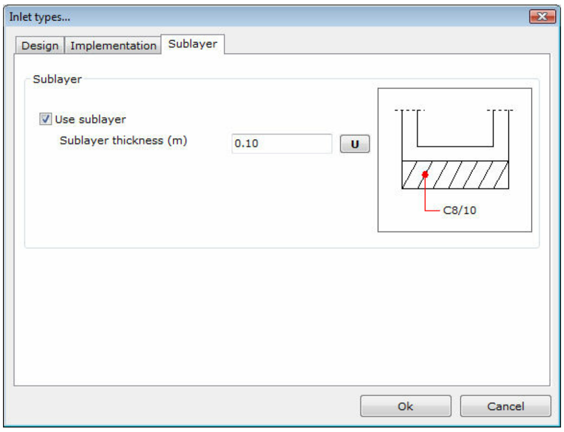
- Enter a unique **Name** for the specification.
- Optionally, you can add some **Comments.** Use the ellipsis button to edit multiline text.
- Enter the external **inlet height** in ft or m.
- Select tine **inlet geometry**. This can be one of **circular** or **rectangular.**
- Enter the external dimensions of the inlet in ft or m. In case of circular inlet, only one dimension is required.
- Check **inlet has neck** if the inlet has a neck that extents from its maximum height to the ground. In this case, you must provide the neck's external diameter in ft or m.

Implementation Tab



- In the **Thickness** frame, enter the **thicknesses** of the inlet in ft or m by typing into the corresponding text boxes.
- In the **Materials** frame, select the concrete grade for each element and check whether you want to calculate **wood form** and/or **asphalt** in quantities.

Sublayer Tab



- Optionally, you can use a layer of concrete below the inlet. In this case, check **sublayer is made of** in the previous tab and provide the thickness of the layer in ft or m by typing in the text box.

6.15.3 Delete

To delete an existing specification:

1. Select the specification from the list on the left.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The specification is deleted from the list.

6.15.4 Edit

To edit an existing specification:

1. Select the specification from the list on the left.
2. Press **Edit**. The data form appears.

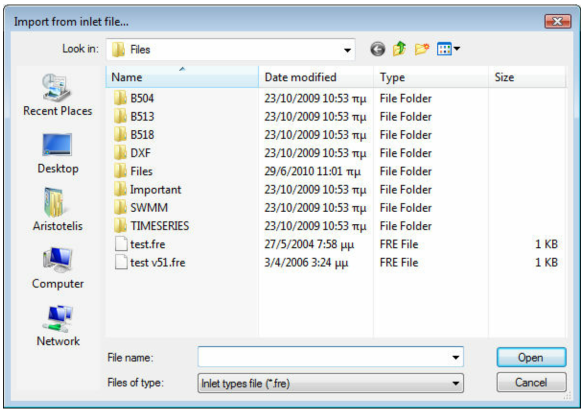
3. Make the appropriate selections as described in the add specification topic.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.15.5 Import

With this option, you can import specifications from an external file. Existing specifications are erased.

To import specifications from an external file:

1. Select **Import.** The file selection dialog box will appear:



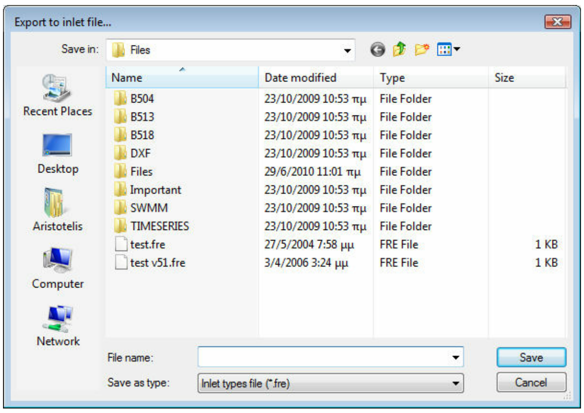
2. Select the path of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Inlet type file" with the extension .fre.
4. Select the file by clicking on it.
5. Select **Open** to import the specifications.

6.15.6 Export

With this option, you can export specifications to an external file. These can be imported at a later time.

To export pipe specifications to an external file:

1. Select **Export**. The following form will appear:



2. Select the path of the file.
3. Type the filename in the **File name** text box.
4. Select **Save** to export pipe specifications with the selected filename and path. Select **Cancel** to cancel the operation.

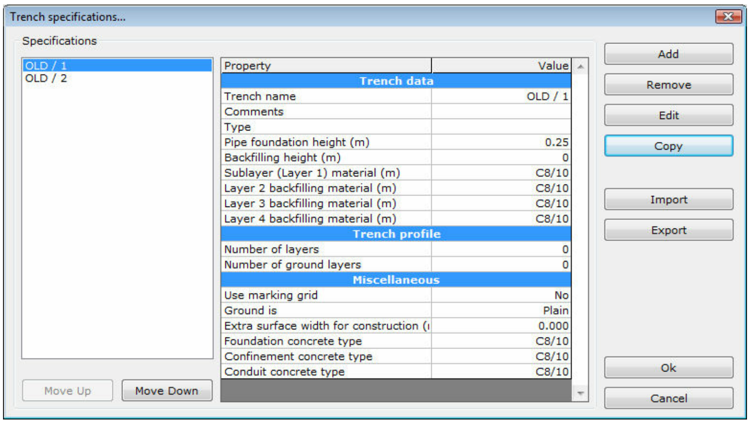
6.16 Trench specifications

6.16.1 Management

With this option, you can add or modify trench specifications. After defining trench specifications, you can use them in the data table of the main form.

To add or modify trench specifications:

1. Select **Trench specifications** from the **Data** menu. The following form will appear:



2. Enter the trench specifications.
3. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Trench specifications can be imported from a file or exported to a file, as described later in this chapter.

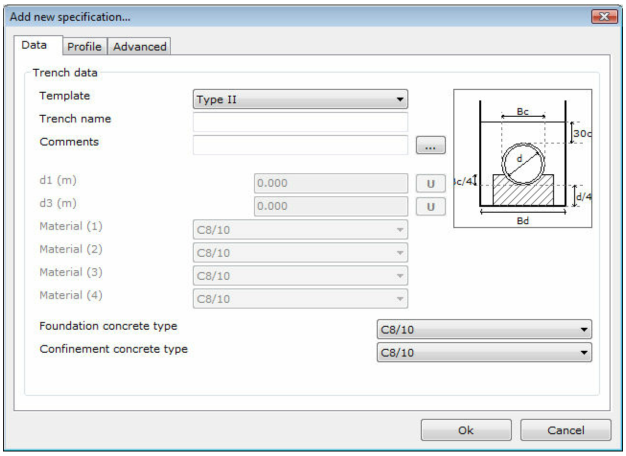
The next sections describe the process of adding, modifying or deleting a trench specification.

6.16.2 Add

With this option, you can add a trench specification.

To add a trench specification:

1. Select **Add**. The following form appears:

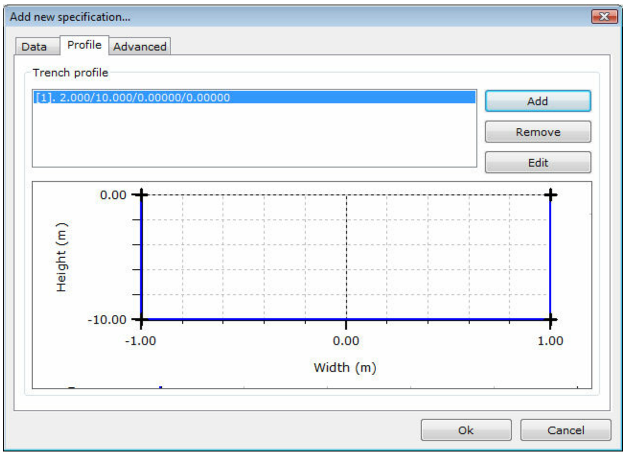


2. Make the appropriate selections as described below.
3. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Data Tab

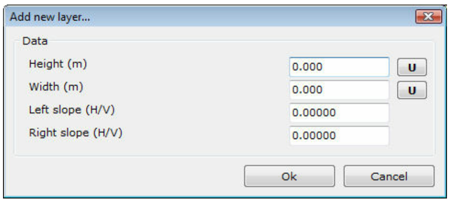
- Select a **template** for the trench.
- Enter a unique **Name** for the trench specification.
- Optionally, you can add some **Comments.** Use the ellipsis button to edit multiline text.
- Depending on the selected trench template, enter the required dimensions and / or material types. Unnecessary fields are disabled.

Profile Tab



Enter the trench profile. The profile is defined as a set of trapezoidal layers, from bottom to top. For example, a vertical trench may be defined for the first 6 ft, followed by a trapezoidal layer for stability reasons. There is no restriction in the combination of layers. If the total specified depth is insufficient, the program extents the topmost layer as necessary.

To add a layer, press **Add.** The layer data form appears:



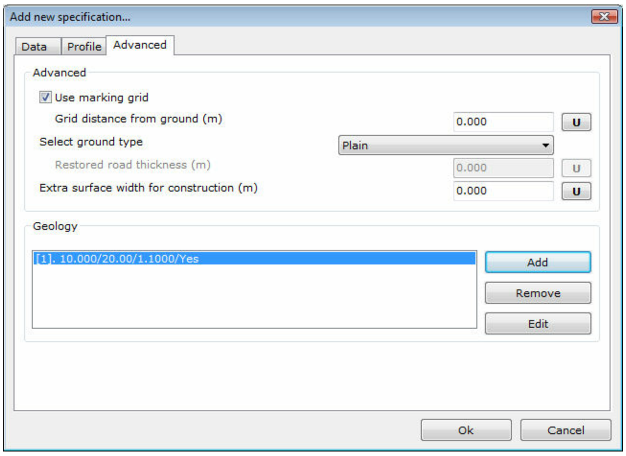
Enter the **height**, **width** and **slopes** of the layer. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Press **Remove** to delete the currently selected layer.

Press **Edit** to edit the currently selected layer.

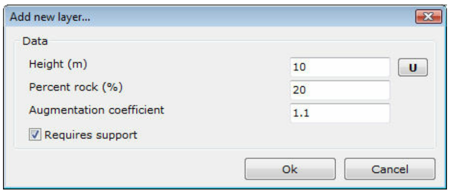
A sketch of the current trench profile is drawn in the form.

Advanced Tab



- Check **Use marking grid** if you want to take into account the calculation of marking grid in quantities.
- Select the appropriate **ground type** from the drop-down list. This can be one of **plain, concrete** or **asphalt.** In the latter two cases, enter the thickness in ft or m. These are needed for the calculation of the restoration and backfill volumes in the quantities.
- Enter the **extra surface width** that is necessary for the construction in ft or m.
- Enter the **geological profile**. The geological profile is defined in layers, from top to bottom. For example, a single layer with height of 6ft, 20% rock percentage, augmentation coefficient equal to 1.15 and need for support can be defined. If the total height of the profile is insufficient, the program automatically extends the bottommost layer as necessary.

To add a layer, press **Add.** The layer data form appears:



Enter the **height**, **rock percentage** and **augmentation factor** of the layer. Check

Requires support if the specified layer requires support. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Press **Remove** to delete the currently selected layer.

Press **Edit** to edit the currently selected layer.

6.16.3 Delete

To delete an existing specification:

1. Select the specification from the list on the left.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The specification is deleted from the list.

6.16.4 Edit

To edit an existing specification:

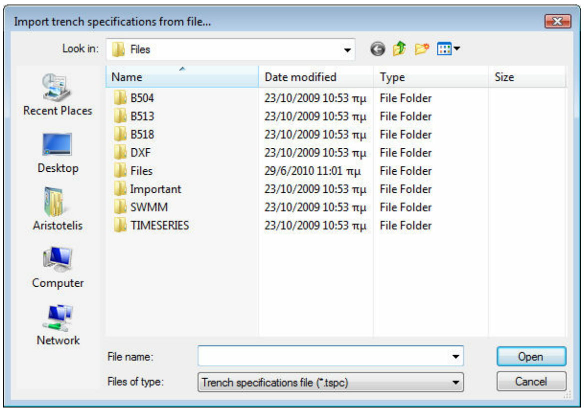
1. Select the specification from the list on the left.
2. Press **Edit**. The data form appears.
3. Make the appropriate selections as described in the add specification topic.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.16.5 Import

With this option, you can import specifications from an external file. Existing specifications are erased.

To import specifications from an external file:

1. Select **Import.** The file selection dialog box will appear:



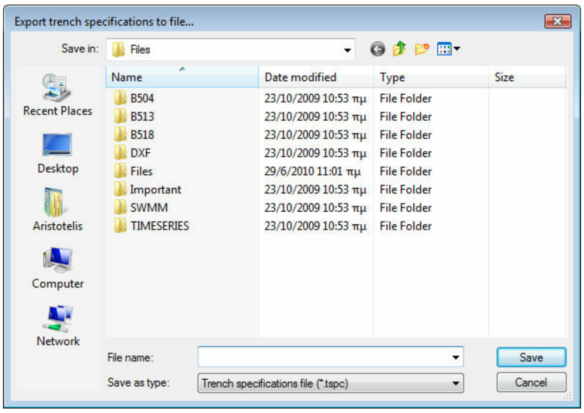
2. Select the path of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Trench specifications file" with the extension .tspc.
4. Select the file by clicking on it.
5. Select **Open** to import the specifications.

6.16.6 Export

With this option, you can export specifications to an external file. These can be imported at a later time.

To export pipe specifications to an external file:

1. Select **Export**. The following form will appear:



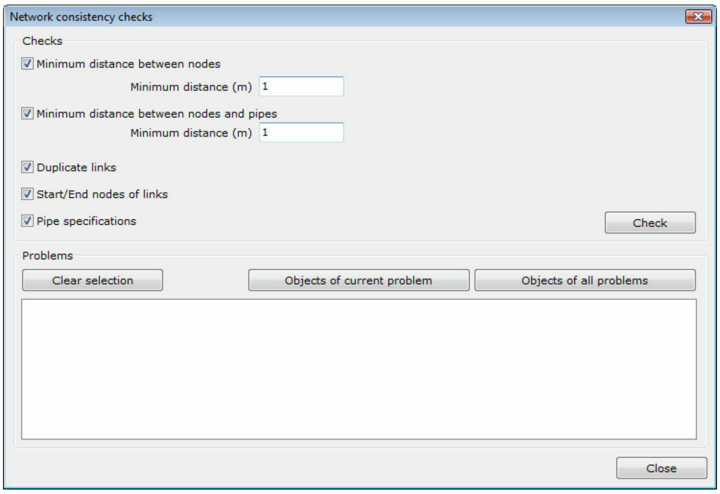
2. Select the path of the file.
3. Type the filename in the **File name** text box.
4. Select **Save** to export pipe specifications with the selected filename and path. Select **Cancel** to cancel the operation.

6.17 Network consistency

When importing a network from an external source such as a DXF file or entering graphically its data, there can be a number of hard-to-locate inconsistencies. This form facilitates the location and correction of several types of problems such as duplicate nodes and pipes.

To use the network consistency check tool:

1. Select **Network Consistency** from the **Tools** menu.
2. Select the type of problems you wish to look for and optionally correct.
3. Click on **Check** to look for specific problems.
4. Highlight one problem in the list and click **Objects of current problem** to highlight the location of the erroneous input on the map.
5. Optionally you may want to click on **Objects of all problems** to highlight the location of all erroneous input on the map.
6. Click on **Clear selection** to hide all highlighted objects on the map.
7. Click **Close** to close the form.



6.18 Options

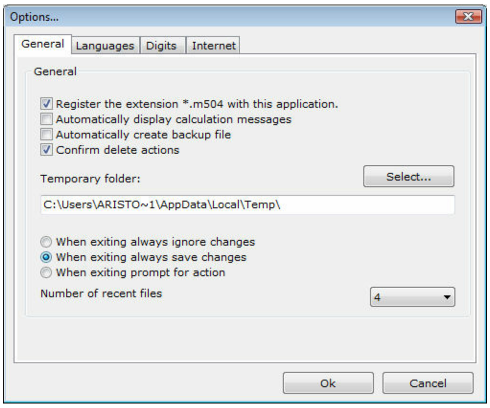
6.18.1 General preferences

With this option, you can modify the general preferences of the program.

To modify the general preferences:

1. Select **Options** from the **Data** menu.
2. Select **General preferences** from the **Options** menu.
3. The general preferences dialog box appears. The preferences are grouped into four tabs. You can select a tab by clicking on its name.

General Tab



This tab contains general preferences regarding the usage of the program.

Check **Register the extension *.m504 with this application** to associate the extension .m504 with this program. This extension is used by the program when saving a project. In this way, you will be able to run the program and load a project by double-clicking on the project filename in Windows Explorer.

Check **Automatically display calculation messages** if you want the report details to be automatically displayed when you calculate the results.

Check **Automatically create backup file** if you want a backup file (with the extension .bck) to be created every time a project is loaded. By default, this file is created in the temporary folder of Windows.

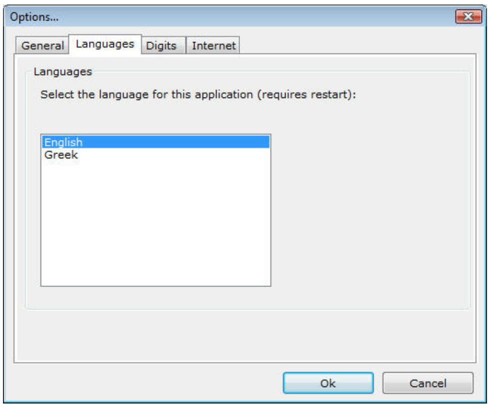
Check **Confirm delete actions** if you want to be asked for confirmation each time an object is about to be deleted. This setting affects the behaviour of all delete actions, for example the deletion of an object.

You can also modify the temporary folder that will be used for the creation of backup files. By default, this folder is the temporary folder of Windows.

Finally, there are three options regarding the termination of the program:

- **When exiting always ignore changes** - All changes since the last save of the project are ignored.
- **When exiting always save changes** - All changes in the current project are automatically saved. If the filename of the project is not set, a dialog box will appear that allows the selection of the filename, as when selecting Save project as from the **File** menu.
- **When exiting prompt for action** - If there are changes in the current project, then a dialog box will appear. You can choose to save or ignore the changes. If the filename of the project is not set, a dialog box will appear that allows the selection of the filename, as when selecting Save project as from the **File** menu.

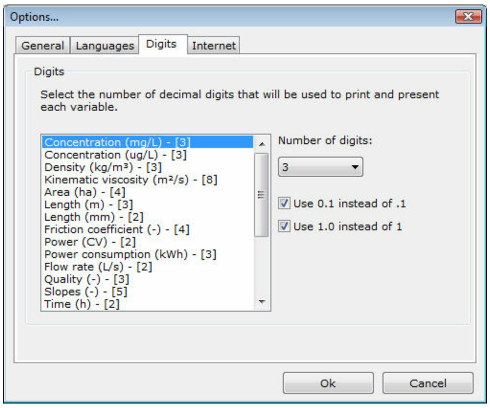
Languages Tab



If more than one language packs have been installed, then you can choose the language of the program. In the above case, there are two language packs; English (that are already selected) and Greek. If you change the language, all forms, menus, messages, help files will reflect the chosen language.

In order for the changes to take effect, you must restart the program.

Digits Tab



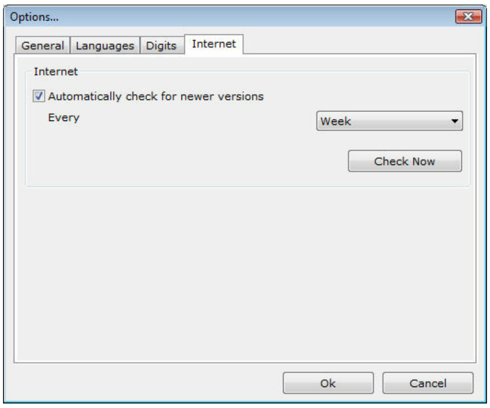
With this tab, you can modify the way the results are presented. All values used in the program are displayed in the list on the left.

For each value, you can select the number of decimal digits using the **Number of digits** drop-down list.

Check **Use 0.1 instead of .1** to use a preceding zero when displaying numbers between -1 and 1, for example -0.08 instead of -.08 and 0.98 instead of .98.

Check **Use 1.0 instead of 1** to use trailing zeros (when necessary) in order to display a number with the decimal digits selected in the **Number of digits** drop-down list, for example 1.1600 instead of 1.16 (when the number of digits is set to 4).

Internet Tab



The program can automatically check for newer versions over the Internet. Check **Automatically check for newer versions** to enable this feature. The check is automatically performed at an interval specified in the **Every** drop-down list. Select **Check now** to manually check for newer versions.

When a newer version is found, you will be prompted to download and install the latest version.

NOTE: TechnoLogismiki protects your privacy. During the check for newer versions, no data is transferred from your computer to the Internet.

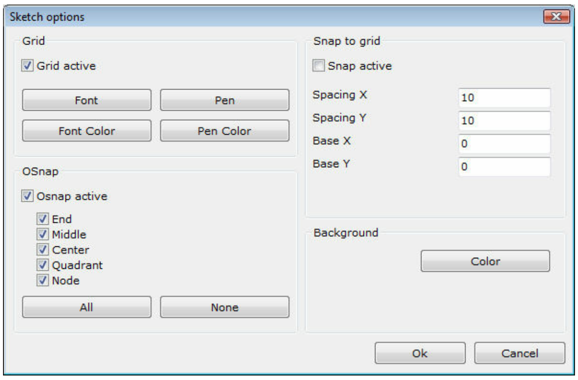
Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

6.18.2 Sketch

With this option, you can modify the profile sketch.

To modify the profile sketch:

1. Select **Options** from the **Data** menu.
2. Select **Sketch** from the **Options** menu. The profile sketch options dialog box appears:



2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

<u>Grid</u>

- Select **Grid active** if you want the dynamic grid to be displayed.
- Press the **Font** button to select the font that will be used by the grid.
- Press the **Font color** button to select the color of the font that will be used by the grid.
- Press the **Pen** button to select the style and width of the grid line.
- Press the **Pen color** button to select the color of the grid line.

<u>OSnap</u>

- Select **OSnap active** if you want the snap to objects to be active.
- Select one or more OSnaps to be active: **End, Middle, Center, Quadrant, Node.** Press **All** to select all OSnaps. Press **None** to select none.

<u>Snap to grid</u>

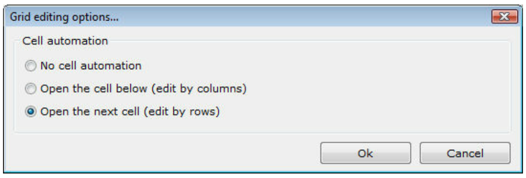
- Select **Snap active** if you want the snap to grid to be active.
- Select the appropriate **Spacing X** and **Spacing Y** values.
- Select the appropriate **Base X** and **Base Y** values.

<u>Background</u>

- Press the **Color** button to select the background color of the sketch

6.18.3 Grid editiing

With this option, you can modify the behavior of grids.



The behaviour of all editable grids is controlled by the preferences in this dialog box.

Select **No cell automation** if you want the active cell to remain the same when hitting ENTER.

Select **Open the cell below (edit by columns)** if you want to activate the cell below when hitting ENTER. This is particularly useful when editing tables by columns.

Select **Open the next cell (edit by rows)** if you want to activate the next cell on the right when hitting ENTER. This is particularly useful when editing tables by rows.

NOTE: These preferences affect all projects, both old and new.

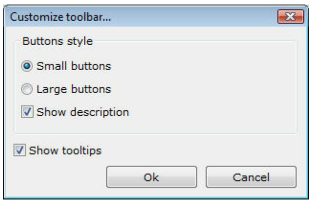
Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

6.18.4 Customize toolbar

With this option, you can customize the toolbar of the main form.

To customize the toolbar of the main form:

1. Select **Options** from the **Data** menu.
2. Select **Customize toolbar** from the **Options** menu.
3. Make the appropriate changes.
4. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



The toolbar may contain small or large buttons.

Check **Show description** if you want a small description to be displayed under the buttons.

Check **Show tooltips** if you want tooltips to be displayed when the mouse pointer hovers over a button for 2-3 seconds.

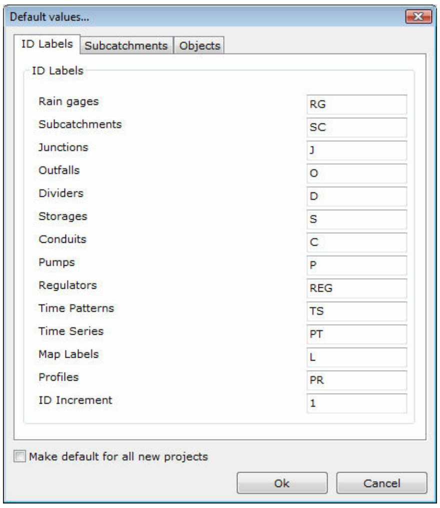
NOTE: These preferences affect all projects, both old and new.

6.18.5 Default values

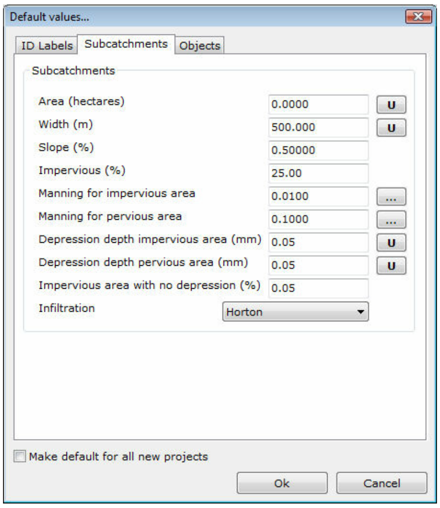
With this option, you can modify the default values for new linear or point objects, curves etc.

To modify the default values:

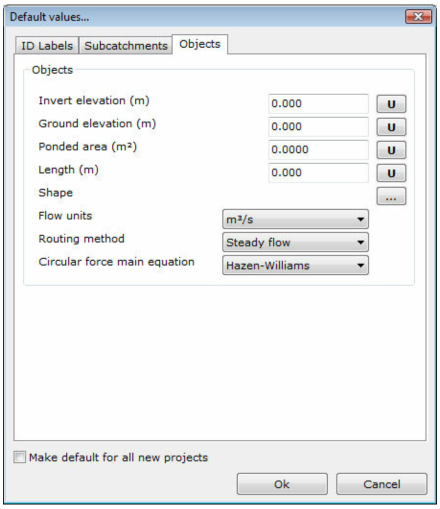
1. Select **Options** from the **Data** menu.
2. Select **Default values** from the **Options** menu.
3. In the **ID Labels** tab, select the prefixes of new object names.



4. In the **Subcatchments** tab, select the default values for new subcatchments.



5. In the **Objects** tab select the default values for other new objects.



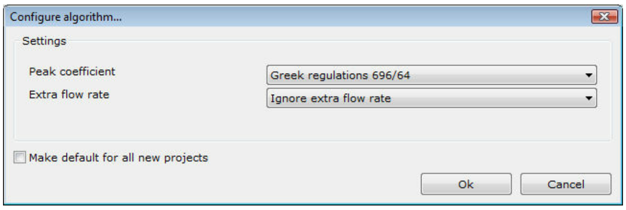
6. Select **Make default for all new projects** if you wish to use these settings for all new projects.
7. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.18.6 Algorithm

With this option, you can select the method of estimation of the peak flow factor and the extra flow rate.

To select the method of estimation of the peak flow factor and the extra flow rate:

1. Select **Options** from the **Data** menu.
2. Select **Algorithm** from the **Options** menu. The following form appears:



3. Select one of the following methods for calculating the peak coefficient:
- Greek Regulations 696/74
- EYDAP specifications
- Analytical statistical method
- Gifft
- Harmon
- Metcalf & Eddy
- Babbit

4. Select one of the following methods for calculating the extra flow:
- Ignore extra flow rate
- EYDAP (high aquifer)
- EYDAP (low aquifer)
- Percentage of peak flow Qp
- Percentage of mean daily flow Qd

- Metcalf & Eddy (New networks)
- Metcalf & Eddy (Old networks)
- EYDAP (per area, high aquifer)
- EYDAP (per area, low aquifer)
- Per area
- Per pipe length
- Per pipe length and diameter

5. Select **Make default for all new projects** if you wish to use these settings for all new projects. It is stressed that these settings are saved with the project. Thus, older projects are not affected by changes.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Chapter VII

7 Objects

7.1 Objects menu

With this option, you can add and modify objects. In the **Objects** menu you can select one of the following options:

- Add
  - Raingage
  - Subcatchment
  - Junction
  - Outfall
  - Divider
  - Storage
  - Conduit
  - Pump
  - Orifice
  - Weir
  - Outlet
  - Profile
  - Label
- Properties
  - Raingage
  - Subcatchment: Infiltration, Groundwater, Land use, Initial buildup
  - Node Inflows
  - Node Treatment
  - Junction
  - Outfall
  - Divider
  - Storage
  - Link internal vertices
  - Conduit
  - Pump
  - Orifice
  - Weir
  - Outlet
  - Profile
  - Label
- Object conversion
- Add vertex
- Add vertex by distance
- Delete vertex
- Stretch vertex
- Convert vertex to junction
- Labels
- Swap link ends
- Transects: Management, Add, Delete, Edit, Move, Sort
- Control rules: Management, Add, Delete, Edit, Move, Sort

7.2 Add

7.2.1 Raingage

With this option, you can add one or more raingages.

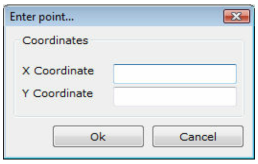
To add a raingage:

1. Select **Add** from the **Objects** menu.
2. Select **Raingage** from the **Add** menu.
3. Click onto the drawing to define the position of the raingage.

To add more than one raingages:

1. Select **Add** from the **Objects** menu.
2. Select **Raingage** from the **Add** menu while holding down CTRL key.
3. Click onto the drawing to define the position of the raingage while holding down CTRL key.
4. Repeat step 3 as many times as required.
5. Hit ESC when you have finished.

When the program expects a point, you can provide the coordinates analytically by hitting CTRL+2. The following form appears:



1. Enter the coordinates by typing into the corresponding text box.
2. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes. The program resumes the previous action.

NOTE: When selecting points graphically, you can use Snap and / or OSnap. These options can be configured using the menu **Data > Options > Sketch** or by hitting **CTRL + 1**.

7.2.2 Subcatchment

With this option, you can add one or more subcatchments. Subcatchments defined by one or two points are considered symbolical and you need to provide their area explicitly.

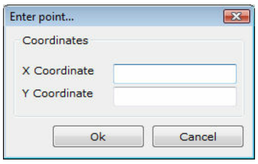
To add a subcatchment:

1. Select **Add** from the **Objects** menu.
2. Select **Subcatchment** from the **Add** menu.
3. Click successively onto the drawing to define the vertices of the subcatchment.
4. Press **ENTER** to close the polyline and define the subcatchment. Press **BACKSPACE** to erase the last defined vertex.

To add more than one subcatchments:

1. Select **Add** from the **Objects** menu.
2. Select **Subcatchment** from the **Add** menu while holding down CTRL key.
3. Click successively onto the drawing to define the vertices of the subcatchment.
4. Press **ENTER** while holding down CTRL key to close the polyline and define the subcatchment. Press **BACKSPACE** to erase the last defined vertex.
5. Repeat steps 3 and 4 as many times as required.
6. Hit ESC when you have finished.

When the program expects a point, you can provide the coordinates analytically by hitting CTRL+2. The following form appears:



1. Enter the coordinates by typing into the corresponding text box.
2. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes. The program resumes the previous action.

NOTE: When selecting points graphically, you can use Snap and / or OSnap. These options can be configured using the menu **Data > Options > Sketch** or by hitting **CTRL + 1**.

7.2.3 Junction

With this option, you can add one or more junctions.

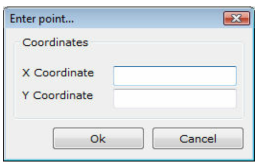
To add a junction:

1. Select **Add** from the **Objects** menu.
2. Select **Junction** from the **Add** menu.
3. Click onto the drawing to define the position of the junction.

To add more than one junctions:

1. Select **Add** from the **Objects** menu.
2. Select **Junction** from the **Add** menu while holding down CTRL key.
3. Click onto the drawing to define the position of the junction while holding down CTRL key.
4. Repeat step 3 as many times as required.
5. Hit ESC when you have finished.

When the program expects a point, you can provide the coordinates analytically by hitting CTRL+2. The following form appears:



1. Enter the coordinates by typing into the corresponding text box.
2. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes. The program resumes the previous action.

NOTE: When selecting points graphically, you can use Snap and / or OSnap. These options can be configured using the menu **Data > Options > Sketch** or by hitting **CTRL + 1**.

7.2.4 Outfall

With this option, you can add one or more outfalls.

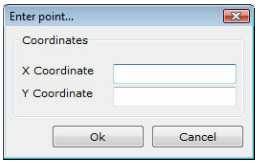
To add a outfall:

1. Select **Add** from the **Objects** menu.
2. Select **Outfall** from the **Add** menu.
3. Click onto the drawing to define the position of the outfall.

To add more than one outfalls:

1. Select **Add** from the **Objects** menu.
2. Select **Outfall** from the **Add** menu while holding down CTRL key.
3. Click onto the drawing to define the position of the outfall while holding down CTRL key.
4. Repeat step 3 as many times as required.
5. Hit ESC when you have finished.

When the program expects a point, you can provide the coordinates analytically by hitting CTRL+2. The following form appears:



1. Enter the coordinates by typing into the corresponding text box.
2. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes. The program resumes the previous action.

NOTE: When selecting points graphically, you can use Snap and / or OSnap. These options can be configured using the menu **Data > Options > Sketch** or by hitting **CTRL + 1**.

7.2.5 Divider

With this option, you can add one or more dividers.

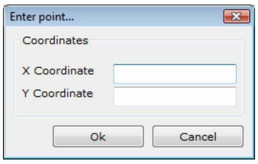
To add a divider:

1. Select **Add** from the **Objects** menu.
2. Select **Divider** from the **Add** menu.
3. Click onto the drawing to define the position of the divider.

To add more than one dividers:

1. Select **Add** from the **Objects** menu.
2. Select **Divider** from the **Add** menu while holding down CTRL key.
3. Click onto the drawing to define the position of the divider while holding down CTRL key.
4. Repeat step 3 as many times as required.
5. Hit ESC when you have finished.

When the program expects a point, you can provide the coordinates analytically by hitting CTRL+2. The following form appears:



1. Enter the coordinates by typing into the corresponding text box.
2. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes. The program resumes the previous action.

NOTE: When selecting points graphically, you can use Snap and / or OSnap. These options can be configured using the menu **Data > Options > Sketch** or by hitting **CTRL + 1**.

7.2.6 Storage

With this option, you can add one or more storages.

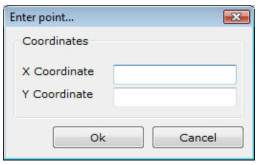
To add a storage:

1. Select **Add** from the **Objects** menu.
2. Select **Storage** from the **Add** menu.
3. Click onto the drawing to define the position of the storage.

To add more than one storages:

1. Select **Add** from the **Objects** menu.
2. Select **Storage** from the **Add** menu while holding down CTRL key.
3. Click onto the drawing to define the position of the storage while holding down CTRL key.
4. Repeat step 3 as many times as required.
5. Hit ESC when you have finished.

When the program expects a point, you can provide the coordinates analytically by hitting CTRL+2. The following form appears:



1. Enter the coordinates by typing into the corresponding text box.

2. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes. The program resumes the previous action.

NOTE: When selecting points graphically, you can use Snap and / or OSnap. These options can be configured using the menu **Data > Options > Sketch** or by hitting **CTRL + 1**.

7.2.7 Conduit

With this option, you can add one or more conduits. The ends of the conduit are always point objects (Junctions, Outfalls, Dividers, Storages). Although there is no such restriction, it is recommended that the direction of conduits matches the flow direction.

To add a conduit:

1. Select **Add** from the **Objects** menu.
2. Select **Conduit** from the **Add** menu.
3. Click onto the point object that will be the start point of the conduit.
4. Click onto the point object that will be the end point of the conduit. The conduit is drawn.

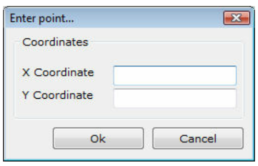
To add more than one conduits:

1. Select **Add** from the **Objects** menu.
2. Select **Conduit** from the **Add** menu while holding down CTRL key.
3. Click onto the point object that will be the start point of the conduit.
4. Click onto the point object that will be the end point of the conduit while holding down CTRL key. The conduit is drawn.
5. Repeat steps 3 and 4 to add the next conduit.
6. Hit ESC when you have finished.

NOTE: At least two point objects are required.

If you wish to define intermediate vertices for a conduit, define their successive coordinates either graphically or analytically **prior** to clicking on the end point object. These intermediate vertices define piecewise linear objects and can be fully manipulated (including addition, deletion, displacement) after the object is created. They can be taken into account when calculating the length of the conduit, in the calculation of excavation volumes etc.

When the program expects a point, you can provide the coordinates analytically by hitting CTRL+2. The following form appears:



1. Enter the coordinates by typing into the corresponding text box.
2. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes. The program resumes the previous action.

NOTE: When selecting points graphically, you can use Snap and / or OSnap. These options can be configured using the menu **Data > Options > Sketch** or by hitting **CTRL + 1**.

7.2.8 Pump

With this option, you can add one or more pumps. The ends of the pump are always point objects (Junctions, Outfalls, Dividers, Storages). Although there is no such restriction, it is recommended that the direction of pumps matches the flow direction.

To add a pump:

1. Select **Add** from the **Objects** menu.
2. Select **Pump** from the **Add** menu.
3. Click onto the point object that will be the start point of the pump.
4. Click onto the point object that will be the end point of the pump. The pump is drawn.

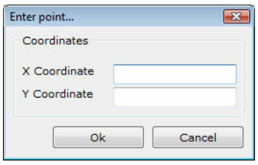
To add more than one pumps:

1. Select **Add** from the **Objects** menu.
2. Select **Pump** from the **Add** menu while holding down CTRL key.
3. Click onto the point object that will be the start point of the pump.
4. Click onto the point object that will be the end point of the pump while holding down CTRL key. The pump is drawn.
5. Repeat steps 3 and 4 to add the next pump.
6. Hit ESC when you have finished.

NOTE: At least two point objects are required.

If you wish to define intermediate vertices for a pump, define their successive coordinates either graphically or analytically **prior** to clicking on the end point object. These intermediate vertices define piecewise linear objects and can be fully manipulated (including addition, deletion, displacement) after the object is created.

When the program expects a point, you can provide the coordinates analytically by hitting CTRL+2. The following form appears:



1. Enter the coordinates by typing into the corresponding text box.
2. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes. The program resumes the previous action.

NOTE: When selecting points graphically, you can use Snap and / or OSnap. These options can be configured using the menu **Data > Options > Sketch** or by hitting **CTRL + 1**.

7.2.9 Orifice

With this option, you can add one or more orifices. The ends of the orifice are always point objects (Junctions, Outfalls, Dividers, Storages). Although there is no such restriction, it is recommended that the direction of orifices matches the flow direction.

To add a orifice:

1. Select **Add** from the **Objects** menu.
2. Select **Orifice** from the **Add** menu.
3. Click onto the point object that will be the start point of the orifice.
4. Click onto the point object that will be the end point of the orifice. The orifice is drawn.

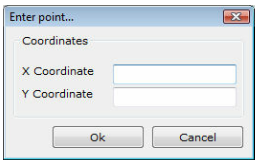
To add more than one orifices:

1. Select **Add** from the **Objects** menu.
2. Select **Orifice** from the **Add** menu while holding down CTRL key.
3. Click onto the point object that will be the start point of the orifice.
4. Click onto the point object that will be the end point of the orifice while holding down CTRL key. The orifice is drawn.
5. Repeat steps 3 and 4 to add the next orifice.
6. Hit ESC when you have finished.

NOTE: At least two point objects are required.

If you wish to define intermediate vertices for a orifice, define their successive coordinates either graphically or analytically **prior** to clicking on the end point object. These intermediate vertices define piecewise linear objects and can be fully manipulated (including addition, deletion, displacement) after the object is created.

When the program expects a point, you can provide the coordinates analytically by hitting CTRL+2. The following form appears:



1. Enter the coordinates by typing into the corresponding text box.
2. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes. The program resumes the previous action.

NOTE: When selecting points graphically, you can use Snap and / or OSnap. These options can be configured using the menu **Data > Options > Sketch** or by hitting **CTRL + 1**.

7.2.10 Weir

With this option, you can add one or more weirs. The ends of the weir are always point objects (Junctions, Outfalls, Dividers, Storages). Although there is no such restriction, it is recommended that the direction of weirs matches the flow direction.

To add a weir:

1. Select **Add** from the **Objects** menu.
2. Select **Weir** from the **Add** menu.
3. Click onto the point object that will be the start point of the weir.
4. Click onto the point object that will be the end point of the weir. The weir is drawn.

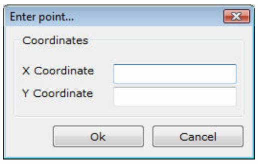
To add more than one weirs:

1. Select **Add** from the **Objects** menu.
2. Select **Weir** from the **Add** menu while holding down CTRL key.
3. Click onto the point object that will be the start point of the weir.
4. Click onto the point object that will be the end point of the weir while holding down CTRL key. The weir is drawn.
5. Repeat steps 3 and 4 to add the next weir.
6. Hit ESC when you have finished.

NOTE: At least two point objects are required.

If you wish to define intermediate vertices for a weir, define their successive coordinates either graphically or analytically **prior** to clicking on the end point object. These intermediate vertices define piecewise linear objects and can be fully manipulated (including addition, deletion, displacement) after the object is created.

When the program expects a point, you can provide the coordinates analytically by hitting CTRL+2. The following form appears:



1. Enter the coordinates by typing into the corresponding text box.
2. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes. The program resumes the previous action.

NOTE: When selecting points graphically, you can use Snap and / or OSnap. These options can be configured using the menu **Data > Options > Sketch** or by hitting **CTRL + 1**.

7.2.11 Outlet

With this option, you can add one or more outlets. The ends of the outlet are always point objects (Junctions, Outfalls, Dividers, Storages). Although there is no such restriction, it is recommended that the direction of outlets matches the flow direction.

To add a outlet:

1. Select **Add** from the **Objects** menu.
2. Select **Outlet** from the **Add** menu.
3. Click onto the point object that will be the start point of the outlet.
4. Click onto the point object that will be the end point of the outlet. The outlet is drawn.

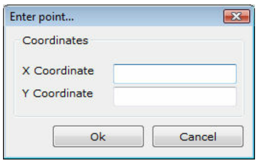
To add more than one outlets:

1. Select **Add** from the **Objects** menu.
2. Select **Outlet** from the **Add** menu while holding down CTRL key.
3. Click onto the point object that will be the start point of the outlet.
4. Click onto the point object that will be the end point of the outlet while holding down CTRL key. The outlet is drawn.
5. Repeat steps 3 and 4 to add the next outlet.
6. Hit ESC when you have finished.

NOTE: At least two point objects are required.

If you wish to define intermediate vertices for a outlet, define their successive coordinates either graphically or analytically **prior** to clicking on the end point object. These intermediate vertices define piecewise linear objects and can be fully manipulated (including addition, deletion, displacement) after the object is created.

When the program expects a point, you can provide the coordinates analytically by hitting CTRL+2. The following form appears:



1. Enter the coordinates by typing into the corresponding text box.
2. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes. The program resumes the previous action.

NOTE: When selecting points graphically, you can use Snap and / or OSnap. These options can be configured using the menu **Data > Options > Sketch** or by hitting **CTRL + 1**.

7.2.12 Profile

With this option, you can add one or more profiles. Profiles are not actual objects but rather a series of references to alternating nodes and links. Although there is no restriction in the definition of profiles, it is recommended that

- they follow the flow direction,
- they do not overlap each other.

To add a profile:

1. Select **Add** from the **Objects** menu.
2. Select **Profile** from the **Add** menu.
3. Click successively onto the drawing to select the nodes of the profile. It is not necessary to click onto all nodes. If two successive nodes are not connected by a single link, the program automatically seeks and selects the **shortest path** (in terms of number of links) connecting these two nodes. Thus, in most cases, you can fully define a profile by clicking on the first and last node.
4. Press **ENTER** to define the profile. Press **BACKSPACE** to erase the reference to the

last defined point object.

To add more than one profiles:

1. Select **Add** from the **Objects** menu.
2. Select **Profile** from the **Add** menu while holding down CTRL key.
3. Click successively onto the drawing to select the nodes of the profile. It is not necessary to click onto all nodes. If two successive nodes are not connected by a single link, the program automatically seeks and selects the **shortest path** (in terms of number of links) connecting these two nodes. Thus, in most cases, you can fully define a profile by clicking on the first and last node.
4. Press **ENTER** while holding down CTRL key to define the profile. Press **BACKSPACE** to erase the last defined vertex.
5. Repeat steps 3 and 4 as many times as required.
6. Hit ESC when you have finished.

7.2.13 Label

With this option, you can add one or more labels.

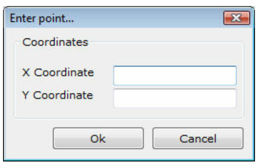
To add a label:

1. Select **Add** from the **Objects** menu.
2. Select **Label** from the **Add** menu.
3. Click onto the drawing to define the position of the label.

To add more than one labels:

1. Select **Add** from the **Data** menu.
2. Select **Label** from the **Add** menu while holding down CTRL key.
3. Click onto the drawing to define the position of the label while holding down CTRL key.
4. Repeat step 3 as many times as required.
5. Hit ESC when you have finished.

When the program expects a point, you can provide the coordinates analytically by hitting CTRL+2. The following form appears:



1. Enter the coordinates by typing into the corresponding text box.
2. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes. The program resumes the previous action.

NOTE: When selecting points graphically, you can use Snap and / or OSnap. These options can be configured using the menu **Data > Options > Sketch** or by hitting **CTRL + 1**.

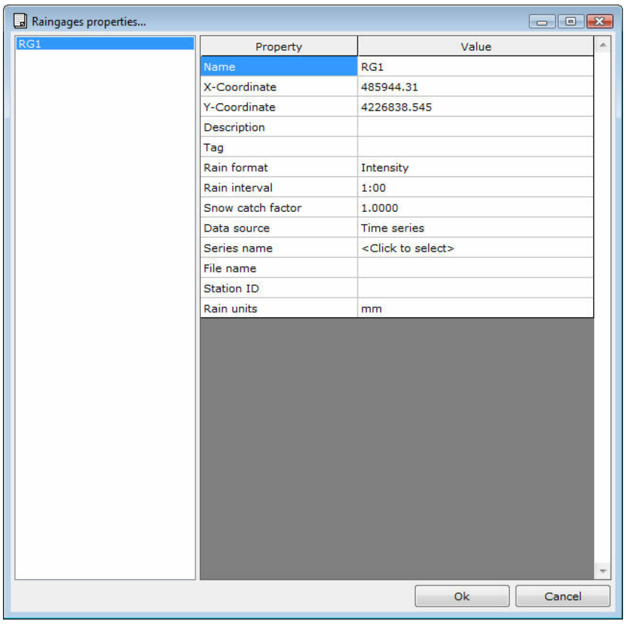
7.3 Properties

7.3.1 Raingage

With this option, you can view and modify the properties of raingages. Raingage data may be available as time series or by an external file.

To view and modify the properties of raingages:

1. Select **Properties** from the **Objects** menu.
2. Select **Raingage** from the **Properties** menu. The following form appears:



3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. Make the appropriate changes, as described below. The new property values are assigned to all selected objects in the list.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

<u>Properties</u>

- **Name**: enter the name of the raingage. Two or more raingages may share the same name, but this is not recommended since there will be confusion in the results.
- **X coordinate**: enter the X coordinate of the raingage.
- **Y coordinate**: enter the Y coordinate of the raingage.
- **Description**: enter the description of the raingage.
- **Tag**: enter a tag for the raingage. This appears neither in the input data nor the results.

- **Rain format:** Select the format in which the rain data are supplied. This can be one of the following:
  - **Intensity**: each value is an average rate over the recording interval in mm/h or in/h.
  - **Volume**: each value is the volume of rain divided by the area in mm or in.
  - **Cumulative**: each value is the cumulative volume of rain divided by the area mm or in.
- **Rain interval:** Select the time interval between the midpoints of successive events in decimal hours or HH:MM format.
- **Snow catch factor**: enter the factor that corrects gage readings for snowfall.
- **Data source**: enter the source of rainfall data; either **Time Series** for user-supplied time series data or **File** for an external data file.
- **Series name**: enter the name of time series with rainfall data if Data Source selection was **Time Series.** Leave blank otherwise.
- **File name**: enter the name of external file containing rainfall data.
- **Station ID**: enter the recording gage station number.
- **Rain units**: enter the depth units for rainfall values in the file. This can be one of **in** or **mm.**

Rainfall files

The program accepts the following rainfall formats:

- **DSI-3240** and related formats (hourly observations) from the U.S. National Weather Service (NWS) and Federal Aviation Agency stations. These are available online from the National Climatic Data Center (NCDC) in USA.
- **DSI-3260** and related formats (15-minute observations) from the U.S. National Weather Service (NWS). These are available online from the National Climatic Data Center (NCDC) in USA.
- **HLY03** and **HLY21** formats for hourly rainfall at Canadian stations, available online from Environment Canada.
- **FIF21** format for fifteen minute rainfall at Canadian stations, available online from Environment Canada.
- 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, as follows:

```
ST001  2004  12  01  00  00  0.12
ST001  2004  12  01  00  20  0.08
ST001  2004  12  01  01  05  0.02
ST001  2004  12  01  18  35  0.10
```

Each value may be intensity, volume or cumulative in English or metric system.

7.3.2 Subcatchment

With this option, you can view and modify the properties of subcatchments.

Subcatchments are hydrologic units of land whose topography and drainage system elements direct surface runoff to a single discharge point. The user is responsible for dividing a study area into an appropriate number of subcatchments, and for identifying the outlet point of each subcatchment. Discharge outlet points can be either nodes of the drainage system or other subcatchments.

Subcatchments can be divided into pervious and impervious subareas. Surface runoff can infiltrate into the upper soil zone of the pervious subarea, but not through the impervious subarea. Impervious areas are themselves divided into two subareas - one that contains depression storage and another that does not. Runoff flow from one subarea in a subcatchment can be routed to the other subarea, or both subareas can drain to the subcatchment outlet.

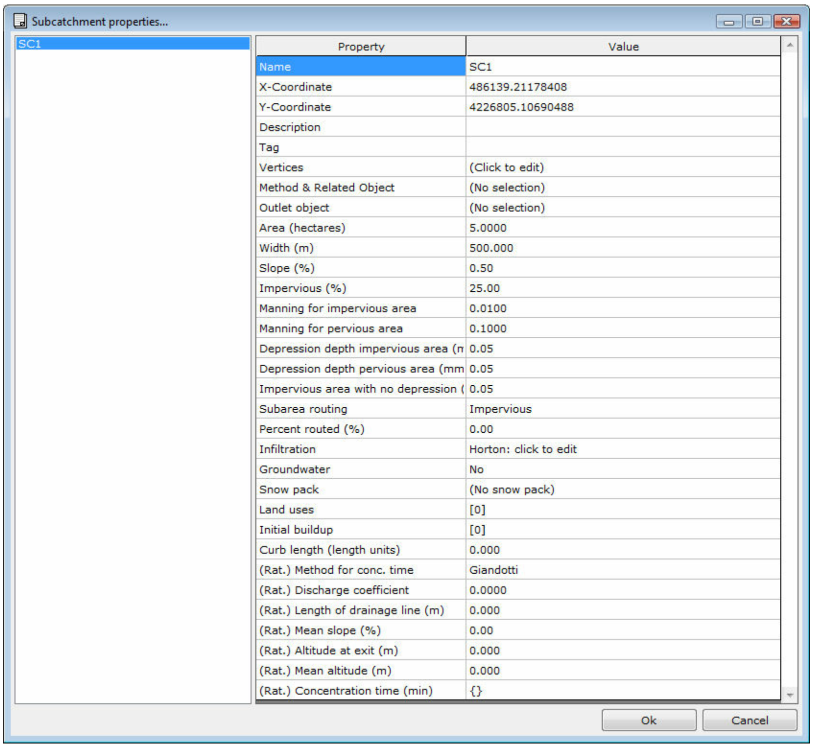
Infiltration of rainfall from the pervious area of a subcatchment into the unsaturated upper soil
zone can be described using three different models:

- Horton
- Green - Ampt
- SCS Curve Number

To model the accumulation, re-distribution, and melting of precipitation that falls as snow on a subcatchment, it must be assigned a Snow Pack object. To model groundwater flow between an aquiferunderneath the subcatchment and a node of the drainage system, the subcatchment must be assigned a set of Groundwater parameters. Pollutant buildup and washoff from subcatchments are associated with the Land Uses assigned to the subcatchment.

To view and modify the properties of subcatchments:

1. Select **Properties** from the **Objects** menu.
2. Select **Subcatchment** from the **Properties** menu. The following form appears:



3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. Make the appropriate changes, as described below. The new property values are assigned to all selected objects in the list.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Properties

- **Name**: enter the name of the subcatchment. Two or more subcatchments may share the same name, but this is not recommended since there will be confusion in the results.
- **X coordinate**: enter the X coordinate of the subcatchment centroid.
- **Y coordinate**: enter the Y coordinate of the subcatchment centroid.
- **Description**: enter the description of the subcatchment.
- **Tag**: enter a tag for the subcatchment. This appears neither in the input data nor the results.
- **Vertices**: click the ellipsis button to define the vertices analytically.
- **Raingage**: select the associated raingage from the drop-down list.
- **Outlet object**: select one junction, outfall, divider, storage or subcatchment which receives the subcatchment's runoff.
- **Area (ac or ha)**: enter the area of the subcatchment. Leave the field blank if you want the program to calculate the area from plan view. In this case the result is displayed within curly braces {}.
- **Width (ft or m)**: enter the characteristic width of the overland flow path for sheet flow runoff. An initial estimate of the characteristic width is given by the subcatchment area divided by the average maximum overland flow length. The maximum overland flow length is the length of the flow path from the inlet to the furthest drainage point of the subcatchment. Maximum lengths from several different possible flow paths should be averaged. These paths should reflect slow flow, such as over pervious surfaces, more than rapid flow over pavement, for example. Adjustments should be made to the width parameter to produce good fits to measured runoff hydrographs.
- **Slope (%)**: enter the average percent slope of the subcatchment.
- **Impervious (%)**: enter the percentage of land area which is impervious.
- **Manning for impervious area**: enter Manning's n for overland flow over the impervious portion of the subcatchment.
- **Manning for pervious area**: enter Manning's n for overland flow over the pervious portion of the subcatchment.
- **Depression depth impervious area (in or mm)**: enter the depth of depression storage on the impervious portion of the subcatchment. You can find typical values in the Appendix.
- **Depression depth pervious area (in or mm)**: enter the depth of depression storage on the pervious portion of the subcatchment. You can find typical values in the Appendix.
- **Impervious area with no depression (%)**: enter the percentage of the impervious area with no depression storage.
- **Subarea routing**: select the internal routing of runoff between pervious and impervious areas. This can be one of:

  - **Impervious**: runoff from pervious area flows to impervious area
  - **Pervious**: runoff from impervious area flows to pervious area
  - **Outlet**: runoff from both areas flows directly to outlet.
- **Percent routed (%)**: enter the percentage of runoff routed between subareas.
- **Infiltration**: enter data regarding infiltration.
- **Ground water**: enter data regarding ground water.
- **Snow pack**: select the snow pack associated with the subcatchment.
- **Land use**: enter the percentages of the land uses that are associated with the subcatchment.
- **Initial buildup**: enter the initial pollutant concentrations that are associated with the subcatchment.
- **Curb length (length units)**: enter the total curb length within the subcatchment. This variable is used only when pollutant buildup is normalized to curb length.

7.3.2.1 Infiltration

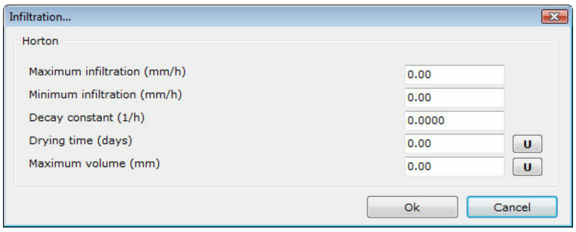
The Infiltration form is used to specify values for the parameters that describe the rate at which rainfall infiltrates into the upper soil zone in a subcatchment's pervious area.

It is invoked when editing the **Infiltration** property of a subcatchment. The infiltration parameters depend on which infiltration model was selected for the project: Horton, Green-Ampt, or Curve Number.

To edit the **Infiltration** property of a subcatchment (referring to the Subcatchment property form):

1. Double click on the **Infiltration** property.
2. Click the ellipsis button.
3. Make the appropriate changes.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Horton



Maximum infiltration (in/h or mm/h): enter the maximum infiltration. Some characteristic values are:

1. Dry soils (with little or no vegetation):

   - Sandy soils 5 in/h or 130 mm/h
   - Loam soils 3 in /h or 75 mm/h
   - Clay soils 1 in/h ή 25 mm/h

2. Dry soils (with dense vegetation):

- Multiply the values of (1) by 2.

3. Moist soils:

- Soils which have drained but not dried out (i.e., field capacity): Divide values from (1) and (2) by 3.
- Soils close to saturation: Choose value close to min. infiltration rate.
- Soils which have partially dried out: Divide values from (1) and (2) by 1.5 - 2.5.

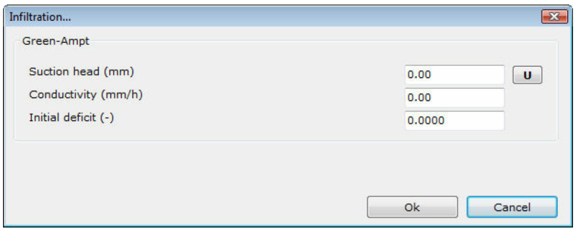
Minimum infiltration (in/h or mm/h): enter the minimum infiltration rate on the Horton curve. This is equivalent to the soil’s saturated hydraulic conductivity. Typical values are provided in theAppendix.

Decay constant (1/h): enter the decay constant. Typical values range between 2 and 7.

Drying time (d): Time in days for a fully saturated soil to dry completely. Typical values range from 2 to 14 days.

Maximum volume (in or mm): enter the maximum infiltration volume possible. It can be estimated as the difference between a soil's porosity and its wilting point times the depth of the infiltration zone.

<u>Green-Ampt</u>

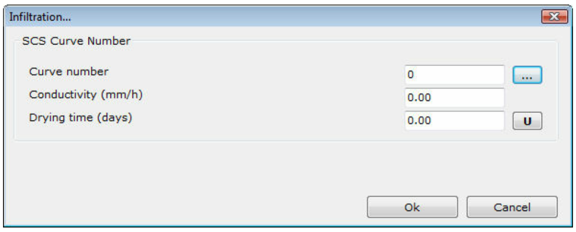


Suction head (in or mm): enter the average value of soil capillary suction along the wetting front.

Conductivity (in/h or mm/h): enter the soil saturated hydraulic conductivity.

Initial deficit: enter the fraction of soil volume that is initially dry (i.e., difference between soil porosity and initial moisture content). For a completely drained soil, it is the difference between the soil's porosity and its field capacity. Typical values are provided in the Appendix.

<u>SCS Curve Number</u>



Curve number: enter the SCS curve number which is tabulated in the publication SCS Urban Hydrology for Small Watersheds, 2nd Ed., (TR-55), June 1986. Press the ellipsis button to invoke the corresponding database.

Conductivity (in/h or mm/h): enter the soil's saturated hydraulic conductivity. This is used to estimate the minimum number of dry hours that must occur before a new storm is considered to begin using the equation: dry hours = 4.5 / (conductivity, in/hr)1/2.

Drying time (d): enter the number of days it takes a fully saturated soil to dry. Typical values range between 2 and 14 days.

7.3.2.2 Groundwater

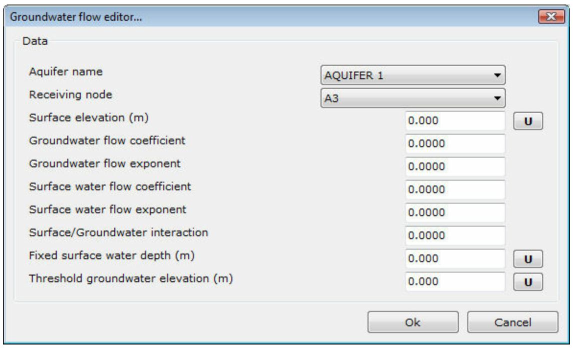
The Groundwater form is used to specify values for the parameters that describe the groundwater of a subcatchment. It is invoked when editing the **Groundwater** property of a subcatchment. It is used to link a subcatchment to both an aquifer and to a node of the drainage system that exchanges groundwater with the aquifer. It also specifies coefficients that determine the rate of groundwater flow between the aquifer and the node. These coefficients (A1, A2, B1, B2, and A3) appear in the following equation that computes groundwater flow as a function of groundwater and surface water heads:

$$Q_{GW} = A_1 \left(H_{GW} - E\right)^{B_1} - A_2 \left(H_{SW} - E\right)^{B_2} + A_3 H_{GW} H_{SW}$$

Where Q_{GW} groundwater flow (ft^3/s/ac or m^3/s/ha)
H_{GW} elevation of groundwater table (ft or m)
H_{SW} elevation of surface water at receiving node (ft or m)
E threshold groundwater elevation or node invert elevation (ft or m).

To edit the **Groundwater** property of a subcatchment (referring to the Subcatchment property form):

1. Double click on the **Groundwater** property.
2. Click the ellipsis button.
3. Make the appropriate changes.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



Aquifer name: select the aquifer that supplies groundwater from the drop-down list.

Receiving node: select the name of node that receives groundwater from the aquifer.

Surface elevation (ft or m): enter the elevation of ground surface for the subcatchment that lies above the aquifer.

Groundwater flow coefficient: enter the value of coefficient A_1 of the above equation.

Groundwater flow exponent: enter the value of coefficient B_1 of the above equation.

Surface water flow coefficient: enter the value of coefficient A_2 of the above equation.

Surface water flow exponent: enter the value of coefficient B_2 of the above equation.

Surface / Groundwater interaction: enter the value of coefficient A_3 of the above equation.

Fixed surface water depth (ft or m): enter the fixed depth of surface water at the receiving node. Set to zero if surface water depth will vary as computed by flow routing.

Threshold groundwater elevation (ft or m): enter the aquifer water table elevation which must be reached before any ground water flow occurs. Leave blank to use the receiving node's invert elevation.

NOTE: The values of the flow coefficients must be in units that are consistent with the groundwater flow units of cfs/acre for English unit system or cms/ha for metric .

NOTE: If groundwater flow is simply proportional to the difference in groundwater and surface water heads, then :

- B_1 and B_2 are set equal to 1
- A_1 equals the proportionality factor
- A_2 equals A_1

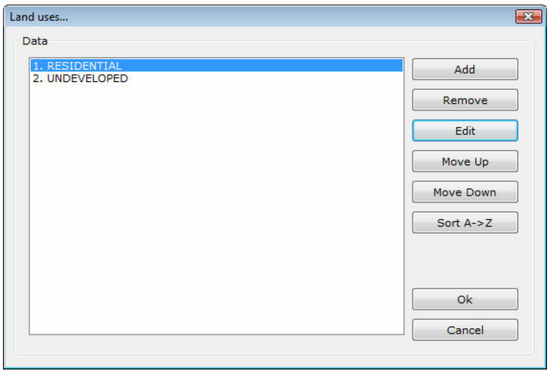
- A_3 equals 0

7.3.2.3 Land use

For each subcatchment, you must enter the associated percentage of each Land Use. It is not necessary to associate all land uses with a specified subcatchment.

To edit the **Land Use** property of a subcatchment (referring to the Subcatchment property form):

1. Double click on the **Land Uses** property.
2. Click the ellipsis button.
3. Make the appropriate changes.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

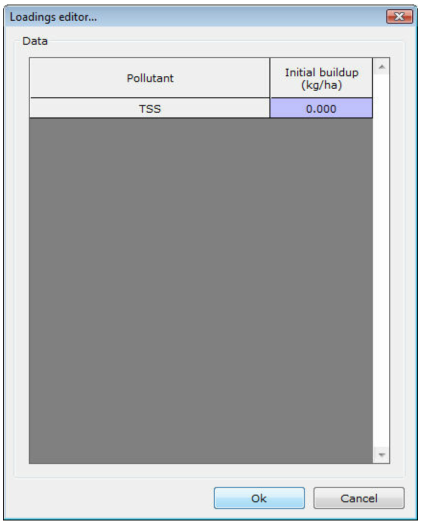


7.3.2.4 Initial buildup

For each subcatchment, you must enter the initial buildup of each pollutant in lb/ac or kg/ha. This specifies the amount of pollutant buildup existing over the subcatchment at the start of the simulation. If a non-zero value is specified for the initial buildup of a pollutant, it will override any initial buildup computed from the antecedent dry days parameter in general data.

To edit the **Initial buildup** property of a subcatchment (referring to the Subcatchment property form):

1. Double click on the **Initial buildup** property.
2. Click the ellipsis button.
3. Make the appropriate changes.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



7.3.3 Node Inflows

Apart from inflows from subcatchments and groundwater, junctions accept four types of inflows:

1. Direct. These are user-defined time series of inflows added directly into a node. They can be used to perform flow and water quality routing in the absence of any runoff computations (as in a study area where no subcatchments are defined).

2. Dry weather. These are continuous inflows that typically reflect the contribution from sanitary sewage in sewer systems or base flows in pipes and stream channels. They are represented by an average inflow rate that can be periodically adjusted on a monthly, daily, and hourly basis by applying time pattern multipliers to this average value is used to specify a continuous source of dry weather flow entering a node of the drainage system.

3. RDII. These are stormwater flows that enter sanitary or combined sewers due to "inflow" from direct connections of down spouts, sump pumps, foundation drains, etc. as well as "infiltration" of subsurface water through cracked pipes, leaky joints, poor manhole connections, etc. RDII can be computed for a given rainfall record based on set of triangular unit hydrographs (UH) that determine a short-term, intermediate-term, and long-term inflow response for each time period of rainfall. Any number of UH sets can be supplied for different sewershed areas and different months of the year. RDII flows can also be specified in an external RDII interface file.

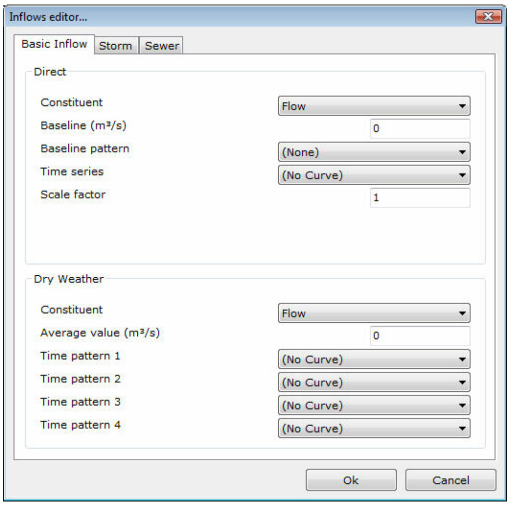
4. Areas. The corresponding values of Runoff Areas are entered. In this case, the IDF curve and the runoff coefficient is required. For sewage, only the flow per unit area is required. There is no restriction in the combination and number of runoff areas.

To enter inflow data:

1. Select the properties form of a node (junction, outfall, divider or storage).
2. Double click on the **Inflows** property.
3. Click the ellipsis button.
4. Make the appropriate changes.

5. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Basic Inflow



Direct

Constituent: select the constituent (**Flow** or one of the project's specified pollutants) whose direct inflow will be described.

Baseline: specifies the value of the constant baseline component of the constituent's inflow. For **Flow**, the units are the project's flow units. For pollutants, the units are the pollutant's concentration units if inflow is a concentration, or can be any mass flow units if the inflow is a mass flow. If left blank then no baseline inflow is assumed.

Baseline Pattern: An optional Time Pattern whose factors adjust the baseline inflow on either an hourly, daily, or monthly basis (depending on the type of time pattern specified).

Time series: select the name of the time series that contains inflow data for the selected constituent. If left blank then no direct inflow will occur for the selected constituent at the node in question.

Scale factor: enter a multiplier used to adjust the values of the constituent's inflow time series. The baseline value is not adjusted by this factor. The scale factor can have several uses, such as allowing one to easily change the magnitude of an inflow hydrograph while keeping its shape the same, without having to re-edit the entries in the hydrograph's time series. Or it can allow a group of nodes sharing the same time series to have their inflows behave in a time-synchronized fashion while letting their individual magnitudes be different.

Inflow type: (for pollutants) select the type of inflow data contained in the time series as being either a concentration (mass/volume) or mass flow rate (mass/time). This field does not appear for **Flow** inflow.

Mass unit conversion: A numerical factor used to convert the units of pollutant mass flow rate in the time series data into concentration mass units per second. For example, if the time series data were in pounds per day and the pollutant concentration defined in the project was mg/L, then the conversion factor value would be (453,590 mg/lb) / (86400 sec/day) = 5.25 (mg/sec) per (lb/day).

NOTE: If a pollutant is assigned a direct inflow in terms of concentration, then one must also assign a direct inflow to flow, otherwise no pollutant inflow will occur. If pollutant inflow is defined in terms of mass, then a flow inflow time series is not required,

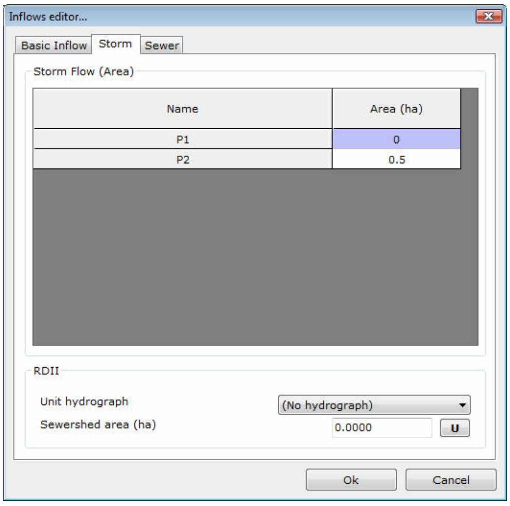
Dry weather

Constituent: select the constituent (**Flow** or one of the project's specified pollutants) whose dry weather inflow will be described..

Average value: specifies the average (or baseline) value of the dry weather inflow of the constituent in the relevant units (flow units for flow, concentration units for pollutants). Leave blank if there is no dry weather flow for the selected constituent.

Time pattern: Specifies the names of the time patterns to be used to allow the dry weather flow to vary in a periodic fashion by month of the year, by day of the week, and by time of day (for both weekdays and weekends). One can either type in a name or select a previously defined pattern from the drop down list of each combo box. Up to four different types of patterns can be assigned.

Storm



Storm flow (area)

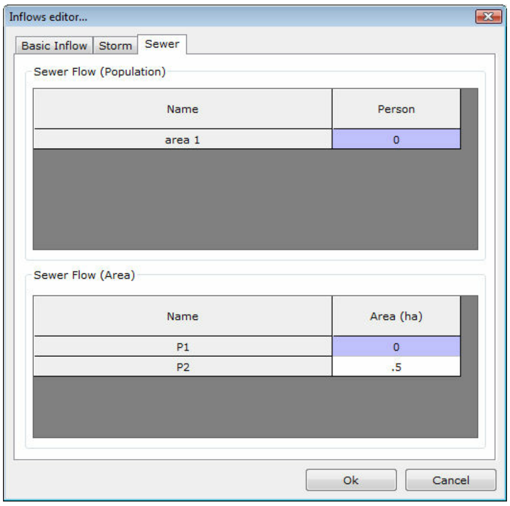
For each runoff area defined in the project, you can enter the real area which flows directly in the node. Areas are expected in hectares or acres depending on the unit system.

RDII

Unit hydrograph: select the name of the Unit Hydrograph group that applies to the node in question. The unit hydrographs in the group are used in combination with the group's assigned rain gage to develop a time series of RDII inflows per unit area over the period of the simulation. Leave this field blank to indicate that the node receives no RDII inflow.

Sewershed area (ac or ha): enter the area of the sewershed that contributes RDII to the node in question. Note this area will typically be only a small, localized portion of the subcatchment area that contributes surface runoff to the node.

Sewer



Sewer flow (population)

For each sewer area per person defined in the project, you can enter the number of persons of that particular area whose sewage end in the selected node.

Sewer flow (area)

For each sewer area per unit area defined in the project, you can enter the area which flows directly in the node. Areas are expected in hectares or acres depending on the unit system.

7.3.4 Node Treatment

Removal of pollutants from the flow streams entering any drainage system node is modeled by assigning a set of treatment functions to the node. A treatment function can be any well-formed mathematical expression involving

- the **pollutant concentration** of the mixture of all flow streams entering the node
- the **removals** of other pollutants
- any of the following process variables:
  - **FLOW** for flow rate into node (in user-defined flow units)
  - **DEPTH (ft or m)** for water depth above node invert.
  - **AREA (ft^2 or m^2)** for node surface area
  - **DT (sec)** for routing time step

- **HRT (h)** for hydraulic residence time

The result of the treatment function can be either a concentration (denoted by the letter C) or a fractional removal (denoted by R). For example, a first-order decay expression for BOD exiting from a storage node might be expressed as:

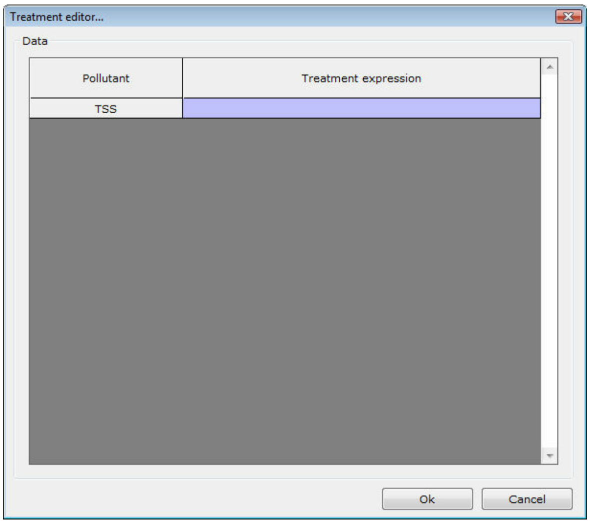
C = BOD * exp (-0.05*HRT)

or the removal of some trace pollutant that is proportional to the removal of total suspended solids (TSS) could be expressed as:

R = 0.75 * R_TSS

To enter treatment data:

1. Select the properties form of a node (junction, outfall, divider or storage).
2. Double click on the **Treatment** property.
3. Click the ellipsis button.
4. Make the appropriate changes.
5. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

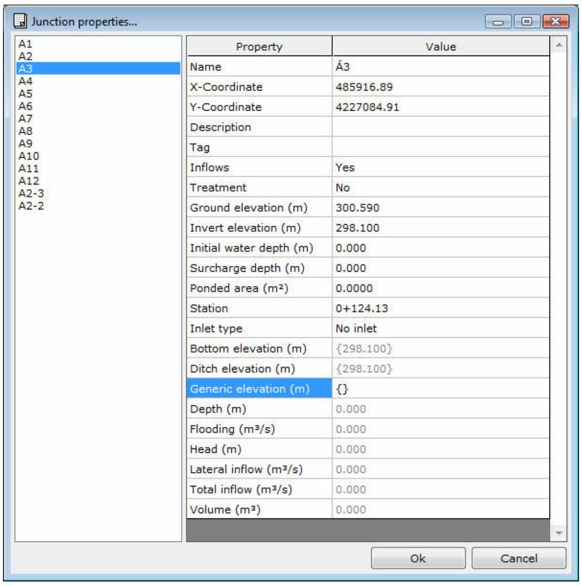


7.3.5 Junction

Junctions are drainage system nodes where links join together. Physically they can represent the confluence of natural surface channels, manholes in a sewer system, or pipe connection fittings. External inflows can enter the system at junctions. Excess water at a junction can become partially pressurized while connecting conduits are surcharged and can either be lost from the system or be allowed to pond atop the junction and subsequently drain back into the junction.

To view and modify the properties of junctions:

1. Select **Properties** from the **Objects** menu.
2. Select **Junction** from the **Properties** menu. The following form appears:



3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. Make the appropriate changes, as described below. The new property values are assigned to all selected objects in the list.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Properties

- **Name**: enter the name of the junction. Two or more junctions may share the same name, but this is not recommended since there will be confusion in the results.
- **X coordinate**: enter the X coordinate of the junction.
- **Y coordinate**: enter the Y coordinate of the junction.
- **Description**: enter the description of the junction.
- **Tag**: enter a tag for the junction. This appears neither in the input data nor the results.
- **Inflows:** click the ellipsis button to assign time series, dry weather, or RDII inflows to the junction.
- **Treatment:** click the ellipsis button to edit a set of treatment functions for pollutants entering the junction.
- **Ground elevation (ft or m):** enter the ground elevation of the junction. This field can be auto-filled from the program, if the junction belongs to a profile in which upstream and downstream ground elevations have been defined.
- **Invert elevation (ft or m):** enter the invert elevation of the junction.
- **Initial water depth (ft or m):** enter the depth of water at the junction at the start of the simulation.
- **Surcharge depth (ft or m):** enter the additional depth of water beyond the maximum depth that is allowed before the unction floods. This parameter can be used to simulate bolted manhole covers or force main connections.
- **Ponded area (ft^2 or m^2):** enter the area occupied by ponded water atop the

junction after flooding occurs. If the ponding option is turned on, a non-zero value of this parameter will allow ponded water to be stored and subsequently returned to the conveyance system when capacity exists.

- **Station:** click the ellipsis button to enter the station of the junction.
- **Inlet type:** select the inlet type for the junction from the drop-down list. The inlet types are entered as manhole specifications.
- **Bottom elevation (ft or m):** enter the bottom elevation of the junction.
- **Ditch elevation (ft or m):** enter the ground elevation of the junction.
- **Generic elevation (ft or m):** enter a generic elevation of the junction. This elevation appears in the profile drawing only.

Results

The results refer to the current time frame, selected from the drop-down list at the bottom-left corner of the main form.

- **Depth (ft or m):** the water depth in the junction.
- **Flooding:** the flow that is lost when inflows exceed storage and conveyance capacity, in user-defined flow units.
- **Head (ft or m):** the absolute elevation of hydraulic head
- **Lateral inflow:** the sum of runoff and all other inflows, in user-defined flow units .
- **Total inflow:** the sum of lateral inflow and upstream inflow, in user-defined flow units.
- **Volume (ft^3 or m^3):** the water volume held in storage (including ponded water).

7.3.6 Outfall

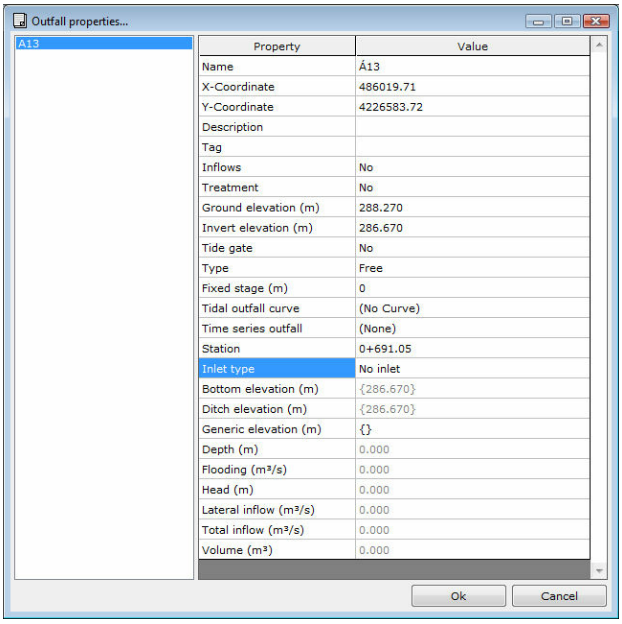
Outfalls are terminal nodes of the drainage system used to define final downstream boundaries under Dynamic Wave flow routing. For other types of flow routing they behave as a junction. Only a single link can be connected to an outfall node.

The boundary conditions at an outfall can be described by any one of the following stage relationships:

- the critical or normal flow depth in the connecting conduit
- a fixed stage elevation
- a tidal stage described in a curve of tide height versus hour of the day
- a user-defined time series of stage versus time.

To view and modify the properties of outfalls:

1. Select **Properties** from the **Objects** menu.
2. Select **Outfall** from the **Properties** menu. The following form appears:



3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. Make the appropriate changes, as described below. The new property values are assigned to all selected objects in the list.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Properties

- **Name**: enter the name of the outfall. Two or more outfalls may share the same name, but this is not recommended since there will be confusion in the results.
- **X coordinate**: enter the X coordinate of the outfall.
- **Y coordinate**: enter the Y coordinate of the outfall.
- **Description**: enter the description of the outfall.
- **Tag**: enter a tag for the outfall. This appears neither in the input data nor the results.
- **Inflows:** click the ellipsis button to assign time series, dry weather, or RDII inflows to the outfall.
- **Treatment:** click the ellipsis button to edit a set of treatment functions for pollutants entering the outfall.
- **Ground elevation (ft or m):** enter the ground elevation of the outfall. This field can be auto-filled from the program, if the outfall belongs to a profile in which upstream and downstream ground elevations have been defined.
- **Invert elevation (ft or m):** enter the invert elevation of the outfall.
- **Tide gate:** select **Yes** if tide gate is present to prevent backflow and **No** when no tide gate is present.
- **Type:** select one of the following types:
  - **Free**: outfall stage determined by minimum of critical flow depth and normal flow depth in the connecting conduit.

- ○ **Normal**: outfall stage based on normal flow depth in connecting conduit.
- ○ **Fixed**: outfall stage set to a fixed value.
- ○ **Tidal**: outfall stage given by a tidal curve of tide elevation versus time of day.
- ○ **Time series**: outfall stage supplied from a time series of elevations.

- **Fixed stage (ft or m):** enter the water elevation for a **Fixed** type of outfall.
- **Tidal outfall curve:** select the tidal curve relating water elevation to hour of the day for a **Tidal** outfall.
- **Time series outfall:** select the time series containing time history of outfall elevations for a **Time series** outfall.
- **Station:** click the ellipsis button to enter the station of the outfall.
- **Inlet type:** select the inlet type for the outfall from the drop-down list. The inlet types are entered as manhole specifications.
- **Bottom elevation (ft or m):** enter the bottom elevation of the outfall.
- **Ditch elevation (ft or m):** enter the ground elevation of the outfall.
- **Generic elevation (ft or m):** enter a generic elevation of the outfall. This elevation appears in the profile drawing only.

<u>**Results**</u>

The results refer to the current time frame, selected from the drop-down list at the bottom-left corner of the main form.

- **Depth (ft or m):** the water depth in the outfall.
- **Flooding:** the flow that is lost when inflows exceed storage and conveyance capacity, in user-defined flow units.
- **Head (ft or m):** the absolute elevation of hydraulic head
- **Lateral inflow:** the sum of runoff and all other inflows, in user-defined flow units .
- **Total inflow:** the sum of lateral inflow and upstream inflow, in user-defined flow units.
- **Volume (ft^3 or m^3):** the water volume held in storage (including ponded water).

7.3.7 Divider

Dividers are drainage system nodes that divert inflows to a specific conduit in a prescribed manner. A flow divider can have no more than two conduit links on its discharge side. Flow dividers are only active under Kinematic Wave routing and are treated as simple junctions under Dynamic Wave routing.

There are four types of flow dividers, defined by the manner in which inflows are diverted:

- **Cutoff**: diverts all inflow above a defined cutoff value.
- **Tabular:** uses a table that expresses diverted flow as a function of total inflow.
- **Overflow:** diverts all inflow above the flow capacity of the on-diverted conduit.
- **Weir:** uses a weir equation to compute diverted flow.

The flow diverted through a weir divider is computed by the following equation:

$$Q_{DIV} = C_W \left(f \cdot H_W \right)^{1.5}$$

where Q_{DIV} diverted flow

C_W weir coefficient
H_W weir height
f coefficient calculated as:

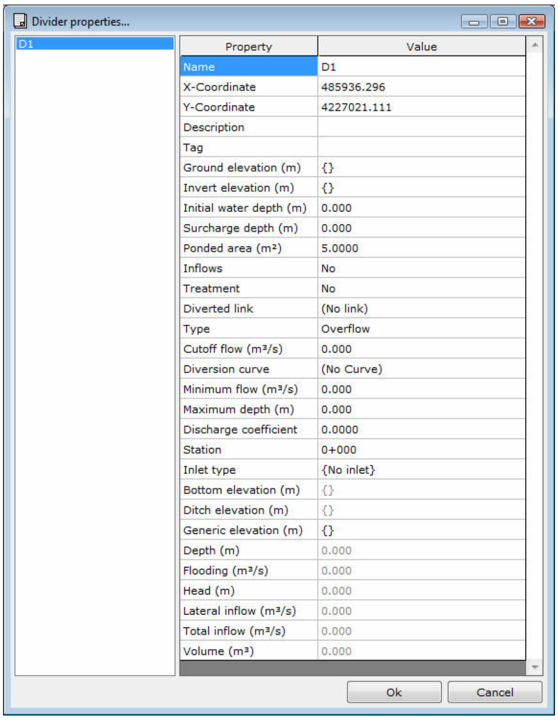
$$f = \frac{Q_{in} - Q_{min}}{Q_{max} - Q_{min}}$$

where Q_{IN} inflow to the divider
Q_{MIN} flow at which diversion begins
Q_{MAX} flow given by:

$$Q_{max} = C_W H_W^{1.5}$$

To view and modify the properties of dividers:

1. Select **Properties** from the **Objects** menu.
2. Select **Divider** from the **Properties** menu. The following form appears:



3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. Make the appropriate changes, as described below. The new property values are assigned to all selected objects in the list.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Properties

- **Name**: enter the name of the divider. Two or more dividers may share the same name, but this is not recommended since there will be confusion in the results.
- **X coordinate**: enter the X coordinate of the divider.
- **Y coordinate**: enter the Y coordinate of the divider.
- **Description**: enter the description of the divider.
- **Tag**: enter a tag for the divider. This appears neither in the input data nor the results.
- **Ground elevation (ft or m):** enter the ground elevation of the divider. This field can be auto-filled from the program, if the divider belongs to a profile in which upstream and downstream ground elevations have been defined.
- **Invert elevation (ft or m):** enter the invert elevation of the divider.
- **Initial water depth (ft or m):** enter the depth of water at the divider at the start of the simulation.
- **Surcharge depth (ft or m):** enter the additional depth of water beyond the maximum depth that is allowed before the divider floods. This parameter can be used to simulate bolted manhole covers.
- **Ponded area (ft^2 or m^2):** enter the area occupied by ponded water atop the divider after flooding occurs. If the ponding option is turned on, a non-zero value of this parameter will allow ponded water to be stored and subsequently returned to the conveyance system when capacity exists.
- **Inflows:** click the ellipsis button to assign time series, dry weather, or RDII inflows to the divider.
- **Treatment:** click the ellipsis button to edit a set of treatment functions for pollutants entering the divider.
- **Diverted link**: select the link which receives the diverted flow.
- **Type:** select one of the following types:
    - **Cutoff**: diverts all inflow above a defined cutoff value.
    - **Overflow**: diverts all inflow above the flow capacity of the on-diverted conduit.
    - **Tabular**: uses a table that expresses diverted flow as a function of total inflow.
    - **Weir**: uses a weir equation to compute diverted flow.
- **Cutoff flow**: enter the cutoff flow value used for a **Cutoff** divider, in user-defined flow units.
- **Diversion curve**: select the diversion curve used with a **Tabular** divider.
- **Minimum flow**: enter the minimum flow at which diversion begins for a **Weir** divider, in user-defined flow units.
- **Maximum depth (ft or m)**: enter the vertical height of **Weir** opening.
- **Discharge coefficient**: enter the product of **Weir's** coefficient and its length. Discharge coefficients are typically in the range of 2.65 to 3.10 per foot, for flows in CFS
- **Station:** click the ellipsis button to enter the station of the divider.
- **Inlet type:** select the inlet type for the divider from the drop-down list. The inlet types are entered as manhole specifications.
- **Bottom elevation (ft or m):** enter the bottom elevation of the divider.
- **Ditch elevation (ft or m):** enter the ground elevation of the divider.
- **Generic elevation (ft or m):** enter a generic elevation of the divider. This elevation appears in the profile drawing only.

Results

The results refer to the current time frame, selected from the drop-down

list at the bottom-left corner of the main form.

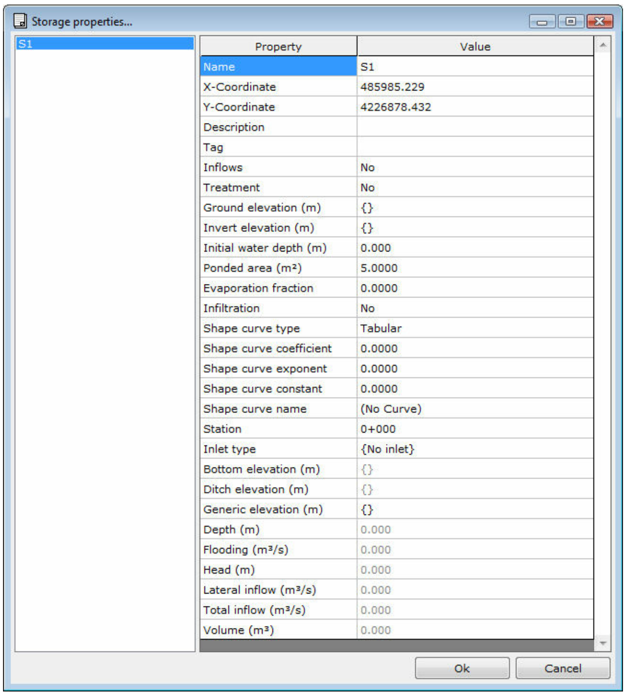
- **Depth (ft or m):** the water depth in the divider.
- **Flooding:** the flow that is lost when inflows exceed storage and conveyance capacity, in user-defined flow units.
- **Head (ft or m):** the absolute elevation of hydraulic head
- **Lateral inflow:** the sum of runoff and all other inflows, in user-defined flow units .
- **Total inflow:** the sum of lateral inflow and upstream inflow, in user-defined flow units.
- **Volume (ft^3 or m^3):** the water volume held in storage (including ponded water).

7.3.8 Storage

Storage Units are drainage system nodes that provide storage volume. Physically they could represent storage facilities as small as a catch basin or as large as a lake. The volumetric properties of a storage unit are described by a function or storage curve of surface area versus height.

To view and modify the properties of storages:

1. Select **Properties** from the **Objects** menu.
2. Select **Storage** from the **Properties** menu. The following form appears:



3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. Make the appropriate changes, as described below. The new property values are assigned to all selected objects in the list.

6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Properties

- **Name**: enter the name of the storage. Two or more storages may share the same name, but this is not recommended since there will be confusion in the results.
- **X coordinate**: enter the X coordinate of the storage.
- **Y coordinate**: enter the Y coordinate of the storage.
- **Description**: enter the description of the storage.
- **Tag**: enter a tag for the storage. This appears neither in the input data nor the results.
- **Inflows:** click the ellipsis button to assign time series, dry weather, or RDII inflows to the storage.
- **Treatment:** click the ellipsis button to edit a set of treatment functions for pollutants entering the storage.
- **Ground elevation (ft or m):** enter the ground elevation of the storage. This field can be auto-filled from the program, if the outfall belongs to a profile in which upstream and downstream ground elevations have been defined.
- **Invert elevation (ft or m):** enter the invert elevation of the storage.
- **Initial water depth (ft or m):** enter the depth of water at the storage at the start of the simulation.
- **Ponded area (ft^2 or m^2):** enter the surface area occupied by ponded water atop the storage unit once the water depth exceeds the maximum depth. If the ponding option is turned on, a non-zero value of this parameter will allow ponded water to be stored and subsequently returned to the conveyance system when capacity exists.
- **Evaporation fraction**: enter the fraction of the potential evaporation from the storage unit’s water surface that is actually realized.
- **Shape curve type**: select the type of curve describing the storage:
  - **Functional**: the following relation holds: Surface = A x $(\text{Depth})^B$ + C
  - **Tabular**: a storage curve is used.
- **Shape curve coefficient**: enter the value of A in the functional relationship between surface area and storage depth.
- **Shape curve exponent**: enter the value of B in the functional relationship between surface area and storage depth.
- **Shape curve constant**: enter the value of C in the functional relationship between surface area and storage depth.
- **Shape curve name**: select the storage curve containing the relationship between surface area and storage depth.
- **Station:** click the ellipsis button to enter the station of the storage.
- **Inlet type:** select the inlet type for the storage from the drop-down list. The inlet types are entered as manhole specifications.
- **Bottom elevation (ft or m):** enter the bottom elevation of the storage.
- **Ditch elevation (ft or m):** enter the ground elevation of the storage.
- **Generic elevation (ft or m):** enter a generic elevation of the storage. This elevation appears in the profile drawing only.

Results

The results refer to the current time frame, selected from the drop-down list at the bottom-left corner of the main form.

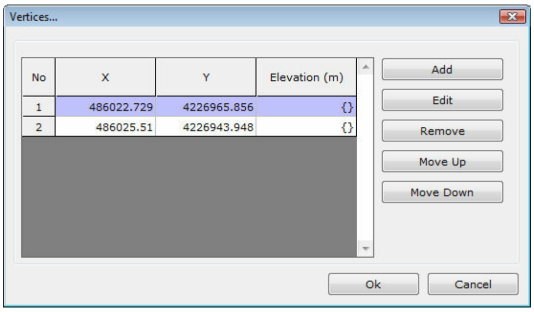
- **Depth (ft or m):** the water depth in the storage.
- **Flooding:** the flow that is lost when inflows exceed storage and conveyance capacity, in user-defined flow units.
- **Head (ft or m):** the absolute elevation of hydraulic head
- **Lateral inflow:** the sum of runoff and all other inflows, in user-defined flow units .
- **Total inflow:** the sum of lateral inflow and upstream inflow, in user-defined flow units.
- **Volume (ft^3 or m^3):** the water volume held in storage (including ponded water).

7.3.9 Link internal vertices

All links may include internal vertices. These are used to describe a more complex plan view, so that calculation of quantities and profile drawings are more precise. There is no restriction in the number of intermediate vertices.

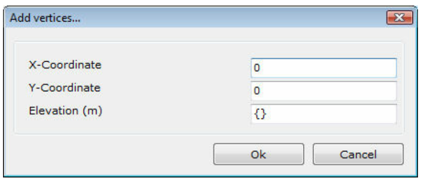
To manage the internal vertices, you can use the corresponding buttons of the toolbar or the internal vertices form:

1. Select the properties form of a link (conduit, pump, orifice, weir or outlet).
2. Double click on the **Internal vertices** property.
3. Click the ellipsis button.
4. Make the appropriate changes.
5. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



To add a new internal vertex:

1. Press **Add**. The following form appears:



2. Make the appropriate selections as described below.
3. Enter the **X-coordinate**, the **Y-coordinate**, and the **ground elevation** (in ft or m). Leaving the latter field empty activates auto-filling. If the specified link belongs to a profile in which upstream and downstream ground elevation data exist, the program calculates the elevation based on linear interpolation. In this case, the result appears

within curly braces "{}".
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To edit an existing internal vertex:

1. Select the vertex from the list on the left.
2. Press **Add**. The data form appears.
3. Make the appropriate selections as described below.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To delete an existing internal vertex:

1. Select the internal vertex from the list on the left.
2. Press **Remove**. The snow pack is deleted from the list.

To move an existing internal vertex upwards in the list:

1. Select the internal vertex from the list on the left.
2. Press **Move Up**.
3. The internal vertex is moved one place upwards.

To move an existing internal vertex downwards in the list:

1. Select the internal vertex from the list on the left.
2. Press **Move Down**.
3. The internal vertex is moved one place downwards.

7.3.10 Conduit

Conduits are pipes or channels that move water from one node to another in the conveyance system. Their cross-sectional shapes can be selected from a variety of standard open and closed geometries as listed in the following table.

Name	**Parameters**	**Name**	**Parameters**
Circular	Full height	Circular force main	Full height, Roughness
Filled Circular	Full height, Filled depth	Rectangular - Closed	Full height, Width
Rectangular - Open	Full height, Width	Trapezoidal	Full height, Base width, Side slopes
Triangular	Full height, Top width	Horizontal Ellipse	Full height, Maximum width
Vertical Ellipse	Full height, Max. width	Standardized Arch	Type no.
Arch	Full height, Max. width	Parabolic	Full height, Top width
Power	Full height, Top width, Exponent	Rectangular - Triangular	Full height, Top width, Triangle height

Rectangular - Round	Full height, Top width, Bottom radius	Modified basket handle	Full height, Top width, Top radius
Egg	Full height	Horseshoe	Full height
Gothic	Full height	Catenary	Full height
Semi-Elliptical	Full height	Basket handle	Full height
Semi-Circular	Full height	Irregular	Transect
Custom	Shape curve	Dummy	

Most open channels can be represented with a rectangular, trapezoidal, or user-defined irregular cross-section shape. For the latter, a transect object is used to define how depth varies with distance across the cross-section. The most common shapes for new drainage and sewer pipes are circular, elliptical, and arch pipes. They come in standard sizes that are published by the American Iron and Steel Institute in Modern Sewer Design and by the American Concrete Pipe Association in the Concrete Pipe Design Manual. The Filled Circular shape allows the bottom of a circular pipe to be filled with sediment and thus limit its flow capacity. The Custom Closed Shape allows any closed geometrical shape that is symmetrical about the center line to be defined by supplying a shape curve for the cross section.

The program uses the Manning equation to express the relationship between flow rate (Q), cross sectional area (E), hydraulic radius (R), and slope (J) in all conduits:

Metric system

$$Q = \frac{E}{n} R^{2/3} \sqrt{J}$$

English system

$$Q = \frac{1.49}{n} E R^{2/3} \sqrt{J}$$

where n is Manning's friction coefficient. The slope J is interpreted as either the conduit slope or the friction slope (i.e., head loss per unit length), depending on the flow routing method used. For pipes with Circular Force Main cross-sections either the Hazen-Williams or Darcy-Weisbach formula is used in place of the Manning equation for fully pressurized flow:

Darcy - Weisbach

$$Q = E\sqrt{\frac{8g}{f} RJ}$$

Hazen - Williams

Metric system

$$Q = 0.849 \cdot C \cdot E \cdot R^{0.63} J^{0.54}$$

English system

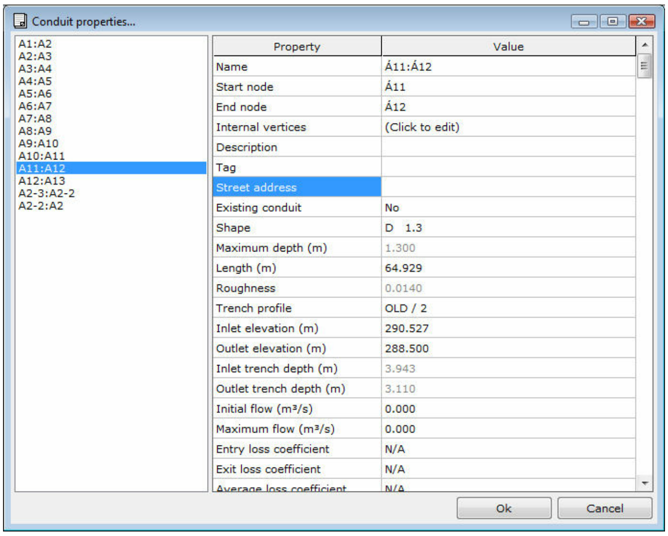
$$Q = 1.318 \cdot C \cdot E \cdot R^{0.63} J^{0.54}$$

where, g is the acceleration of gravity, f is Darcy - Weisbach friction coefficient and C is Hazen-Williams friction coefficient. For turbulent flow, the latter is determined from the height of the roughness elements on the walls of the pipe (supplied as an input parameter) and the flow’s Reynolds Number using the Colebrook-White equation.

A conduit does not have to be assigned a Force Main shape for it to pressurize. Any of the closed cross-section shapes can potentially pressurize and thus function as force mains that use the Manning equation to compute friction losses

To view and modify the properties of conduits:

1. Select **Properties** from the **Objects** menu.
2. Select **Conduit** from the **Properties** menu. The following form appears:



3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. Make the appropriate changes, as described below. The new property values are assigned to all selected objects in the list.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

<u>Properties</u>

- **Name**: enter the name of the storage. Two or more storages may share the same name, but this is not recommended since there will be confusion in the results.
- **Start Node**: select the start node (node, divider or storage) from the drop-down list.
- **End node**: select the end node (node, outfall, divider or storage) from the drop-down list.
- **Internal vertices**: click the ellipsis button to modify the internal vertices of the conduit.
- **Description**: enter the description of the conduit.
- **Tag**: enter a tag for the conduit. This appears neither in the input data nor the results.
- **Street address**: optionally, enter the street name of the conduit
- **Existing conduit**: select **Yes** if the conduit is existing (not necessarily old). The difference is that existing conduits are checked with different maximum capacity. The maximum capacities are defined in the checks.
- **Shape**: select the appropriate conduit shape. If none is selected, the program uses the default shape or the shape that was explicitly defined upstream in a profile that includes the conduit. In the latter case, the name of the conduit shape appears within curly braces.
- **Maximum depth (ft or m, read only)**: this is the maximum depth of the

selected conduit shape.

- **Length (ft or m):** leave this field empty to activate auto-fill and let the program calculate the length from the plan view. In this case, the result appears within curly braces "{}". Alternatively, specify an explicit value that will be used.
- **Roughness (read only)**: this is the conduit roughness, as entered in conduit shapes.
- **Trench profile:** select the trench profile from the drop-down list. T
- **Inlet elevation (ft or m):** leave this field empty to activate auto-fill and let the program calculate the correct elevation. This can be achieved if the conduit belongs to a profile in which enough data for automatic calculation of elevations exist. The calculation is based on user-defined design options and in this case the result appears within curly braces "{}". Alternatively, specify an explicit value that will be used.
- **Outlet elevation (ft or m):** leave this field empty to activate auto-fill and let the program calculate the correct elevation. This can be achieved if the conduit belongs to a profile in which enough data for automatic calculation of elevations exist. The calculation is based on user-defined design options and in this case the result appears within curly braces "{}". Alternatively, specify an explicit value that will be used.
- **Initial flow:** the initial flow rate in the conduit, in user-defined flow units.
- **Maximum flow:** enter the maximum flow allowed in the conduit under any routing method (as of version 7.0), in user-defined flow units. Use 0 if not applicable.
- **Entry loss coefficient**: enter the head loss coefficient associated with energy losses at the entrance of the conduit.
- **Exit loss coefficient**: enter the head loss coefficient associated with energy losses at the exit of the conduit.
- **Average loss coefficient**: enter the coefficient associated with energy losses along the length of the conduit.
- **Flap gate**: select **Yes** if a flap gate exists that prevents backflow through the conduit. Select **No** if no flap gate exists.
- **Culvert**: select a culvert type from the drop-down list.

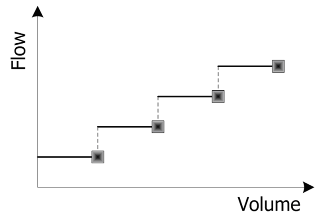
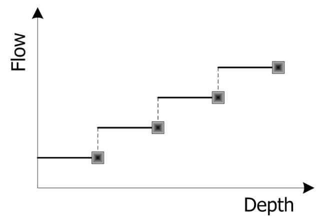
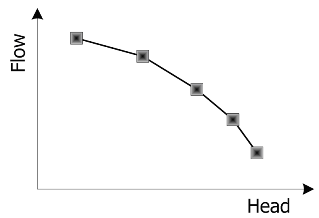
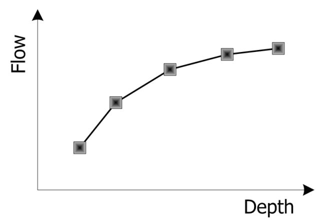
Results

The results refer to the current time frame, selected from the drop-down list at the bottom-left corner of the main form.

- **Slope:** the slope of the conduit.
- **Depth (ft or m):** the depth of the water in the conduit.
- **Froude No :** the Froude no of the flow in the conduit.
- **Capacity:** the capacity of the conduit.
- **Flow:** the flow rate in the conduit, in user-defined flow units.
- **Velocity (ft/s or m/s):** the flow velocity in the conduit.

7.3.11 Pump

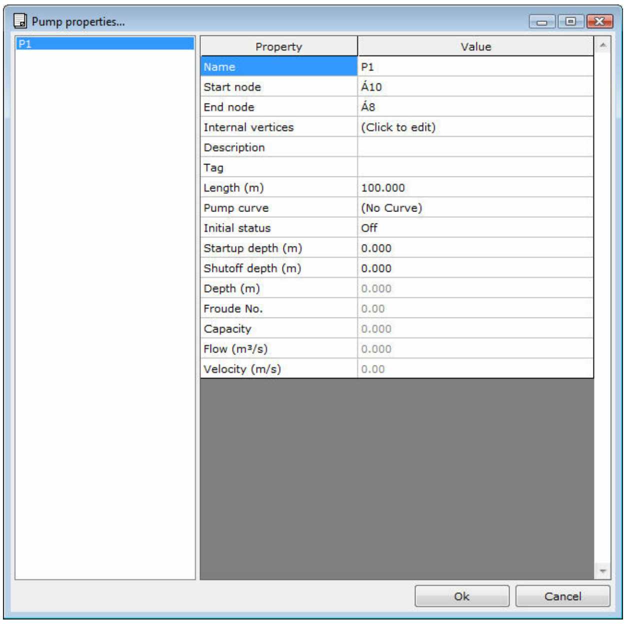
Pumps are links used to lift water to higher elevations. A pump curve describes the relation between a pump's flow rate and conditions at its inlet and outlet nodes. Four different types of pump curves are supported:

Type 1: An off-line pump with a wet well where flow increases incrementally with available wet well volume.	
Type 2: An in-line pump where flow increases incrementally with inlet node depth.	
Type 3: An in-line pump where flow varies continuously with head difference between the inlet and outlet nodes.	
Type 4: A variable speed in-line pump where flow varies continuously with inlet node depth.	
Ideal: An "ideal" transfer pump whose flow rate equals the inflow rate at its inlet node. No curve is required. The pump must be the only outflow link from its inlet node. Used mainly for preliminary design.	

The on/off status of pumps can be controlled dynamically by specifying startup and shutoff water depths at the inlet node or through user-defined control rules. Rules can also be used to simulate variable speed drives that modulate pump flow.

To view and modify the properties of pumps:

1. Select **Properties** from the **Objects** menu.
2. Select **Pump** from the **Properties** menu. The following form appears:



3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. Make the appropriate changes, as described below. The new property values are assigned to all selected objects in the list.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Properties

- **Name**: enter the name of the pump. Two or more pumps may share the same name, but this is not recommended since there will be confusion in the results.
- **Start node**: select the start node (node, divider or storage) from the drop-down list.
- **End node**: select the end node (node, outfall, divider or storage) from the drop-down list.
- **Internal vertices**: click the ellipsis button to modify the internal vertices of the pump.
- **Description**: enter the description of the pump.
- **Tag**: enter a tag for the pump. This appears neither in the input data nor the results.
- **Pump curve**: optionally, select an appropriate pump curve from the drop-down list. If no curve is selected, the pump is assumed to be ideal.
- **Initial status**: select the status of the pump (**On** or **Off**) at the beginning of the simulation.
- **Startup depth (ft or m)**: enter the depth at inlet node when pump turns on.
- **Shutoff depth (ft or m)**: enter the depth at inlet node when pump shuts off.

Results

The results refer to the current time frame, selected from the drop-down

list at the bottom-left corner of the main form.

- **Depth (ft or m):** the depth of the water in the pump.
- **Froude No :** the Froude no of the flow in the pump.
- **Capacity:** the capacity of the pump.
- **Flow:** the flow rate in the pump, in user-defined flow units.
- **Velocity (ft/s or m/s):** the flow velocity in the pump.

7.3.12 Orifice

Orifices are used to model outlet and diversion structures in drainage systems, which are typically openings in the wall of a manhole, storage facility, or control gate. They are internally represented in the program as a link connecting two nodes. An orifice can have either a circular or rectangular shape, be located either at the bottom or along the side of the upstream node, and have a flap gate to prevent backflow.

Orifices can be used as storage unit outlets under all types of flow routing. If not attached to a storage unit node, they can only be used in drainage networks that are analyzed with Dynamic Wave flow routing.

The flow through a fully submerged orifice is computed as:

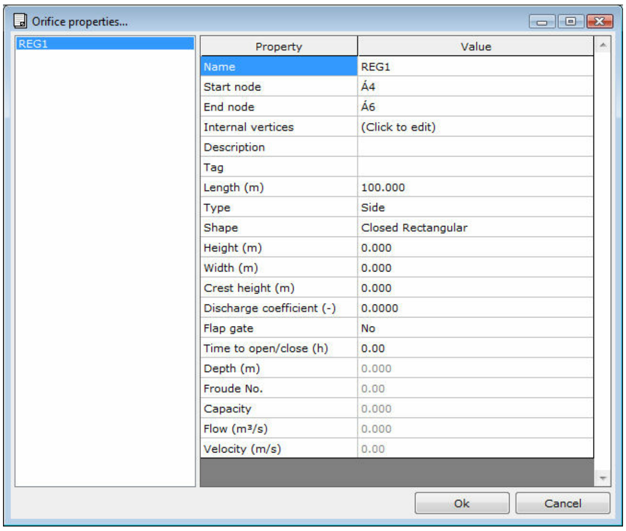
$$Q = CA\sqrt{2gh}$$

where	Q	flow rate
	C	discharge coefficient
	A	area of orifice opening
	g	acceleration or gravity
	h	head difference across the orifice

The height of an orifice's opening can be controlled dynamically through user-defined control rules. This feature can be used to model gate openings and closings.

To view and modify the properties of orifices:

1. Select **Properties** from the **Objects** menu.
2. Select **Orifice** from the **Properties** menu. The following form appears:



3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. Make the appropriate changes, as described below. The new property values are assigned to all selected objects in the list.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Properties

- **Name**: enter the name of the orifice. Two or more orifices may share the same name, but this is not recommended since there will be confusion in the results.
- **Start node**: select the start node (node, divider or storage) from the drop-down list.
- **End node**: select the end node (node, outfall, divider or storage) from the drop-down list.
- **Internal vertices**: click the ellipsis button to modify the internal vertices of the orifice.
- **Description**: enter the description of the orifice.
- **Tag**: enter a tag for the orifice. This appears neither in the input data nor the results.
- **Type**: select the type of the orifice (**Side** or **Bottom**)
- **Shape**: select the shape of the orifice (**Circular** or **Closed Rectangular**)
- **Height (ft or m)**: enter the height of orifice opening when fully open. Corresponds to the diameter of a circular orifice or the height of a rectangular orifice.
- **Width (ft or m)**: enter the width of rectangular orifice when fully opened.
- **Crest height (ft or m)**: enter the height of bottom of orifice above invert of inlet node.
- **Discharge coefficient**: enter the discharge coefficient. A typical value is 0.65.
- **Flap gate**: select **Yes** if a flap gate exists that prevents backflow through the orifice. Select **No** if no flap gate exists.
- **Time to open/close**: enter the time it takes to open a close (or close an open)

gate orifice in decimal hours. Use 0 or leave blank if timed openings/closings do not apply. Use control rules to adjust gate position.

Results

The results refer to the current time frame, selected from the drop-down list at the bottom-left corner of the main form.

- **Depth (ft or m):** the depth of the water in the orifice.
- **Froude No :** the Froude no of the flow in the orifice.
- **Capacity:** the capacity of the orifice.
- **Flow:** the flow rate in the orifice, in user-defined flow units.
- **Velocity (ft/s or m/s):** the flow velocity in the orifice.

7.3.13 Weir

Weirs, like orifices, are used to model outlet and diversion structures in a drainage system. Weirs are typically located in a manhole, along the side of a channel, or within a storage unit. They are internally represented in the program as a link connecting two nodes, where the weir itself is placed at the upstream node. A flap gate can be included to prevent backflow.

Four varieties of weirs are available, each incorporating a different formula for computing flow across the weir as listed in the following table:

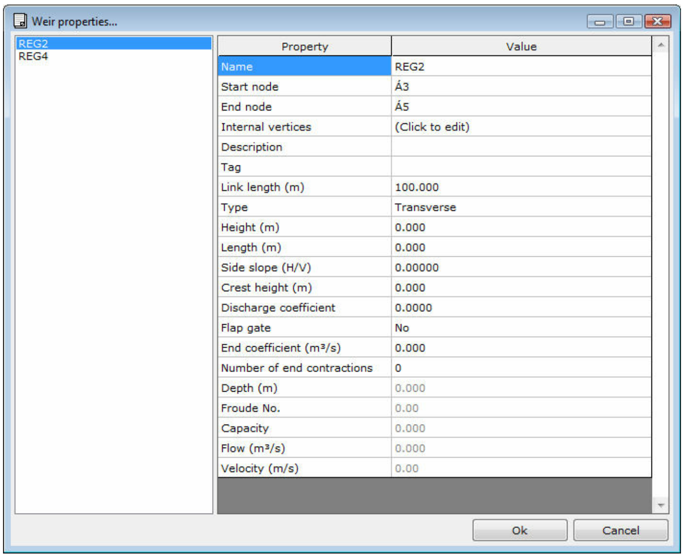
Weir type	Cross section shape	Flow formula
Transverse	Rectangular	$Q = C_W L h^{3/2}$
Side flow	Rectangular	$Q = C_W L h^{5/3}$
V-notch	Triangular	$Q = C_W S h^{5/2}$
Trapezoidal	Trapezoidal	$Q = C_W S h^{3/2} + C_{WS} S h^{5/2}$

Weirs can be used as storage unit outlets under all types of flow routing. If not attached to a storage unit, they can only be used in drainage networks that are analyzed with Dynamic Wave flow routing.

The height of the weir crest above the inlet node invert can be controlled dynamically through user-defined control rules. This feature can be used to model inflatable dams.

To view and modify the properties of weirs:

1. Select **Properties** from the **Objects** menu.
2. Select **Weir** from the **Properties** menu. The following form appears:



3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. Make the appropriate changes, as described below. The new property values are assigned to all selected objects in the list.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Properties

- **Name**: enter the name of the weir. Two or more weirs may share the same name, but this is not recommended since there will be confusion in the results.
- **Start node**: select the start node (node, divider or storage) from the drop-down list.
- **End node**: select the end node (node, outfall, divider or storage) from the drop-down list.
- **Internal vertices**: click the ellipsis button to modify the internal vertices of the weir.
- **Description**: enter the description of the weir.
- **Tag**: enter a tag for the weir. This appears neither in the input data nor the results.
- **Type**: select the type of the weir (**Transverse**, **Side flow**, **V-notch** or **Trapezoidal**).
- **Height (ft or m)**: enter the vertical height of weir opening.
- **Length (ft or m)**: enter the horizontal length of weir opening.
- **Side slope (H/V)**: enter the side slope (width to height) for **V-notch** or **Trapezoidal** weir.
- **Crest height (ft or m)**: enter the height of bottom of weir opening from invert of inlet node.
- **Discharge coefficient**: enter the discharge coefficient for flow through the central portion of the weir (for flow in CFS when using English units or CMS when using Metric units). Typical values are: 3.33 (English) 1.84 (Metric) for sharp crested transverse weirs, 2.5 - 3.3 (English) 1.38 - 1.83 (Metric) for broad

crested rectangular weirs, 2.4 - 2.8 (English) 1.35 - 1.55 (Metric) for V-notch triangular weirs.

- **Flap gate**: select **Yes** if a flap gate exists that prevents backflow through the orifice. Select **No** if no flap gate exists.
- **End coefficient**: enter the discharge coefficient for flow through the triangular ends of a **Trapezoidal** weir. See the recommended values for V-notch weirs listed above.
- **Number of end contractions**: enter the number of end contractions for a **Transverse** or **Trapezoidal** weir whose length is shorter than the channel it is placed in. Values will be either 0, 1, or 2 depending on if no ends, one end, or both ends are beveled in from the side walls.

Results

The results refer to the current time frame, selected from the drop-down list at the bottom-left corner of the main form.

- **Depth (ft or m):** the depth of the water in the weir.
- **Froude No :** the Froude no of the flow in the weir.
- **Capacity:** the capacity of the weir.
- **Flow:** the flow rate in the weir, in user-defined flow units.
- **Velocity (ft/s or m/s):** the flow velocity in the weir.

7.3.14 Outlet

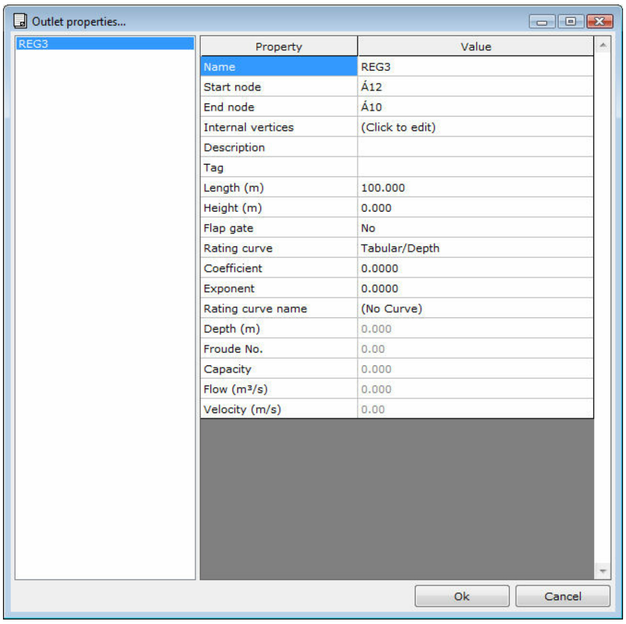
Outlets are flow control devices that are typically used to control outflows from storage units. They are used to model special head-discharge relationships that cannot be characterized by pumps, orifices, or weirs. Outlets are internally represented in the program as a link connecting two nodes. An outlet can also have a flap gate that restricts flow to only one direction.

Outlets attached to storage units are active under all types of flow routing. If not attached to a storage unit, they can only be used in drainage networks analyzed with Dynamic Wave flow routing.

A user-defined rating curve determines an outlet's discharge flow as a function of the head difference across it. Control rules can be used to dynamically adjust this flow when certain conditions exist.

To view and modify the properties of outlets:

1. Select **Properties** from the **Objects** menu.
2. Select **Outlet** from the **Properties** menu. The following form appears:



3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. Make the appropriate changes, as described below. The new property values are assigned to all selected objects in the list.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Properties

- **Name**: enter the name of the outlet. Two or more outlets may share the same name, but this is not recommended since there will be confusion in the results.
- **Start node**: select the start node (node, divider or storage) from the drop-down list.
- **End node**: select the end node (node, outfall, divider or storage) from the drop-down list.
- **Internal vertices**: click the ellipsis button to modify the internal vertices of the outlet.
- **Description**: enter the description of the outlet.
- **Tag**: enter a tag for the outlet. This appears neither in the input data nor the results.
- **Height (ft or m)**: enter the height of outlet above inlet node invert.
- **Flap gate**: select **Yes** if a flap gate exists that prevents backflow through the outlet. Select **No** if no flap gate exists.
- **Rating curve**: Method of defining flow (Q) as a function of freeboard depth or head (y) across the outlet
  - **FUNCTIONAL/DEPTH -** uses a power function $Q = Ay^B$ where y is the freeboard depth above the outlet's opening.
  - **FUNCTIONAL/HEAD -** uses a power function $Q = Ay^B$ where y is the head difference across the outlet.

- TABULAR/DEPTH - uses a tabulated curve of flow versus freeboard depth values.
- TABULAR/HEAD - uses a tabulated curve of flow versus head difference values.

- **Coefficient**: enter the coefficient A for the functional relationship between head and flow rate with **Functional** curve.
- **Exponent**: enter the coefficient B for the functional relationship between head and flow rate with **Functional** curve.
- **Rating curve name**: select the rating curve containing the relationship between head and flow rate with **Tabular** curve.

Results

The results refer to the current time frame, selected from the drop-down list at the bottom-left corner of the main form.

- **Depth (ft or m):** the depth of the water in the outlet.
- **Froude No :** the Froude no of the flow in the outlet.
- **Capacity:** the capacity of the outlet.
- **Flow:** the flow rate in the outlet, in user-defined flow units.
- **Velocity (ft/s or m/s):** the flow velocity in the outlet.

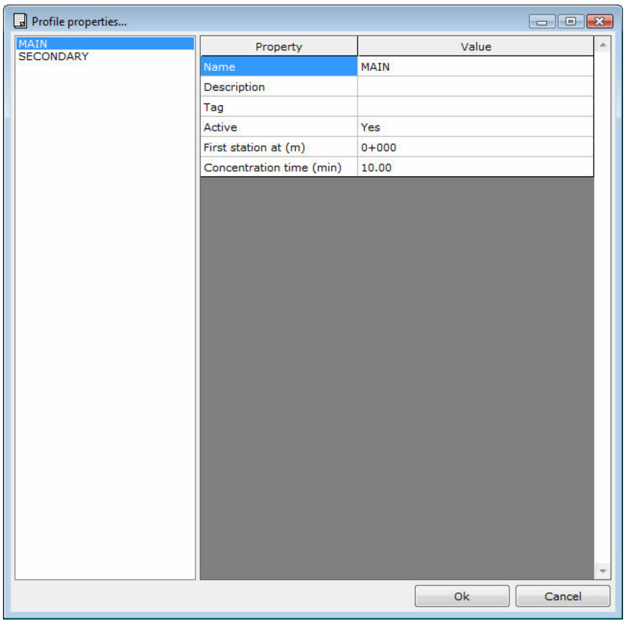
7.3.15 Profile

Profiles are not actual objects but rather a series of references to alternating nodes and links. Data input becomes very easy using profiles since:

- data are input in tabular form,
- the elevations are displayed in a visual and comprehensive way,
- the creation of profile DXF drawings is easy,
- a network may be defined without plan view data.

To view and modify the properties of profiles:

1. Select **Properties** from the **Objects** menu.
2. Select **Profile** from the **Properties** menu. The following form appears:



3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. Make the appropriate changes, as described below. The new property values are assigned to all selected objects in the list.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Properties

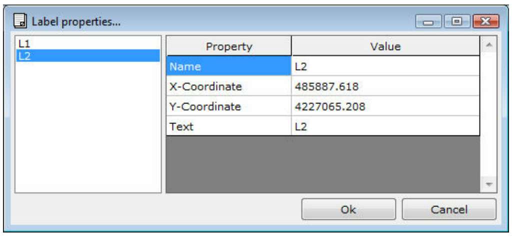
- **Name**: enter the name of the profile. Two or more profiles may share the same name, but this is not recommended since there will be confusion in the results.
- **Description**: enter the description of the profile.
- **Tag**: enter a tag for the profile. This appears neither in the input data nor the results.
- **Active**: select whether the profile will appear in the profile drawing. This setting can be changes using the profile drawing form.
- **First station**: enter the first station of the profile. This option is used with the semi-automatic data input of profile stations.
- **Concentration time (min)**: enter the concentration time. This field is needed only when an IDF curve is used.

7.3.16 Label

Labels are optional text objects that can be used to enrich the plan view with important data.

To view and modify the properties of labels:

1. Select **Properties** from the **Objects** menu.
2. Select **Label** from the **Properties** menu. The following form appears:



3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. Make the appropriate changes, as described below. The new property values are assigned to all selected objects in the list.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Properties

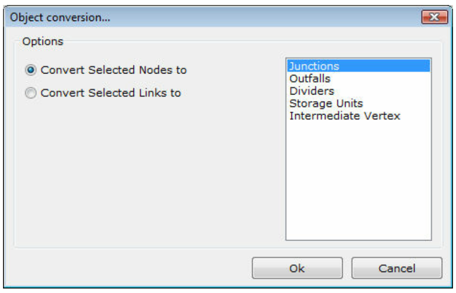
- **Name**: enter the name of the label (not the text).
- **X-coordinate**: enter the X-coordinate of the label.
- **Y-coordinate**: enter the Y-coordinate of the label.
- **Text**: enter the text that will be displayed.

7.4 Object conversion

With this option, you can convert objects from one type to another.

To convert objects from one type to another:

1. Select (in plan view) the objects you wish to convert.
2. Select **Object conversion** from the **Objects** menu. The following form appears:



3. Select if you wish to convert nodes (point objects) or links (linear objects).
4. Select the target object type from the list on the right.
5. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

NOTE: Non-common properties are lost during object conversion.

NOTE: To convert a vertex back to a junction, see the convert vertex to junction function.

7.5 Add vertex

With this option, you can add an intermediate vertex to an existing link: conduit, pump, orifice, weir, outlet. This procedure is described also in the section of link internal vertices.

To add an intermediate vertex to an existing link:

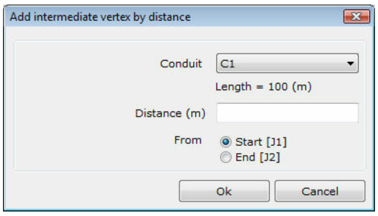
1. Select **Add vertex** from the **Objects** menu.
2. Click on the link to add a vertex.

7.6 Add vertex by distance

With this option, you add a vertex on a conduit, by specifying a distance from either end of the conduit. For example, if the conduit has a length equal to 20 m and you need to add a vertex that be located 5 m from its end (station 0+015) then select its ending node and enter 5 or its starting node and enter 15.

To add a vertex by distance:

1. Select **Add Vertex By Distance** from the **Objects** menu.
2. Select the **conduit** from the drop-down list. Its length appears on the form.



3. Enter the desired distance from either starting or ending node.
4. Select whether this distance is measured from the starting or ending node.
5. Select **Ok** to add a vertex on the specified conduit. Select **Cancel** to close the dialog box without applying any changes.

7.7 Delete vertex

With this option, you can delete an intermediate vertex of an existing link: conduit, pump, orifice, weir, outlet. This procedure is also described in the section of link internal vertices.

To delete an intermediate vertex of an existing link:

1. Select **Delete vertex** from the **Objects** menu.
2. Click on the link vertex to delete it.

7.8 Stretch vertex

With this option, you can move an intermediate vertex of an existing link: conduit, pump, orifice, weir, outlet. This procedure is also described in the section of link internal vertices.

To delete an intermediate vertex to an existing link:

1. Select **Stretch vertex** from the **Objects** menu.
2. Click on the link vertex you wish to move.
3. Click again to define the new vertex position.

7.9 Convert vertex to junction

To convert one or more vertices back to junctions, one must you this function instead of the generic conversion form.

To convert one vertex to a junction:

1. From the **Objects** menu, click on **Convert Vertex To Junction**.
2. On the map, click on the vertex you wish to convert back to a junction.
3. The vertex is converted.

To convert multiple vertices to junctions:

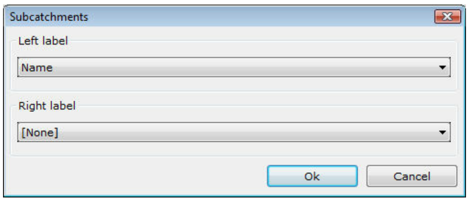
1. From the **Objects** menu, click on **Convert Vertex To Junction**.
2. Hold **CTRL** down and click on the map the vertex you wish to convert back to a junction.
3. Repeat step 2 until all vertices are converted.

7.10 Labels

With this option, you can select which object properties will be displayed. Up to two properties can be displayed simultaneously, one on the left label and one on the right.

To select which object properties will be displayed:

1. Select **Labels** from the **Objects** menu.
2. Select the object type from the **Labels** menu. The following form appears:



3. Select the property that will be displayed on the left.
4. Select the property that will be displayed on the right.
5. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

7.11 Swap link ends

All links have a certain direction. With this option, you can swap the ends of the selected links and invert their direction.

To swap the ends of the selected links:

1. Select the links in the drawing.
2. Select **Swap link ends** from the **Objects** menu. The ends are swapped.

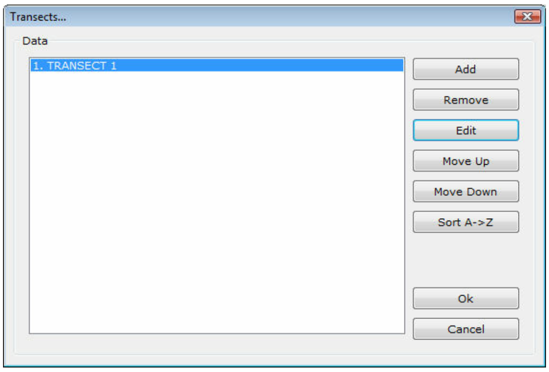
7.12 Transects

7.12.1 Management

Transects refer to the geometric data that describe how bottom elevation varies with horizontal distance over the cross section of a natural channel or irregular-shaped conduit.

To manage transects:

1. Select **Transects** from the **Objects** menu. The following form appears:



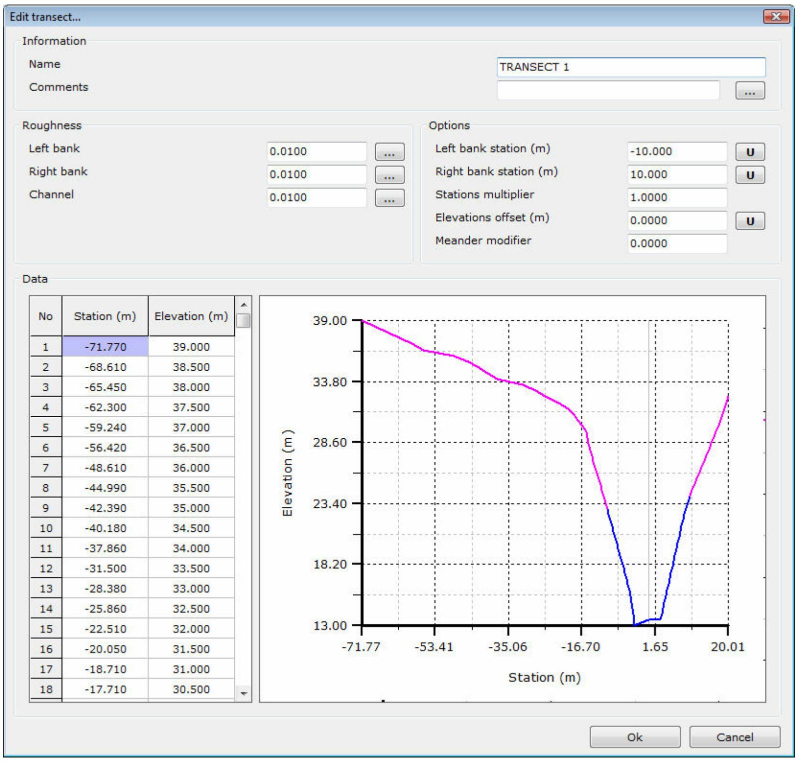
2. Make the necessary modifications.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

7.12.2 Add

With this option, you can add a new transect.

To add a new transect:

1. Press **Add**. The data form appears:



2. Enter a user-assigned **name**. This name cannot be null or used for another transect in the project.
3. Optionally, add some **comments** to the curve. Press the ellipsis button to edit multiline text.
4. Enter the Manning friction coefficient for the left bank.
5. Enter the Manning friction coefficient for the right bank.
6. Enter the Manning friction coefficient for the channel.
7. Enter the **station** of the left bank in ft or m. If no left bank exists, enter the station of the first point of the transect.
8. Enter the **station** of the right bank in ft or m. If no right bank exists, enter the station of the last point of the transect.
9. Enter the **stations multiplier** which will be applied to the stations specified. Enter 1.0 if you wish to disable this feature.
10. Enter the **elevation offset** in ft or m which will be added to the elevations specified.
11. Type the data in the data list.
12. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

7.12.3 Delete

To delete an existing transect:

1. Select the transect from the list on the left.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The transect is deleted from the list.

7.12.4 Edit

To edit an existing transect:

1. Select the transect from the list on the left.
2. Press **Edit**. The data form appears.
3. Make the appropriate selections as described in the add transect topic.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

7.12.5 Move

To move an existing transect upwards in the list:

1. Select the transect from the list on the left.
2. Press **Move Up**.
3. The transect is moved one place upwards.

To move an existing transect downwards in the list:

1. Select the transect from the list on the left.
2. Press **Move Down**.
3. The transect is moved one place downwards.

7.12.6 Sort

To sort the transect list:

1. Press **Sort A->Z**.
2. The list is sorted alphabetically.

7.13 Control rules

7.13.1 Management

Control Rules determine how pumps and regulators (orifices, weirs and outlets) in the drainage system will be adjusted over the course of a simulation.

Each control rule is a series of statements of the form:

RULE ruleID

IF condition_1
AND condition_2
OR condition_3
AND condition_4
Etc.

THEN action_1
AND action_2
Etc.

ELSE action_3
AND action_4
Etc.

PRIORITY value

Only the **RULE**, **IF** and **THEN** portions of a rule are required; the **ELSE** and **PRIORITY** portions are optional.

When mixing **AND** and **OR** clauses, the **OR** operator has higher precedence than **AND**, i.e.,

IF A **OR** B **AND** C

is equivalent to:

IF (A **OR** B) **AND** C.

If the interpretation was meant to be

IF A **OR** (B **AND** C)

then this can be expressed using two rules as in

IF A **THEN** ...
IF B **AND** C **THEN** ...

The **PRIORITY** value is used to determine which rule applies when two or more rules require that conflicting actions be taken on a link. A rule without a priority value always has a lower priority than one with a value. For two rules with the same priority value, the rule that appears first is given the higher priority.

<u>Condition clauses</u>

A condition clause of a control rule has the following format:

object	**id**	**attribute**	**relation**	**value**

where	**object**	object type
	id	object name
	attribute	attribute or property of the object
	relation	relational operator (=, <>, <, <=, =>, >)
	value	property value

Some examples of condition clauses are:

NODE	N1	DEPTH	>	10
PUMP	A1	STATUS	=	OFF
SIMULATION	CLOCKTIME		=	22:45:00

The objects and attributes that can appear in a condition clause are as follows:

Object	Property	Value
Node	Depth Head Inflow	numerical value numerical value numerical value

Link	Flow Depth	numerical value numerical value
Pump	Status Flow Setting	On/Off numerical value modulated control
Orifice	Setting	fraction open or modulated control
Weir	Setting	fraction open or modulated control
Simulation	Time Date Clocktime (00:00:00) Month Day	elapsed time in decimal hours or hr: min:sec month/day/year time of day in hr:min:sec number of month (1-12), 1 being January number of day (1-7), 1 being Sunday

Action clauses

An action clause of a control rule can have one of the following formats:

- PUMP id STATUS = ON/OFF
- PUMP/ORIFICE/WEIR/OUTLET id SETTING = value

where the meaning of SETTING depends on the object being controlled:

- for pumps it is a multiplier applied to the flow computed from the pump curve.
- for orifices it is the fractional amount that the orifice is fully open.
- for weirs it is the fractional amount of the original freeboard that exists (i.e., weir control is accomplished by moving the crest height up or down).
- for outlets it is a multiplier applied to the flow computed from the outlet's rating curve.

Some examples of action clauses are:

PUMP P67 STATUS = OFF
ORIFICE O212 SETTING = 0.5

Modulated controls

Modulated controls are control rules that provide for a continuous degree of control applied to a pump or flow regulator as determined by the value of some controller variable, such as water depth at a node, or by time. The functional relation between the control setting and the controller variable is specified by using a control curve, a timeseries or a PID controller (see below). Some examples of modulated control rules are:

RULE MC1
IF NODE N2 DEPTH >= 0
THEN WEIR W25 SETTING = CURVE C25

RULE MC2
IF SIMULATION TIME > 0
THEN PUMP P12 SETTING = TIMESERIES TS101

RULE MC3
IF LINK L33 FLOW <> 1.6
THEN ORIFICE O12 SETTING = PID 0.1 0.0 0.0

Note how a modified form of the action clause is used to specify the name of the control curve, time series or PID parameter set (Gain, Integral, and Derivative coefficients) that defines the degree of control. Also, by convention the controller variable used in a Control Curve or PID Controller will always be the object and attribute named in the last condition clause of the rule. As an example, in rule MC1 above Curve C25 would define how the fractional setting at Weir W25 varied with the water depth at Node N2. In rule MC3, the PID controller adjusts the opening of Orifice O12 to maintain a flow of 1.6 in Link L33.

PID Controller

A PID (Proportional-Integral-Derivative) Controller is a generic closed-loop control scheme that tries to maintain a desired set-point on some process variable by computing and applying a corrective action that adjusts the process accordingly. In the context of a hydraulic conveyance system a PID controller might be used to adjust the opening on a gated orifice to maintain a target flow rate in a specific conduit or to adjust a variable speed pump to maintain a desired depth in a storage unit. The classical PID controller has the form:

$$m(t) = K_p\left[e(t) + \frac{1}{T_i}\int e(\tau)\,d\tau + T_d\frac{de(t)}{dt}\right]$$

m(t) controller output
K_p proportional coefficient (gain)
T_i integral time (minutes)
T_d derivative time (minutes)
e(t) error (difference between setpoint and observed variable value)
t time

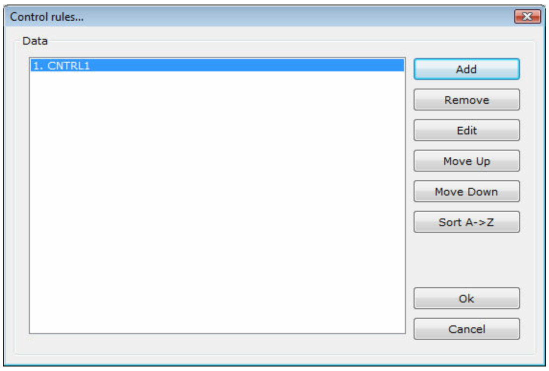
The performance of a PID controller is determined by the values assigned to the coefficients K_p, T_i, and T_d. The controller output *m(t)* has the same meaning as a link setting used in a rule's Action Clause while *dt* is the current flow routing time step in seconds. The error *e(t)* is the difference between the control variable setpoint *x** and its value at time *t*, *x(t)*, normalized to the setpoint value:

$$e(t) = \frac{x^* - x(t)}{x^*}$$

Note that for direct action control, where an increase in the link setting causes an increase in the controlled variable, the sign of *Kp* must be positive. For reverse action control, where the controlled variable decreases as the link setting increases, the sign of *Kp* must be negative. The user must recognize whether the control is direct or reverse action and use the proper sign on *Kp* accordingly. For example, adjusting an orifice opening to maintain a desired downstream flow is direct action. Adjusting it to maintain a downstream water level is reverse action while adjusting it to maintain an upstream water level is direct action. Controlling a pump to maintain a fixed wet well water level would be reverse action while using it to maintain a fixed downstream flow is direct action.

To manage control rules:

1. Select **Control rules** from the **Objects** menu. The following form appears:



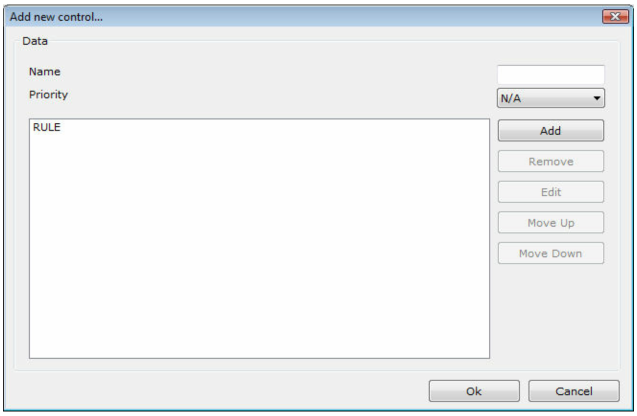
2. Make the necessary modifications.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

7.13.2 Add

With this option, you can add a new rule.

To add a new rule:

1. Press **Add**. The data form appears:

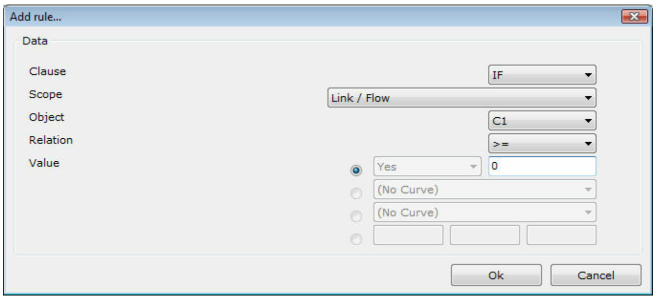


2. Enter a user-assigned **name**. This name cannot be null or used for another control rule in the project.
3. Optionally, you can assign a priority level for the rule.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the

dialog box without saving any changes.

To enter a new logical statement to the rule:

1. Press **Add** to enter a new logical statement to the rule. The following form appears:



2. Select the **Clause**, **Scope**, **Object**, allowed **Relation** and **Value**. Value may be one of the following:

- Option 1: a boolean (YES/NO) or numeric value
- Option 2: a control curve, if one or more control curves are present
- Option 3: a time series, if one or more time series are present
- Option 4: a PID controller. Enter values for gain, integral and derivative coefficient respectively.

Not all options are available for a selected scope. If more than one option is available, the user must specify which one will be used by activating the radio button on the left of the option.

3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes

To delete an existing logical statement of a rule:

1. Select the statement from the list.
2. Select **Delete**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. If you select **Yes**, the statement is deleted.

To modify an existing logical statement of a rule:

1. Select the statement from the list.
2. Select **Edit**.
3. Make the appropriate changes.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

To modify the position of an existing logical statement of a rule:

1. Select the statement from the list.
2. Select **Move Up** to move the statement up by one row. Select **Move Down** to move the statement down by one row.

7.13.3 Delete

To delete an existing control rule:

1. Select the control rule from the list on the left.
2. Press **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The control rule is deleted from the list.

7.13.4 Edit

To edit an existing control rule:

1. Select the control rule from the list on the left.
2. Press **Edit**. The data form appears.
3. Make the appropriate selections as described in the add control rule topic.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

7.13.5 Move

To move an existing control rule upwards in the list:

1. Select the control rule from the list on the left.
2. Press **Move Up**.
3. The control rule is moved one place upwards.

To move an existing control rule downwards in the list:

1. Select the control rule from the list on the left.
2. Press **Move Down**.
3. The control rule is moved one place downwards.

7.13.6 Sort

To sort the control rule list:

1. Press **Sort A->Z**.
2. The list is sorted alphabetically.

Chapter VIII

8 Profiles

8.1 Profiles menu

With this menu, you can perform various operations regarding profiles and network design. These are available in case at least one profile has been defined. In the **Profiles** menu you can select one of the following options:

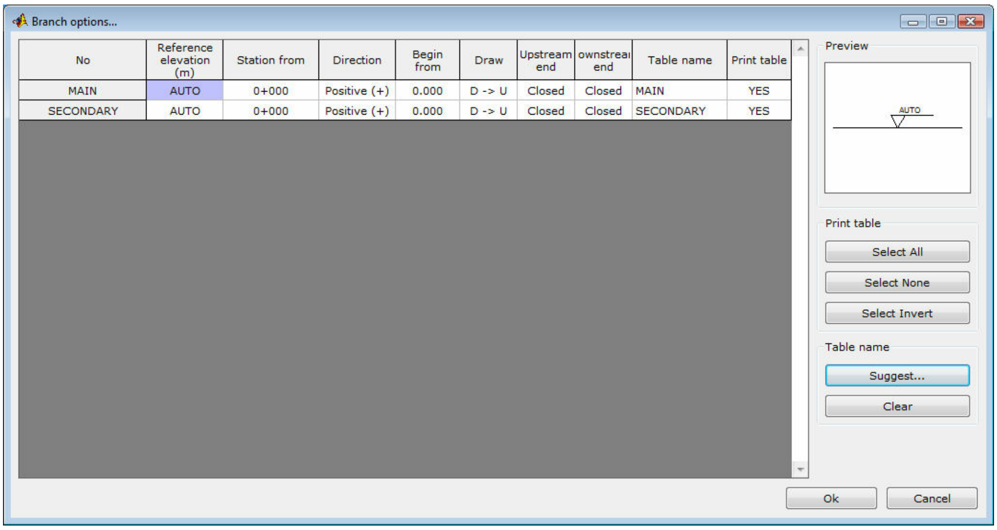
- Profile options
- Renaming of profiles nodes
- Elevation calculations
- Automated branch design
- Force constant slope
- Uniform inflow
- Stations
- Inlets
- Street addresses
- Vertical street addresses
- Other junctions
- Special devices

8.2 Profile options

With this option, you can modify the way the branches are displayed.

To modify the way the branches are displayed:

1. Select **Profile options** from the **Profiles** menu. The following form will appear:



2. Make the appropriate changes as described below. By clicking on each column, a sketch of the setting is displayed on the right.

- **Reference elevation**: Enter the reference elevation in m. However, it is recommended that you leave this field empty, in which case a label "AUTO" will appear and the reference elevation will be automatically calculated by the program.

- **Station from**: enter the initial station if this is not zero. This affects only the drawing of the profiles; it is not related with the stations of the data table.
- **Direction**: enter one of **positive**, **negative** if you want the stations to be increasing or decreasing, respectively.
- **Begin from:** enter the distance from the first station in m. Usually this value will coincide the **Station from** field, but it will be displayed in a different row in the profile.
- **Draw:** select one of **D->U, U->D** if you want the profile to be drawn downstream to upstream or vice versa, respectively. This affects the profile drawing; in the data table of the main form and the profile sketch, the branch is drawn upstream to downstream.
- **Upstream end**: select one of **open, closed** if you want the end to be drawn open or closed, respectively, in the profile drawing.
- **Downstream end:** select one of **open, closed** if you want the end to be drawn open or closed, respectively, in the profile drawing.
- **Table name**: enter the title of the branch. This is optional and it will be printed above the corresponding table in the profile drawing. When editing this value, the buttons **Suggest** and **Clear** of the **Title** frame become enabled. The former sets the name equal to the title of the branch; the latter clears all table names.
- **Print table**: select one of **Yes, No** if you want a table with row descriptions to be included for the specific branch in the profile drawing. When editing this value, the buttons **Select all, Select None** and **Select Invert** of the **Table** frame become enabled. With these, you can set **Yes** or **No** to all branches with one click.

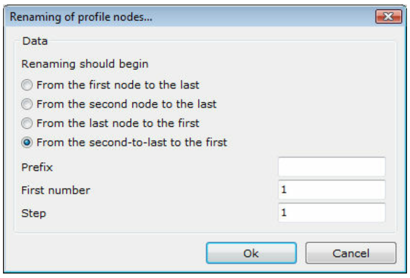
3. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

8.3 Renaming of profiles nodes

Since nodes have various names and a profile usually consists of several nodes, this tool can be used to rename all nodes belonging to a particular profile. The rename procedure can create chained names of the nodes, making it easy for the engineer or the reviewer to navigate through a profile. For example, a profile consisting of nodes KP1, N2, N5, N7 and Z1 would be KP1-N2-N5-N7-Z1, while after renaming it could be J1-J2-J3-J4-J5.

To rename the nodes of the selected profile:

1. Select the profile in the profile list.
2. From the **Tools** menu select **Renaming Of Profile Nodes**.
3. Select where renaming begins (first to last node, second to last node, and vice versa)
4. Optionally enter a **prefix**, i.e. J.
5. Enter the **first** number.
6. Enter the **step**. For example if the first number is 2 and the step is 3, then nodes will be renamed as J2, J5, J8, etc.
7. Press **Ok** to rename the nodes as described above or **Cancel** to close the form and ignore all changes.



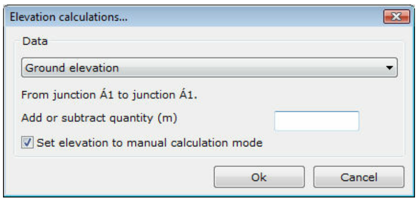
NOTE: You can easily work through an entire network. Suppose you have two profiles, J1 to J10 and P1 to P8. The second profile intersects the first at junction J5. Rename the second profile use J5- as the prefix, first number 1 and step 1. It will be renamed as J5-1 to J5-8.

8.4 Elevation calculations

With this option, you can add or subtract a value to or from a specified elevation.

To add or subtract a value to or from a specified elevation:

1. Select the branch from the list of the main form.
2. Select two or more rows in the data table (when viewing the profile) by clicking and dragging the mouse. The first and last row signify the first and last station, respectively, of the part that the calculations will be applied.
3. Select **Elevation calculations** from the **Profiles** menu. The following form will appear:



4. The form displays the first and last station.
5. Select the type of elevation that will be modified from the drop-down list. The available options are:

- **Ground elevation**
- **Upstream bottom elevation**
- **Downstream bottom elevation**
- **Trench elevation**
- **Custom elevation**
- **Manhole bottom elevation**

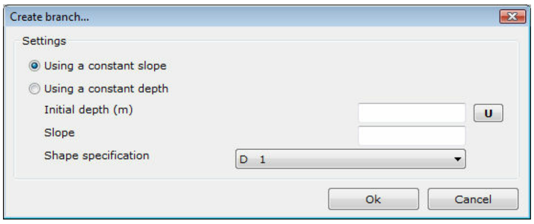
6. Enter the value to be added or subtracted by typing into the corresponding text box.
7. If some values are calculated automatically by the program using linear interpolation then, after this procedure, these will be recalculated, thus canceling the effect of the above modifications. Check **Convert automatically computed data** to fix the new values and prevent the program from recalculating them.
8. Select **Ok** to proceed with the operation and close the dialog box. Select **Cancel** to close the dialog box and cancel the operation.

8.5 Automated branch design

With this option, you can design a part of a network based on constant slope of the bottom elevation or constant depth from the ground elevation.

To design a part of a network:

1. Select the profile from the list of the main form.
2. Select **Automated branch design** from the **Profiles** menu. The following form will appear:



3. Select one of **constant slope** or **constant depth**.
4. If you select **constant slope,** enter the **initial depth** of the first inlet and the **constant slope** by typing in the corresponding text boxes.
5. If you select **constant depth,** enter the **depth** (ft or m) by typing in the corresponding text box.
6. In both cases, click the button with the ellipses (...) to select the **section** that will be used for the pipes.
7. Select **Ok** to proceed with the operation and close the dialog box. Select **Cancel** to close the dialog box and cancel the operation.

8.6 Force constant slope

With this option, you can apply a uniform slope to a specified part of a profile.

To apply a uniform slope to a specified part of a profile:

1. Select the profile from the list of the main form.
2. Select two or more rows in the data table (when viewing the profile) by clicking and dragging the mouse. The first and last row signify the first and last station, respectively, of the part that the uniform slope will be applied.
3. Select **Force constant slope** from the **Profiles** menu. The following form will appear:

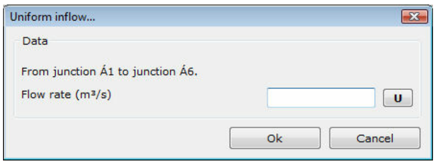
4. The form displays the first and last station.
5. Enter the uniform slope by typing in the corresponding text box.
6. Select **Ok** to proceed with the operation and close the dialog box. Select **Cancel** to close the dialog box and cancel the operation.

8.7 Uniform inflow

With this option, you can apply a flow rate uniformly along a part of a profile.

To apply a flow rate uniformly along a part of a profile:

1. Select the profile from the list of the main form.
2. Select two or more rows in the data table (when viewing the profile) by clicking and dragging the mouse. The first and last row signify the first and last station, respectively, of the part that the flow rate will be applied uniformly.
3. Select **Uniform flow** from the **Profiles** menu. The following form will appear:



4. The form displays the first and last station.
5. Enter the inflow value in user selected flow units by typing in the corresponding text box.
6. Select **Ok** to proceed with the operation and close the dialog box. Select **Cancel** to close the dialog box and cancel the operation.

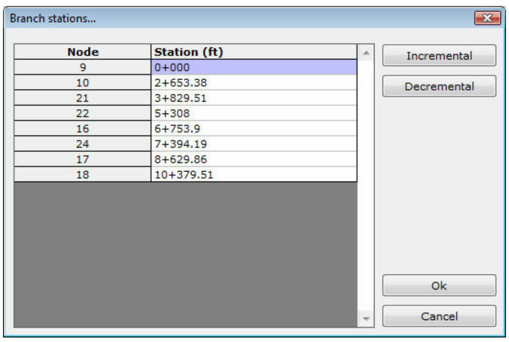
NOTE: If the last inlet is selected then it is ignored because is does not affect the flow rate of the branch. The inflow is distributed uniformly, taking into account the length of the segments between the stations.

8.8 Stations

With this option, you can enter station data for the selected profile. This option is very helpful if you want to enter the data of all stations, especially if these are uniformly spaced (e.g. every 20m). Alternatively, the station data can be input using the data table of the main form.

To enter station data for the selected profile:

1. Select the profile from the list.
2. Select **Stations** from the **Data** menu. The following form appears:



3. Enter the station data.
4. You can optionally use the following quick buttons:

- **Incremental**: beginning with the first station, add the distance between two stations.
- **Decremental**: beginning with the first station, subtract the distance between two stations.

5. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

NOTE: The first station is defined in the profile options form.

8.9 Inlets

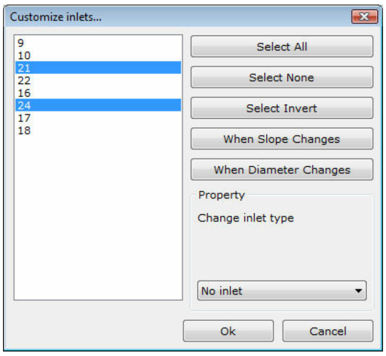
With this option, you can set the type of the inlets. The default type is manhole; however, the inlet may be just a computational node. You can use this option to:

- Define the simple computational nodes and the inlets with manholes.
- Define the type of inlet.

Prior to using this option, you must enter the inlet types that will be used in the network. In general, these are depended on the pipe diameter. This can be accomplished using **manhole specifications** found in the **Data** menu.

To set the type of inlets:

1. Select the profile from the list.
2. Select **Inlets** from the **Profiles** menu. The following form will appear:



3. Select **Select All** to set the type of all inlets to manhole.
4. Select **Select None** to set the type of all inlets to simple computational node. These are displayed in different color in the data table.
5. Select **Select Invert** to toggle the type of inlets between the above two types.
6. Select **When slope changes** to set the type of all inlets where slope changes to manhole.
7. Select **When diameter changes** to set the type of all inlets where size or shape of the pipe changes to manhole.

8. You can change the type of the inlet manually as follows:

- Select one or more inlets from the list on the left. In order to select multiple inlets, hold down CTRL while selecting.
- From the **Inlet types** frame, select one of **Without inlet, Without type,** or select a specified type of inlet.
- Repeat the above procedure for all inlets.

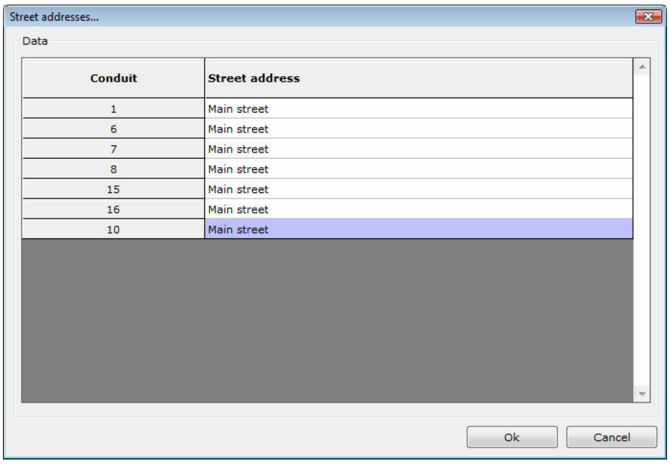
9. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

8.10 Street addresses

With this option, you can add or modify street addresses to the profile drawing. This data does not affect calculations.

To add or modify street addresses:

1. Select the profile from the list of the main form.
2. Select **Street addresses** from the **Profiles** menu. The following form will appear:



3. Enter the street addresses by typing directly onto the grid.
4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

For your convenience, the following options are available from the **Edit** menu:

- **Select all:** All cells are selected.
- **Cut:** The data of the selected cells are deleted from the grid and copied to the clipboard.
- **Copy:** The data of the selected cells are copied to the clipboard.
- **Paste:** The data of the clipboard is pasted into the grid.
- **Clear selection:** The data of the selected cells are deleted.

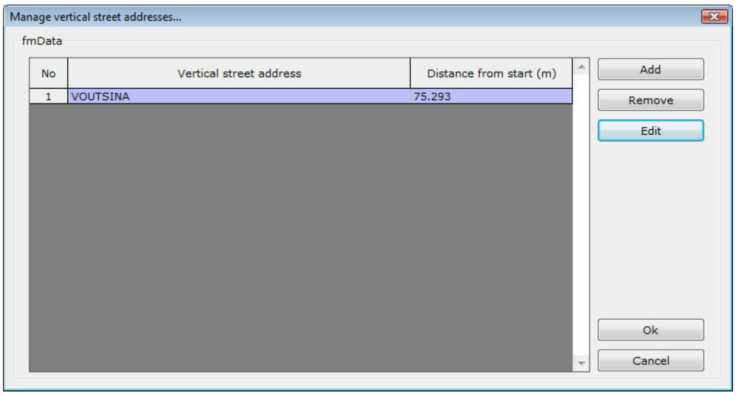
NOTE: If the same street address spans more than one sections, the program will merge the street addresses into one.

8.11 Vertical street addresses

With this option, you can add or modify vertical street addresses to the profile drawing. This data does not affect calculations.

To add or modify vertical street addresses:

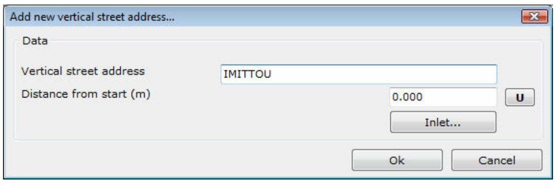
1. Select the profile from the list of the main form.
2. Select **Vertical street addresses** from the **Profiles** menu. The following form will appear:



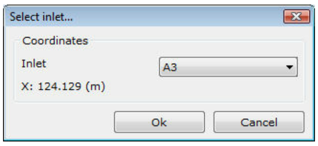
3. Make the appropriate changes.
4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To add a vertical street address:

1. Select **Add.** The following form will appear:



2. Enter the **Street address** by typing in the corresponding text box.
3. In the **Coordinates** frame, enter the **distance from start** of the vertical street address. For your convenience, you can select **Inlet**. The following form appears:



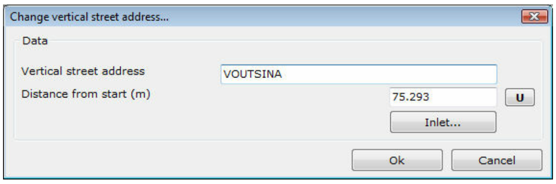
- Select the inlet from the drop-down list.

Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To modify an existing vertical street address:

1. Select the vertical street address from the list.
2. Select **Modify**. The following form will appear:



3. Make the appropriate changes.
4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To delete an existing vertical street address:

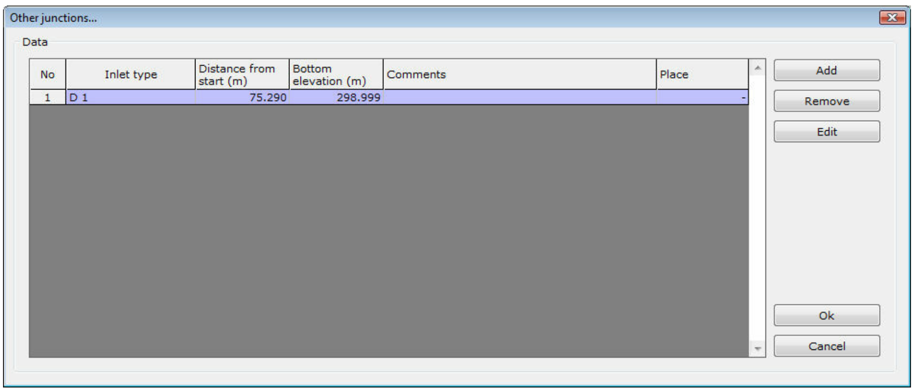
1. Select the vertical street address from the list.
2. Select **Delete**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The vertical street address is deleted.

8.12 Other junctions

With this option, you can add pipe junctions from other networks to the profile drawing and the profile sketch. These do not affect calculations.

To add pipe junctions to the profile drawing and the profile sketch:

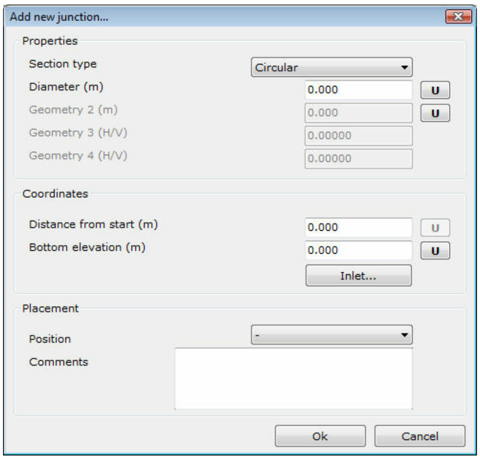
1. Select the profile from the list of the main form.
2. Select **Other junctions** from the **Profiles** menu. The following form will appear:



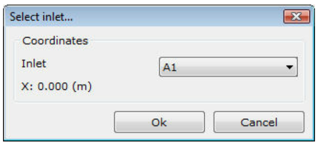
3. Make the appropriate changes.
4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To add a pipe junction:

1. Select **Add.** The following form will appear:



2. Select the **Section type** and **Diameter** of the pipe in m.
3. In the **Coordinates** frame, enter the **Distance from start** of the pipe junction and the **Bottom elevation** in m. For your convenience, you can select **Inlet**. The following form appears:



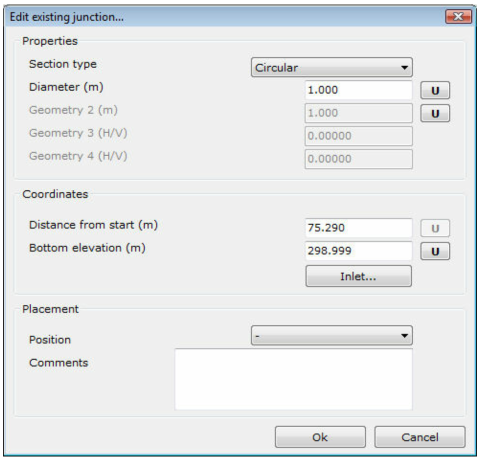
- Select the inlet from the drop-down list.
- Check **Match invert** if you wish to match the invert elevation of the specified inlet.

Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

4. In the **Placement** frame, you can select the **position** of the pipe junction i.e. whether the junction comes from the left or right, and you can add comments to the pipe junction. This data is used only for the profile drawings.
5. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To modify an existing junction:

1. Select the junction from the list.
2. Select **Modify**. The following form will appear:



3. Make the appropriate changes.
4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To delete an existing junction:

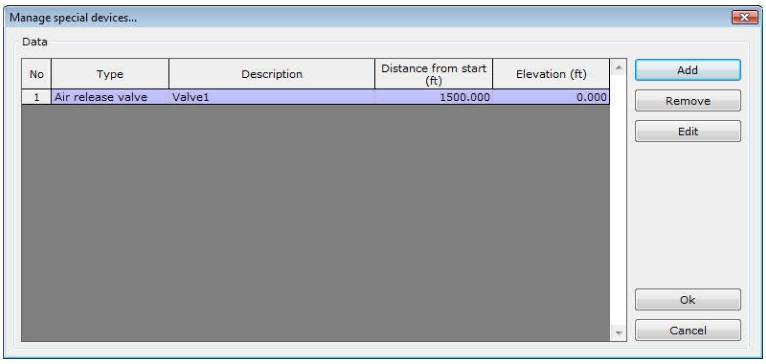
1. Select the junction from the list.
2. Select **Delete**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The junction is deleted.

8.13 Special devices

With this option, you can add special devices to the profile drawing and the profile sketch. These do not affect calculations.

To add special devices to the profile drawing and the profile sketch:

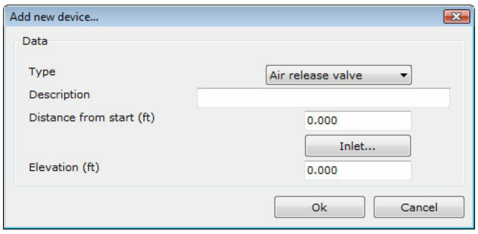
1. Select the profile from the list of the main form.
2. Select **Special devices** from the **Profiles** menu. The following form will appear:



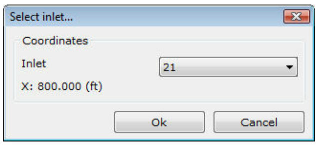
3. Make the appropriate changes.
4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To add a special device:

1. Select **Add.** The following form will appear:



2. In the **Properties** frame, select one of **Air release valve, Vacuum valve** and **Valve** as the type of the special device. You can also provide a **Description** of the special device by typing into the corresponding text box.
3. In the **Coordinates** frame, enter the **Distance from start** of the special device and the **Elevation** in m. For your convenience, you can select **Inlet**. The following form appears:



- Select the inlet from the drop-down list.
- Check **Match invert** if you wish to match the invert elevation of the specified inlet.

Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To modify an existing special device:

1. Select the special device from the list.
2. Select **Modify**. The following form will appear:

3. Make the appropriate changes.
4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To delete an existing special device:

1. Select the special device from the list.
2. Select **Delete**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The special device is deleted.

Chapter IX

9 Tools

9.1 Tools menu

With this menu, you can perform advanced operations regarding network's design and integrity design. In the **tools** menu you can select one of the following options:

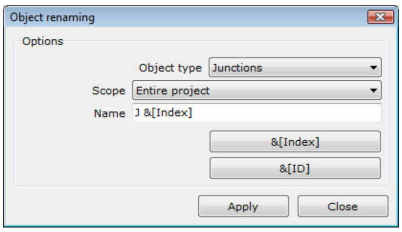
- Object renaming
- Placement of conduits at constant depth
- Vertical displacement
- Inflow distribution
- Sewer area distribution
- Exact total sewer area
- Delete all inflows
- Subcatchments from DXF
- Automatic design
- Contours

9.2 Object renaming

To conveniently rename objects of a certain type such as junctions, conduits, pumps, etc, you may use the object renaming tool.

To rename objects of a certain type:

1. Select **Object Renaming** from the **Tools** menu.
2. Select the object type from the drop-down list.
3. Select the **scope** from the drop-down list. The scope can either be the entire project or the current selection.
4. Enter the renaming command (see below).
5. Press **Apply** to rename the objects described above or **Cancel** to close the form and ignore all changes.



Renaming command

The renaming command is a string that dictates how the objects will be renamed. This command is consisted of keywords and strings. Keyword vary depending on the object type selected and appear as buttons for your convenience.

Examples:

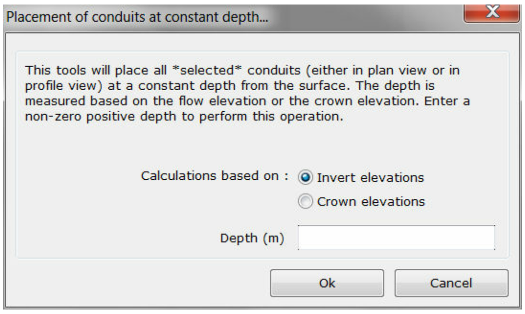
J&[Index]: will rename all objects as J1, J2, J3, etc.
&[StartNodeName] -> &[EndNodeName]: will rename all pipes as J1 -> J2, J2 -> J3, etc.

9.3 Placement of conduits at constant depth

This tools places the selected conduits at a constant (given) depth. The depth may be entered in m or ft, depending on the unit system, and can be measured from the invert elevation of the crown elevation. To place all conduits in a network at a constant depth, select them all before invoking this command by pressing CTRL+A.

To place the selected conduits at a constant depth:

1. Select the conduits to be placed at a constant depth.
2. From the **Tools** menu, select **Placement Of Conduits At Constant Depth**.
3. Select whether the calculations will be made based on the invert or crown elevations.
4. Enter the depth in ft or m.
5. Press **Ok** to place the selected conduits at the specified depth or **Cancel** to ignore changes.



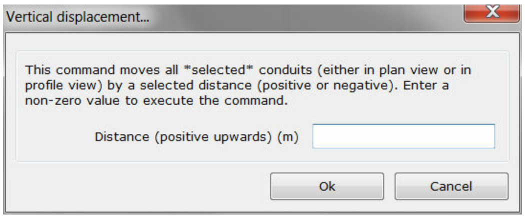
NOTE: This tool is available as a separate product. It is enabled once the "Sewer Networks Toolpack" product is purchased.

9.4 Vertical displacement

Use this tool to move the selected conduits upwards or downwards. Positive offsets move the network towards the ground level while negative values have the opposite effect, sinking the network.

To move the whole network:

1. Select the conduits you wish to move, either in plan-view or in profile view.
2. From the **Tools** menu, select **Vertical Displacement**.
3. Enter the distance in ft or m. A positive distance will move the network towards the ground. The distance requested is the absolute difference in the new elevation minus the current elevation of any object.
4. Press **Ok** to place the selected conduits at the specified depth or **Cancel** to ignore changes.



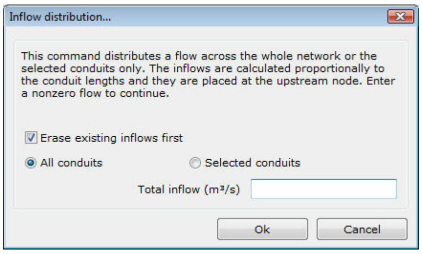
NOTE: This tool is available as a separate product. It is enabled once the "Sewer Networks Toolpack" product is purchased.

9.5 Inflow distribution

This tool is useful when one wants to distribute a know inflow (say 60 L/s) in all the network's conduits, depending on their length. The program will calculate the total conduits length and will distribute proportionally the total inflow to their upstream nodes.

To distribute a known inflow:

1. From the **Tools** menu select **Inflow Distribution**.
2. If you wish to erase all existing inflows prior to distribute the total inflow, check the option **Erase existing inflows first**. Otherwise, the portion of the total inflow at every node will be added to the existing inflow (if any).
3. Select whether the inflow distribution will be performed based on all conduits or the selected conduits only.
4. Enter the total inflow in flow units.
5. Press **Ok** to distribute the total inflow to all nodes in the network or **Cancel** to ignore any changes and hide the form.



NOTE: This tool is available as a separate product. It is enabled once the "Sewer Networks Toolpack" product is purchased.

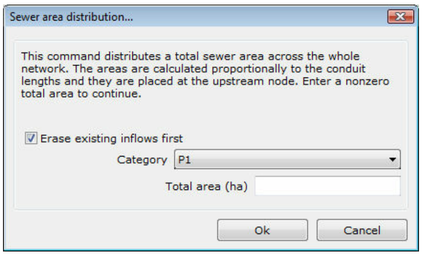
9.6 Sewer area distribution

Use sewer area distribution tool to divide a user specified sewer area across the network. This division will be proportional to the conduit lengths and the inflows (in terms of areas) will be placed at the upstream node of each conduit. Optionally, you can remove any existing inflows from the nodes prior to distributing a sewer area.

To distribute a known sewer area:

1. From the **Tools** menu select **Sewer Area Distribution**.
2. If you wish to erase all existing inflows prior to distribute the total area, check the

option **Erase existing inflows first**. Otherwise, the portion of the total area at every node will be added to the existing inflow area (if any).
3. Select the area category from the drop-down list.
4. Enter the total area in ha or ac depending on the unit system.
5. Press **Ok** to distribute the total area to all nodes in the network or **Cancel** to ignore any changes and hide the form.



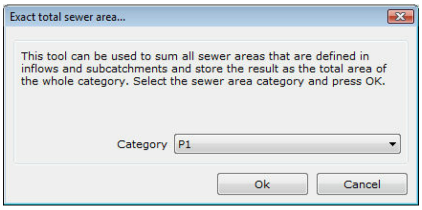
NOTE: This tool is available as a separate product. It is enabled once the "Sewer Networks Toolpack" product is purchased.

9.7 Exact total sewer area

This tool calculates the total area of the area objects for a selected area category and automatically transfers the result to the category's data.

To calculate the area of a specified area category:

1. Select **Exact Total Sewer Area** from the **Tools** menu.
2. Select the area category whose area will be computed.
3. Click **Ok** to compute the total area and transfer the result to the area category. Click **Cancel** to hide the tool.



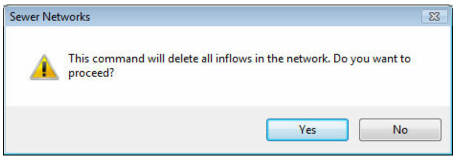
NOTE: This tool is available as a separate product. It is enabled once the "Sewer Networks Toolpack" product is purchased.

9.8 Delete all inflows

Use this tool to remove all inflows from all nodes.

To delete all inflows:

1. From the **Tools** menu select **Delete** All Inflows.
2. Click on **Yes** to remove all inflows or **No** to cancel the command without removing the inflows.



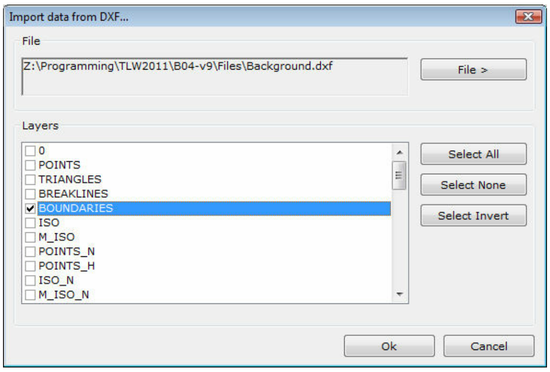
NOTE: This tool is available as a separate product. It is enabled once the "Sewer Networks Toolpack" product is purchased.

9.9 Subcatchments from DXF

If you do not wish to specify on screen each area separately via the program's user interface, it is possible to import one or more areas from external dxf files. The areas have to be closed polygons (polylines). This tool, apart from reading and plotting areas from external dxf files, scans for the nearest junction to every area and connects them.

To import subcatchments from DXF:

1. Select **Subcatchments From DXF** from the **Tools** menu.
2. The following form appears, where the input file can be selected.



3. Select one or more layers containing the polygon data.
4. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
5. Select **Ok** to import the polygons, display them on the drawing and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

NOTE: This tool is available as a separate product. It is enabled once the "Sewer Networks Toolpack" product is purchased.

9.10 Automatic design

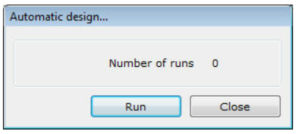
Use this tool to automatically select the appropriate conduit shape for each conduit, depending on a wide variety of criteria. To use this tool, you must first specify all conduit shapes that will be used in the automatic design and have them sorted by size (not mandatory, but logical). The program will start from the first specification and move to the last, solving the network and verifying that the following conditions are met:

a) The velocity at each conduit is between the minimum and maximum values of its specification.
b) The percent full at each conduit is not above the maximum limit set in its specification.
c) The network operates under gravity forces only.

Please note that once this operation finishes, the previous specifications at each conduit will be replaced by the results of the automatic design.

To automatically select specifications for the whole network:

1. Enter the appropriate conduit shapes and check those that will be used in the design.
2. From the **Tools** menu select **Automatic Design**.
3. Click on **Run** to start the procedure or **Close** to hide the tool without applying any changes.



NOTES:

1. Although the result will be optimal, it will be dependent on the specifications entered. Please ensure that all data in the specifications are correct.
2. This operation may take a lot of time, especially in large networks.
3. You can always undo this operation using the undo button or command.

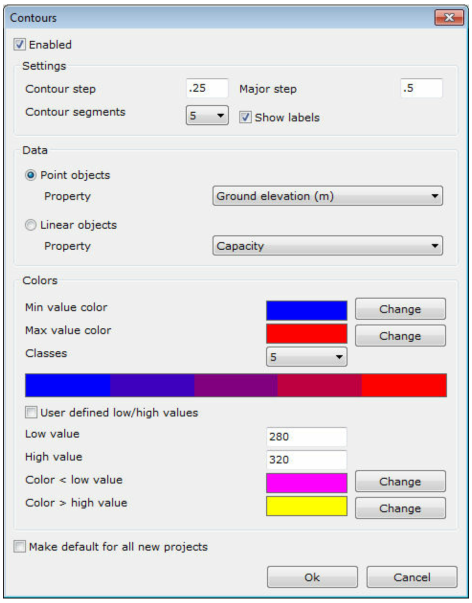
NOTE: This tool is available as a separate product. It is enabled once the "Sewer Networks Toolpack" product is purchased.

9.11 Contours

Use this tool to create contours for a specific property of point objects.

To create contours for a specific property of point objects:

1. From the **Tools** menu select **Contours**. The following form appears:



2. Check **Enabled** to enable the contours.
3. Select the **Contour step** (e.g. for the ground elevation of nodes, this is expressed in units of length), the step for the **major contours** (which are printed thicker and they are accompanied by their value) and the **Contour segments**, which controls the smoothness of curves.
4. Select **Point objects** and the corresponding property.
5. Select the colors that correspond to the minimum and maximum values by clicking on the corresponding **Change** button. If you want to use custom (user defined) high and low values, check the corresponding field. In this case you need to provide the colors for the values that are higher than the high value or lower than the low value.
6. Select the number of classes for the classification of objects.
7. Select **Ok** to close the dialog box and save changes. Select **Cancel** to close the dialog box and ignore changes.

NOTES:

1. The settings for the contour coloring are the same (and affect) those of object coloring. To create contours though, you must select **Point objects**.
2. Each time you perform calculations, the contours are evaluated anew. This operation may take a lot of time.
3. The contours are exported, in color, when you perform File > Export > Plan view to DXF.

NOTE: This tool is available as a separate product. It is enabled once the "Sewer Networks Toolpack" product is purchased.

Chapter X

10 Results

10.1 Results menu

With this menu, you can perform calculations and view the results. In the **Results** menu you can select one of the following options:

- Perform calculations
- Results report
- Tabulated report
  - By object
  - By variable
- Graphical report
  - Graph
  - System
  - Scatter
- Colors
- Total conduit lengths
- Special devices count
- Quality
- Profiles
  - Options
  - Design
- Quantities

10.2 Perform calculations

With this option, you can perform calculations.

To perform calculations:

1. Select **Perform calculations** from the **Results** menu.
2. The calculations are performed.

10.3 Results report

After the completion of calculations, a report is prepared that contains a list with possible issues.

To show this report:

1. Select **Perform calculations** from the **Results** menu.
2. Select **Results report** from the **Results** menu. If an error report is available, it is displayed.
3. Hit **ESC** to close the form.

NOTE: The error codes are described in detail in the Appendix. For each error code, common troubleshooting options are provided.

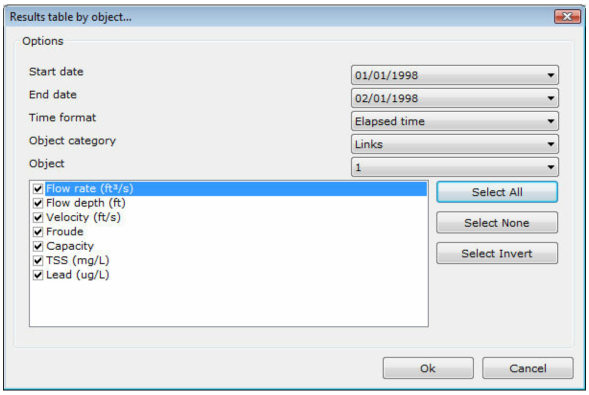
10.4 Tabular report

10.4.1 By object

With this option, the results of the calculations are displayed in tabular form an a per-object basis.

To display the results of the calculations in tabular form an a per-object basis:

1. Make sure that the calculations have been completed successfully.
2. Select **Tabular report** from the **Results** menu.
3. Select **By object** from the **Tabular report** menu. The following form appears:



4. Select the **Start date** from the drop-down list.
5. Select the **End date** from the drop-down list.
6. Select whether you wish to view the result with **Elapsed time** or in the format **Date/Time**.
7. Select the **object category** from the drop-down list.
8. Select the **Name** of the object. This step is omitted when **System** is selected in the previous step.
9. Select one or more results from the list. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
10. Press **Ok** to create the table. Press **Cancel** to close the form without creating the table.


Table - Link 1

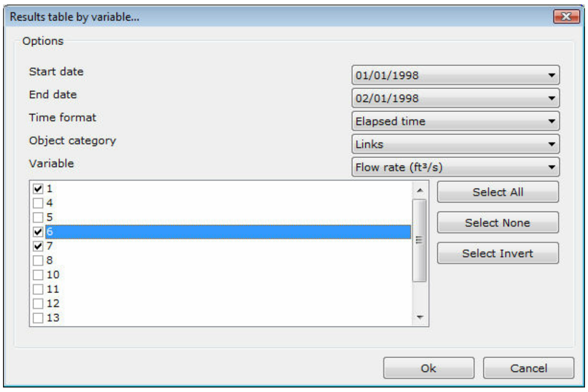
Days	Hours	Flow rate (ft³/s)	Flow depth (ft)	Velocity (ft/s)	Froude	Capacity	TSS (mg/L)	Lead (ug/L)
0	01:00	0.000	0.000	0.00	0.00	0.00	0.000	0.000
0	02:00	1.240	0.289	5.20	2.04	0.19	15.721	3.144
0	03:00	2.596	0.419	6.44	2.07	0.28	15.390	3.078
0	04:00	4.649	0.568	7.58	2.06	0.38	14.810	2.962
0	05:00	2.675	0.425	6.49	2.07	0.28	14.339	2.868
0	06:00	0.843	0.239	4.65	2.02	0.16	14.122	2.824
0	07:00	0.112	0.090	2.60	1.86	0.06	14.060	2.812
0	08:00	0.012	0.032	1.33	1.60	0.02	14.052	2.810
0	09:00	0.005	0.022	1.00	1.38	0.01	0.007	0.001
0	10:00	0.003	0.016	0.78	1.16	0.01	0.000	0.000
0	11:00	0.002	0.013	0.64	0.99	0.01	0.000	0.000
0	12:00	0.001	0.011	0.53	0.85	0.01	0.000	0.000
0	13:00	0.001	0.009	0.00	0.00	0.01	0.000	0.000
0	14:00	0.001	0.008	0.00	0.00	0.01	0.000	0.000
0	15:00	0.000	0.007	0.00	0.00	0.00	0.000	0.000
0	16:00	0.000	0.006	0.00	0.00	0.00	0.000	0.000
0	17:00	0.000	0.006	0.00	0.00	0.00	0.000	0.000
0	18:00	0.000	0.005	0.00	0.00	0.00	0.000	0.000
0	19:00	0.000	0.005	0.00	0.00	0.00	0.000	0.000
0	20:00	0.000	0.004	0.00	0.00	0.00	0.000	0.000
0	21:00	0.000	0.004	0.00	0.00	0.00	0.000	0.000
0	22:00	0.000	0.004	0.00	0.00	0.00	0.000	0.000
0	23:00	0.000	0.003	0.00	0.00	0.00	0.000	0.000
1	00:00	0.000	0.003	0.00	0.00	0.00	0.000	0.000
1	01:00	0.000	0.003	0.00	0.00	0.00	0.000	0.000
1	02:00	0.000	0.003	0.00	0.00	0.00	0.000	0.000
1	03:00	0.000	0.003	0.00	0.00	0.00	0.000	0.000
1	04:00	0.000	0.003	0.00	0.00	0.00	0.000	0.000
1	05:00	2.029	0.369	6.00	2.07	0.25	14.917	2.983
1	06:00	1.014	0.263	4.87	2.01	0.18	14.719	2.944
1	07:00	0.055	0.064	2.08	1.76	0.04	14.659	2.932
1	08:00	0.014	0.034	1.38	1.60	0.02	14.654	2.931
1	09:00	0.006	0.023	1.04	1.41	0.02	0.067	0.013
1	10:00	0.003	0.017	0.81	1.19	0.01	0.000	0.000
1	11:00	0.002	0.013	0.65	1.01	0.01	0.000	0.000

10.4.2 By variable

With this option, the results of the calculations are displayed in tabular form an a per-variable basis.

To display the results of the calculations in tabular form an a per-variable basis:

1. Make sure that the calculations have been completed successfully.
2. Select **Tabular report** from the **Results** menu.
3. Select **By variable** from the **Tabular report** menu. The following form appears:



4. Select the **Start date** from the drop-down list.
5. Select the **End date** from the drop-down list.
6. Select whether you wish to view the result with **Elapsed time** or in the format **Date/Time**.
7. Select the **object category** from the drop-down list.
8. Select the **Variable** of the object.
9. Select one or more objects from the list. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.

10. Press **Ok** to create the table. Press **Cancel** to close the form without creating the table.

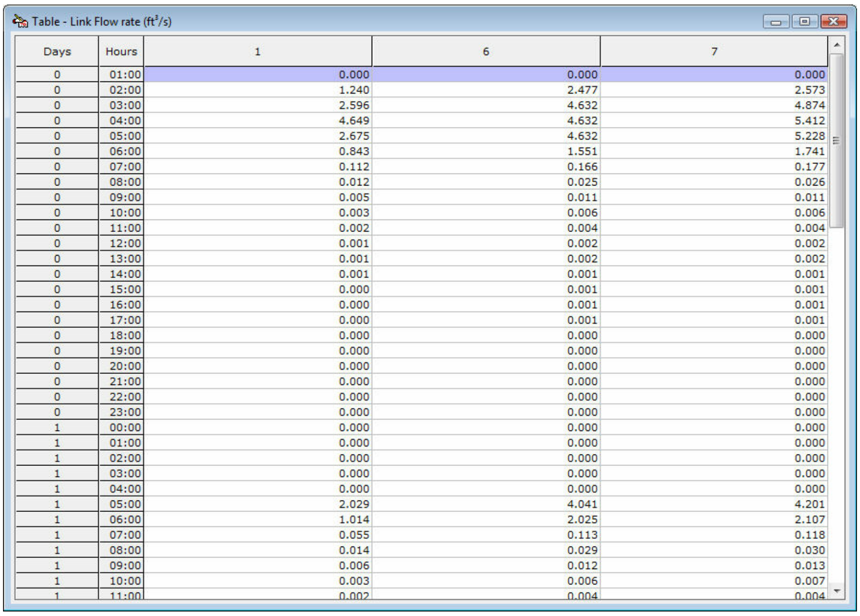


Table - Link Flow rate (ft^3/s)

Days	Hours	1	6	7
0	01:00	0.000	0.000	0.000
0	02:00	1.240	2.477	2.573
0	03:00	2.596	4.632	4.874
0	04:00	4.649	4.632	5.412
0	05:00	2.675	4.632	5.228
0	06:00	0.843	1.551	1.741
0	07:00	0.112	0.166	0.177
0	08:00	0.012	0.025	0.026
0	09:00	0.005	0.011	0.011
0	10:00	0.003	0.006	0.006
0	11:00	0.002	0.004	0.004
0	12:00	0.001	0.002	0.002
0	13:00	0.001	0.002	0.002
0	14:00	0.001	0.001	0.001
0	15:00	0.000	0.001	0.001
0	16:00	0.000	0.001	0.001
0	17:00	0.000	0.001	0.001
0	18:00	0.000	0.000	0.000
0	19:00	0.000	0.000	0.000
0	20:00	0.000	0.000	0.000
0	21:00	0.000	0.000	0.000
0	22:00	0.000	0.000	0.000
0	23:00	0.000	0.000	0.000
1	00:00	0.000	0.000	0.000
1	01:00	0.000	0.000	0.000
1	02:00	0.000	0.000	0.000
1	03:00	0.000	0.000	0.000
1	04:00	0.000	0.000	0.000
1	05:00	2.029	4.041	4.201
1	06:00	1.014	2.025	2.107
1	07:00	0.055	0.113	0.118
1	08:00	0.014	0.029	0.030
1	09:00	0.006	0.012	0.013
1	10:00	0.003	0.006	0.007
1	11:00	0.002	0.004	0.004

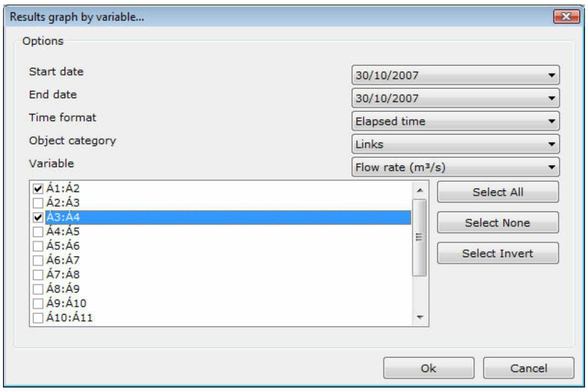
10.5 Graphical report

10.5.1 Graph

With this option, the results of the calculations are displayed in graphical form an a per-variable basis.

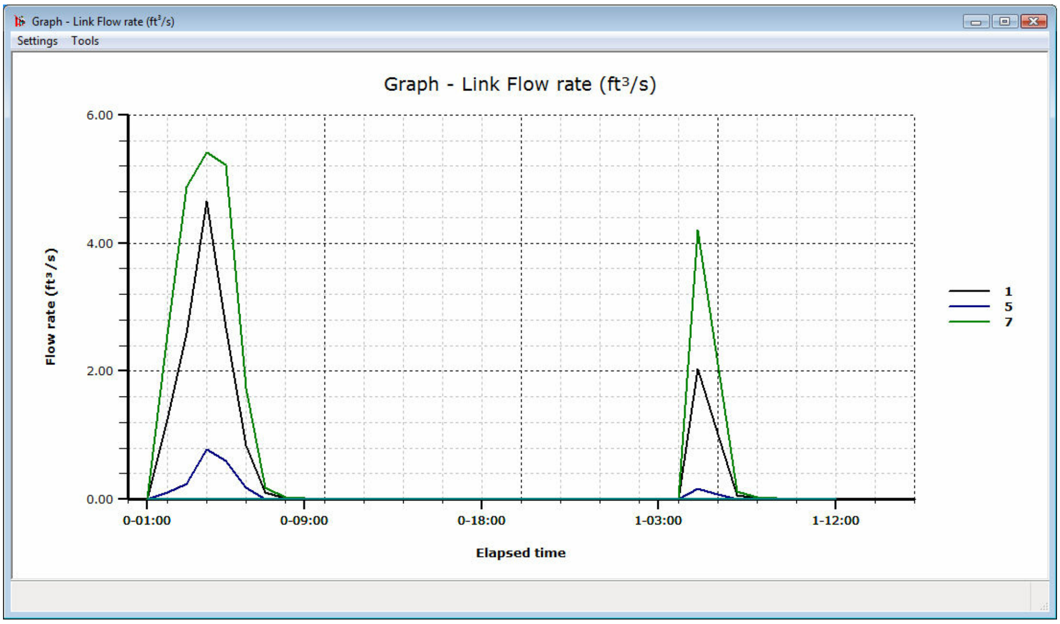
To display the results of the calculations in graphical form an a per-variable basis:

1. Make sure that the calculations have been completed successfully.
2. Select **Graphical report** from the **Results** menu.
3. Select **Graph** from the **Graphical report** menu. The following form appears:



4. Select the **Start date** from the drop-down list.
5. Select the **End date** from the drop-down list.
6. Select whether you wish to view the result with **Elapsed time** or in the format **Date/Time**.

7. Select the **object category** from the drop-down list.
8. Select the **Variable** of the object.
9. Select one or more objects from the list. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
10. Press **Ok** to create the graph. Press **Cancel** to close the form without creating the graph.



In the **Settings** menu, you can select one of the following options:

- **Customize**: you can customize the appearance of the graph (colors, axes, line styles, text etc).
- **Save settings**: the current settings are saved in a file.
- **Load settings**: the settings are loaded from a file.
- **Export to BMP**: the current image is saved in BMP format.

In the **Tools** menu, you can select one of the following options:

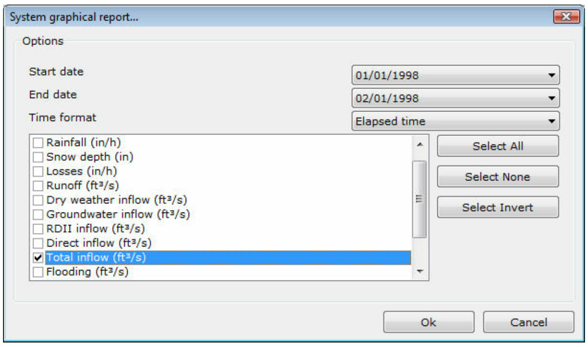
- **Copy to clipboard**: the current image is copied to the clipboard and becomes available to many programs such as Microsoft Word.
- **Set total graph width**: the total image width (in pixels) is set. This is particularly useful when creating images with certain dimensions.
- **Set total graph height**: the total image height (in pixels) is set. This is particularly useful when creating images with certain dimensions.
- **Set graph width**: the internal graph width (in pixels) is set. This is particularly useful when creating images with certain dimensions.
- **Set graph height**: the internal graph height (in pixels) is set. This is particularly useful when creating images with certain dimensions.

10.5.2 System

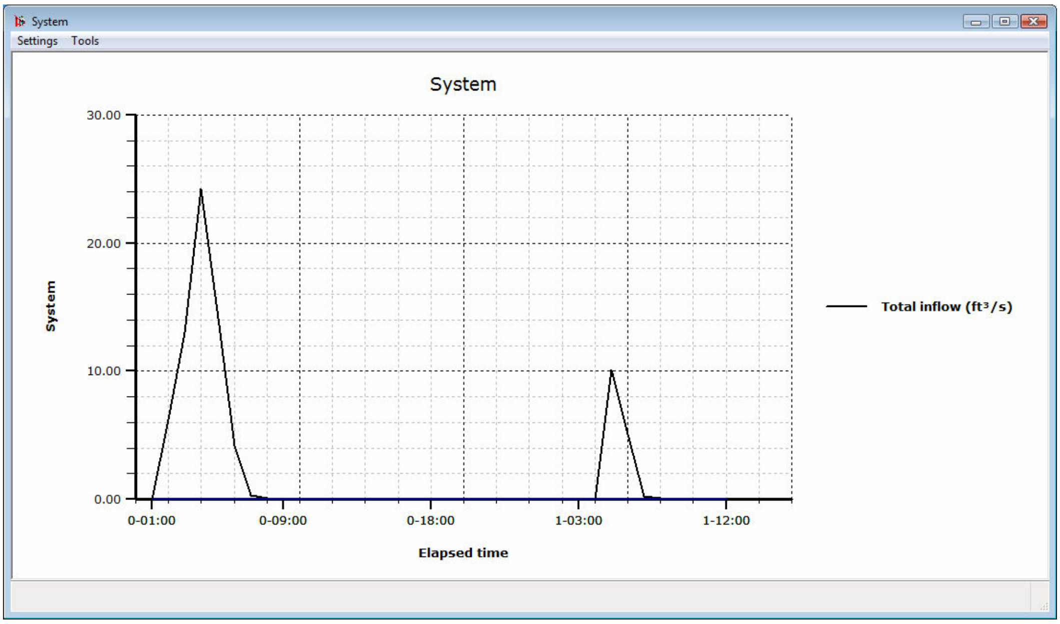
With this option, the results of the calculations regarding system variables are displayed in graphical form.

To display the results of the calculations regarding system variables in graphical form:

1. Make sure that the calculations have been completed successfully.
2. Select **Graphical report** from the **Results** menu.
3. Select **System** from the **Graphical report** menu. The following form appears:



4. Select the **Start date** from the drop-down list.
5. Select the **End date** from the drop-down list.
6. Select whether you wish to view the result with **Elapsed time** or in the format **Date/Time**.
7. Select one or more variables from the list. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
8. Press **Ok** to create the graph. Press **Cancel** to close the form without creating the graph.



In the **Settings** menu, you can select one of the following options:

- **Customize**: you can customize the appearance of the graph (colors, axes, line styles, text etc).

- **Save settings**: the current settings are saved in a file.
- **Load settings**: the settings are loaded from a file.
- **Export to BMP**: the current image is saved in BMP format.

In the **Tools** menu, you can select one of the following options:

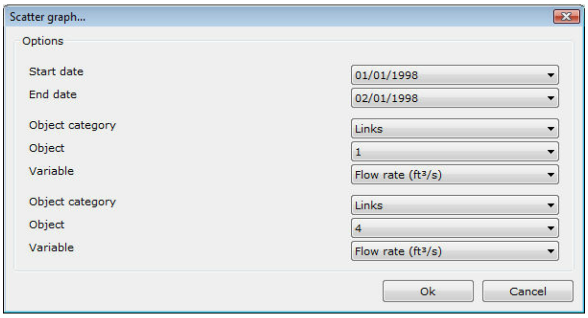
- **Copy to clipboard**: the current image is copied to the clipboard and becomes available to many programs such as Microsoft Word.
- **Set total graph width**: the total image width (in pixels) is set. This is particularly useful when creating images with certain dimensions.
- **Set total graph height**: the total image height (in pixels) is set. This is particularly useful when creating images with certain dimensions.
- **Set graph width**: the internal graph width (in pixels) is set. This is particularly useful when creating images with certain dimensions.
- **Set graph height**: the internal graph height (in pixels) is set. This is particularly useful when creating images with certain dimensions.

10.5.3 Scatter

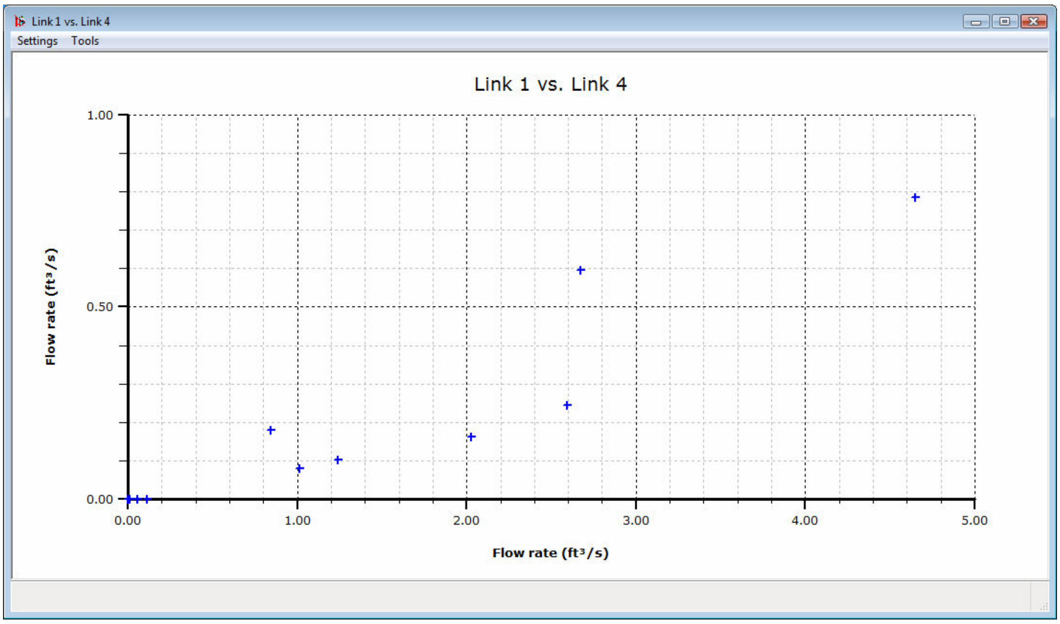
With this option, the results of the calculations regarding two variables are displayed in a scatter graph form.

To display the results of the calculations in a scatter graph form:

1. Make sure that the calculations have been completed successfully.
2. Select **Graphical report** from the **Results** menu.
3. Select **Scatter** from the **Graphical report** menu. The following form appears:



4. Select the **Start date** from the drop-down list.
5. Select the **End date** from the drop-down list.
6. Select the **Object category, Object name** and **Variable** for both objects.
7. Press **Ok** to create the graph. Press **Cancel** to close the form without creating the graph.



In the **Settings** menu, you can select one of the following options:

- **Customize**: you can customize the appearance of the graph (colors, axes, line styles, text etc).
- **Save settings**: the current settings are saved in a file.
- **Load settings**: the settings are loaded from a file.
- **Export to BMP**: the current image is saved in BMP format.

In the **Tools** menu, you can select one of the following options:

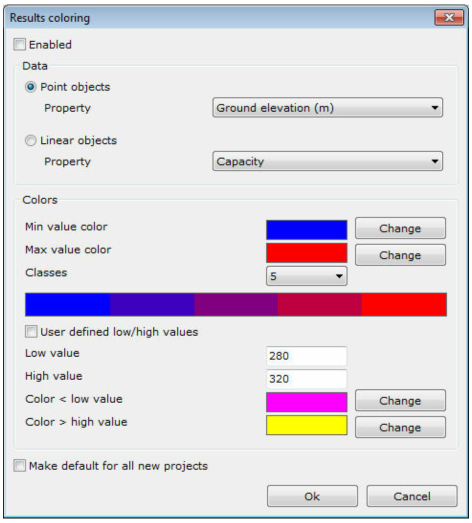
- **Copy to clipboard**: the current image is copied to the clipboard and becomes available to many programs such as Microsoft Word.
- **Set total graph width**: the total image width (in pixels) is set. This is particularly useful when creating images with certain dimensions.
- **Set total graph height**: the total image height (in pixels) is set. This is particularly useful when creating images with certain dimensions.
- **Set graph width**: the internal graph width (in pixels) is set. This is particularly useful when creating images with certain dimensions.
- **Set graph height**: the internal graph height (in pixels) is set. This is particularly useful when creating images with certain dimensions.

10.6 Colors

With this option, you can modify the settings for the coloring of objects depending on their properties.

To modify the settings for the coloring of objects:

1. Select **Colors** from the **Results** menu. The following form appears:



2. Check **Enabled** to enable the coloring of objects.
3. Select whether you want to color the point or the linear objects.
4. Select the colors that correspond to the minimum and maximum values by clicking on the corresponding **Change** button. If you want to use custom (user defined) high and low values, check the corresponding field. In this case you need to provide the colors for the values that are higher than the high value or lower than the low value.
5. Select the number of classes for the classification of objects.
6. Select **Ok** to close the dialog box and save changes. Select **Cancel** to close the dialog box and ignore changes.

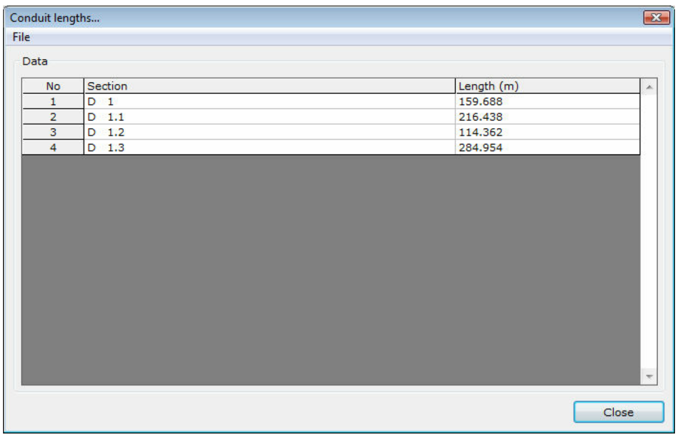
NOTE: In some cases, such as when the calculations have not been completed or when there is no variation of property values, the coloring will not be applied. In these cases, the default coloring is used.

10.7 Total conduit lengths

With this option, you can calculate the conduit lengths per profile that is currently used in the network. The report is based on conduit shapes.

To calculate the conduit lengths:

1. Select **Total conduit lengths** from the **Results** menu. The following form appears:



2. The total conduit lengths are displayed in the list.
3. Select **Ok** or **Cancel** to close the dialog box.

From the **File** menu, the following options are available:

Print, Print to Word, Print to Excel

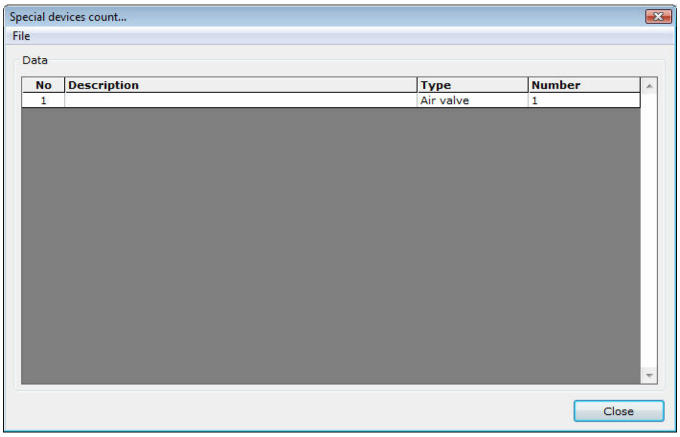
Select the appropriate option to create a report and sent it to the corresponding recipient.

10.8 Special devices count

With this option, you can count the special devices, i.e. air release valves, vacuum valves and valves, that are currently used in the network.

To count the special devices:

1. Select **Special devices count** from the **Results** menu. The following form appears:



2. The total number of special devices are displayed in the list.
3. Select **Ok** or **Cancel** to close the dialog box.

From the **File** menu, the following options are available:

Print, Print to Word, Print to Excel

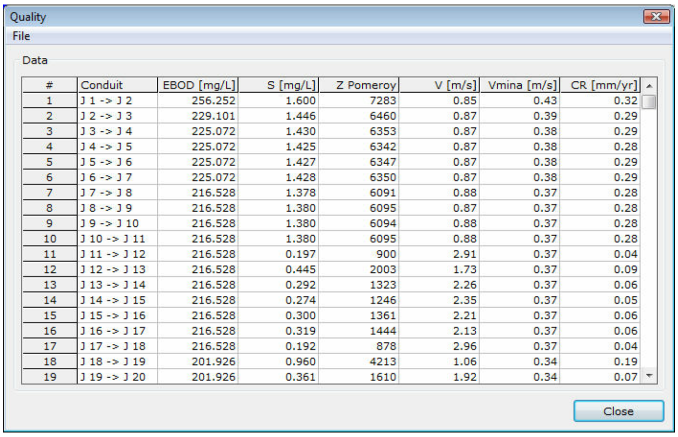
Select the appropriate option to create a report and sent it to the corresponding recipient.

10.9 Quality

With this option, you can display a form with results of quality calculations.

To display a form with results of quality calculations:

1. Select **Quality** from the **Results** menu. The following form appears:



#	Conduit	EBOD [mg/L]	S [mg/L]	Z Pomeroy	V [m/s]	Vmina [m/s]	CR [mm/yr]
1	J 1 -> J 2	256.252	1.600	7283	0.85	0.43	0.32
2	J 2 -> J 3	229.101	1.446	6460	0.87	0.39	0.29
3	J 3 -> J 4	225.072	1.430	6353	0.87	0.38	0.29
4	J 4 -> J 5	225.072	1.425	6342	0.87	0.38	0.28
5	J 5 -> J 6	225.072	1.427	6347	0.87	0.38	0.29
6	J 6 -> J 7	225.072	1.428	6350	0.87	0.38	0.29
7	J 7 -> J 8	216.528	1.378	6091	0.88	0.37	0.28
8	J 8 -> J 9	216.528	1.380	6095	0.87	0.37	0.28
9	J 9 -> J 10	216.528	1.380	6094	0.88	0.37	0.28
10	J 10 -> J 11	216.528	1.380	6095	0.88	0.37	0.28
11	J 11 -> J 12	216.528	0.197	900	2.91	0.37	0.04
12	J 12 -> J 13	216.528	0.445	2003	1.73	0.37	0.09
13	J 13 -> J 14	216.528	0.292	1323	2.26	0.37	0.06
14	J 14 -> J 15	216.528	0.274	1246	2.35	0.37	0.05
15	J 15 -> J 16	216.528	0.300	1361	2.21	0.37	0.06
16	J 16 -> J 17	216.528	0.319	1444	2.13	0.37	0.06
17	J 17 -> J 18	216.528	0.192	878	2.96	0.37	0.04
18	J 18 -> J 19	201.926	0.960	4213	1.06	0.34	0.19
19	J 19 -> J 20	201.926	0.361	1610	1.92	0.34	0.07

2. The results are displayed in the list.
3. Select **Close** to close the dialog box.

The results include:

1. The number and name of the conduit (note that only circular conduits are taken into account).
2. The effective concentration **EBOD** of the active organic content.
3. The limit concentration **S** after a long distance in the conduit.
4. The value of the **Pomeroy index**.
5. The flow velocity **V** in the conduit.
6. The self-ventilating velocity **Vmina**, according to Bielecki & Schremmer.
7. The corrosion rate **CR** of concrete conduits, according to Bielecki & Schremmer.

From the **File** menu, the following options are available:

Print, Print to Word, Print to Excel

Select the appropriate option to create a report and sent it to the corresponding recipient.

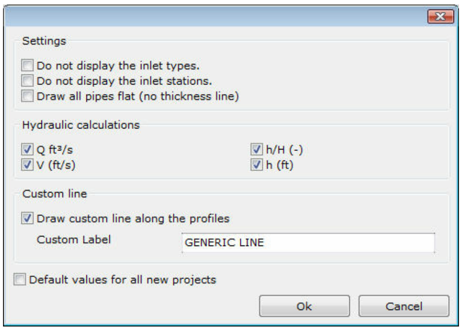
10.10 Profiles

10.10.1 Options

With this option, you can modify the settings of the profile drawings.

To modify the settings of the profile drawings:

1. Select **Profiles** from the **Results** menu.
2. Select **Options** from the **Profiles** menu. The following form will appear:



2. In the **Settings** frame:

- Check **Do not display the inlet types** if you do not want to display the inlet types.
- Check **Do not display the inlet stations** if you do not want to display the inlet stations.
- Check **Draw all pipes flat (no thickness line)** if you do not want to display the pipe thickness in the profile drawing. This does not affect the sketch of the main form where the pipe thickness is not drawn.

3. In the **Hydraulic calculations** frame, select one or more values that you wish to be included in the profile drawing.
4. In the **Custom line** frame:

- Check **Draw custom line along the profiles** if you want a custom line to be drawing along the profiles. In this case, you can provide the title of the custom line in the corresponding text box.

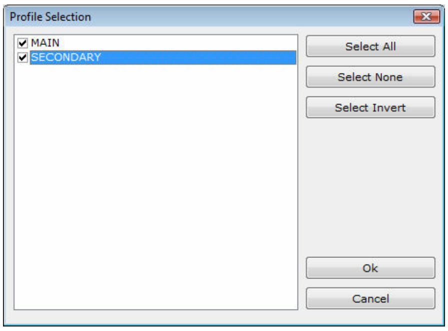
5. Check **Default values for all new projects** if you wish to make the settings default for all new projects. This does not affect existing projects.
6. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

10.10.2 Design

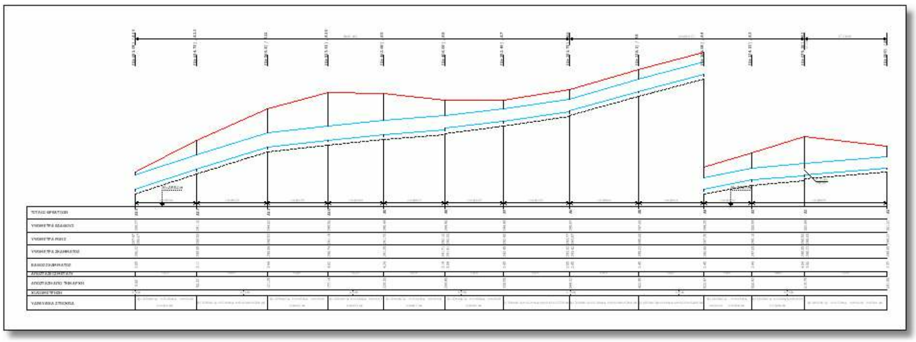
With this option, you can create the profile drawing. The data are prepared and sent to the **Profile designer**. A complete user manual on the capabilities of **Profile designer** can be found in the corresponding help file.

To create a profile:

1. Select **Draw profiles** from the **Results** menu. The following form appears:



2. Select one or more profiles to be included in the profile drawing. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
3. Select **Ok** to sent the data to the **Profile designer.** Select **Cancel** to cancel the operation.



10.11 Quantities

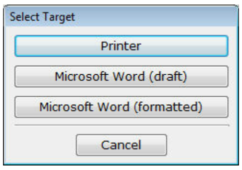
With this option, a report on the network quantities is assembled and prepared for preview.

In the current version of the program, this report includes:

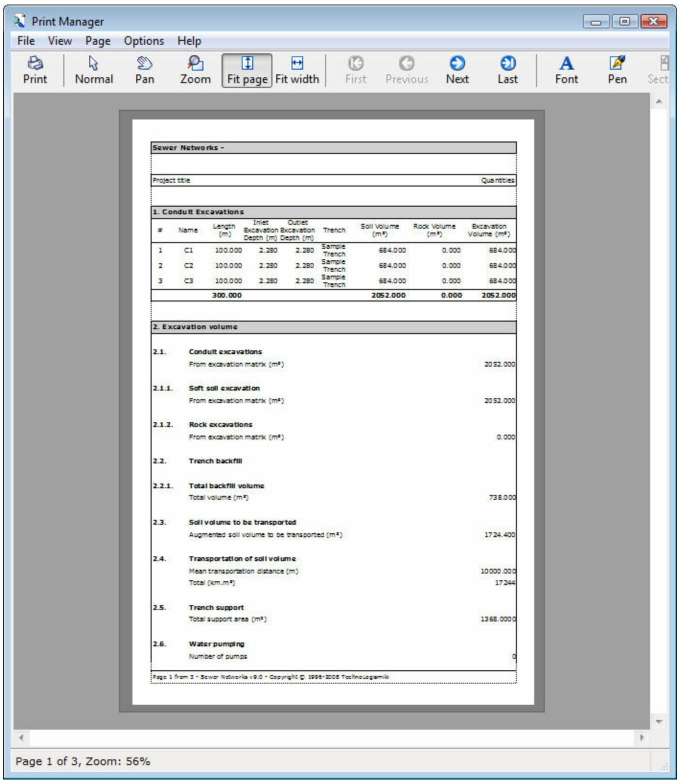
- Excavation tables based on trench profiles
- Excavation/backfill volumes
- Pipe quantities based on conduit shapes
- Manhole quantities based on manhole specifications
- Trench quantities based on trench specifications

To create a report on the network quantities:

1. Select **Quantities** from the **Results** menu.
2. The printer selection form appears:



3. Depending on the printer selected, the relevant object appears:



NOTE: The formatted Microsoft Word file requires the use of the clipboard. During the creation of the file, you should not use the clipboard.

Chapter XI

11 Help

11.1 Help menu

In the **Help** menu you can select one of the following options:

- Contents
- User guide
- Tutorials
- Tip of the day
- Unit conversion
- TechnoLogismiki website
- Buy products
- TechnoLogismiki NOMOS
- TechnoLogismiki Live!
- About the program

11.2 Contents

With this option, you can access the online help which contains detailed information regarding the usage of the program.

To view the online help:

1. Click **Contents** from the **Help** menu.
2. The online help appears.

NOTE: If an error message appears then the online help has not been installed. You can install the online help from the installation CD or the Internet.

11.3 User guide

With this option, you can access the user guide which contains detailed information regarding the usage of the program.

To view the user guide:

1. Click **User Guide** from the **Help** menu.
2. The user guide appears.

NOTE: If an error message appears then the online help has not been installed. You can install the online help from the installation CD or the Internet.

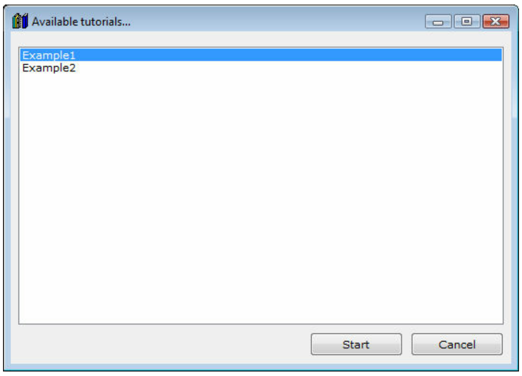
NOTE: Adobe Acrobat Reader or a similar program that can display pdf files is required in order to view or print the user guide.

11.4 Tutorials

With this option, you can access the tutorials of the program. The tutorials are step-by-step examples that allow you to decrease the learning cycle of the programs dramatically.

To access the tutorials:

1. Click **Tutorials** from the **Help** menu.
2. The tutorial selection dialog box appears.
2. Select the appropriate tutorial and click **Start** to proceed. Click **Cancel** to close the dialog box.



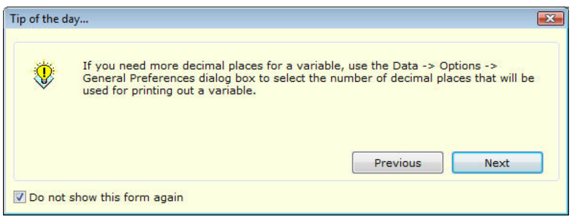
NOTE: The number and content of the tutorials is changed frequently. Use the live update system of TechnoLogismiki's products to download the latest tutorials.

11.5 Tip of the day

With this option, you can access the tip database of the program. The tips are short guidelines regarding the usage of the programs which may be of great help to the user.

To access the tips:

1. Click **Tip of the day** from the **Help** menu.
2. The tip of the day form appears.
3. Check **Do not show this form again** to prevent the program from showing the tip of the day when starting. Press the **Previous/Next** buttons to browse all available tips.
4. Press **Esc** to close the form.



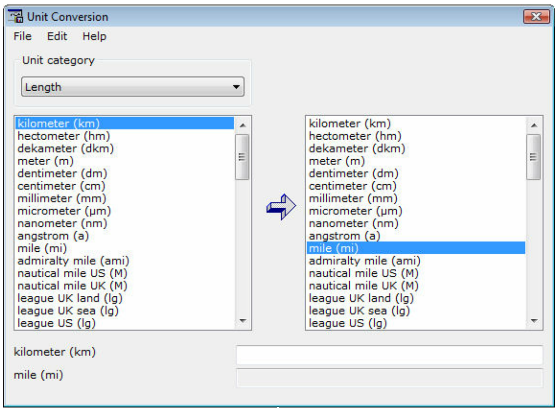
NOTE: The number and content of the tips is changed frequently. Use the live update system of TechnoLogismiki's products to download the latest tips.

11.6 Unit conversion

With this option, you can access the unit conversion tool. You can find more information about its usage in its help system.

To launch the unit conversion tool:

1. Click **Unit conversion** from the **Help** menu.
2. The unit conversion tool is launched.



NOTE: If an error message appears then the unit conversion tool has not been installed. You can install the unit conversion tool from the installation CD or the Internet.

11.7 TechnoLogismiki website

With this option, you can load on your Internet browser the website of TechnoLogismiki's.

11.8 Buy products

With this option, you can load on your Internet browser the main product page of TechnoLogismiki's website.

11.9 TechnoLogismiki NOMOS

With this option, you can load on your Internet browser the **NOMOS** service of Technologismiki.

11.10 TechnoLogismiki Live!

With this option, you can load on your Internet browser the **Live!** service of Technologismiki.

11.11 About the program

With this option, a form containing the name, version and licence information of the program appears.

To show this form:

1. From the **Help** menu, select **About the program**.
2. The form appears.
3. Click anywhere on the form or hit ESC to close the form.

Chapter XII

12 Appendix

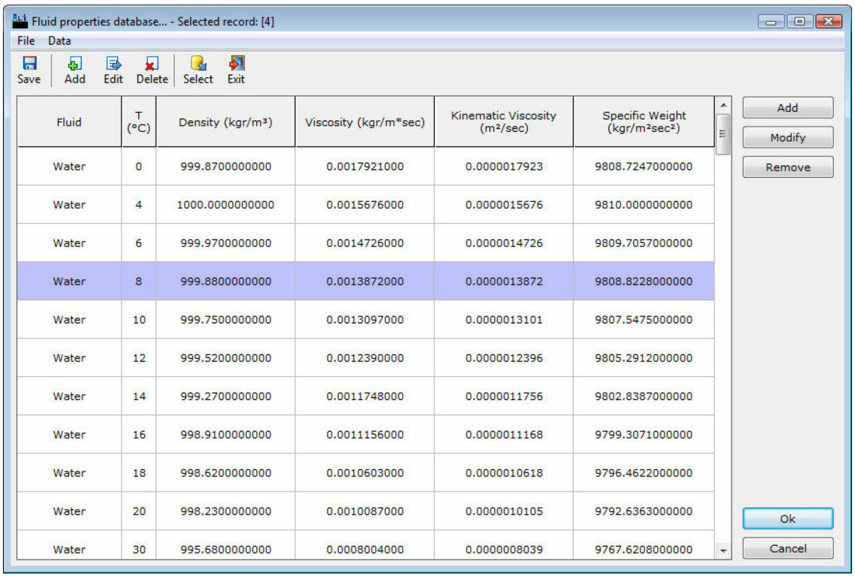
12.1 Unit system

Unit	Metric system	English system
Area	hectares m^2	acres ft^2
Length	m mm	ft in
Rainfall intensity	mm/h	in/h
Flow rate	m^3/s L/s ML/day	ft^3/s g/m Mg/day
Pollutant concentration	mass/hectare	mass/acre
Volume	m^3	ft^3

12.2 Fluid database

For your convenience, a fully customizable fluid database is embedded in the program. The fluid database is invoked in various cases within the program. By selecting an appropriate fluid record and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

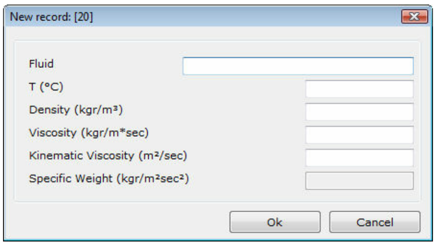
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



To add a new record:

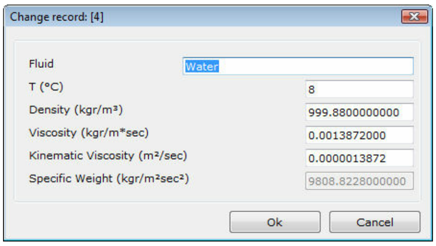
1. Click **Add** to open the new record dialog box.
2. Type the name of the fluid. This field is required.

3. Enter the temperature, density, viscosity and kinematic viscosity of the fluid.
4. The specific weight is calculated automatically.
5. Click **Ok** to close the dialog box and add a new record at the end of the list. Click **Cancel** to close the dialog box without making any changes.



To modify an existing record:

1. Click **Modify** to open the modify record dialog box.
2. Make the appropriate changes.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



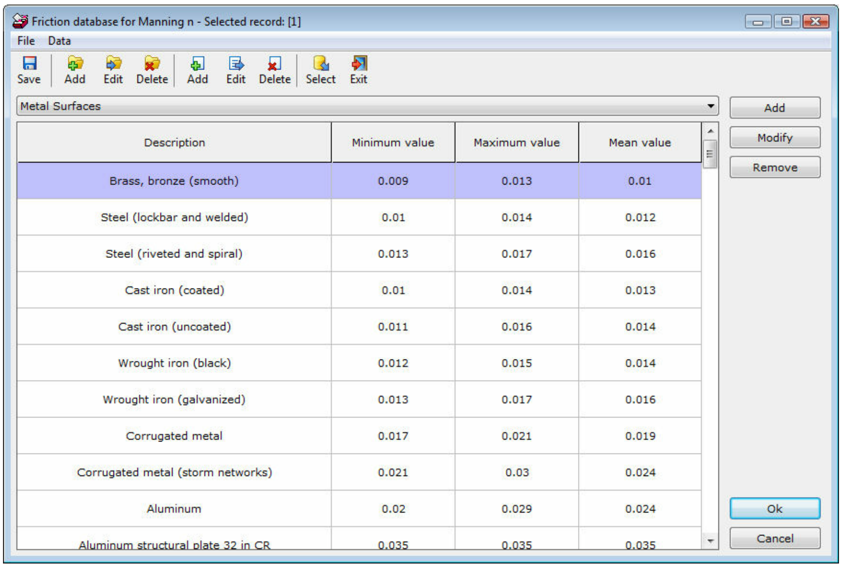
To remove an existing record:

1. Select the record you wish to remove.
2. Click **Remove** to remove the record. You will be asked to confirm the deletion.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.

12.3 Friction database

For your convenience, a fully customizable friction database is embedded in the program. The friction database is invoked in various cases within the program. By selecting an appropriate friction record (which is depended on the selected friction formula) and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

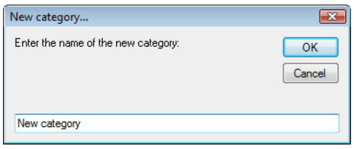
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



The database consists of several categories. Usually, the category defines the material of the surface (e.g. Metal surfaces).

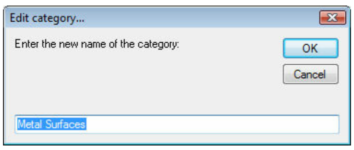
To add a new category:

1. Select **Add category** from the **Data** menu.
2. Type the name of the category in the text box. The name of the category must be unique.
3. Select **Ok** to add the category at the end of the list. Select **Cancel** to cancel the procedure.



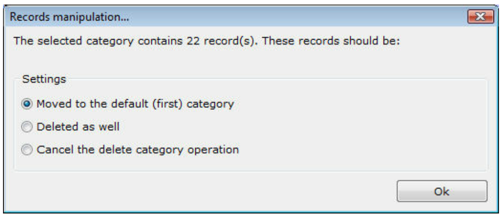
To modify the name of an existing category:

1. Click **Modify** to open the modify category dialog box.
2. Type the name of the category in the text box. The name of the category must be unique.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



To remove an existing category:

1. Select the category you wish to remove from the drop-down list.
2. Click **Remove** to remove the category. You will be asked to confirm the deletion.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.
4. If the category contains records, then the following dialog box appears:

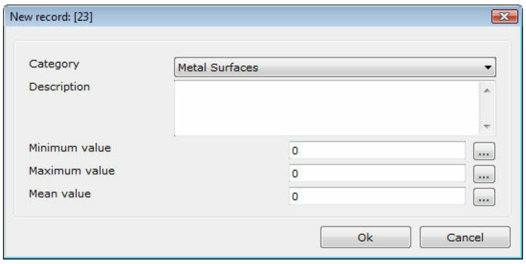


4.1. Select the first option to move the records of the category to the default (first category).
4.2. Select the second option to delete the records.
4.3. Select the third option to cancel the deletion.
5. Click **Ok** to proceed.

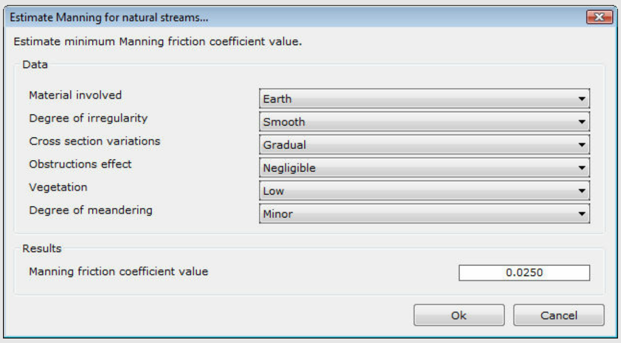
NOTE: The database must contain at least one category.

To add a new record:

1. Click **Add** to open the new record dialog box.
2. Select the category of the new record from the drop-down list.
3. Type the description of the record. This field is required.
4. Enter the minimum, maximum and mean value of the friction.
5. Click **Ok** to close the dialog box and add a new record at the end of the list. Click **Cancel** to close the dialog box without making any changes.



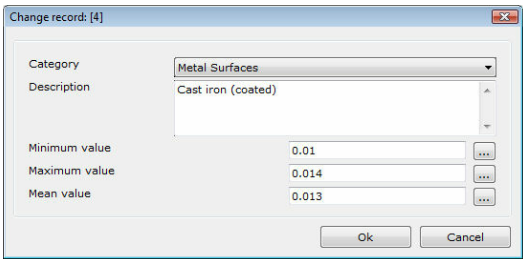
NOTE: In case of Manning friction coefficients in natural streams, you can estimate the values based on several characteristics of the stream. Click on the buttons with the ellipses (...) next to the text boxes to invoke the following dialog box:



Make the appropriate selections. Click **Ok** to close the dialog box and transfer the data to the corresponding text box. Click **Cancel** to close the dialog box without transferring any data.

To modify an existing record:

1. Click **Modify** to open the modify record dialog box.
2. Make the appropriate changes.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



To remove an existing record:

1. Select the record you wish to remove.
2. Click **Remove** to remove the record. You will be asked to confirm the deletion.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.

12.4 Manning friction coefficients

Surface / Material	Mean Value
Aluminum	0.024
Asbestos cement	0.013
Asphalt ditch	0.016
Asphalt pavement	0.016
Asphalt smooth	0.013
Asphalted cast iron	0.012
Natural ground	0.020
Best concrete	0.010
Brick in mortar	0.015
Brick sewer	0.015
Cast iron	0.012
CMP	0.024
Concrete	0.013
PVC	0.010
Centrifugal SPUN	0.013
Concrete (steel forms)	0.011
Concrete (wood forms)	0.015
Concrete gutter (broom finish)	0.016
Concrete gutter (troweled finish)	0.012
Copper	0.011

Fiber glass roving	0.011
Gravel riprap (D=25)	0.033
Gravel riprap (D=50)	0.041
Grouted riprap	0.030
Natural stream (clean)	0.030
Natural stream (stone)	0.050
Natural stream (weedy)	0.035

12.5 Bazin friction coefficients

Surface / Material	Max value	Min value	Mean value
Rough concrete	0.5	0.4	0.46
Smooth concrete	0.08	0.04	0.06
Brick in mortar	0.018	0.014	0.016
Sewer pipes (Greek regulations 696/74)	0.25	0.25	0.25
Storm pipes (Greek regulations 696/74)	0.46	0.46	0.46

12.6 Hazen - Williams friction coefficients

Surface / Material	Mean value
Asbestos cement	140
Asphalted cast iron	130
Best concrete	150
Centrifugal SPUN	135
Concrete (wood forms)	120
Concrete (steel forms)	140
Copper	135
Ductile iron	130
Galvanized iron	120
Glass	140
PVC	150
Riveted steel (new, rough)	80
Riveted steel (new, smooth)	110
Steel	120
Wood (new)	140

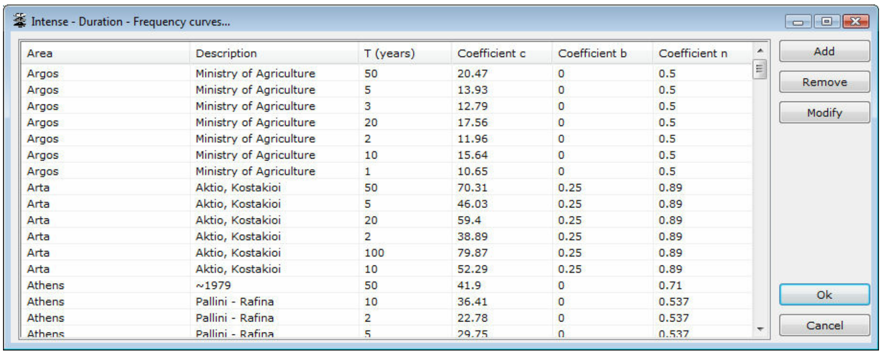
12.7 Darcy - Weisbach friction coefficients

Surface / Material	Mean value (mm)
Aluminum	0.300
Asbestos cement	0.002
Asphalted cast iron	0.120
Best concrete	0.366
Brick in mortar	0.610
Sewer brick	0.610
CMP	0.305
Concrete	0.122
Centrifugal SPUN	0.366
Concrete (steel forms)	1.829
Concrete (wood forms)	0.610
Copper	0.002
Galvanized steel	1.520
Glass	0.001
PVC	0.122
HDPE	0.150

12.8 IDF database

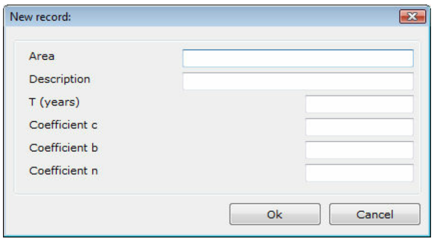
For your convenience, a fully customizable IDF (Intensity - Duration - Frequency) database is embedded in the program. The IDF database is invoked in various cases within the program. By selecting an appropriate record and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database. Note that the database was built using data from various resources in the literature; you need to be cautious when using a curve.



To add a new record:

1. Click **Add** to open the new record dialog box.
2. Enter the **area** and optionally the description of the record.
3. Enter the return period in years. This value was used for the calculation of the IDF curve and is for reference purposes only; it is not used in the calculations.
4. Enter the dimensionless coefficients c, b, n, in such a way that when time is entered in hours, the intensity is given in mm/hr.
5. Click **Ok** to close the dialog box and add a new record at the end of the list. Click **Cancel** to close the dialog box without making any changes.



To modify an existing record:

1. Click **Modify** to open the modify record dialog box.
2. Make the appropriate changes.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.

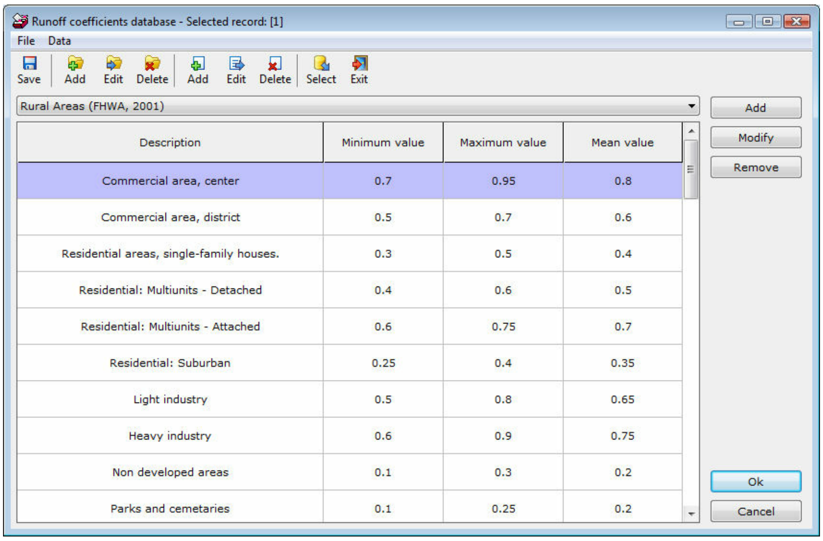
To remove an existing record:

1. Select the record you wish to remove.
2. Click **Remove** to remove the record. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.

12.9 Runoff coefficient database

For your convenience, a fully customizable runoff coefficient database is embedded in the program. The database is invoked in various cases within the program. By selecting an appropriate record and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

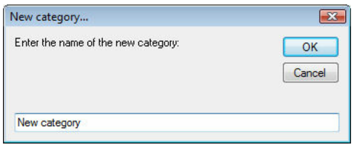
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



The database consists of several categories. Usually, the category defines the regulations (e.g. FHWA, 2001).

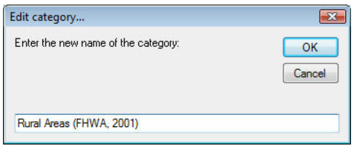
To add a new category:

1. Select **Add category** from the **Data** menu.
2. Type the name of the category in the text box. The name of the category must be unique.
3. Select **Ok** to add the category at the end of the list. Select **Cancel** to cancel the procedure.



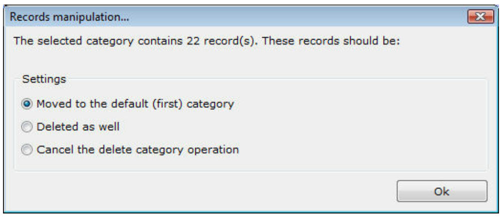
To modify the name of an existing category:

1. Click **Modify** to open the modify category dialog box.
2. Type the name of the category in the text box. The name of the category must be unique.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



To remove an existing category:

1. Select the category you wish to remove from the drop-down list.
2. Click **Remove** to remove the category. You will be asked to confirm the deletion.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.
4. If the category contains records, then the following dialog box appears:

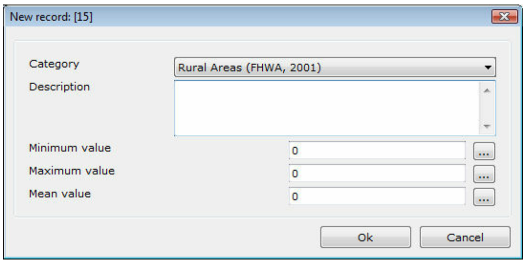


4.1. Select the first option to move the records of the category to the default (first category).
4.2. Select the second option to delete the records.
4.3. Select the third option to cancel the deletion.
5. Click **Ok** to proceed.

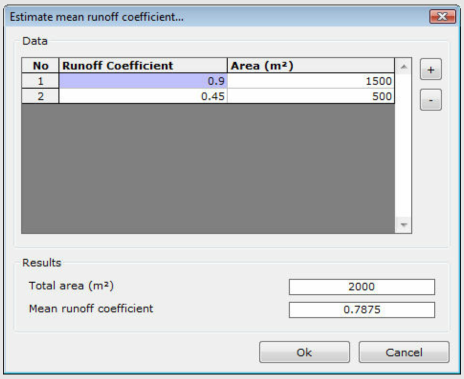
NOTE: The database must contain at least one category.

To add a new record:

1. Click **Add** to open the new record dialog box.
2. Select the category of the new record from the drop-down list.
3. Type the description of the record. This field is required.
4. Enter the minimum, maximum and mean value of the runoff coefficient.
5. Click **Ok** to close the dialog box and add a new record at the end of the list. Click **Cancel** to close the dialog box without making any changes.



NOTE: You can calculate the mean runoff coefficient for complex areas with known runoff coefficients. Click on the buttons with the ellipses (...) next to the text boxes to invoke the following dialog box:

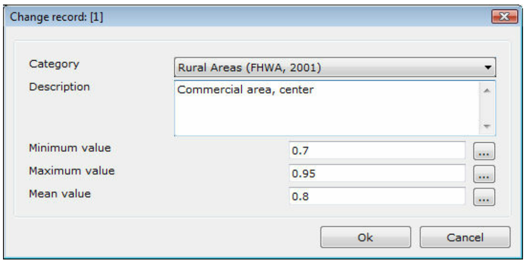


Click the plus sign **(+)** to add a new area. Type the runoff coefficient and the area in m^2. The total area and the mean runoff coefficient is displayed in the **Results** frame. Click the minus sign **(-)** to delete the selected area. The area is deleted with no confirmation.

Make the appropriate selections. Click **Ok** to close the dialog box and transfer the data to the corresponding text box. Click **Cancel** to close the dialog box without transferring any data.

To modify an existing record:

1. Click **Modify** to open the modify record dialog box.
2. Make the appropriate changes.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



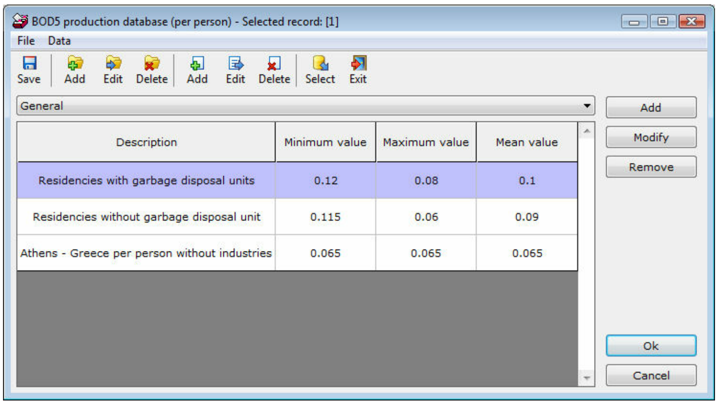
To remove an existing record:

1. Select the record you wish to remove.
2. Click **Remove** to remove the record. You will be asked to confirm the deletion.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.

12.10 BOD5 production

For your convenience, a fully customizable BOD5 production database is embedded in the program. The database is invoked in various cases within the program. By selecting an appropriate record and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

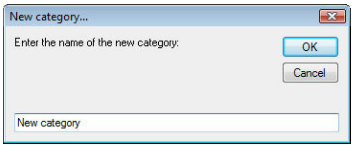
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



The database consists of several categories.

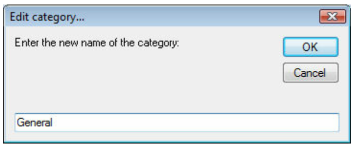
To add a new category:

1. Select **Add category** from the **Data** menu.
2. Type the name of the category in the text box. The name of the category must be unique.
3. Select **Ok** to add the category at the end of the list. Select **Cancel** to cancel the procedure.



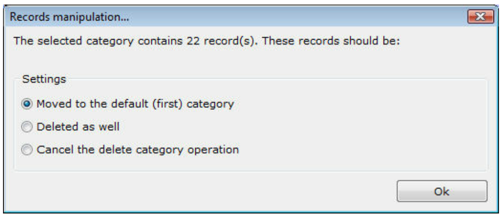
To modify the name of an existing category:

1. Click **Modify** to open the modify category dialog box.
2. Type the name of the category in the text box. The name of the category must be unique.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



To remove an existing category:

1. Select the category you wish to remove from the drop-down list.
2. Click **Remove** to remove the category. You will be asked to confirm the deletion.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.
4. If the category contains records, then the following dialog box appears:

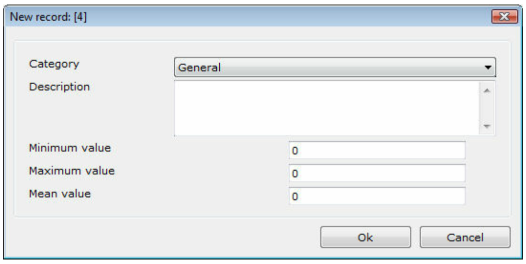


4.1. Select the first option to move the records of the category to the default (first category).
4.2. Select the second option to delete the records.
4.3. Select the third option to cancel the deletion.
5. Click **Ok** to proceed.

NOTE: The database must contain at least one category.

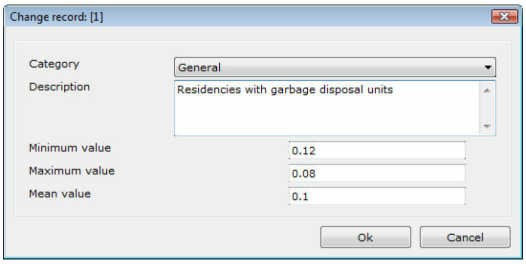
To add a new record:

1. Click **Add** to open the new record dialog box.
2. Select the category of the new record from the drop-down list.
3. Type the description of the record. This field is required.
4. Enter the minimum, maximum and mean value of the BOD5 production.
5. Click **Ok** to close the dialog box and add a new record at the end of the list. Click **Cancel** to close the dialog box without making any changes.



To modify an existing record:

1. Click **Modify** to open the modify record dialog box.
2. Make the appropriate changes.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



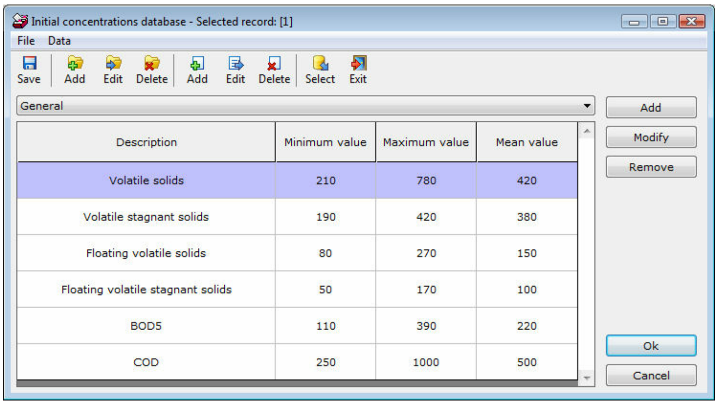
To remove an existing record:

1. Select the record you wish to remove.
2. Click **Remove** to remove the record. You will be asked to confirm the deletion.
3. Select **Yes** to proceed with the deletion. Select **No** to cancel the deletion.

12.11 Initial sulfur concentration

For your convenience, a fully customizable initial sulfur concentration database is embedded in the program. The database is invoked in various cases within the program. By selecting an appropriate record and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

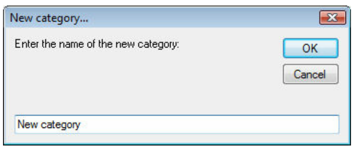
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



The database consists of several categories.

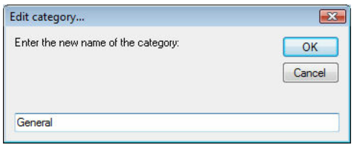
To add a new category:

1. Select **Add category** from the **Data** menu.
2. Type the name of the category in the text box. The name of the category must be unique.
3. Select **Ok** to add the category at the end of the list. Select **Cancel** to cancel the procedure.



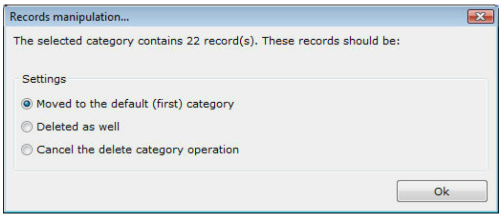
To modify the name of an existing category:

1. Click **Modify** to open the modify category dialog box.
2. Type the name of the category in the text box. The name of the category must be unique.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



To remove an existing category:

1. Select the category you wish to remove from the drop-down list.
2. Click **Remove** to remove the category. You will be asked to confirm the deletion.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.
4. If the category contains records, then the following dialog box appears:

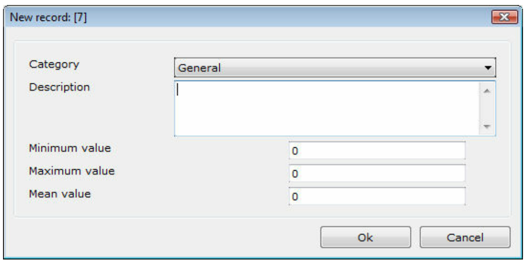


4.1. Select the first option to move the records of the category to the default (first category).
4.2. Select the second option to delete the records.
4.3. Select the third option to cancel the deletion.
5. Click **Ok** to proceed.

NOTE: The database must contain at least one category.

To add a new record:

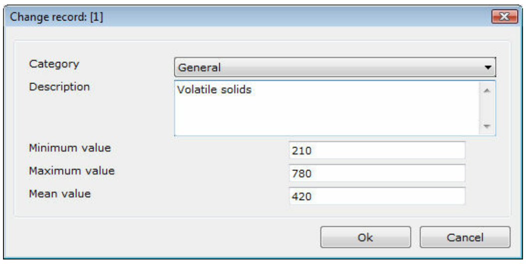
1. Click **Add** to open the new record dialog box.
2. Select the category of the new record from the drop-down list.
3. Type the description of the record. This field is required.
4. Enter the minimum, maximum and mean value of the initial sulfur concentration production.
5. Click **Ok** to close the dialog box and add a new record at the end of the list. Click **Cancel** to close the dialog box without making any changes.



To modify an existing record:

1. Click **Modify** to open the modify record dialog box.
2. Make the appropriate changes.

3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



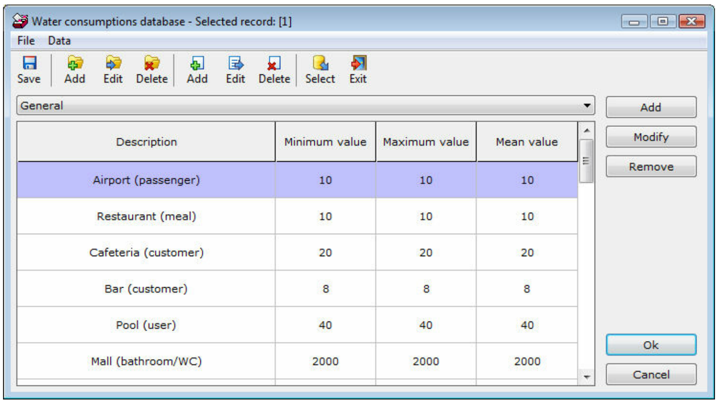
To remove an existing record:

1. Select the record you wish to remove.
2. Click **Remove** to remove the record. You will be asked to confirm the deletion.
3. Select **Yes** to proceed with the deletion. Select **No** to cancel the deletion.

12.12 Water consumption

For your convenience, a fully customizable water consumption database is embedded in the program. The database is invoked in various cases within the program. By selecting an appropriate record and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

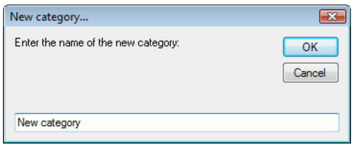
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



The database consists of several categories.

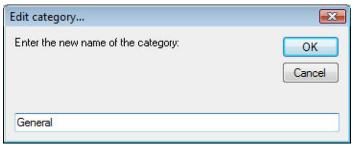
To add a new category:

1. Select **Add category** from the **Data** menu.
2. Type the name of the category in the text box. The name of the category must be unique.
3. Select **Ok** to add the category at the end of the list. Select **Cancel** to cancel the procedure.



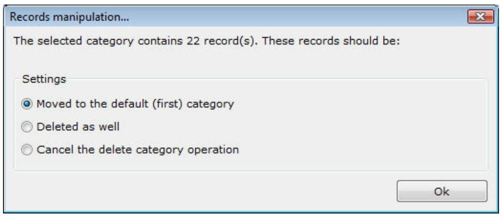
To modify the name of an existing category:

1. Click **Modify** to open the modify category dialog box.
2. Type the name of the category in the text box. The name of the category must be unique.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



To remove an existing category:

1. Select the category you wish to remove from the drop-down list.
2. Click **Remove** to remove the category. You will be asked to confirm the deletion.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.
4. If the category contains records, then the following dialog box appears:

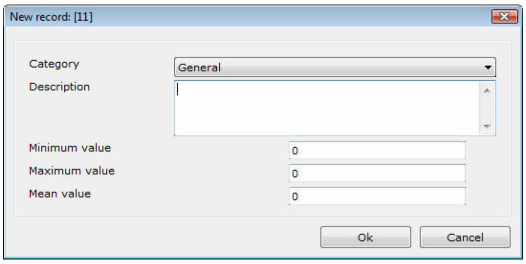


4.1. Select the first option to move the records of the category to the default (first category).
4.2. Select the second option to delete the records.
4.3. Select the third option to cancel the deletion.
5. Click **Ok** to proceed.

NOTE: The database must contain at least one category.

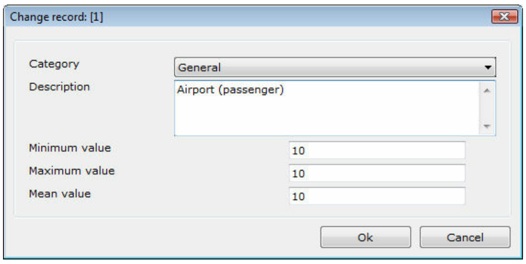
To add a new record:

1. Click **Add** to open the new record dialog box.
2. Select the category of the new record from the drop-down list.
3. Type the description of the record. This field is required.
4. Enter the minimum, maximum and mean value of the water consumption.
5. Click **Ok** to close the dialog box and add a new record at the end of the list. Click **Cancel** to close the dialog box without making any changes.



To modify an existing record:

1. Click **Modify** to open the modify record dialog box.
2. Make the appropriate changes.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



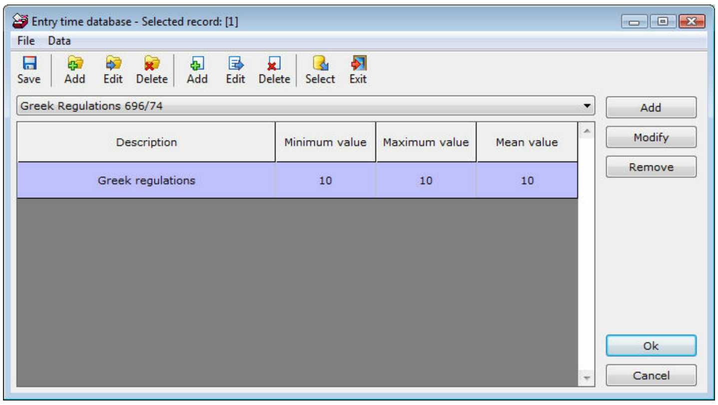
To remove an existing record:

1. Select the record you wish to remove.
2. Click **Remove** to remove the record. You will be asked to confirm the deletion.
3. Select **Yes** to proceed with the deletion. Select **No** to cancel the deletion.

12.13 Initial time

For your convenience, a fully customizable initial time database is embedded in the program. The database is invoked in various cases within the program. By selecting an appropriate record and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

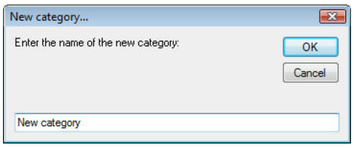
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



The database consists of several categories. Usually, the category defines the regulations e.g. Greek regulations 696/74.

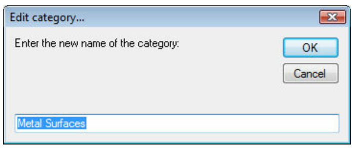
To add a new category:

1. Select **Add category** from the **Data** menu.
2. Type the name of the category in the text box. The name of the category must be unique.
3. Select **Ok** to add the category at the end of the list. Select **Cancel** to cancel the procedure.



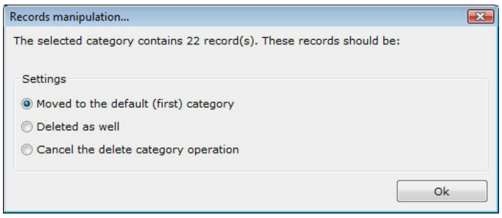
To modify the name of an existing category:

1. Click **Modify** to open the modify category dialog box.
2. Type the name of the category in the text box. The name of the category must be unique.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



To remove an existing category:

1. Select the category you wish to remove from the drop-down list.
2. Click **Remove** to remove the category. You will be asked to confirm the deletion.
3. Select **Yes** to proceed with the deletion. Select **No** to cancel the deletion.
4. If the category contains records, then the following dialog box appears:

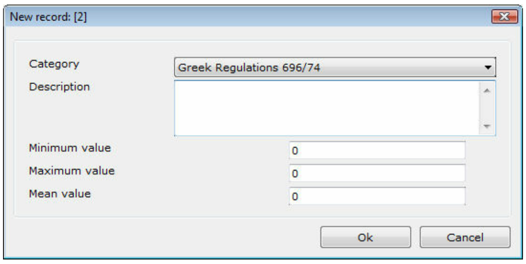


4.1. Select the first option to move the records of the category to the default (first category).
4.2. Select the second option to delete the records.
4.3. Select the third option to cancel the deletion.
5. Click **Ok** to proceed.

NOTE: The database must contain at least one category.

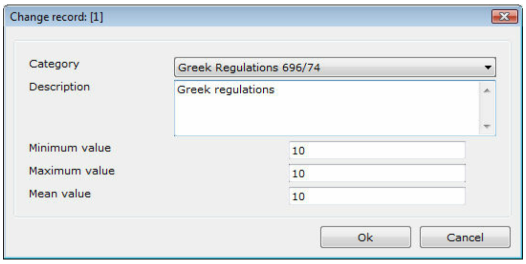
To add a new record:

1. Click **Add** to open the new record dialog box.
2. Select the category of the new record from the drop-down list.
3. Type the description of the record. This field is required.
4. Enter the minimum, maximum and mean value of the initial time in min.
5. Click **Ok** to close the dialog box and add a new record at the end of the list. Click **Cancel** to close the dialog box without making any changes.



To modify an existing record:

1. Click **Modify** to open the modify record dialog box.
2. Make the appropriate changes.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



To remove an existing record:

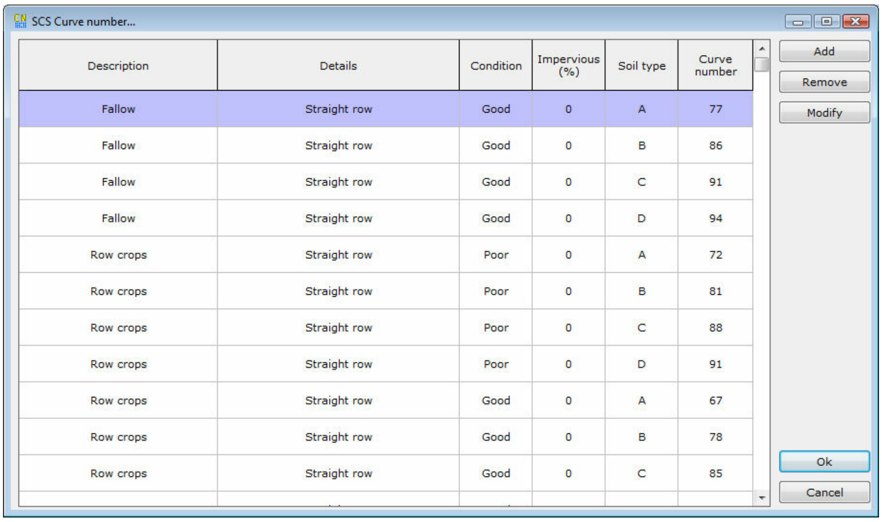
1. Select the record you wish to remove.

2. Click **Remove** to remove the record. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.

12.14 SCS curve number database

For your convenience, a fully customizable SCS curve number database is embedded in the program. The database is invoked in various cases within the program. By selecting an appropriate record and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



Description	Details	Condition	Impervious (%)	Soil type	Curve number
Fallow	Straight row	Good	0	A	77
Fallow	Straight row	Good	0	B	86
Fallow	Straight row	Good	0	C	91
Fallow	Straight row	Good	0	D	94
Row crops	Straight row	Poor	0	A	72
Row crops	Straight row	Poor	0	B	81
Row crops	Straight row	Poor	0	C	88
Row crops	Straight row	Poor	0	D	91
Row crops	Straight row	Good	0	A	67
Row crops	Straight row	Good	0	B	78
Row crops	Straight row	Good	0	C	85

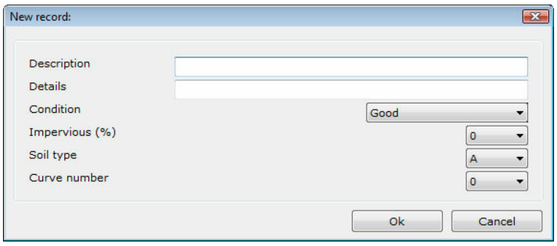
To add a new record:

1. Click **Add** to open the new record dialog box.
2. Type the description and optionally the details.
3. Enter the condition, imperviousness, soil type (A, B, C or D) and curve number (0-100).

- **Soil type A**: Soils having high infiltration rates, even when thoroughly wetted and consisting chiefly of deep, well to excessively-drained sands or gravels. These soils have a high rate of water transmission.
- **Soil type B**: Soils having moderate infiltration rates when thoroughly wetted and consisting chiefly of moderately deep to deep, moderately fine to moderately coarse textures. These soils have a moderate rate of water transmission.
- **Soil type C**: Soils having slow infiltration rates when thoroughly wetted and consisting chiefly of soils with a layer that impedes downward movement of water, or soils with moderately fine to fine texture. These soils have a slow rate of water transmission.
- **Soil type D**: Soils having very slow infiltration rates when thoroughly wetted and consisting chiefly of clay soils with a high swelling potential, soils with a permanent high water table, soils with a clay pan or clay layer at or near the surface, and shallow soils over nearly impervious material. These soils have a very

slow rate of water transmission.

4. Click **Ok** to close the dialog box and add a new record at the end of the list. Click **Cancel** to close the dialog box without making any changes.



To modify an existing record:

1. Click **Modify** to open the modify record dialog box.
2. Make the appropriate changes.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.

To remove an existing record:

1. Select the record you wish to remove.
2. Click **Remove** to remove the record. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.

12.15 Soil characteristics

Soil texture class	K (in/h)	K (mm/h)	Ψ (in)	Ψ (mm)	φ	FC	WP
Sand	4.74	120.4	1.93	49.0	0.437	0.062	0.024
Loamy sand	1.18	30.0	2.40	61.0	0.437	0.105	0.047
Sandy loam	0.43	10.9	4.33	110.0	0.453	0.190	0.085
Loam	0.13	3.3	3.50	88.9	0.463	0.232	0.116
Silt loam	0.26	6.6	6.69	169.9	0.501	0.284	0.135

Sandy clay loam	0.06	1.5	8.66	220.0	0.398	0.244	0.136
Clay loam	0.04	1.0	8.27	210.1	0.464	0.310	0.187
Silty clay loam	0.04	1.0	10.63	270.0	0.471	0.342	0.210
Sandy clay	0.02	0.5	9.45	240.0	0.430	0.321	0.221
Silty clay	0.02	0.5	11.42	290.1	0.479	0.371	0.251
Clay	0.01	0.3	12.60	320.0	0.475	0.378	0.265

K saturated hydraulic conductivity
Ψ suction head
φ porosity, fraction
FC field capacity, fraction
WP wilting point, fraction

Source: Rawls, W.J. et al (1983), Journal of Hydraulic Engineering, 109:1316.

12.16 Water Quality Characteristics of Urban Runoff

Constituent	**Event Mean Concentrations**
TSS (mg/L)	180 - 548
BOD (mg/L)	12 - 19
COD (mg/L)	82 - 178
Total P (mg/L)	0.42 - 0.88
Soluble P (mg/L)	0.15 - 0.28
TKN (mg/L)	1.90 - 4.18
NO_2/NO_3-N (mg/L)	0.86 - 2.2
Total Cu	43 - 118
Total Pb	182 - 443
Total Zn	202 - 633

Source: U.S. Environmental Protection Agency (1983), "Results of the Nationwide Urban Runoff Program (NURP)", Volume I, NTIS PB 94-185552, Water Planning Division, Washington DC, USA.

12.17 Depression storage

Surface	**Depression (in)**	**Depression (mm)**
Impervious surfaces	0.05 - 0.10	1.3 - 2.6
Lawns	0.10 - 0.20	2.6 - 5.1

Pasture	0.20	5.1
Forest litter	0.30	7.6

Source: ASCE (1992), "Design & Construction of Urban Stormwater Management Systems", New York, NY, USA.

12.18 Error messages

12.18.1 Error codes

The error codes that are displayed in the results report consist of a three-digit number and a brief description. In the following, a more detailed description as well as common troubleshooting options are provided.

- Codes 100-199
- Codes 200-299
- Codes 300-399
- Codes 400-499

12.18.2 Codes 1XX

[101] memory allocation error.
There is not enough physical memory in the computer to analyze the study area.

[103] cannot solve KW equations for Link xxx.
The internal solver for Kinematic Wave routing failed to converge for the specified link at some stage of the simulation. Try to change the time step or use another solver.

[105] cannot open ODE solver.
The system could not open its Ordinary Differential Equation solver.

[107] cannot compute a valid time step.
A valid time step for runoff or flow routing calculations (i.e., a number greater than 0) could not be computed at some stage of the simulation. Try to change the time step or use another solver.

[108] ambiguous outlet ID name for Subcatchment xxx.
The name of the element identified as the outlet of a subcatchment belongs to both a node and a subcatchment in the project's data base. This problem may occur only in case you import a SWMM project, as internal procedures in the program cover this case without user intervention.

[109] invalid parameter values for Aquifer xxx.
The properties entered for an aquifer object were either invalid numbers or were inconsistent with one another (e.g., the soil field capacity was higher than the porosity).

[111] invalid length for Conduit xxx.
Conduits cannot have zero or negative lengths.

[112] Elevation drop exceeds length for Conduit xxx.
In order to maintain the numerical stability of the network, the difference in elevation between the upstream and downstream nodes cannot exceed the conduit's length. This corresponds to a slope of 45% (1:1). To correct this problem you can either use a drop well, reduce the elevation difference or increase the conduit's length.

[113] invalid roughness for Conduit xxx.
Conduits cannot have zero or negative roughness values. Correct the roughness coefficient in conduit shape specifications corresponding to the solver invoked.

[114] invalid number of barrels for Conduit xxx.
The number of barrels in conduit shape specifications is zero or negative. Correct the number of barrels to at least 1.

[115] adverse slope for Conduit xxx.
Under Steady or Kinematic Wave routing, all conduits must have positive slopes. This can usually be corrected by reversing the inlet and outlet nodes of the conduit. Adverse slopes are permitted under Dynamic Wave routing.

[116] wrong elevations for Link xxx.
The elevations entered in the properties of the specific link are incorrect. This may occur when the inlet or outlet elevation is higher than the ground elevation.

[117] no cross section defined for Link xxx
Cross section geometry was never defined for the specified link. Select an appropriate conduit shape for the specific link.

[119] invalid cross section for Link xxx
Either an invalid shape or invalid set of dimensions was specified for a link's cross section in conduit shape specifications.

[121] missing or invalid pump curve assigned to Pump xxx.
Either no pump curve or an invalid type of curve was specified for a pump.

[131] the following links form cyclic loops in the drainage system.
The Steady and Kinematic Wave flow routing methods cannot be applied to systems where a cyclic loop exists (i.e., a directed path along a set of links that begins and ends at the same node). Most often the cyclic nature of the loop can be eliminated by reversing the direction of one of its links (i.e., switching the inlet and outlet nodes of the link). The names of the links that form the loop will be listed following this message.

[133] Node xxx has more than one outlet link.
Under Steady and Kinematic Wave flow routing, a junction node can have only a single outlet link.

[134] Node xxx has more than one DUMMY outlet link.
Only a single conduit with a DUMMY cross section can be directed out of a node.

[135] Divider xxx does not have two outlet links.
Flow divider nodes must have two outlet links connected to them.

[136] Divider xxx has invalid diversion link.
The link specified as being the one carrying the diverted flow from a flow divider node was defined with a different inlet node. This problem may occur only in case you import a SWMM project, as internal procedures in the program cover this case without user intervention.

[137] Weir Divider xxx has invalid parameters.
The parameters of a weir-type divider node either are non-positive numbers or are inconsistent (i.e., the value of the discharge coefficient times the weir height raised to the 3/2 power must be greater than the minimum flow parameter).

[138] Node xxx has initial depth greater than maximum depth.
Self-explanatory.

[139] Regulator xxx is the outlet of a non-storage node.
Under Steady and Kinematic Wave flow routing, orifices, weirs, and outlets links can only be used as outflow links from storage nodes.

[141] Outfall xxx has more than 1 inlet link or an outlet link.
An outfall node is only permitted to have one link attached to it.

[143] Regulator xxx has invalid cross-section shape.
An orifice must have either a Circular or Closed Rectangle shape, while a weir must have either an Open Rectangle, Trapezoidal, or Triangular shape.

[145] Drainage system has no acceptable outlet nodes.
Under Dynamic Wave flow routing, there must be at least one node designated as an outfall.

[151] a Unit Hydrograph in set xxx has invalid time base.
The time base of a unit hydrograph must be greater than 0.

[153] a Unit Hydrograph in set xxx has invalid response ratios.
The response ratios for a set of unit hydrographs (the short-, medium-, and long-term response hydrographs) must be between 0 and 1.0 and cannot add up to a value greater than 1.0.

[155] invalid sewer area for RDII at Node xxx.
The sewer area contributing RDII inflow to a node cannot be a negative number.

[157] Inconsistent rainfall format for Rain Gage xxx.
When two or more rain gages reference the same time series data, this fatal error message is generated if the Rainfall Formats (intensity, volume, or cumulative volume) for the gages are not all the same.

[161] cyclic dependency in treatment functions at Node xxx.
An example would be where the removal of pollutant 1 is defined as a function of the removal of pollutant 2 while the removal of pollutant 2 is defined as a function of the removal of pollutant 1.

[171] Curve xxx has its data out of sequence.
The X-values of a curve object must be entered in increasing order.

[173] Time Series xxx has its data out of sequence.
The time (or date/time) values of a time series must be entered in sequential order.

[181] Invalid snow melt climatology parameters.
The ATI Weight or Negative Melt Ratio parameters are not between 0 and 1 or the site latitude is not between -60 and +60 degrees.

[182] Invalid parameters for Snow pack xxx.
A snow pack's minimum melt coefficient is greater than its maximum coefficient; the fractions of free water capacity or impervious plowable area are not between 0 and 1; or the snow removal fractions sum to more than 1.0.

[191] simulation start date comes after ending date.
Self-explanatory.

[193] report start date comes after ending date.
Self-explanatory.

[195] reporting time step is less than routing time step.
Self-explanatory. Change the report time step or the routing time step.

12.18.3 Codes 2XX

[200] one or more errors in input file.
This message appears when one or more input file parsing errors (the 200-series errors) occur.

[201] too many characters in input line.
A line in the input file cannot exceed 1024 characters.

[203] too few items at line n of input file.
Not enough data items were supplied on a line of the input file. This problem may occur only in case you import a SWMM project, as internal procedures in the program cover this case without user intervention.

[205] invalid keyword at line n of input file.
An unrecognized keyword was encountered when parsing a line of the input file. This problem may occur only in case you import a SWMM project, as internal procedures in the program cover this case without user intervention.

[207] duplicate ID name at line n of input file.
An ID name used for an object was already assigned to an object of the same category. This problem may occur only in case you import a SWMM project, as internal procedures in the program cover this case without user intervention.

[209] undefined object xxx at line n of input file.
A reference was made to an object that was never defined. An example would be if node 123 were designated as the outlet point of a subcatchment, yet no such node was ever defined in the study area.

[211] invalid number xxx at line n of input file.
Either a non-numeric character was encountered where a numerical value was

expected or an invalid number (e.g., a negative value) was supplied. This problem may occur only in case you import a SWMM project, as internal procedures in the program cover this case without user intervention.

[213] invalid date/time xxx at line n of input file.
An invalid format for a date or time was encountered. Dates must be entered as month/day/year and times as either decimal hours or as hour:minute:second. This problem may occur only in case you import a SWMM project, as internal procedures in the program cover this case without user intervention.

[217] control rule clause out of sequence at line n of input file.
Errors of this nature can occur when the format for writing control rules is not followed correctly.

[219] data provided for unidentified transect at line n of input file.
This problem may occur only in case you import a SWMM project, as internal procedures in the program cover this case without user intervention.

[221] transect station out of sequence at line n of input file.
The station distances specified for the transect of an irregular cross section must be in increasing numerical order starting from the left bank.

[223] Transect xxx has too few stations.
A transect for an irregular cross section must have at least 2 stations defined for it.

[225] Transect xxx has too many stations.
A transect cannot have more than 1500 stations defined for it. This problem may occur only in case you import a SWMM project, as internal procedures in the program cover this case without user intervention.

[227] Transect xxx has no Manning's N.
No Manning’s N was specified for a transect. Correct the transect specified.

[229] Transect xxx has invalid overbank locations.
The distance values specified for either the left or right overbank locations of a transect do not match any of the distances listed for the transect's stations.

[231] Transect xxx has no depth.
All of the stations for a transect were assigned the same elevation. Correct the transect specified.

[233] invalid treatment function expression at line n of input file.
A treatment function supplied for a pollutant at a specific node is either not a correctly formed mathematical expression or refers to unknown pollutants, process variables, or math functions.

12.18.4 Codes 3XX

[301] files share same names.
The input, report, and binary output files specified on the command line cannot have the same names.

[303] cannot open input file.

The input file either does not exist or cannot be opened (e.g., it might be in use by another program).

[305] cannot open report file.
The report file cannot be opened (e.g., it might reside in a directory to which the user does not have write privileges).

[307] cannot open binary results file.
The binary output file cannot be opened (e.g., it might reside in a directory to which the user does not have write privileges).

[309] error writing to binary results file.
There was an error in trying to write results to the binary output file (e.g., the disk might be full or the file size exceeds the limit imposed by the operating system).

[311] error reading from binary results file.
There was an error in reading results saved to the binary output file when writing results to the report file.

[313] cannot open scratch rainfall interface file.
SWMM could not open the temporary file it uses to collate data together from external rainfall files. Make sure that the temporary folder exists and is accessible.

[315] cannot open rainfall interface file xxx.
The program could not open the specified interface file file, possibly because it does not exist or because the user does not have write privileges to its directory.

[317] cannot open rainfall data file xxx.
An external rainfall data file could not be opened, most likely because it does not exist.

[319] invalid format for rainfall interface file.
The program was trying to read data from a designated rainfall interface file with the wrong format (i.e., it may have been created for some other project or actually be some other type of file).

[321] no data in rainfall interface file for gage xxx.
This message occurs when a project wants to use a previously saved rainfall interface file, but cannot find any data for one of its rain gages in the interface file.

[323] cannot open runoff interface file xxx.
A runoff interface file could not be opened, possibly because it does not exist or because the user does not have write privileges to its directory.

[325] incompatible data found in runoff interface file.
The program was trying to read data from a designated runoff interface file with the wrong format (i.e., it may have been created for some other project or actually be some other type of file).

[327] attempting to read beyond end of runoff interface file.

This error can occur when a previously saved runoff interface file is being used in a simulation with a longer duration than the one that created the interface file.

[329] error in reading from runoff interface file.
A format error was encountered while trying to read data from a previously saved runoff interface file.

[330] different hotstart interface files share the same name.
You have selected a common name for more than one hotstart interface file.

[331] cannot open hotstart interface file xxx.
A hotstart interface file could not be opened, possibly because it does not exist or because the user does not have write privileges to its directory.

[333] incompatible data found in hotstart interface file.
The program was trying to read data from a designated hotstart interface file with the wrong format (i.e., it may have been created for some other project or actually be some other type of file).

[335] error in reading from hotstart interface file.
A format error was encountered while trying to read data from a previously saved hotstart interface file.

[336] no climate file specified for evaporation and/or wind speed.
This error occurs when the user specifies that evaporation or wind speed data will be read from an external climate file, but no name is supplied for the file.

[337] cannot open climate file xxx.
An external climate data file could not be opened, most likely because it does not exist.

[338] error in reading from climate file xxx.
The program was trying to read data from an external climate data file with the wrong format.

[339] attempt to read beyond end of climate file xxx.
The specified external climate does not include data for the period of time being simulated.

[341] cannot open scratch RDII interface file.
The program could not open the temporary file it uses to store RDII flow data. Make sure that the temporary folder exists and is accessible.

[343] cannot open RDII interface file xxx.
An RDII interface file could not be opened, possibly because it does not exist or because the user does not have write privileges to its directory.

[345] invalid format for RDII interface file.
The program was trying to read data from a designated RDII interface file with the wrong format (i.e., it may have been created for some other project or actually be some other type of file).

[351] cannot open routing interface file xxx.

A routing interface file could not be opened, possibly because it does not exist or because the user does not have write privileges to its directory.

[353] invalid format for routing interface file xxx.
The program was trying to read data from a designated routing interface file with the wrong format (i.e., it may have been created for some other project or actually be some other type of file).

[355] mismatched names in routing interface file xxx.
The names of pollutants found in a designated routing interface file do not match the names used in the current project.

[357] inflows and outflows interface files have same name.
In cases where a run uses one routing interface file to provide inflows for a set of locations and another to save outflow results, the two files cannot both have the same name.

[361] Could not open external file used for Time Series xxx.
The program could not open the specified external file, possibly because it does not exist or because the user does not have write privileges to its directory.

[363] Invalid data in external file used for Time Series xxx.
The external time series file has been located but it contains invalid data or data out of order. Please correct the data contained in the external file.

12.18.5 Codes 4XX

[401] Generic solver error.
If the error persists after you have rebooted your computer, contact technical support for further assistance.

[402] Impossible to load project as it is already loaded.
Reboot your computer to correct this problem.

[403] The project cannot be loaded or the last run was not completed.
Reboot your computer to correct this problem.

[405] The number or results exceeds the maximum value.
The number or results is excessive. Select an earlier end date or increase the time step of the report.

Keyword index

- D -

- E -

- F -

- G -

- H -

- I -

- J -

- K -

- L -

- M -

- N -

- O -

- P -

- Q -

- R -

- S -

- T -

- U -

- V -

- W -

- X -

- Y -

- Z -

This page was intentionally left blank.