TECHNO logismiki

Advanced Technical Software

5 Imitou str, 15561, Cholargos, Athens, Greece

Works 2013

River flow

Version 11.0.0

© 2012 Technologismiki

USER GUIDE

River flow

Technologismiki

River flow

© 2012 Technologismiki

Publisher

TechnoLogismiki

Editors

Fotis Fotopoulos
Aristotelis Charalampakis

Technical Assistance

Antigoni Egglezou

Table of Contents

Chapter

I

1   About the program

1.1   What does the program do?

This program calculates the flood lines of a river or stream. You can optionally enter the coordinates of the riverbed from DXF/GIS/GPS and export the results i.e. flood lines, the riverbed etc in DXF/GIS/GPS format.

The following drawings are produced:

- Plan view with table of coordinates.
- Profile of the river.
- Sections including the water level.
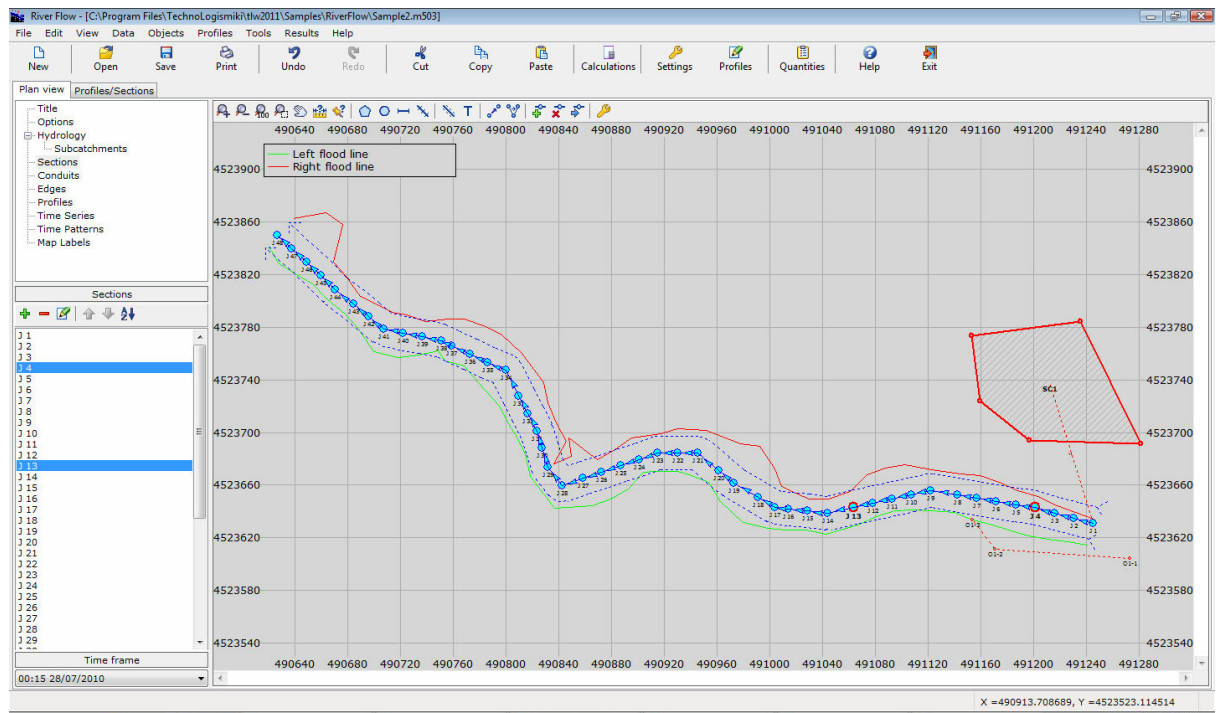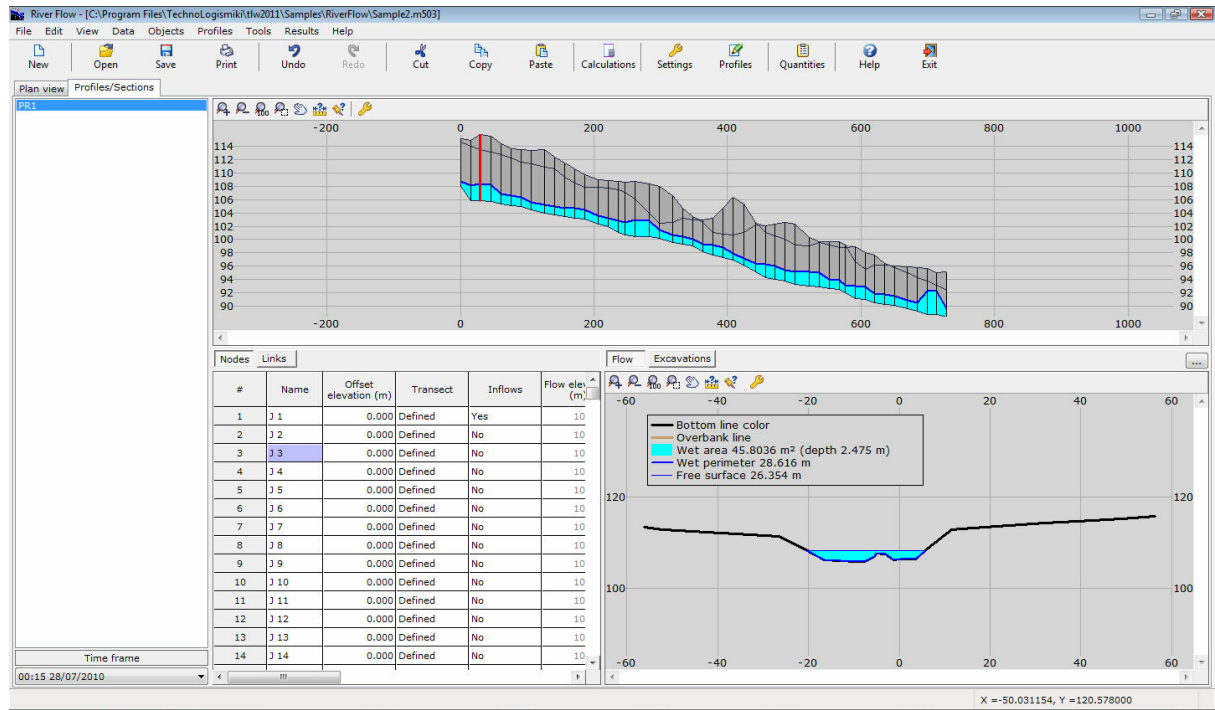
The printouts include the water level, which is calculated based on the energy losses.

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 File

2.1 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
  - Plan view from GTM
  - Plan view from ArcView Shapefile
  - Sections from GRD
  - Section from PCS
  - Sections from ArcView Shapefile
  - River from PROST
  - Background from DXF
  - Contours from DXF
  - Satellite image
- Export
  - Export selection
  - Plan view to DXF
  - Plan view to ArcView Shapefile
  - Plan view GTM
  - Plan view to BMP picture
  - Project to SWMM
  - Sections to DXF
  - Sections to GRD
  - Section to PCS
  - Section to ArcView Shapefile
- Print Setup
- Print
- Print to
  - Print to File
  - Print to Word
  - Print to Word (Formatted)
  - Print to Excel
- Exit

2.2 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.

2.3   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 "River networks project" with the extension .m503.
4. Select the file by clicking on it.
5. Select **Open** to open the selected file. Select **Cancel** to cancel the operation.
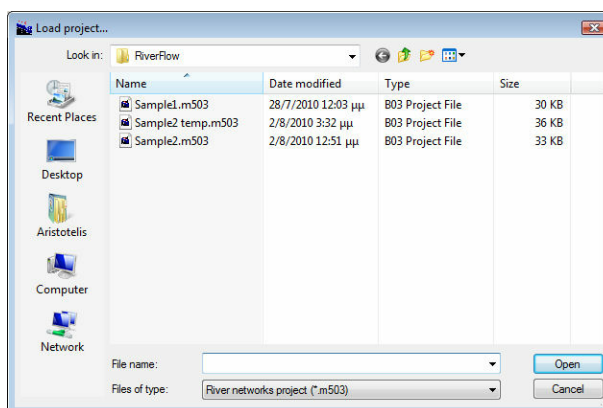
> **NOTE**: You can find sample projects in the installation folder of the program:
> C:\Program Files\TechnoLogismiki\TLW2013\Samples\RiverFlow

Supported file types

- **M03** (River networks project): Files created by versions 2012 and 2013 of River Flow.
- **M503** (River networks project): Files created by versions 2011, 2010, 2008, 2007 and 5 of River Flow.
- **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 backward compatibility (up to version 5); 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.

2.4 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.

2.5 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.
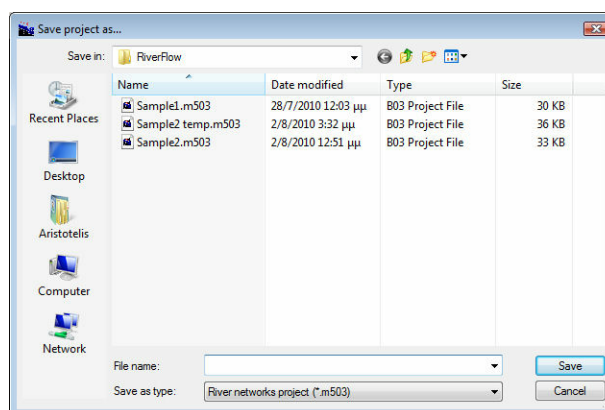
> **NOTE**: If a file with the same name and in the same path already exists, a warning message will appear that asks whether to overwrite the file or not. If you answer Yes,

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.

2.6 Import

2.6.1 Plan view from DXF

With this option, you can import data from a DXF file.

To import plan view data from a DXF file:

1. Select **Import > Plan view from DXF/GTM/GIS** from the **File** menu. The following form appears:
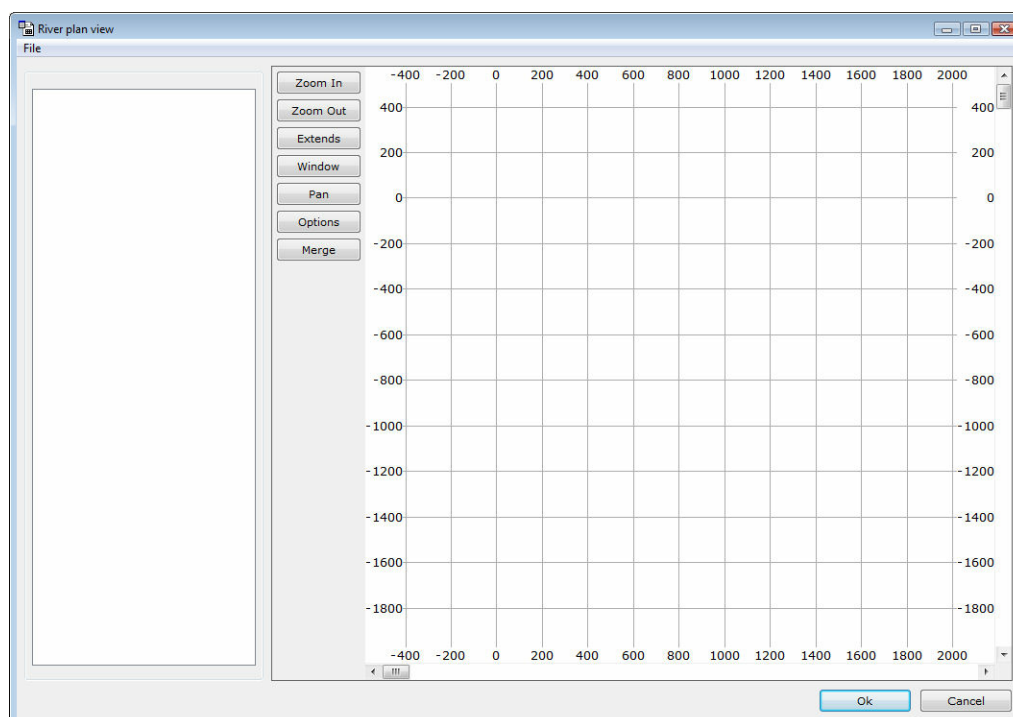
2. Select **Import from DXF** from the **File** menu:
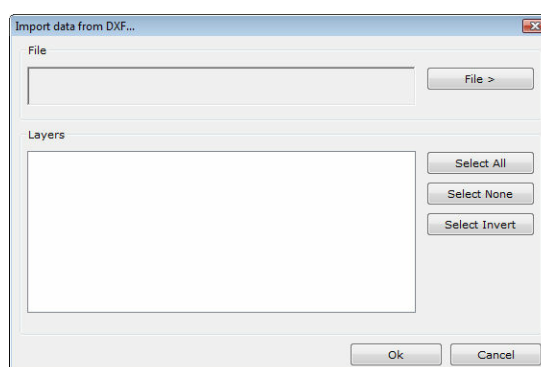
3. Click **File >** to select the DXF file. The file selection dialog box appears:
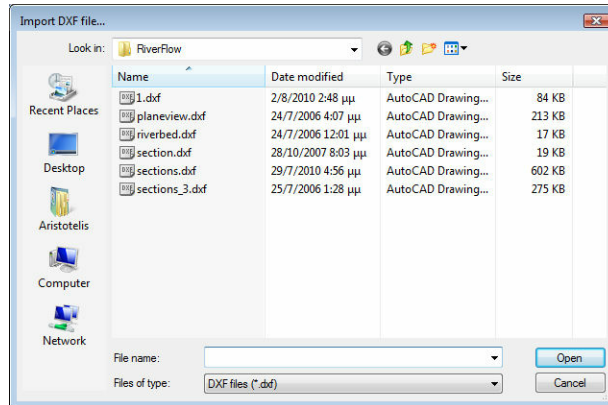
4. Select the path of the file.
5. Select the file type from the **Files of type** drop-down list. The default option is "DXF file" with the extension .dxf.
6. Select the file by clicking on it.
7. Select **Open** to open and analyze the file. The list in the **Layers** frame of step 1 is loaded with the layers contained in the DXF file:
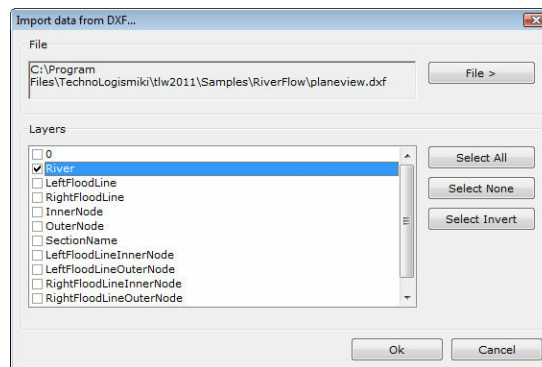
8. Select one or more layers containing the data. The data should be defined in polylines or a series of lines connected to each other. The connection must be exact, therefore you may need to use Snap or OSnap when using CAD software. The program will create stations at the nodes of the polylines and 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.
9. Select **Ok** to import the data and close the dialog box. The data are transferred to the form of step 1. Select **Cancel** to close the dialog box without applying any changes.

10. On the left, there is a tree-view with the river network. In case of secondary branches, you can modify the preselected main routes so that they comply with reality.

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 or more branches:

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

11. Select **Ok** to import the data to the main form of the program 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


2.6.2   Plan view from GTM

With this option, you can import data from a GTM file.

To import plan view data from a GTM file:

1. Select **Import > Plan view from DXF/GTM/GIS** from the **File** menu. The following form appears:


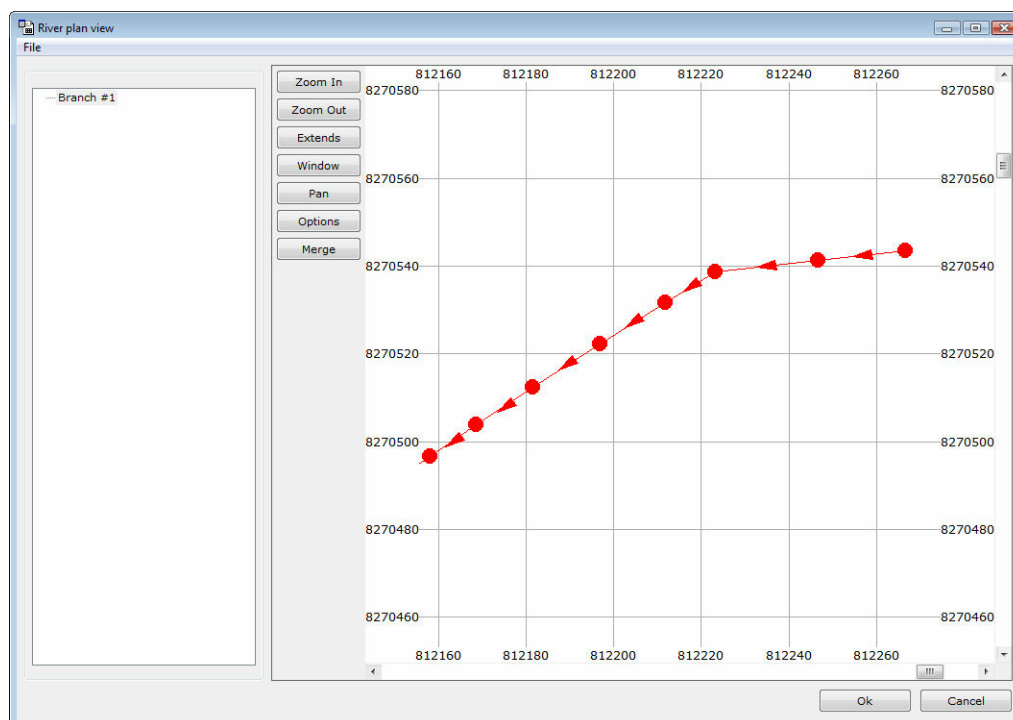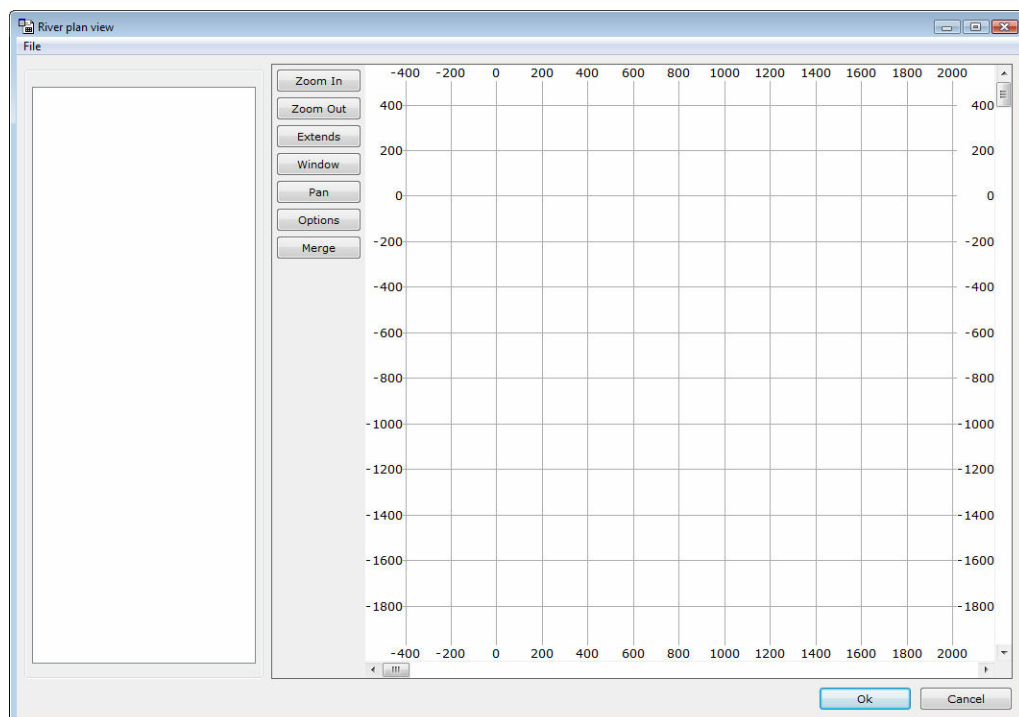**2.** Select **Import from GTM** from the **File** menu:
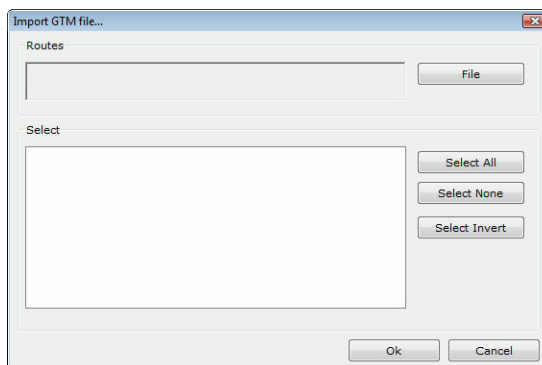
3. Click **File >** to select the GTM file. The file selection dialog box appears:
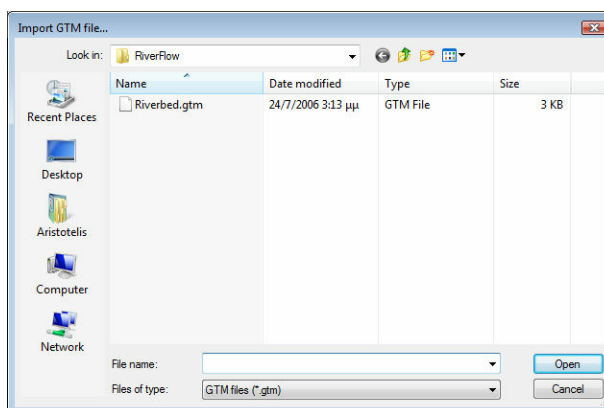
4. Select the path of the file.
5. Select the file type from the **Files of type** drop-down list. The default option is "GTM file" with the extension .gtm.
6. Select the file by clicking on it.
7. Select **Open** to open and analyze the file. The list in the **Select** frame of step 1 is loaded with the layers contained in the DXF file:
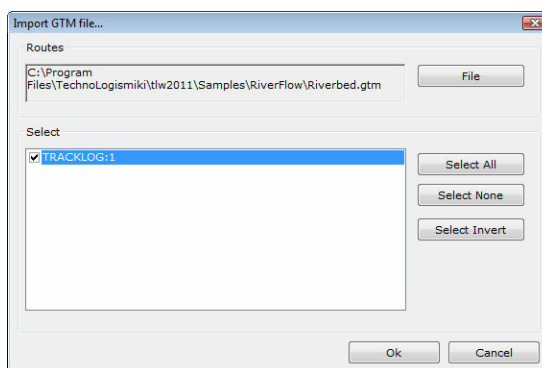
8. Select one or more routes. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
9. Select **Ok** to import the data and close the dialog box. The data are transferred to the form of step 1. Select **Cancel** to close the dialog box without applying any changes.

10. On the left, there is a tree-view with the river network. In case of secondary branches, you can modify the preselected main routes so that they comply with reality.

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 or more branches:

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

11. Select **Ok** to import the data to the main form of the program and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

2.6.3 Plan view from GIS (ArcView Shapefile)

With this option, you can import data from an ArcView Shapefile file.

To import plan view data from an ArcView Shapefile:

1. Select **Import > Plan view from DXF/GTM/GIS** from the **File** menu. The following form appears:
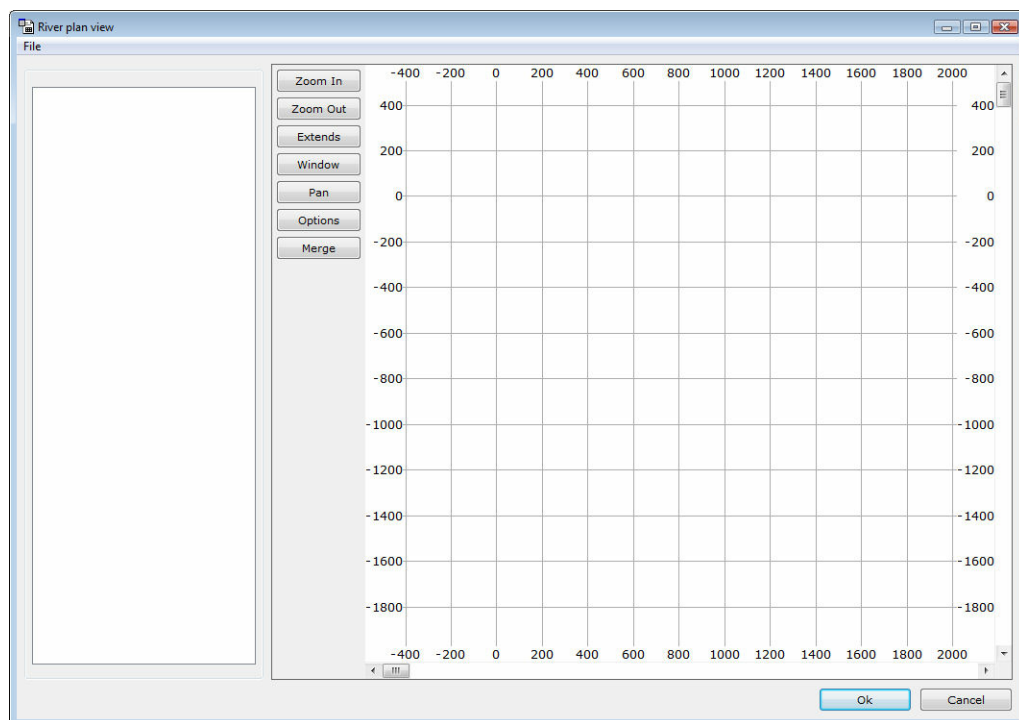
2. Select **Import from Arcview Shapefile** from the **File** menu.
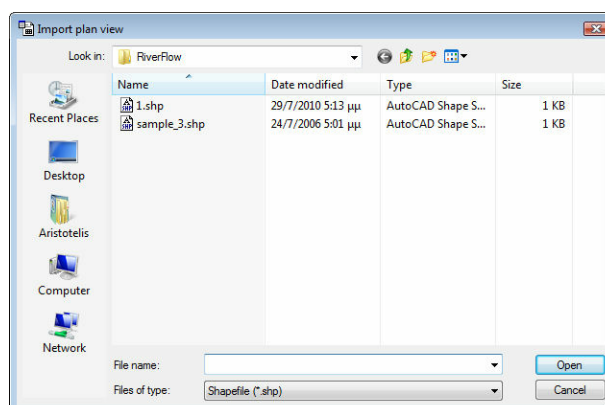
3. Select the path of the file.

4. Select the file type from the **Files of type** drop-down list. The default option is "Shapefile" with the extension .shp.
5. Select the file by clicking on it.
6. Select **Open** to open and analyze the file. The data are transferred to the form of step 1. Select **Cancel** to close the dialog box without applying any changes.

7. On the left, there is a tree-view with the river network. In case of secondary branches, you can modify the preselected main routes so that they comply with reality.

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 or more branches:

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

8. Select **Ok** to import the data to the main form of the program and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

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

2.6.4   Sections from GRD

With this option, you can import sectional data from GRD files. Note that you must have already defined the nodes and profiles prior to importing the sectional data.

To import sectional data from GRD files:

1. Select **Import** from the **File** menu.
2. Select **Sections from GRD** from the **Import** menu. The following form appears:
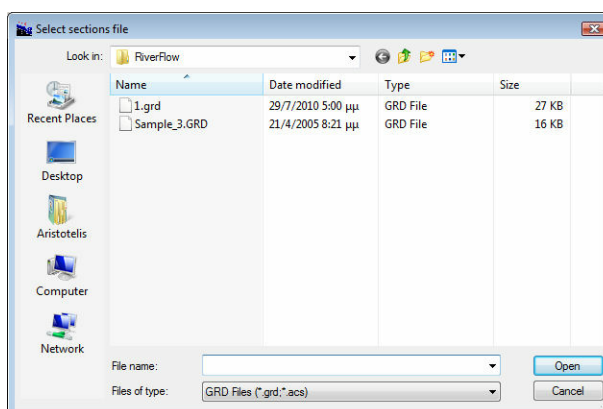

3. Select the path of the file.
4. Select the file type from the **Files of type** drop-down list. The default option is "GRD files" with the extension .grd.
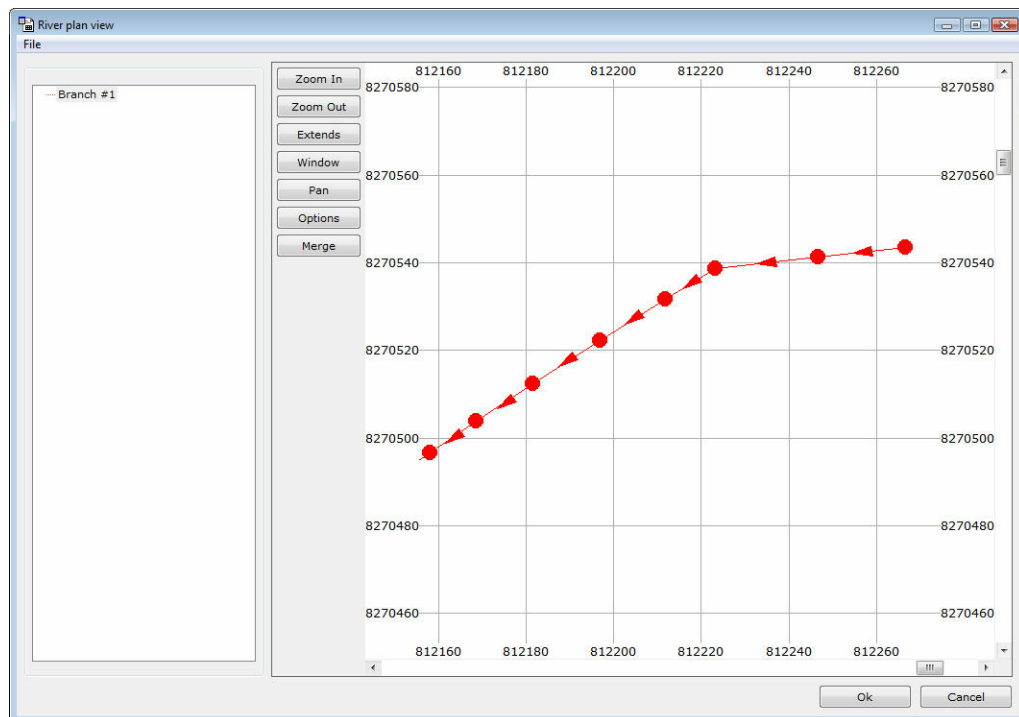**5.** Select the file by clicking on it.
6. Select **Open** to analyze the file.
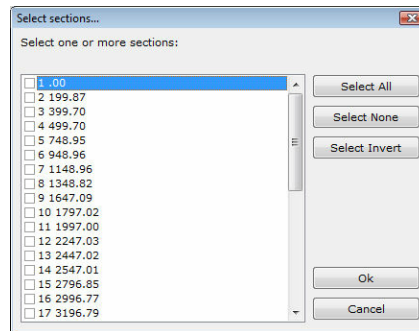
7. Select one or more sections to be imported. Press **Ok** to proceed, or **Cancel** to cancel.
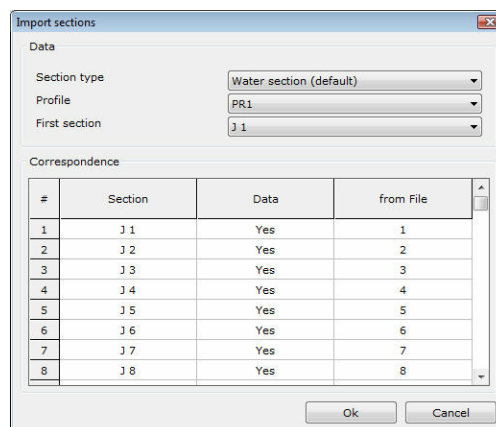
8. Select the section type (water flow, natural ground or excavation line). Select the profile for the application as well as its first section. The correspondence for the overwrite of data is shown in the table below. Press **Ok** to proceed, or **Cancel** to cancel.
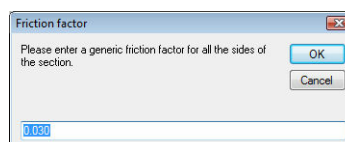
9. Enter a generic friction factor for all section sides. Press **Ok** to finalize and import the data, or **Cancel** to cancel.

2.6.5   Section from PCS

With this option, you can import sectional data from PCS files.

To import sectional data from PCS files:

1. Select a single node for the application of the sectional data.
2. Select **Import** from the **File** menu.
3. Select **Section from PCS** from the **Import** menu. The following form appears:
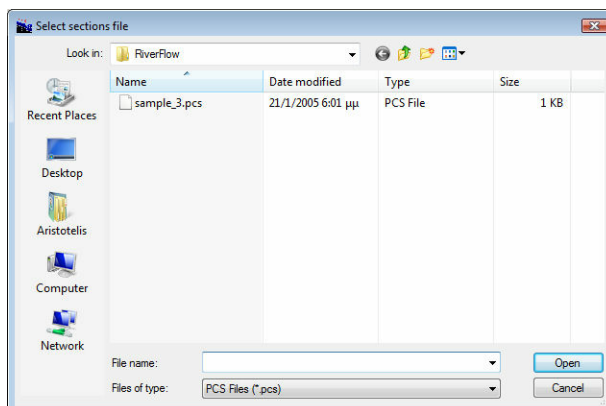
4. Select the path of the file.
5. Select the file type from the **Files of type** drop-down list. The default option is "PCS files" with the extension .pcs.
6. Select the file by clicking on it.
7. Select **Open** to finalize the data import, or **Cancel** to cancel the operation.

2.6.6   Sections from ArcView Shapefile

With this option, you can import sectional data from ArcView shapefiles. The program is able to open and analyze the following types of shapefiles (ArcView v3.x or later, MapInfo v4.x or later):

- Nullshape
- Point/PointM/PointZ
- MultiPoint/MultiPointM/MultiPointZ
- PolyLine/PolyLineM/PolyLineZ

To import sectional data from shapefiles:

1. Select one or more nodes for the application of the sectional data.
2. Select **Import** from the **File** menu.
3. Select **Sections from ArcView Shapefile** from the **Import** menu. The following form appears:
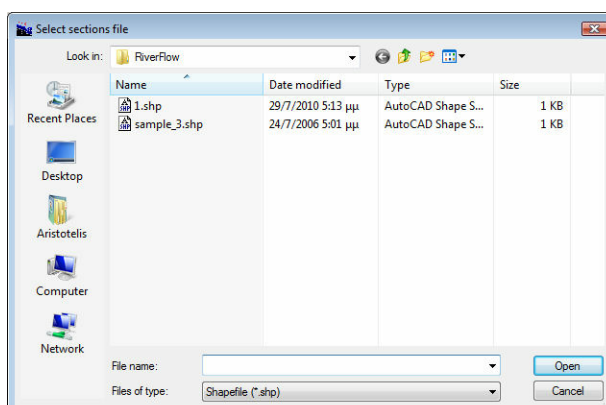
4. Select the path of the file.
5. Select the file type from the **Files of type** drop-down list. The default option is

"Shapefile" with the extension .shp.
6. Select the file by clicking on it.
7. Select **Open** to finalize the data import, or **Cancel** to cancel the operation.

2.6.7 River from PROST

With this option, you can import river data from PROST text files. Note that you must have already defined the nodes and profiles prior to importing the data.

To import river data from PROST text files:

1. Select **Import** from the **File** menu.
2. Select **River from PROST** from the **Import** menu. The following form appears:
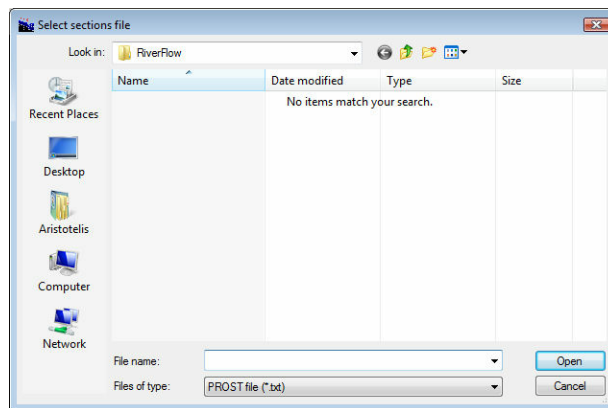
3. Select the path of the file.
4. Select the file type from the **Files of type** drop-down list. The default option is "PCS files" with the extension .pcs.
5. Select the file by clicking on it.
6. Select **Open** to finalize the data import, or **Cancel** to cancel the operation.

2.6.8 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**

2.6.9   Contours from DXF

With this option, you can import elevation contours from DXF files. These can be used to produce the sections easily using an automatic procedure.

To import contour data from DXF:

1. Select **Import** from the **File** menu.
2. Select **Contours 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 contours, from the menu select **Data > Contours.**

2.6.10   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:
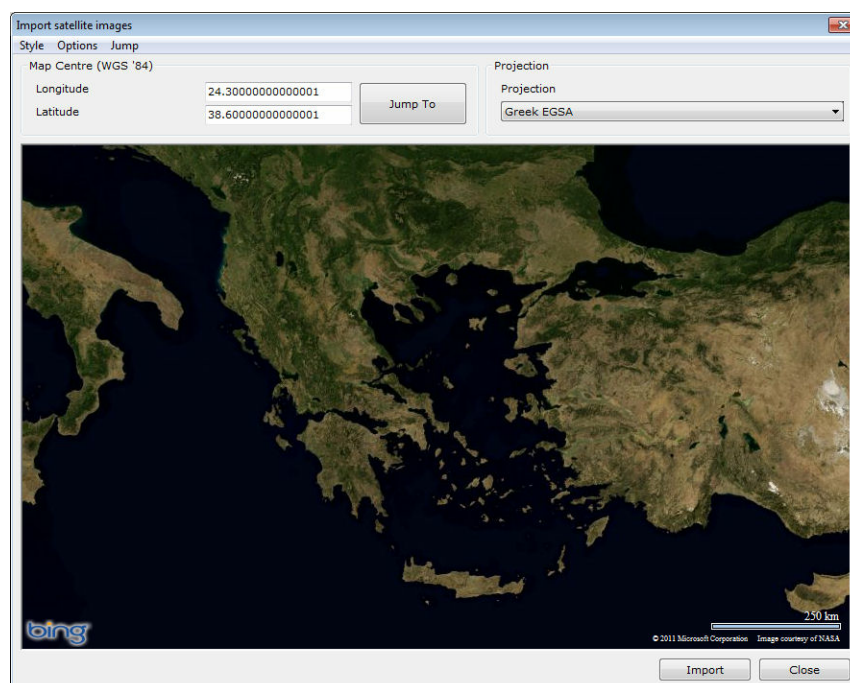
2. Navigate to the area of the project. You can pan the image by holding down the left mouse button. Using the roller you can change the resolution. Alternatively, enter the

longitude and latitude in decimal degrees and press **Jump to:**
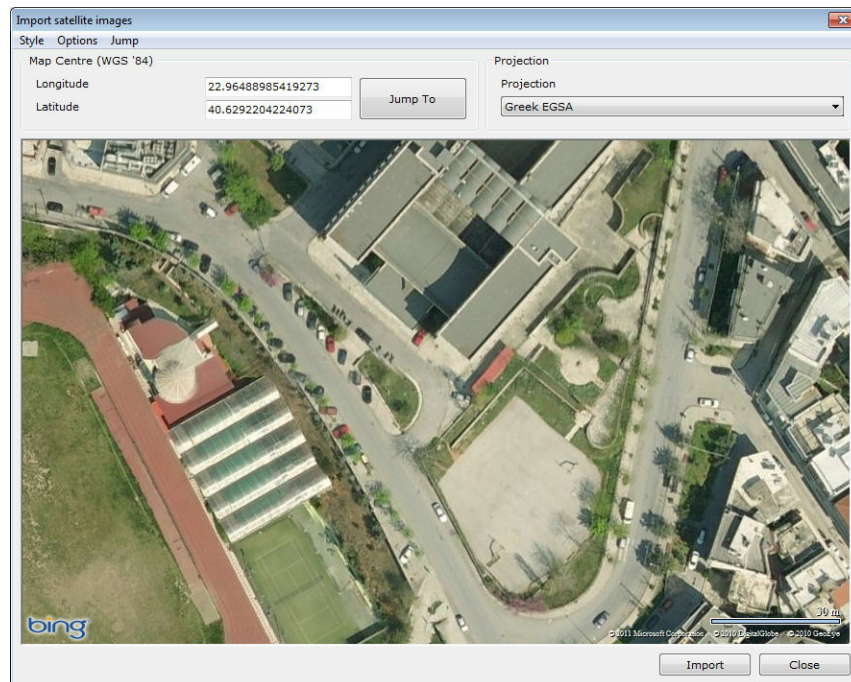
3. When you locate the area, select the appropriate resolution (the resolution varies, depending on the quality of the satellite image), and press Import. The current viewport is imported as a background image:
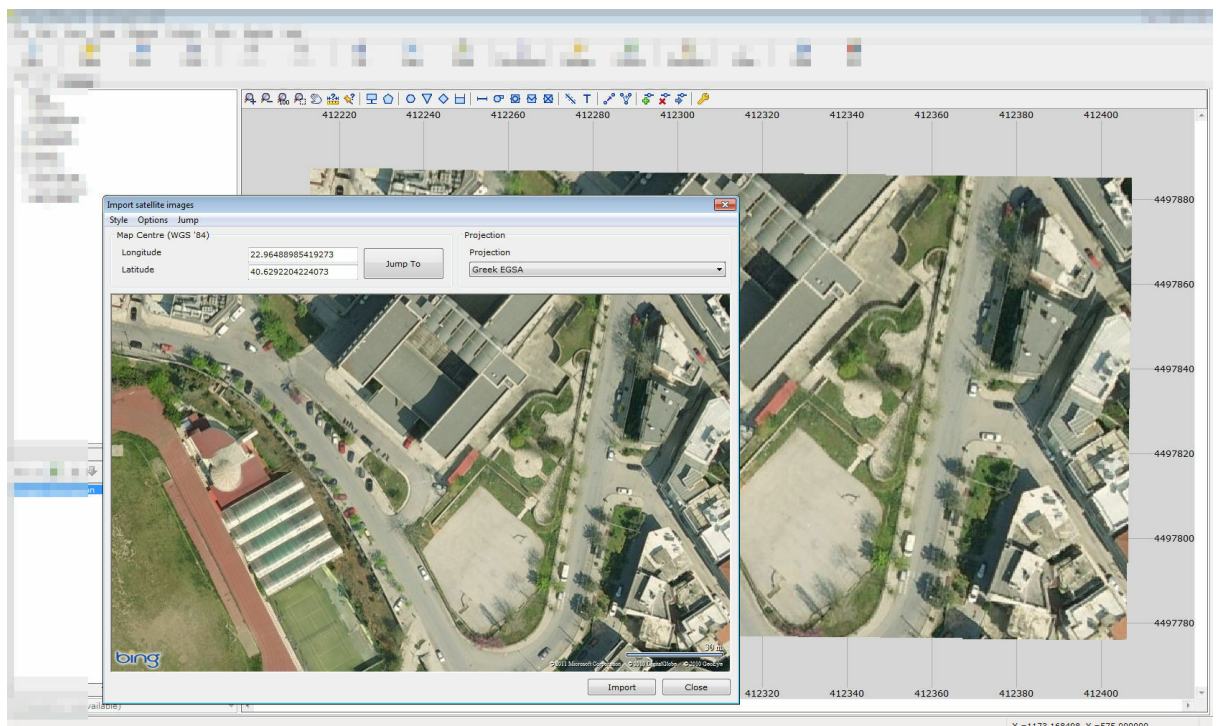
4. Without closing the window, pan the viewport and press Import again. A new image

is imported, which may overlap with the previous one. When you cover the whole area of interest, press **Close**:
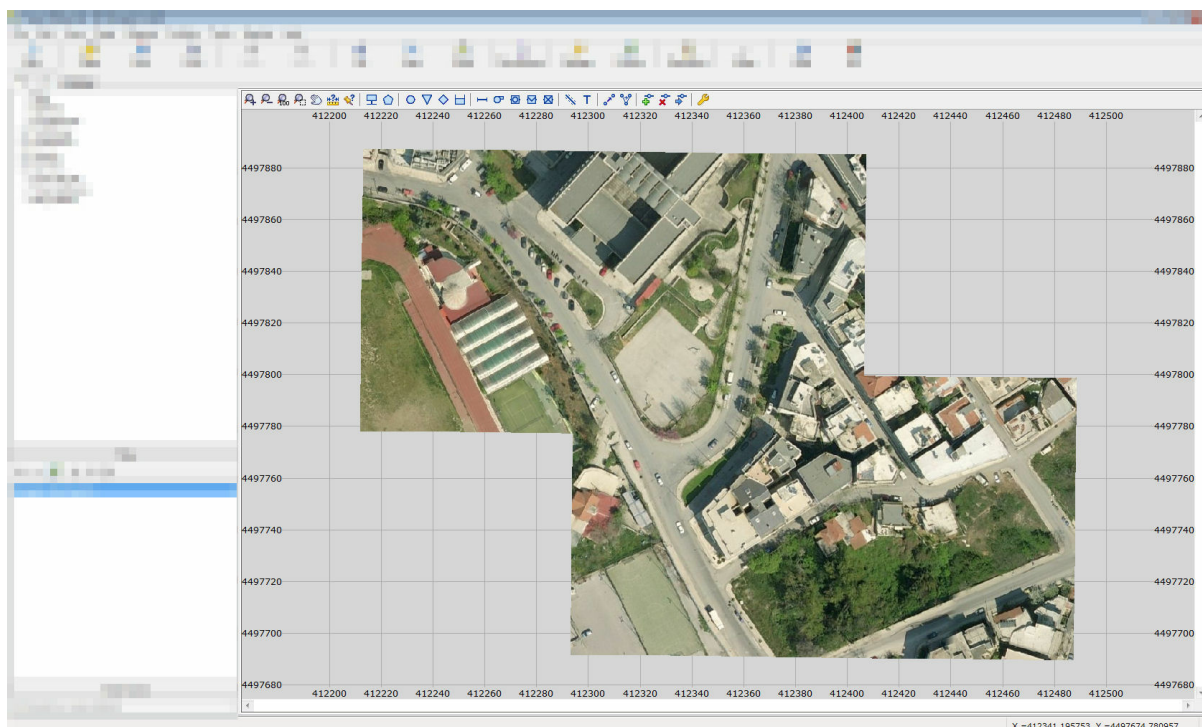
In the satellite image form, the following options are available:

- 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.

2.7 Export

2.7.1 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:
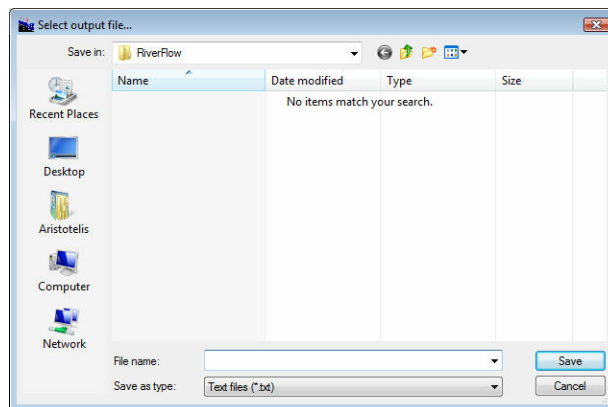
5. Select the path of the file.
6. Type the filename in the **File name** text box.
7. Select **Save** to save the file with the selected filename and path. Select **Cancel** to cancel the operation.

2.7.2 Plan view to DXF

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

- Sections, station names
- Flood lines
- Edge lines, including table of coordinates.

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:
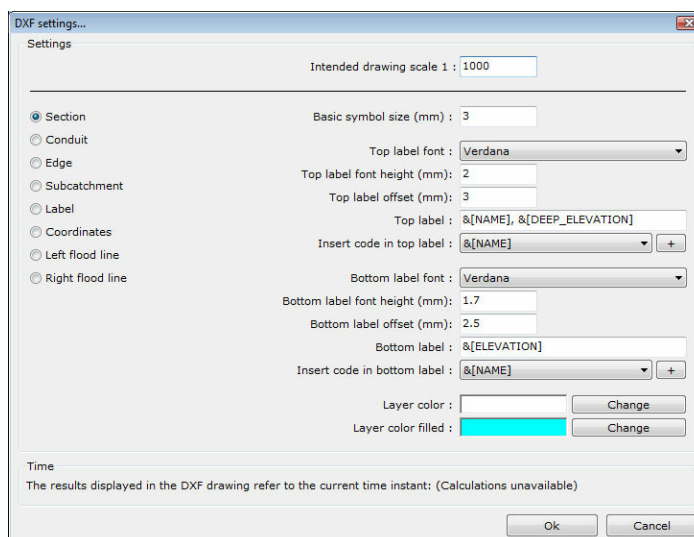
3. Select the appropriate settings:

- **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**.
  - **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.

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.

2.7.3   Plan view to ArcView Shapefile

With this option, two ArcView Shapefiles containing the plan view data are created. The first file contains sectional data and the other contains link 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:
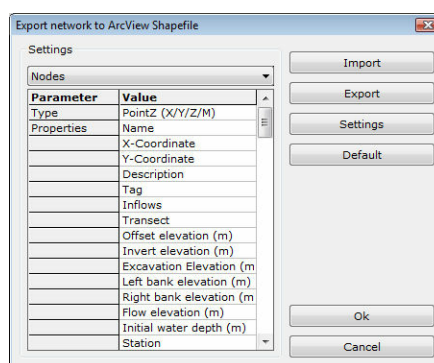
4. Select one of **Nodes**, **Links**, **Subcatchments** from the drop-down list.
5. Select **Settings** to customize the shape type and the properties that will be included:
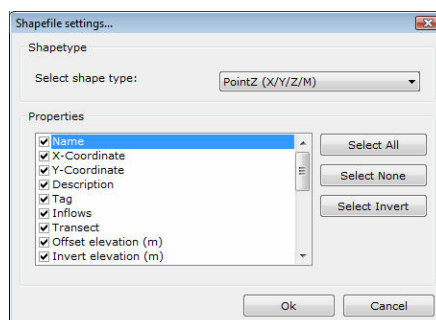
6. Select the appropriate shape type.

> **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.
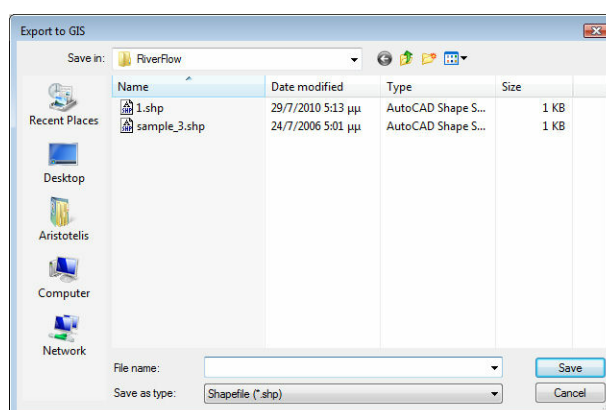
9. Select the path of the file.

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.

2.7.4 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, Node names
- Links

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.
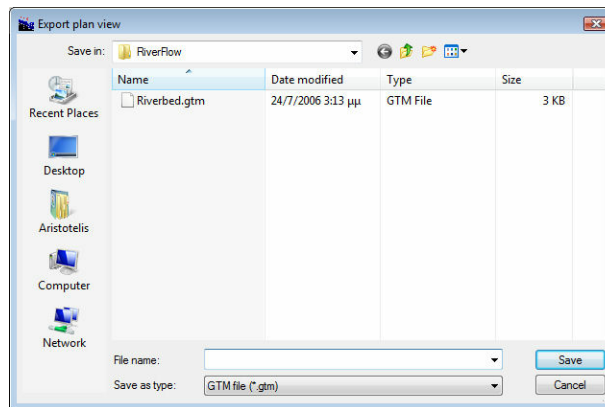
7. Select the path of the file.
8. Type the filename in the **File name** text box.
9. Select **Save** to save the file with the selected filename and path. Select **Cancel** to cancel the operation.
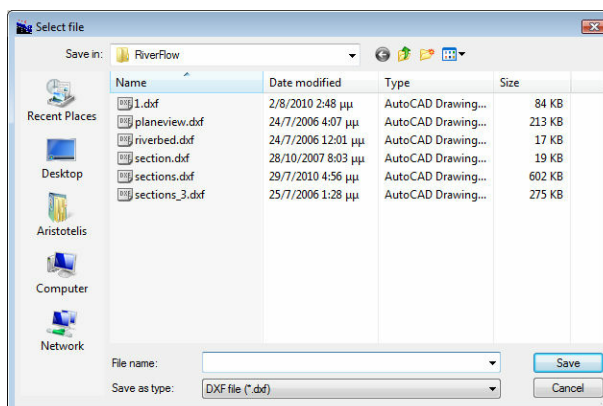
> **NOTE**: If nothing is visible when viewing the newly created GTM file, select **Zoom Extents** to view the whole drawing.

2.7.5 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:
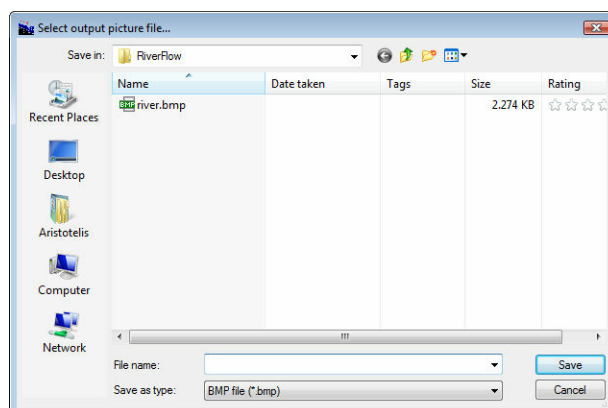
3. Select the path of the file.
4. Type the filename in the **File name** text box.
5. Select **Save** to save the file with the selected filename and path. Select **Cancel** to cancel the operation.

2.7.6 Project to SWMM

With this option, the project is exported to a SWMM file.

To export the project to a SWMM file:

1. Select **Export** from the **File** menu..
2. Select **Project to SWMM** from the **Export** menu. The following form will appear:
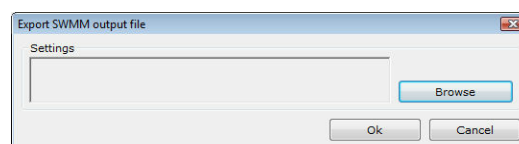
3. Click **Browse** to select a new SWMM file. The file selection dialog box appears.
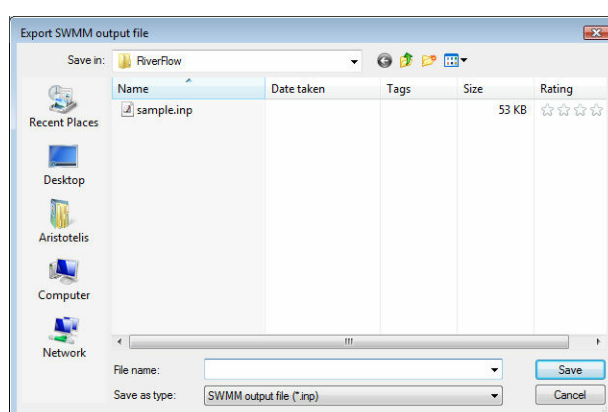
4. Select the path of the file.
5. Type the filename in the **File name** text box to select a new file.
6. Select **Save** to finalize your selection.
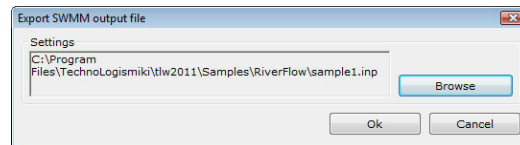
7. Press **Ok** to close the form and finalize the export procedure or press **Cancel** to close the form and cancel the export procedure.

2.7.7 Sections to DXF

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

- Section edges
- Ground line
- Excavation line
- Water area and level
- Table with distances and elevations
- Title and station name

To export the sections to DXF file:

1. Select **Export** from the **File** menu.
2. Select **Sections to DXF file** from the **Export** menu.
3. The following dialog box appears:
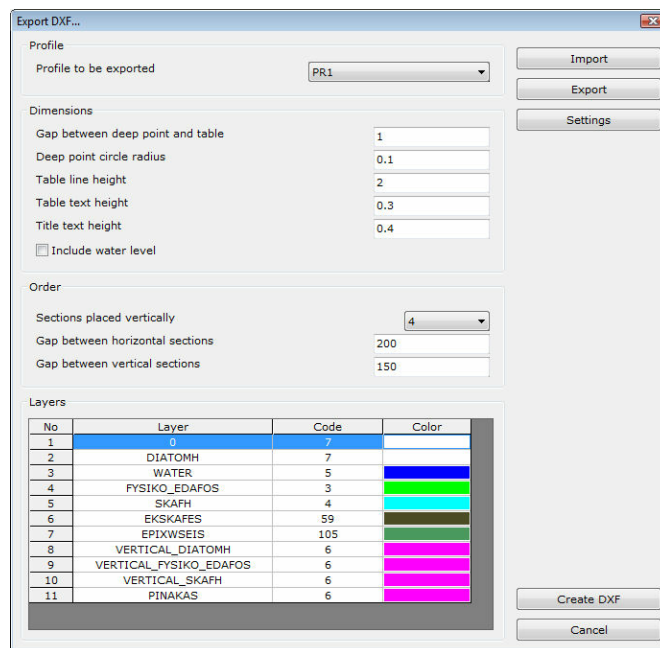
4. Select the appropriate settings (described below).
5. Select **Export** to export the settings to a file. In this way, the settings can be stored in a file and used in other projects. Select **Import** to import and apply such a file.
6. Select **Settings** to modify the DXF driver settings. The following dialog box

appears:
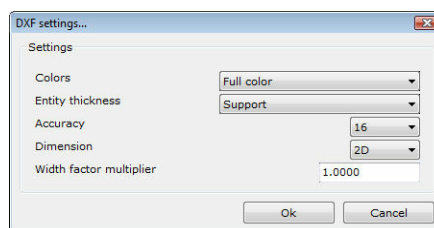
- **Colors**: Select one of Full color or Monochrome. If Monochrome is selected, all color preferences are ignored.
- **Entity thickness**: Select one of Ignore or Support, depending on whether you want to have line thickness in the drawing.
- **Accuracy**: Select the number of decimal places (1 to 16) that will be used when exporting the objects.
- **Dimension**: Select one of 2D or 3D, depending on whether you want elevation information to be included in the drawing.
- **Width factor multiplier**: type a value between 0 and 10.0 that will be used when calculating the width of lines.

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

7. Select the **Profile**, the sections of which you wish to export.
8. Select **Create DXF** to apply the changes and proceed with the creation of the DXF file. Select **Cancel** to close the dialog box without applying any changes.
9. The file selection dialog box appears:
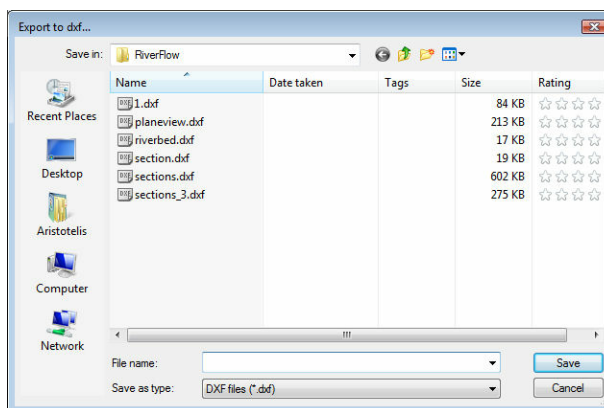
10. Select the path of the file.
11. Type the filename in the **File name** text box.
12. Select **Save** to save the file with the selected filename and path. Select **Cancel** to cancel the operation.

Other drawing settings

The following drawing settings are available in the form of step 3:

- **Gap between deep point and table**: this refers to the vertical gap between the deep point of the section and the table, in drawing units.
- **Deep point circle radius**: this refers to the circle radius at the deep point, in drawing units.
- **Table line height**: this refers to the line height (in drawing units) of the table.
- **Table text height**: this refers to the text height of the table (in drawing units).
- **Title text height**: this refers to the text height of the title and station name (in drawing units).
- **Include water level**: check this option to include the water level in the sections.
- **Sections places vertically**: select the number of columns when placing the sections in the drawing.
- **Gap between horizontal sections**: this refers to the horizontal gap between the sections (in drawing units).
- **Gap between vertical sections**: this refers to the vertical gap between the sections (in drawing units).
- **Layer colors**: double-click on the layer color to change it:

  1. The color selection dialog box appears.
  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.
  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.
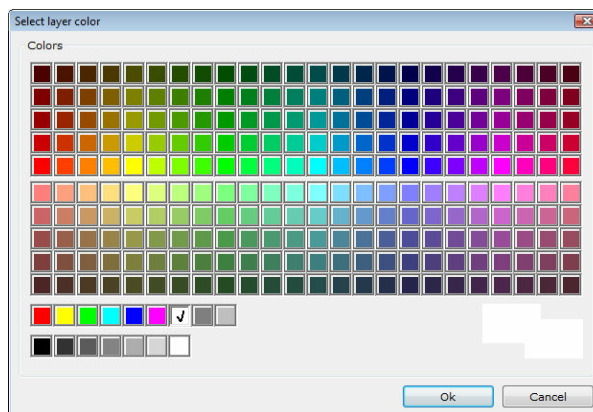
> **NOTE**: If nothing is visible when viewing the newly created DXF file, select Zoom Extent to view the whole drawing.

2.7.8 Sections to GRD

With this option, a GRD file containing one or more of the sections of the current project is created. GRD files contain geometric information on a prismatic cross section with straight edges as well as the title of the section, station data etc.

To export the sections to a GRD file:

1. Select **Export** from the **File** menu.
2. Select **Sections to GRD file** from the **Export** menu. The following dialog box appears:

3. Select the profile you wish to export. Press **Ok** to proceed.
4. The file selection dialog box appears:
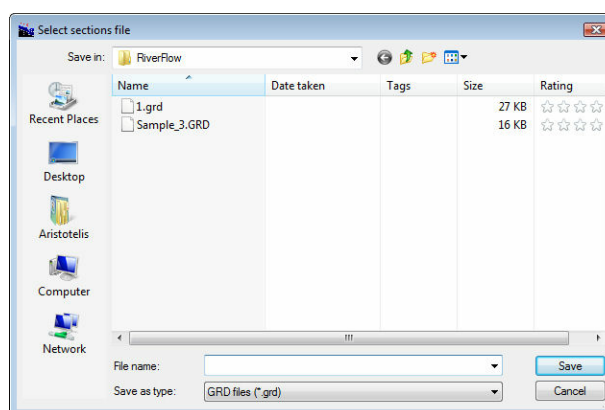
5. Select the path of the file.
6. Type the filename in the **File name** text box.
7. Select **Save** to create the file. Select **Cancel** to cancel the operation.

> **NOTE**: It is recommended that you use standard ANSI characters in the titles of the sections, since many related programs may not function properly.

2.7.9 Section to PCS

With this option, you can create a PCS file containing data on the geometry of the selected section.

To export a section to a PCS file:

1. Select **Export** from the **File** menu.
2. Select **Section to PCS file** from the **Export** menu.

3. Select the section to be exported. Press **Ok** to proceed. The file selection form appears:
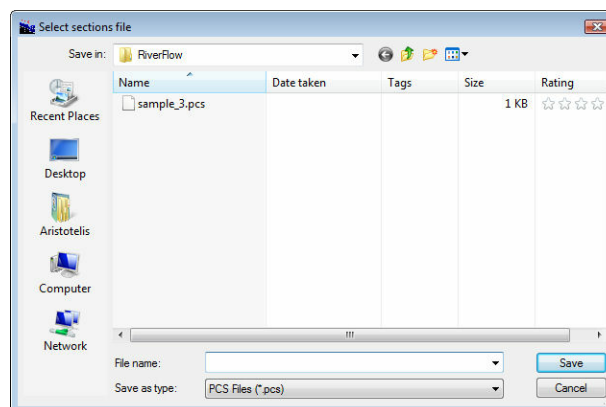
4. Select the path of the file.
5. Type the filename in the **File name** text box.
6. Select **Save** to create the file. Select **Cancel** to cancel the operation.

2.7.10 Section to ArcView Shapefile

With this option, an ArcView Shapefile containing the selected section is created.

To export the selected section to an ArcView Shapefile:

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

4. Select the section to be exported. Press **Ok** to proceed.
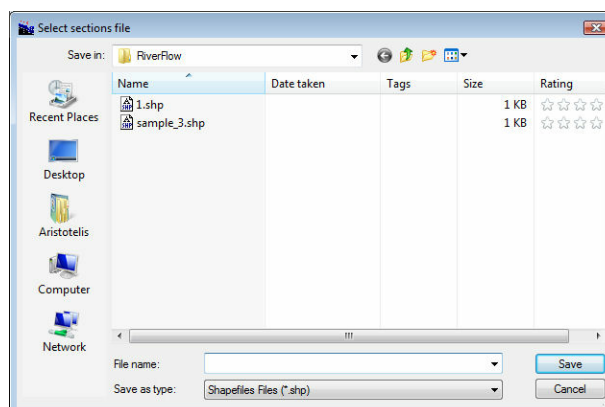
5. Select the path of the file.
6. Type the filename in the **File name** text box.
7. Select **Save** to proceed. Select **Cancel** to cancel the operation.
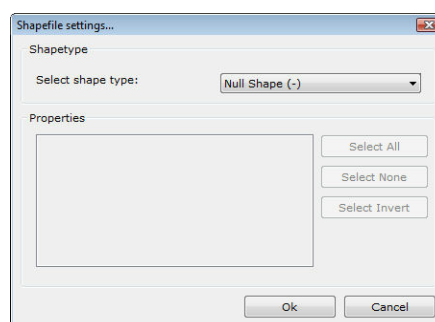
8. Select the appropriate shape type.

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

---

9. Select **Ok** to create the file. Select **Cancel** to abort the operation and close the dialog box.

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

---

2.8   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.
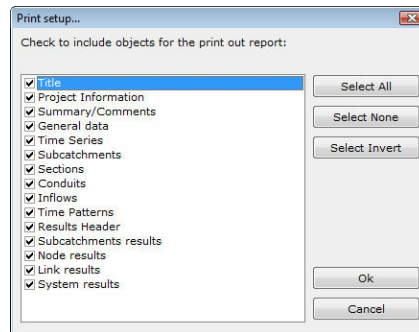
The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection of a list.

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.

2.9   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.
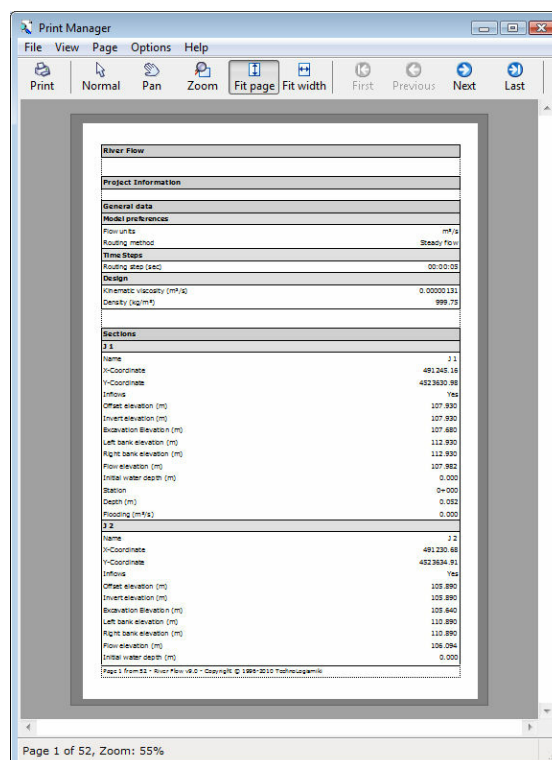
> **NOTE**: A complete user manual on the capabilities of **Print manager** can be found in the corresponding help file.

2.10 Print to

2.10.1 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.
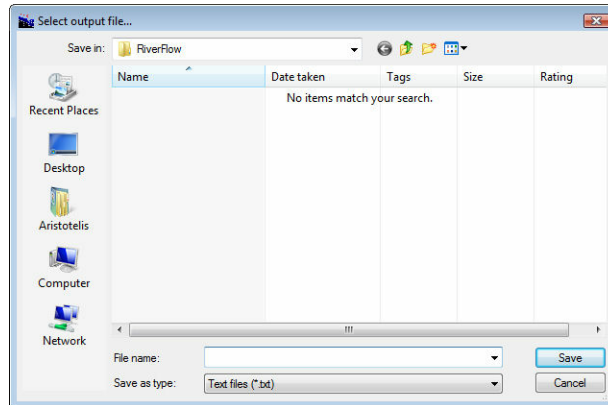
The parts of the project that will be included in the report are determined from print setup.

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.

2.10.2 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.

2.10.3 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.

2.10.4 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.

2.11 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

III

3 Edit

3.1 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

3.2 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.

3.3 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.

3.4   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.

3.5   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.

3.6   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.

3.7   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.

3.8   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.

3.9   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.

3.10 Select objects

3.10.1 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 0bjects** from the **Edit** menu.
3. Select **Create selection** from the **Select 0bjects** 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.

3.10.2 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:
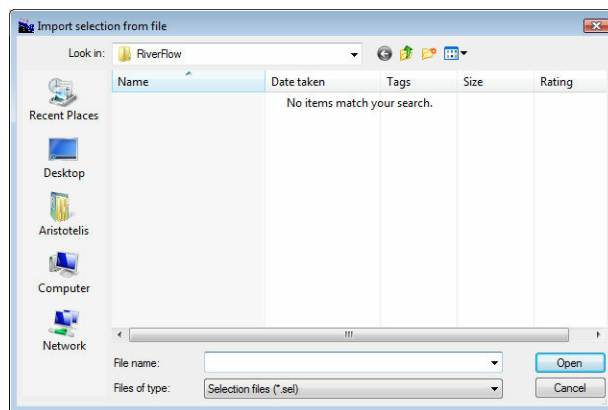
3. Select the path of the file.
4. Select the file type from the **Files of type** drop-down list. The default option is "Selection Files" with the extension .sel.
5. Select the file by clicking on it.
6. Select **Open** to open the selected file. Select **Cancel** to cancel the operation.

3.10.3 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:
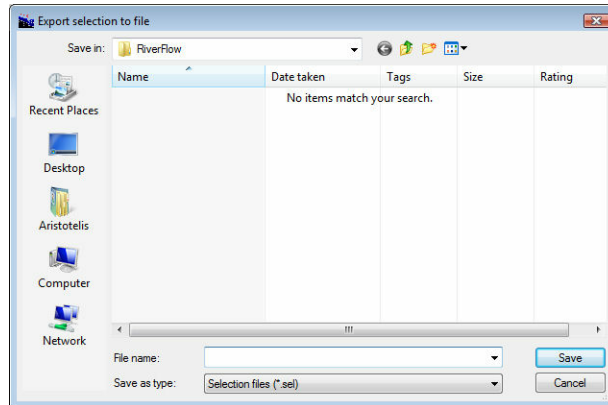
3. Select the path of the file.
4. Type the filename in the **File name** text box.
5. Select **Save** to save the project with the selected filename and path. Select **Cancel** to cancel the operation.

3.10.4 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.

3.11 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:
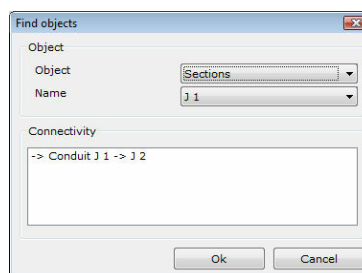
2. Select the **Object type** from the drop-down list.
3. Select the **Name** of the object.
4. The connectivity of the specified object is loaded in the list.
5. Select **Ok** or **Cancel** to close the form.

Chapter

IV

4 View

4.1 View menu

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

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

4.2 Plan view

4.2.1 Options

With this option, you can select the object type(s) that will be visible in plan view and modify their appearance. 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 and modify their appearance:

1. Select **Plan view** from the **View** menu.
2. Select **Options** from the **Plan view** menu. The following form appears:
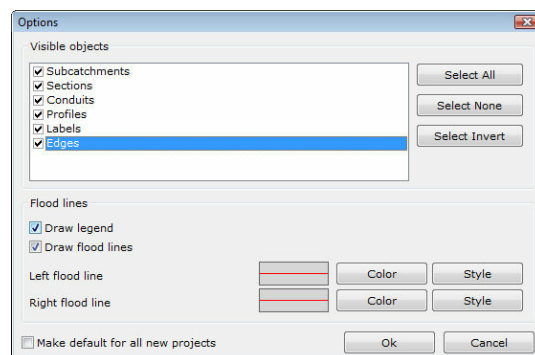
3. Select the object type(s) that you wish to be visible. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
4. Select the settings for the appearance of the flood lines in the corresponding frame. The flood lines become available after successful conclusion of the calculations.
4. Select **Ok** to accept changes and close the form. Select **Cancel** to close the form

without any changes.

4.2.2   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:
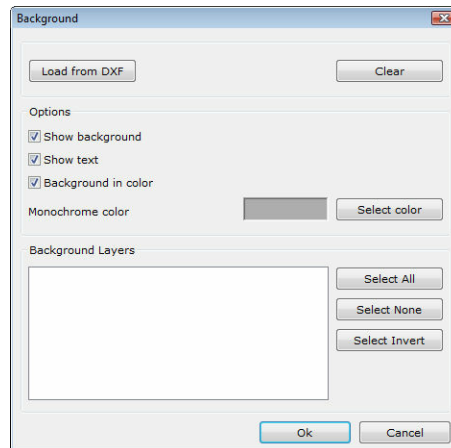
3. To load data from a DXF file, select **Load from DXF.** This process is described in the **File > Import > Background from DXF** section.
4. Select **Clear** to clear the existing background. A confirmation message will be displayed.
5. Select **Show background** to toggle the visibility of the background.
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.
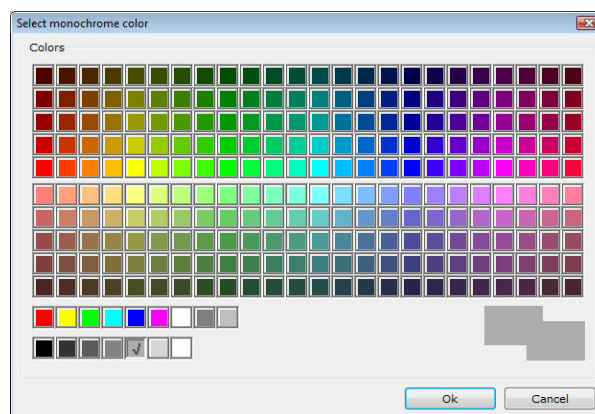
8.2. Select the **color** from the 256 available colors. The currently selected color is marked with a tick. On top of the **Cancel** button, the old and the new color are

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.

4.3 Background pictures

4.3.1 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:
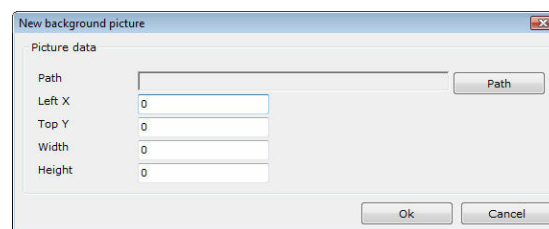
3. Select **Path.** The file selection dialog box will appear:
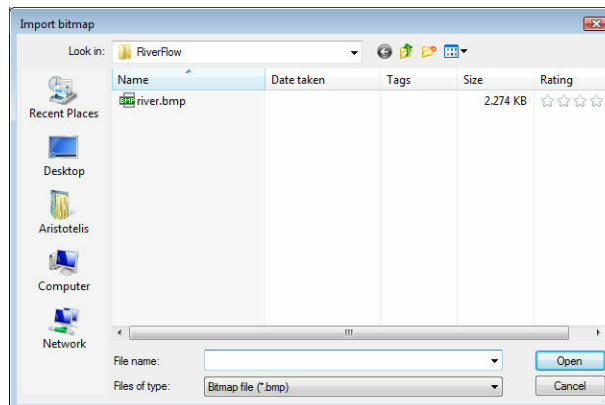
4. Select the path of the file.
5. Select the file type from the **Files of type** drop-down list. The default option is "Bitmap file" with the extension .bmp.
6. Select the file by clicking on it.
7. Select **Open** to open the selected file. Select **Cancel** to cancel the operation.
8. Enter the **Left 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).

4.3.2 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:
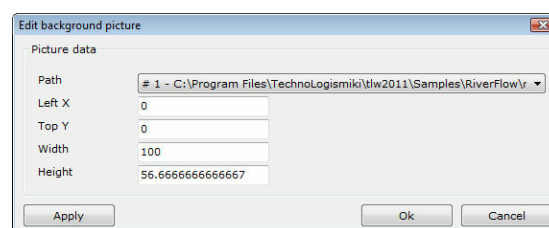
3. Select the picture from the drop-down list.
4. Make the appropriate changes. If you provide only the height or the width, the ratio of the source picture will be used to calculate the missing data.
5. Select **Apply** to apply the changes without closing the form. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

4.3.3 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:
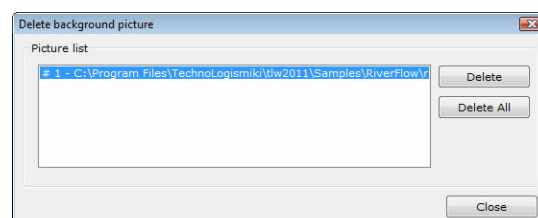
3. Select the picture from the list.
4. Select **Delete** to delete the selected picture.
5. Select **Delete all** to delete all pictures.
6. Select **Close** to close the dialog box.

4.3.4 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.

4.4 Profile

4.4.1 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 Profiles as the axis in order to force the profiles to be drawn in the axis (else, the deep point is selected).
5. Select an appropriate **Height stretch factor**. This factor magnifies distances in the Y direction so that small elevation differences are clear.
6. Select the appropriate settings for the appearance of the objects in the following frames.
7. Select **Make default for all new projects** to use these options for all new projects.
**8.** Select **Ok** to accept changes and close the form. Select **Cancel** to close the form without any changes.

4.5   Section

4.5.1   Options

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

To modify the appearance of the section sketch:

1. Select **Section** from the **View** menu.
2. Select **Options** from the **Section** menu. The following form appears:
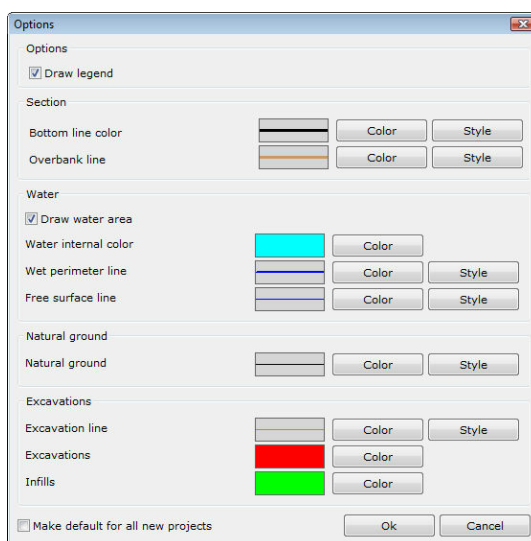
3. Make the appropriate selections for the appearance of the objects.
4. Select **Make default for all new projects** to use these options for all new projects.
5. Select **Ok** to accept changes and close the form. Select **Cancel** to close the form without any changes.

Chapter

V

5   Data

5.1   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
- General data
- Contours
- Time series: Management, Add, Delete, Edit, Move, Sort
- Time patterns: Management, Add, Delete, Edit, Move, Sort
- Network consistency
- Options
  - General preferences
  - Sketch
  - Grid editing
  - Customize toolbar
  - Default values

5.2   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.
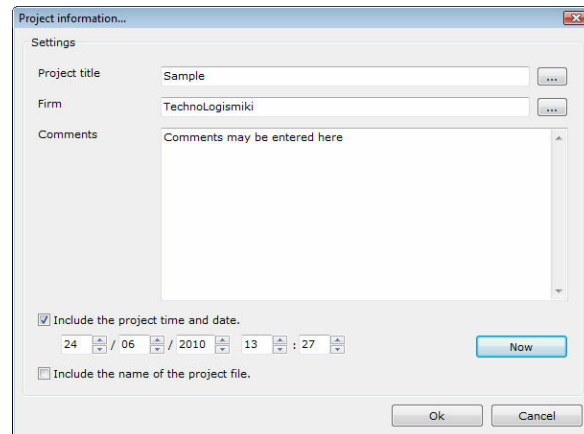
By selecting the buttons with the ellipses (...) next to the **project title** and **firm**, you can access the corresponding databases.

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.
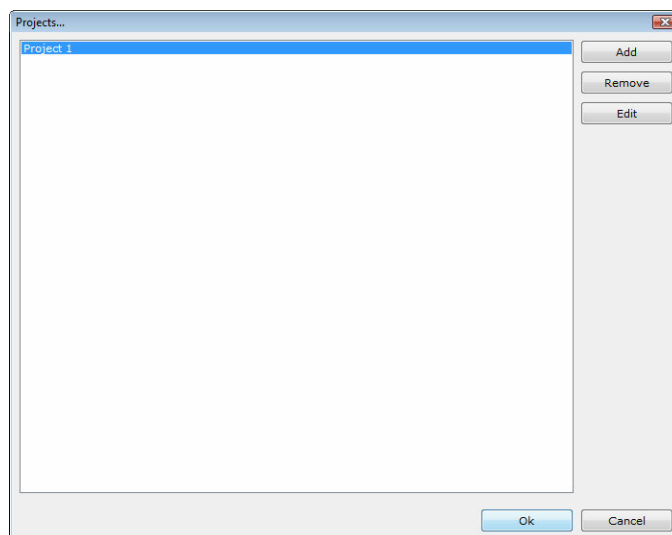
Firm database

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.
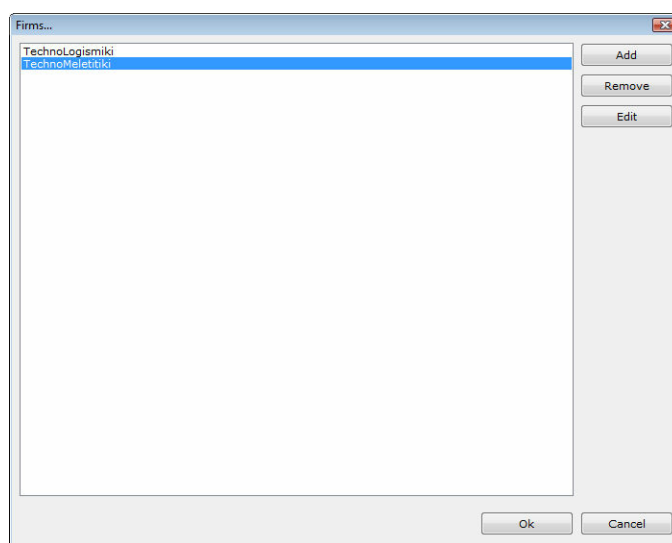
5.3 General data

With this option, you can input general data.

To input general data:

1. Select **General data** 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.

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.

Routing (sec): Enter the time step length in decimal seconds used for routing flows and water quality constituents through the conveyance system. Values around 5 seconds are generally sufficient.

Kinematic viscosity (m^2/s): select the kinematic viscosity of the fluid. Press the button with the ellipses "..." to invoke the corresponding database. This property is for future reference.

Density (kg/m^3): select the density of the fluid. Press the button with the ellipses "..." to invoke the corresponding database. This property is for future reference.

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.

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.

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.

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.

5.4 Hydrology

5.4.1 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. IDF curves are used in conjunction with subcatchments.

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.

5.4.2   Subcatchments

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

5.5   Contours

With this option, you can manage the contours.

To manage the contours:

1. Select **Contours** from the **Data** menu. The following form appears:
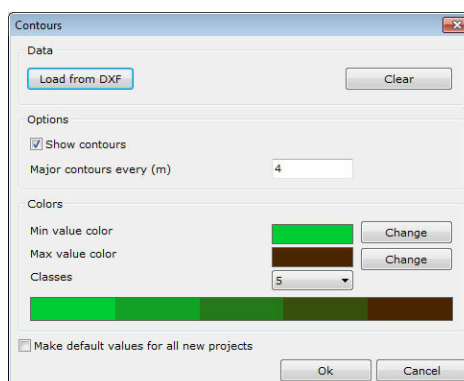
2. Make the appropriate selections as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Data

Load from DXF: Click this button to import contour data from a DXF file. The process is described in the menu **File > Import > Contours from DXF**.

Clear: Click this button to clear the contour data from the current project.

Options:

Show contours: Check this option to show or hide the contours.

Major contour every: Select the interval of the major contours. These contours are

shown thicker and they are accompanied by their value.

Colors

Select the colors that correspond to the minimum and the maximum value of the contours, as well as the number of color classes.

Make default values for all new projects: Check this option to make the settings of this form default for all new projects.

5.6   Time series

5.6.1   Management

Time Series objects are used to describe how certain object properties vary with time.

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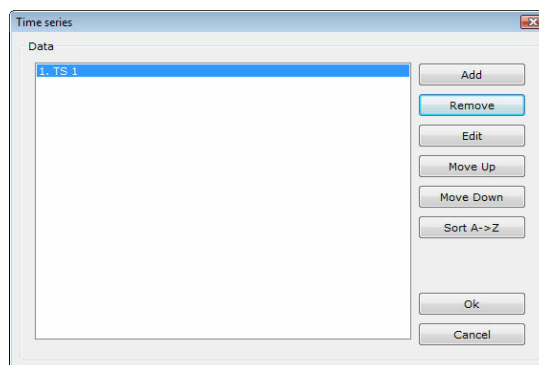
2. Make the necessary modifications.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

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
```

5.6.2   Add

To add a new time series:

1. Press **Add**. The data form appears:
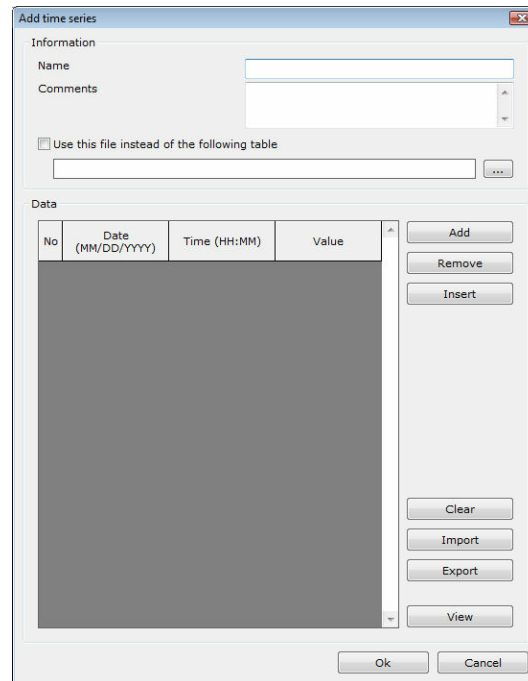
2. Enter a user-assigned **name**. This name cannot be null or used for another time series in the project.

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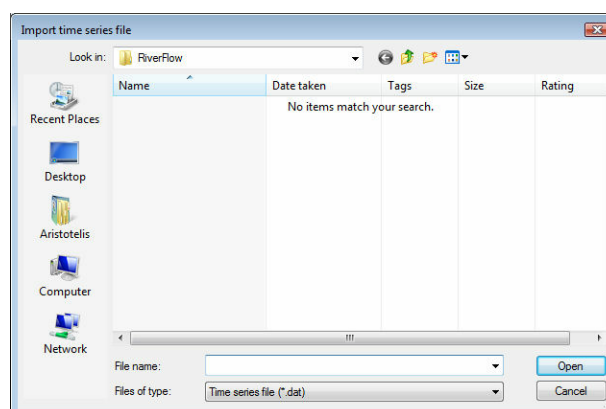
2. Select the path of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Time series file" with the extension .dat.
4. Select the file by clicking on it.
5. Select **Open** to open and analyze the file.

To export a time series to an external file:

1. Press **Export.** The file selection dialog box appears:
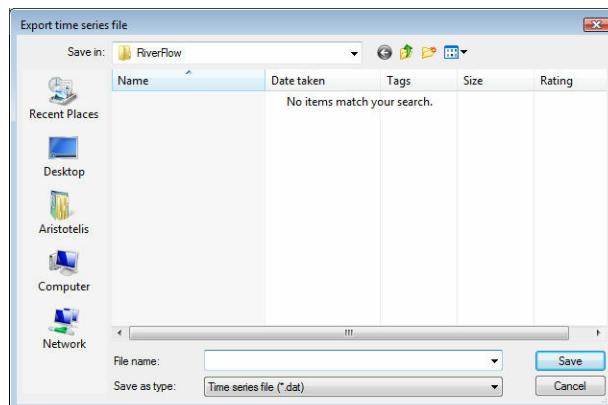
6. Select the path of the file.
7. Type the filename in the **File name** text box.
8. Select **Save** to save the file with the selected filename and path. Select **Cancel** to cancel the operation.

To view a time series:

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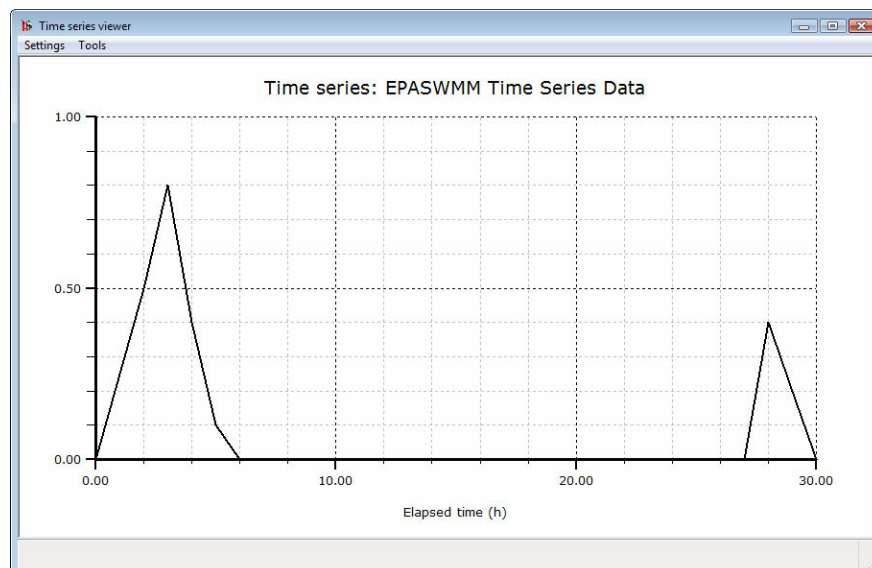
5.6.3 Delete

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.

5.6.4 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.

5.6.5 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.

5.6.6 Sort

To sort the time series list:

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

5.7 Time patterns

5.7.1 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. 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:
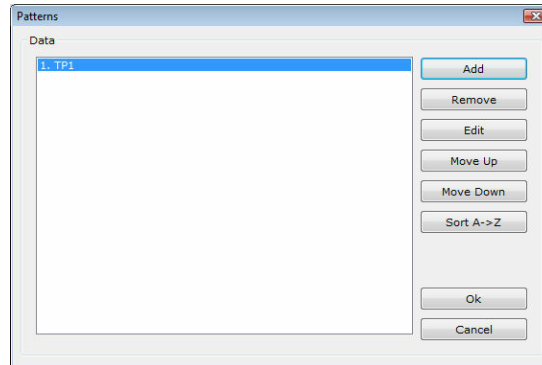
2. Make the necessary modifications.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

5.7.2   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 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".

5.7.3 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.

5.7.4 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.

5.7.5 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.

5.7.6 Sort

To sort the time pattern list:

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

5.8 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.
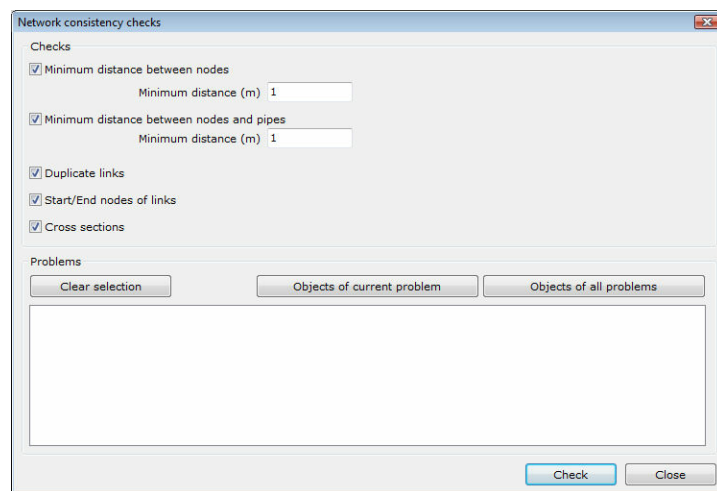
5.9   Options

5.9.1   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
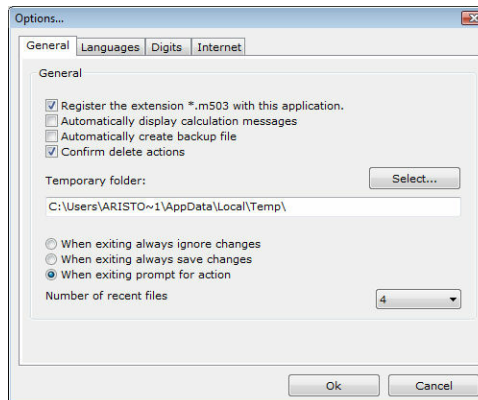
This tab contains general preferences regarding the usage of the program.

Check **Register the extension *.m503 with this application** to associate the extension .m503 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
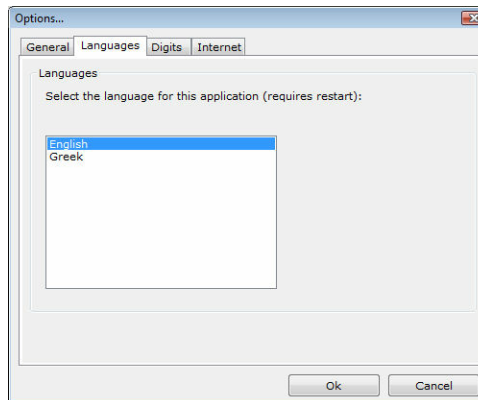
If more than one language packs have been installed, then you can choose the language of the program. In the above case, there are two language packs; English (that are already selected) and Greek. If you change the language, all forms, menus, messages, help files will reflect the chosen language.

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

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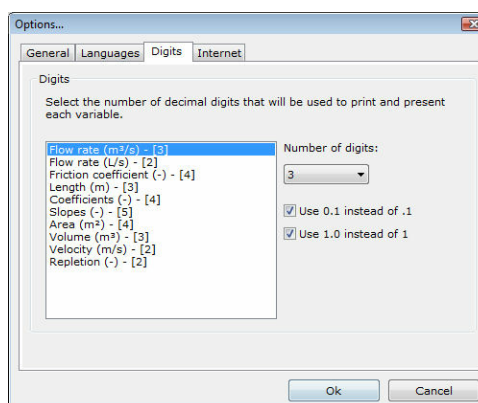
With this tab, you can modify the way the results are presented. All values used in the program are displayed in the list on the left.

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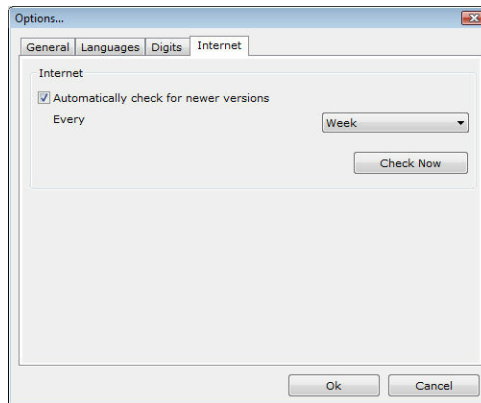
The program can automatically check for newer versions over the Internet. Check **Automatically check for newer versions** to enable this feature. The check is automatically performed at an interval specified in the **Every** drop-down list. Select **Check now** to manually check for newer versions.

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.

5.9.2 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.

Grid

- 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.

OSnap

- 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.

Snap to grid

- 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.

Background

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

5.9.3 Grid editing

With this option, you can modify the behavior of grids.
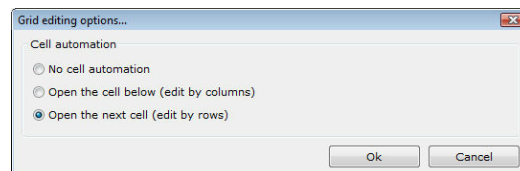
The behaviour of all editable grids is controlled by the preferences in this dialog box.

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.

5.9.4 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.
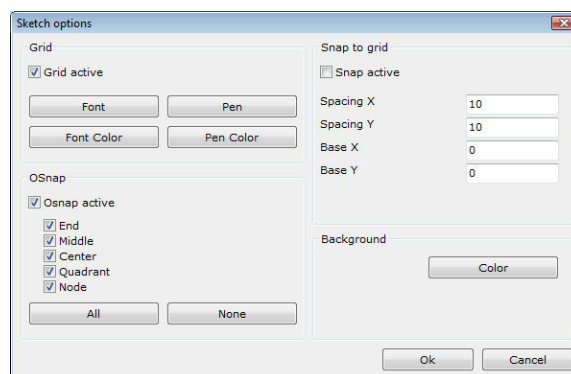**3.** Make the appropriate changes.
4. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.
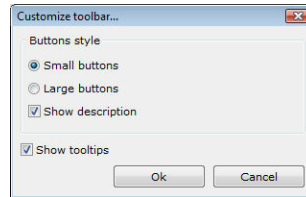
The toolbar may contain small or large buttons.

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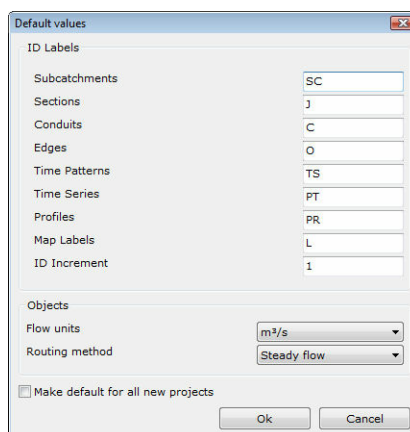
NOTE: These preferences affect all projects, both old and new.

5.9.5 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** frame, select the prefixes of new object names.
4. In the **Objects** frame, select the flow units and the routing method.
5. Select **Make default for all new projects** if you wish to use these settings for all new projects.

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

Chapter

VI

6   Objects

6.1   Objects menu

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

- Add
  - o Subcatchment
  - o Section
  - o Conduit
  - o Pump
  - o Profile
  - o Label
- Properties
  - o Subcatchment
 - o Section
  - o Conduit
  - o Edge line
  - o Profile
  - o Label
- Add vertex
- Delete vertex
- Stretch vertex
- Labels
- Swap conduit ends
- Swap edge line ends

6.2   Add

6.2.1   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**.

6.2.2 Section

With this option, you can add one or more sections (nodes).

To add a section:

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

To add more than one sections:
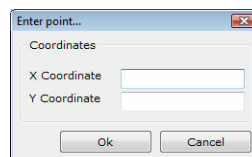
1. Select **Add** from the **Objects** menu.
2. Select **Section** from the **Add** menu while holding down CTRL key.
3. Click onto the drawing to define the position of the section while holding down CTRL key.
4. Repeat step 3 as many times as required.
5. Hit ESC when you have finished.

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**.

6.2.3 Conduit

With this option, you can add one or more conduits. The ends of the conduit are always section objects. 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.
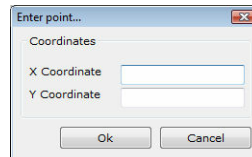
To add more than one conduits:

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**.

6.2.4 Edge line

With this option, you can add one or more edge lines. The edge lines do not participate in the calculations. However, they are exported along an associated table of coordinates when you create a DXF file with the plan view.

To add an edge line:

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

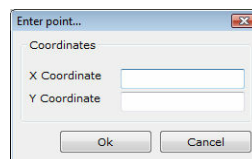To add more than one edge line:

1. Select **Add** from the **Objects** menu.
2. Select **Edge** from the **Add** menu while holding down CTRL key.
3. Click successively onto the drawing to define the vertices.
4. Press **ENTER** while holding down CTRL key to finalize the polyline and define the edge. 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**.

6.2.5 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 sections of the profile. It is not necessary to click onto all sections. If two successive sections 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 section.
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 sections of the profile. It is not necessary to click onto all sections. If two successive sections 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 sections. Thus, in most cases, you can fully define a profile by clicking on the first and last section.
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.

6.2.6 Label

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

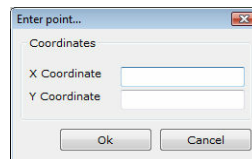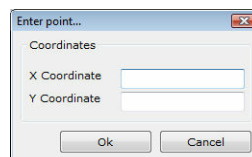To add a label:

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**.

6.3   Properties

6.3.1   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 sections or other subcatchments.

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.
- **Outlet object**: select one section 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 {}.
- **Method for concentration time**: select the method for the evaluation of the concentration time (Giandotti, Kirpich, FAA).
  - for Giandotti, the following data are required: area, length of drainage line, mean altitude, and the altitude at the exit.
  - for Kirpich the following data are required: length of drainage line and mean slope.
  - for FAA the following data are required: length of drainage line, mean slope and the discharge coefficient.
- **Discharge coefficient**: enter the discharge coefficient.
- **Length of drainage line (ft or m)**: enter the length of the main drainage line.
- **Slope (%)**: enter the mean slope of the subcatchment.
- **Altitude at exit (ft or m)**: enter the altitude at the exit of the subcatchment.
- **Mean altitude (ft or m)**: enter the mean altitude of the subcatchment.
- **Concentration time (min)**: enter the concentration time. If left blank, the time is estimated based on the method selected in the aforementioned properties.

6.3.2 Section

Sections are drainage system nodes where conduits join together. Physically they represent the confluence of natural surface channels. External inflows can enter the system at sections.

To view and modify the properties of junctions:

1. Select **Properties** from the **Objects** menu.
2. Select **Section** from the **Properties** menu. The following form appears:
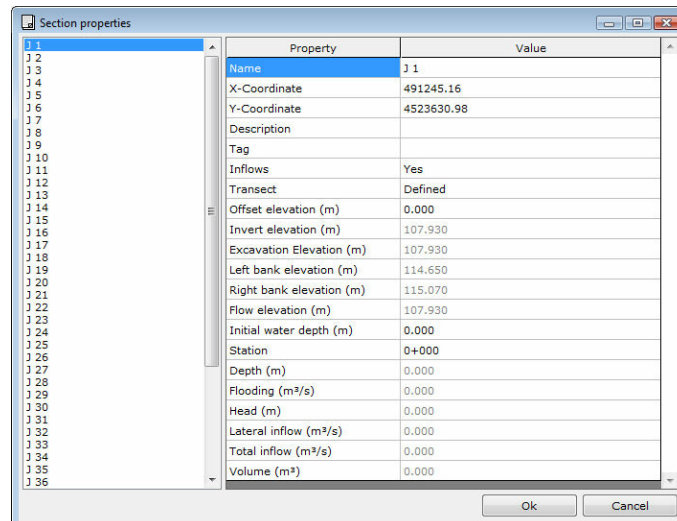
3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. 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 inflows to the section.
- **Transect:** click the ellipsis button to edit the transect of the section.
- **Offset elevation (ft or m):** enter the offset elevation of the section. This value is added to the elevation of all points in the flow line of the section and is particularly useful when using e.g. a single rectangular section to multiple stations.
- **Invert elevation (ft or m):** displays the invert elevation of the section.
- **Excavation elevation (ft or m):** displays the excavation elevation of the section.
- **Left bank elevation (ft or m):** displays the left bank elevation of the section.
- **Right bank elevation (ft or m):** displays the right bank elevation of the section.
- **Flow elevation (ft or m):** displays the flow elevation of the section.
- **Initial water depth (ft or m):** enter the depth of water at the junction at the start of the simulation.
- **Station:** click the ellipsis button to enter the station of the section.

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 section.
- **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).
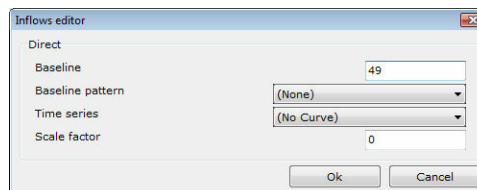
6.3.2.1 Inflows

Apart from inflows from subcatchments, sections also accept direct inflow.

To enter direct inflow data:

1. Select the properties form of a section.
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.

Direct

Baseline: specifies the value of the constant baseline component of the constituent's inflow. The units are the project's flow units. 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.

6.3.2.2   Transect

With this option, you can define the transect of the section. There are actually three transects per section: flow transect (for hydraulic calculations), natural ground transect and trench transect (for calculation of excavations and infills). The offset elevation of the section affects the flow and trench transects.

To define the transect:

1. Select the properties form of a section.
2. Double click on the **Transect** 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.
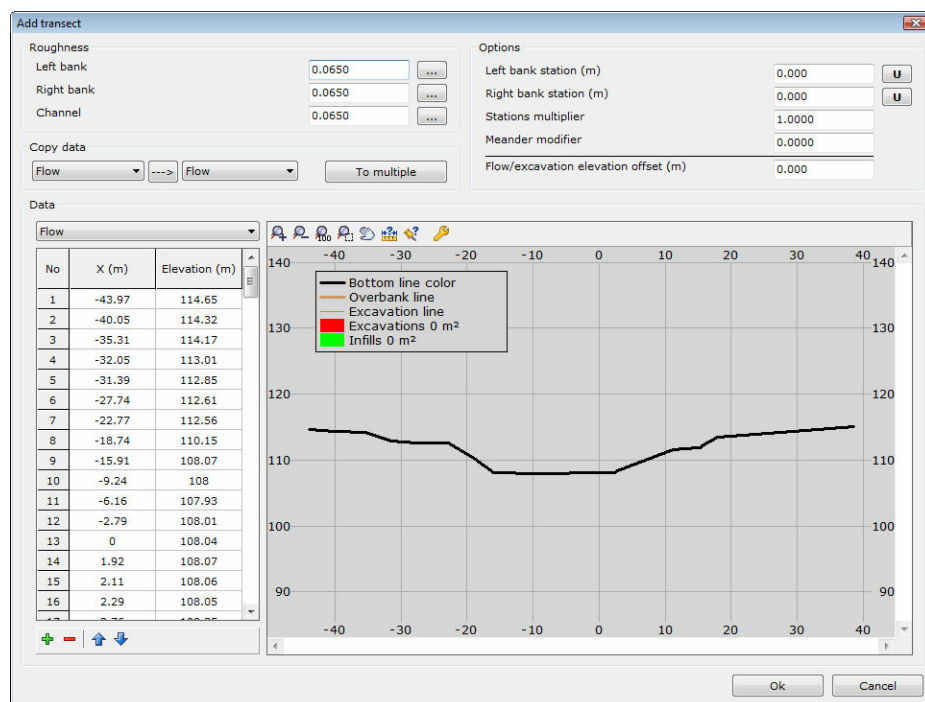
<u>In the **Roughness** frame:</u>

1. Enter the Manning friction coefficient for the left bank.
2. Enter the Manning friction coefficient for the right bank.
3. Enter the Manning friction coefficient for the channel.

<u>In the **Options** frame:</u>

1. 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.
**2**. 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.
3. Enter the **stations multiplier** which will be applied to the stations specified. Enter 1.0 if you wish to disable this feature.
4. Enter the **meander modifier** which will be applied to the stations specified. Enter 1.0 or 0.0 if you wish to disable this feature.
5. Enter the **flow/excavation elevation offset** in ft or m which will be added to the elevations specified.

In the **Data** frame:

1. Select the transect type from the drop-down list.
2. Enter the data as (X,Y) pairs, where X=0 is the center of the section.
3. Press "+" to add a line in the table. Press "-" to erase the selected lines in the table. Select "up" and "down" to move the selected line up/down by one place.

In the **Copy Data** frame:

1. To copy data **between the three transects of the same section**, select the source and target transects and press "--->".
2. To copy data **between the current section and other sections** press **To Multiple**. The following form appears:
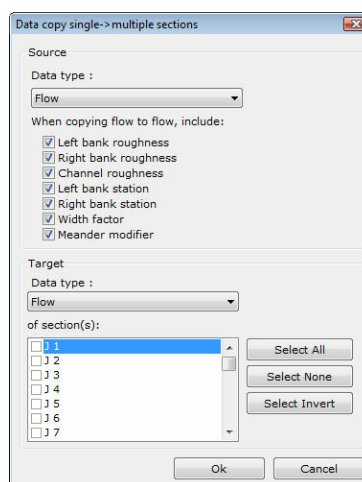
3. Select the source and target data types, as well as the target sections which will receive the copy of the transect.
4. Press **Ok** to proceed with the copy. Press **Cancel** to cancel the procedure.

NOTE:
- The hydraulic calculations are performed based on the flow transect only. The natural ground and trench transect are used for the evaluation of excavations/infills only.
- The X-coordinates of all transects must be in ascending order.

6.3.3 Conduit

Conduits are channels that transport water from one section to another in a conveyance system. Each conduit uses the transect of the upstream section. Therefore, you need to use more dense section points if you need greater accuracy.

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:

$$
\text{Metric system} \qquad\qquad \text{English system}
$$

$$
Q = \frac{E}{n} R^{2/3} \sqrt{J} \qquad\qquad Q = \frac{1.49}{n} E R^{2/3} \sqrt{J}
$$

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.

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.

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.

- **Start Node**: select the start section from the drop-down list.
- **End node**: select the end section from the drop-down list.
- **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
- **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.
- **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.

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.

6.3.4   Edge

The edge lines do not participate in the calculations. However, they are exported along an associated table of coordinates when you create a DXF file with the plan view.

To view and modify the properties of edge lines:

1. Select **Properties** from the **Objects** menu.
2. Select **Edge** from the **Properties** menu. The following form appears:
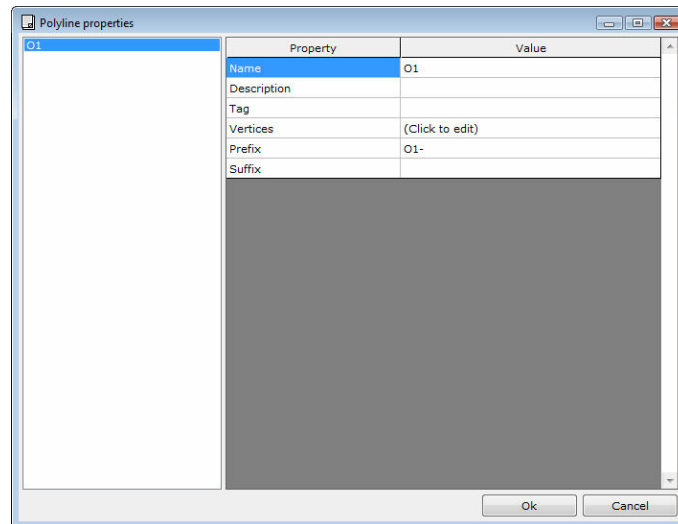
3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. 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 edge line.
- **Description**: enter the description of the edge line.
- **Tag**: enter a tag for the edge line. This appears neither in the input data nor the results.
- **Vertices:** click the ellipsis button to define the vertices analytically.
- **Prefix**: enter a prefix for all vertices. This need not be the same as the name of the edge line.
- **Suffix**: enter a suffix for all vertices.

6.3.5   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,

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:
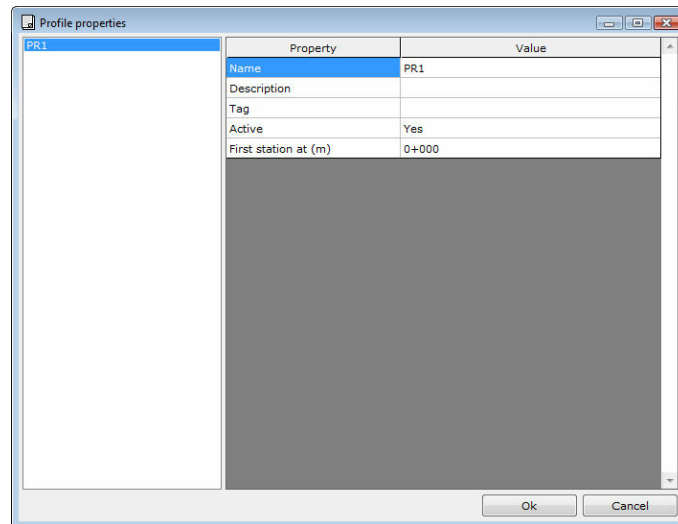
3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. 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.

<u>**Properties**</u>

- **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.

6.3.6   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:
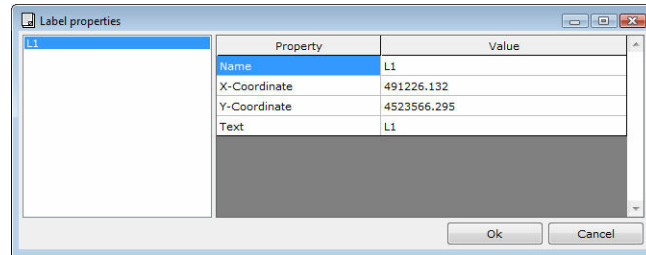
3. Select one or more objects from the list on the left. To select more than one objects, hold down **CTRL** while selecting. The objects that are selected in the plan view are preselected in the list.
4. If more than one objects are selected in the list, only the common properties are displayed.
5. 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.

6.4   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:
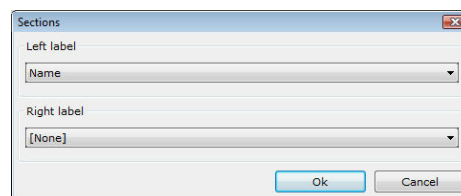
3. Select the property that will be displayed on the left.
4. Select the property that will be displayed on the right.
5. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

6.5   Add vertex

With this option, you can add an intermediate vertex to an existing subcatchment or edge line.

To add an intermediate vertex:

1. Select **Add vertex** from the **Objects** menu.
2. Click on the appropriate object to add a vertex.

6.6 Delete vertex

With this option, you can delete an existing intermediate vertex of a subcatchment or edge line.

To delete an intermediate vertex:

1. Select **Delete vertex** from the **Objects** menu.
2. Click on the vertex to delete it.

6.7 Stretch vertex

With this option, you can move an existing intermediate vertex of a subcatchment or edge line.

To move an existing intermediate vertex:

1. Select **Stretch vertex** from the **Objects** menu.
2. Click on the vertex you wish to move.
3. Click again to define the new vertex position.

6.8 Swap conduit ends

All conduits have a certain direction. With this option, you can swap the ends of the selected conduits and invert their direction.

To swap the ends of the selected conduits:

1. Select the conduits in the drawing.
2. Select **Swap Conduit Ends** from the **Objects** menu. The ends are swapped.

6.9 Swap edge line ends

All edge lines have a certain direction. With this option, you can swap the ends of the selected edge lines and invert their direction.

To swap the ends of the selected edge lines:

1. Select the links in the drawing.
2. Select **Swap Edge Line Ends** from the **Objects** menu. The ends are swapped.

Chapter

VII

7 Profiles

7.1 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
- Uniform inflow
- Stations

7.2 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.

7.3 Section renaming

Since sections have various names and a profile usually consists of several sections, this tool can be used to rename all sections belonging to a particular profile. The rename procedure can create chained names of the sections, making it easy for the engineer or the reviewer to navigate through a profile. For example, a profile consisting of sections 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 Sections**.
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.
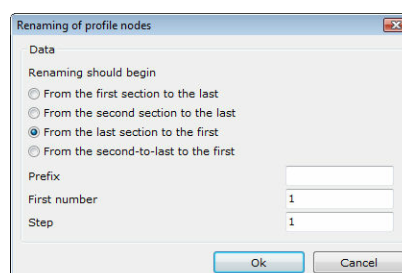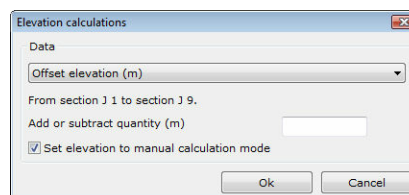
> **NOTE**: You can easily work through an entire network. Suppose you have two profiles, J1 to J10 and P1 to P8. The second profile intersects the first at section J5. Rename the second profile use J5- as the prefix, first number 1 and step 1. It will be renamed as J5-1 to J5-8.

7.4   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.
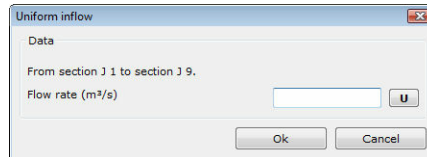**3.** Select **Elevation calculations** from the **Profiles** menu. The following form will appear:

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:
   • **Offset 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 **Set elevation to manual calculation mode** 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.

7.5   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 section, 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.
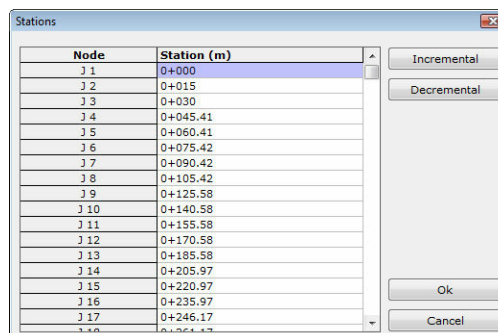
> **NOTE**: If the last section is selected then it is ignored because is does not affect the flow rate of the branch. The inflow is distributed uniformly, taking into account the length of the segments between the stations.

7.6   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 **Profiles** 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.

Chapter

VIII

8 Tools

8.1 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:
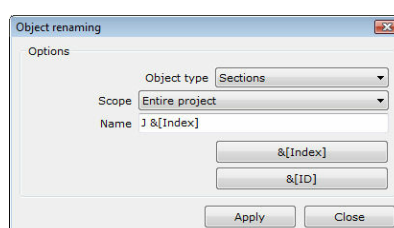
- Object renaming
- Subcatchments from DXF
- Copy sectional data
- Delete sectional data
- Common sectional data
- Section offset from ground
- Sections from contours
- Inflow distribution
- Delete all inflows

8.2 Object renaming

To conveniently rename objects of a certain type such as sections and conduits, 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 sections as J1, J2, J3, etc.
&[StartNodeName] -> &[EndNodeName]: will rename all conduits as J1 -> J2, J2 -> J3, etc.

8.3   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 section 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:
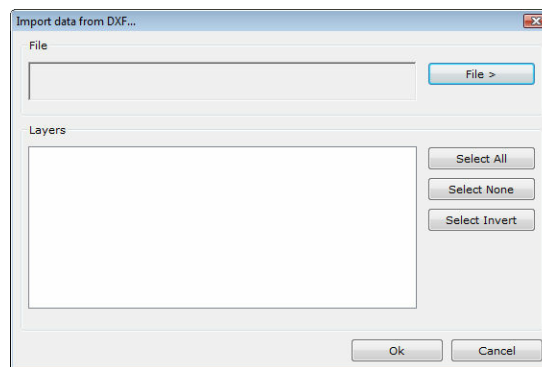
3. Select the input file by clicking the **File >** button.
4. Select one or more layers containing the polygon data.
5. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
6. Select **Ok** to import the polygons, display them on the drawing and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

8.4   Copy sectional data

With this option, you can batch-copy sectional data between the transects of multiple sections. This is particularly useful e.g. when you need to copy the ground transect of each section to the corresponding flow transect (in order to analyze the natural flow in the channel).

To batch-copy sectional data between the transects of multiple sections:

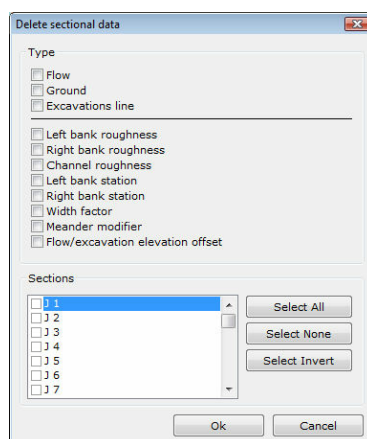**1.** From the **Tools** menu, select **Copy Sectional Data**.
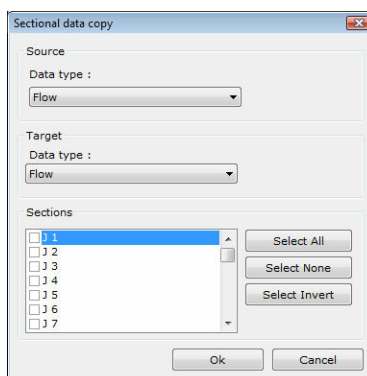
2. Select the source and target transects.
3. Select the sections to apply the data copy.
4. Press **Ok** to proceed or **Cancel** to cancel the operation.

8.5   Delete sectional data

With this option, you can batch-delete sectional data.

To batch-delete sectional data:

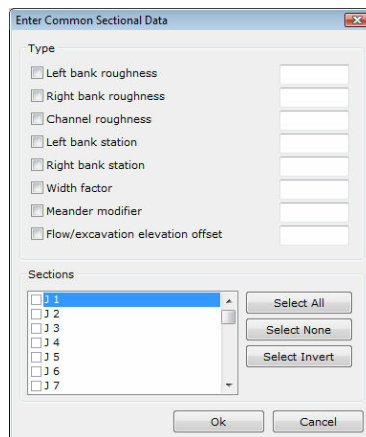**1.** From the **Tools** menu, select **Delete Sectional Data**.

2. Select the data type you wish to delete.
3. Select the sections to apply the deletion.
4. Press **Ok** to proceed or **Cancel** to cancel the operation.

8.6   Common sectional data

With this option, you can set common data to multiple sections.

To set common data to multiple sections:

**1.** From the **Tools** menu, select **Common Sectional Data**.

2. Select the data type you wish to assign and their corresponding common values.
3. Select the sections to apply the assignment.
4. Press **Ok** to proceed or **Cancel** to cancel the operation.

8.7   Section offset from ground

Use this tool to automatically set the offset elevation of the flow/trench transects. This tool sets the origin of the flow transect to the elevation of the center of the ground transect.

To automatically set the offset elevation of the flow/trench transects:

**1.** From the **Tools** menu, select **Section Offset From Ground**.
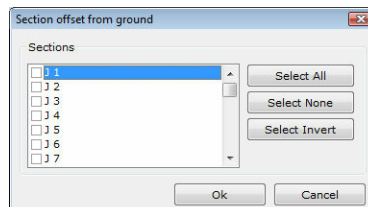
2. Select the sections to apply the assignment.
3. Press **Ok** to proceed or **Cancel** to cancel the operation.

8.8   Sections from contours

Use this tool to automatically obtain the sectional data from the contours.

To automatically obtain the sectional data from the contours:

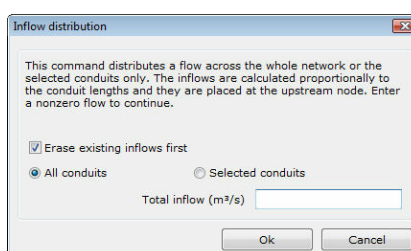**1.** From the **Tools** menu, select **Sections from contours**.
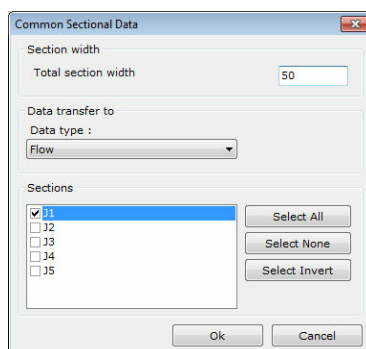
2. Select the total section width, the type of section into which the data will be copied and the sections to analyze.
3. Press **Ok** to proceed or **Cancel** to cancel the operation.

8.9   Inflow distribution

This tool is useful when one wants to distribute a know inflow 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 sections.

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 sections in the network or **Cancel** to ignore any changes and hide the form.
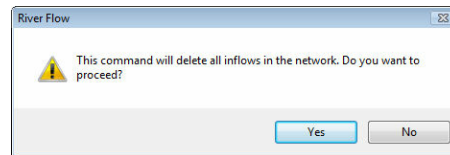

8.10   Delete all inflows

Use this tool to remove all direct inflows from all sections.

To delete direct 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.

Chapter

IX

9 Results

9.1 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
  - o By object
  - o By variable
- Graphical report
  - o Graph
  - o System
  - o Scatter
- Profiles
  - Options
  - Design
- Quantities

9.2 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.

9.3 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.

9.4   Tabular report

9.4.1   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.

9.4.2   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:

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.
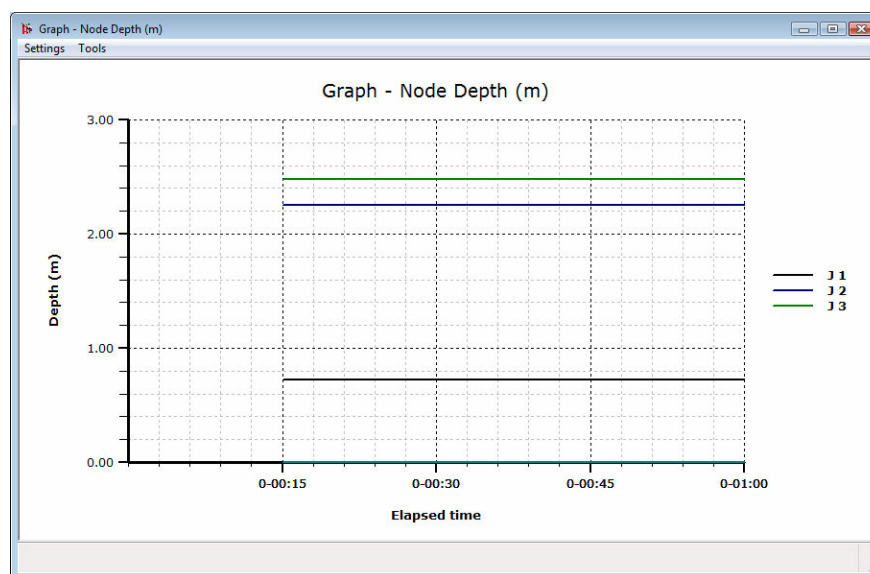
9.5   Graphical report

9.5.1   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.
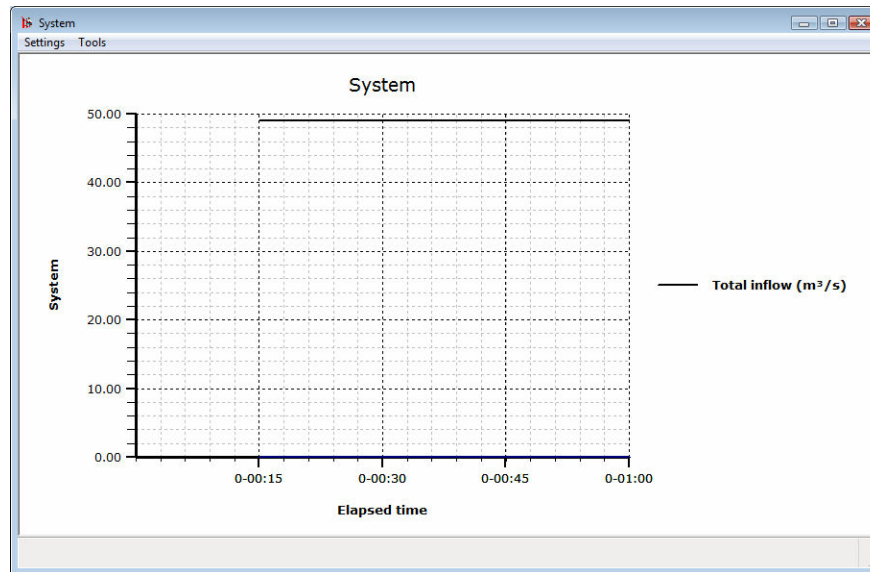
9.5.2   System

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.

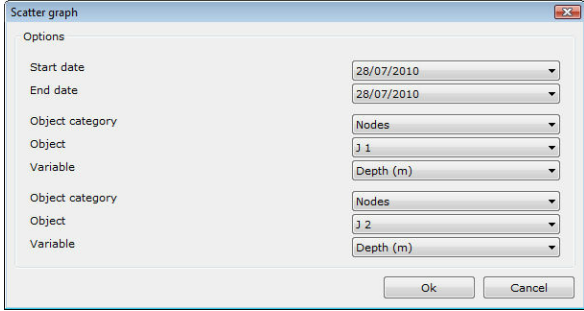In the **Tools** menu, you can select one of the following options:

- **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.

9.5.3   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.
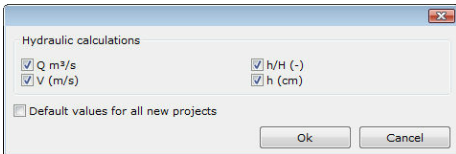
9.6   Profiles

9.6.1   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:

3. In the **Hydraulic calculations** frame, select one or more values that you wish to be included in the profile drawing.
4. 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.
5. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

9.6.2   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:
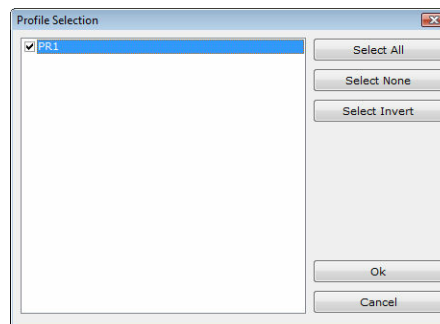
2. Select one or more profiles to be included in the profile drawing. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
3. Select **Ok** to sent the data to the **Profile designer.** Select **Cancel** to cancel the operation.
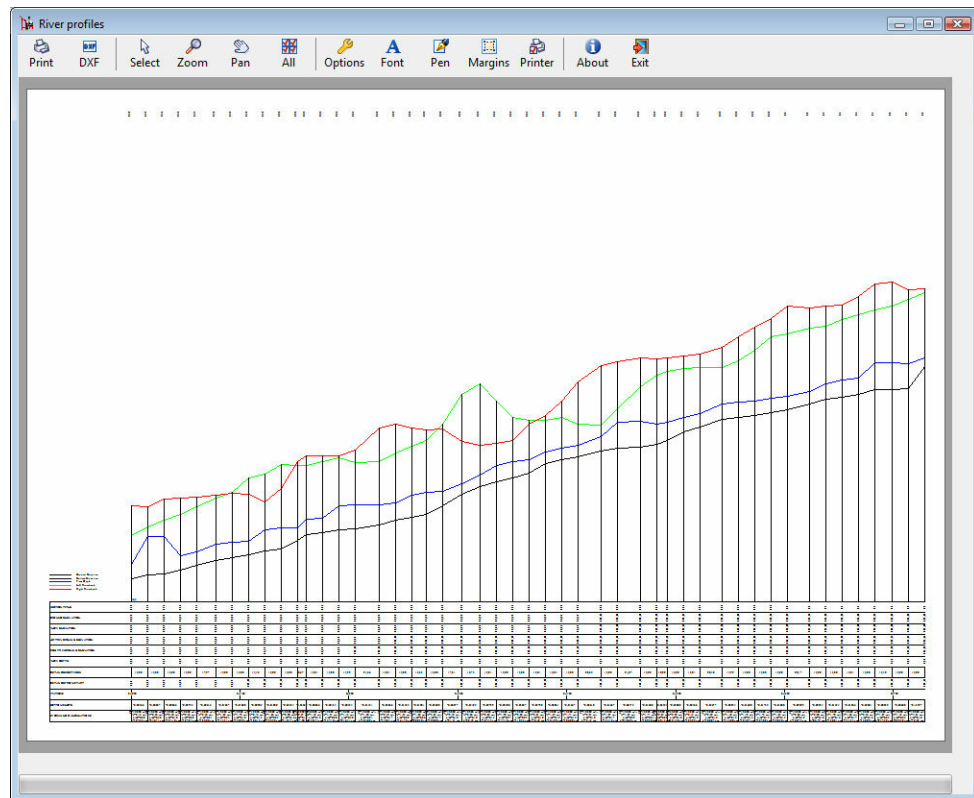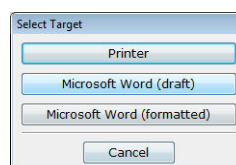
9.7 Quantities

With this option, a report on the excavation/infill quantities is assembled and prepared for preview.

To create a report on the excavation/infill quantities:

1. Select **Quantities** from the **Results** menu.
2. The printer selection form appears:

3. Depending on the printer selected, the relevant object appears:

Print Manager

File View Page Options Help

| Print | Normal | Pan | Zoom | Fit page | Fit width | First | Previous | Next | Last |

39	J 39			7.0228	0.0000	7.0228
40	J 40			3.8729	0.0000	3.8729
41	J 41			6.7997	0.0000	6.7997
42	J 42			3.9639	0.0000	3.9639
43	J 43			3.1787	0.0000	3.1787
44	J 44			4.5492	0.0000	4.5492
45	J 45			3.9825	0.0000	3.9825
46	J 46			6.3155	0.0000	6.3155
47	J 47			3.6885	0.0000	3.6885
48	J 48			4.3312	0.0000	4.3312

2. Conduits

#	Conduit	Upstream Junction	Downstream Junction	Length (m)	Excavation volume (m³)	Infill volume (m³)	Total volume (m³)
1	J 1 -> J 2	J 1	J 2	15.004	162.195	0.000	162.195
2	J 2 -> J 3	J 2	J 3	15.001	214.977	0.000	214.977
3	J 3 -> J 4	J 3	J 4	15.416	152.658	0.000	152.658
4	J 4 -> J 5	J 4	J 5	14.996	121.515	0.000	121.515
5	J 5 -> J 6	J 5	J 6	15.006	115.637	0.000	115.637
6	J 6 -> J 7	J 6	J 7	14.996	76.645	0.000	76.645
7	J 7 -> J 8	J 7	J 8	14.996	72.675	0.000	72.675
8	J 8 -> J 9	J 8	J 9	20.167	98.075	0.000	98.075
9	J 9 -> J 10	J 9	J 10	15.000	66.223	0.000	66.223
10	J 10 -> J 11	J 10	J 11	15.002	71.439	0.000	71.439
11	J 11 -> J 12	J 11	J 12	15.000	63.916	0.000	63.916
12	J 12 -> J 13	J 12	J 13	14.992	66.164	0.000	66.164
13	J 13 -> J 14	J 13	J 14	20.396	84.087	0.000	84.087
14	J 14 -> J 15	J 14	J 15	15.007	63.992	0.000	63.992
15	J 15 -> J 16	J 15	J 16	14.996	90.685	0.000	90.685
16	J 16 -> J 17	J 16	J 17	10.198	68.023	0.000	68.023
17	J 17 -> J 18	J 17	J 18	14.998	154.036	0.000	154.036
18	J 18 -> J 19	J 18	J 19	21.074	201.614	0.000	201.614
19	J 19 -> J 20	J 19	J 20	14.996	55.528	0.000	55.528
20	J 20 -> J 21	J 20	J 21	20.627	72.920	0.000	72.920
21	J 21 -> J 22	J 21	J 22	15.000	62.519	0.000	62.519
22	J 22 -> J 23	J 22	J 23	15.041	55.475	0.000	55.475
23	J 23 -> J 24	J 23	J 24	15.005	72.762	0.000	72.762
24	J 24 -> J 25	J 24	J 25	14.999	87.729	0.000	87.729

Page 2 from 3 - River Flow v9.0 - Copyright © 1996-2010 Technologismiki

Page 2 of 3, Zoom: 55%

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

X

10   Help

10.1   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

10.2   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.

10.3   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.

10.4   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.
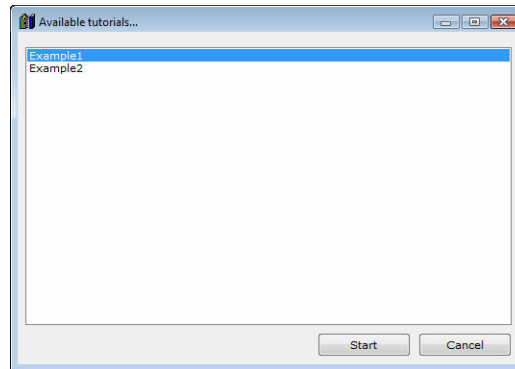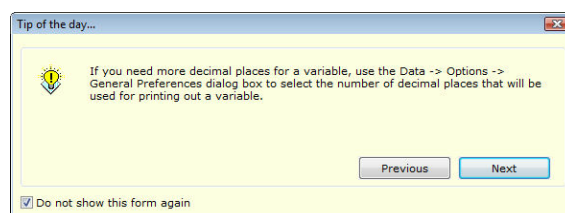

NOTE: The number and content of the tutorials is changed frequently. Use the live update system of TechnoLogismiki's products to download the latest tutorials.

10.5 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.

10.6   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.
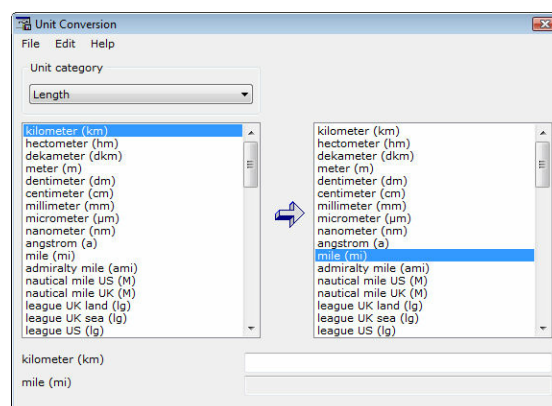
> **NOTE**: If an error message appears then the unit conversion tool has not been installed. You can install the unit conversion tool from the installation CD or the Internet.

10.7   TechnoLogismiki website

With this option, you can load on your Internet browser the website of TechnoLogismiki's.

10.8   Buy products

With this option, you can load on your Internet browser the main product page of TechnoLogismiki's website.

10.9   TechnoLogismiki NOMOS

With this option, you can load on your Internet browser the **NOMOS** service of Technologismiki.

10.10   TechnoLogismiki Live!

With this option, you can load on your Internet browser the **Live!** service of Technologismiki.

10.11   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

XI

11 Appendix
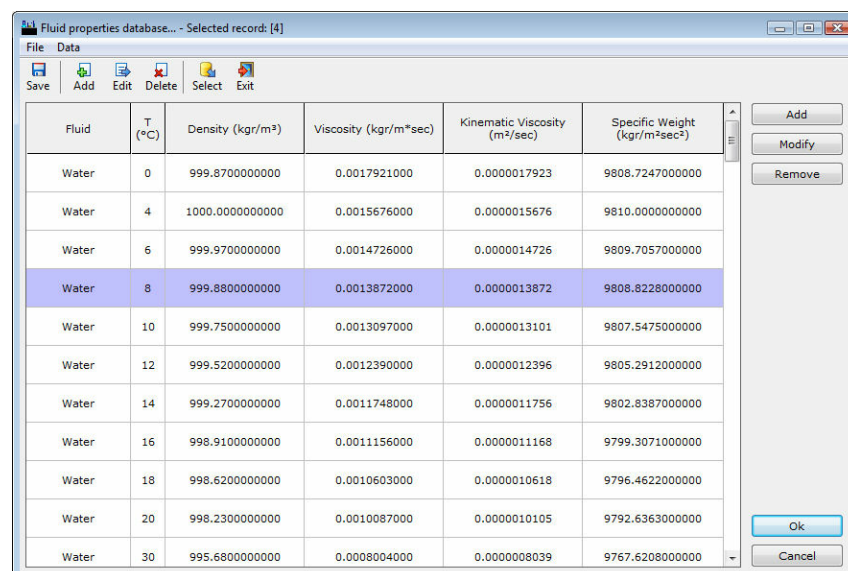
11.1 Unit system

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

11.2 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.
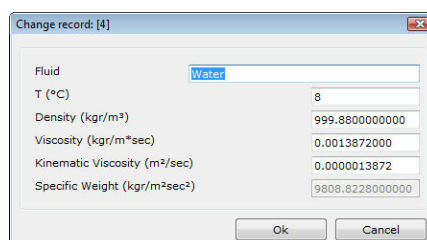

To add a new record:

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.
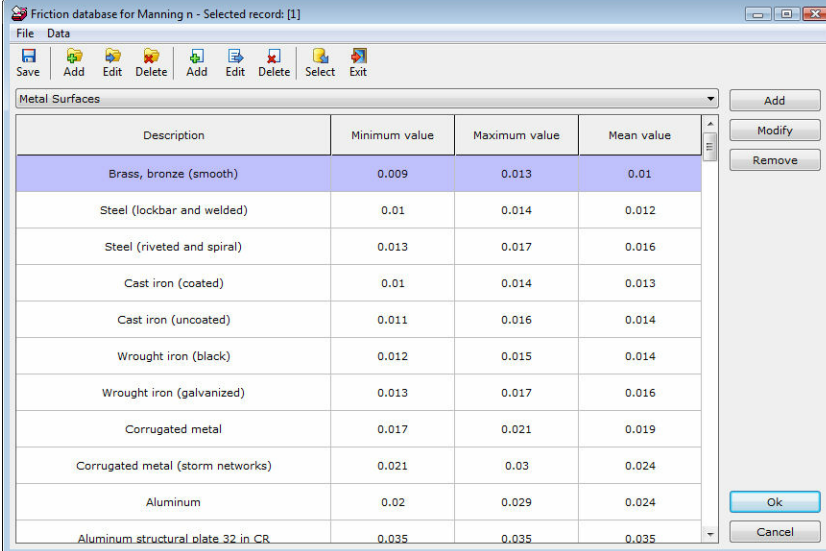
To remove an existing record:

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.

11.3 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.
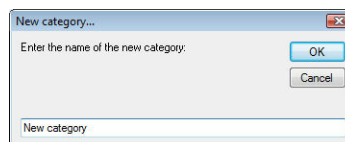
The database consists of several categories. Usually, the category defines the material of the surface (e.g. Metal surfaces).
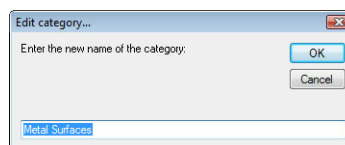
To add a new category:

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.
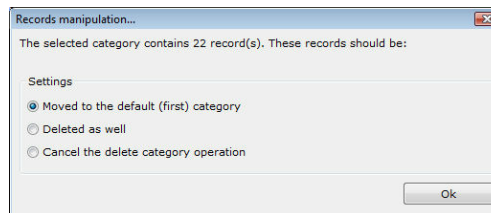**3.** Select **Ok** to add the category at the end of the list. Select **Cancel** to cancel the procedure.

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.
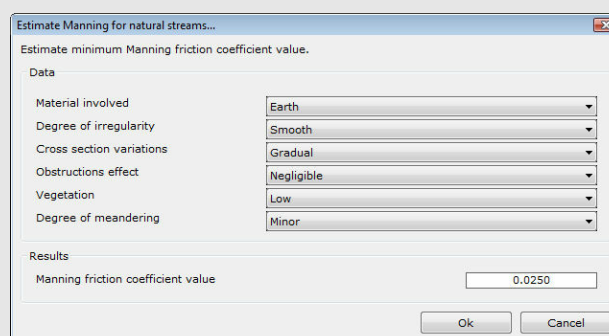**5.** Click **Ok** to proceed.

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.
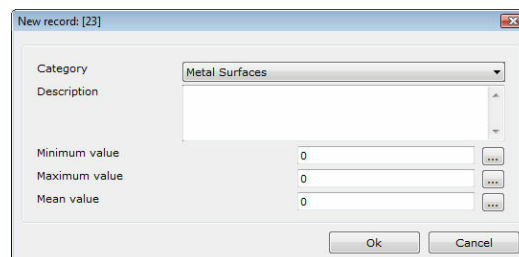
NOTE: In case of Manning friction coefficients in natural streams, you can estimate the values based on several characteristics of the stream. Click on the buttons with the ellipses (...) next to the text boxes to invoke the following dialog box:

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.

11.4   **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

11.5 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

11.6 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
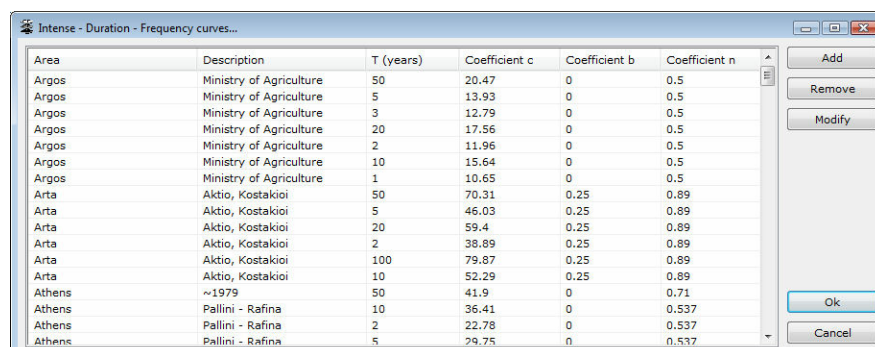
11.7 Darcy - Weisbach friction coefficients

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

11.8 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.
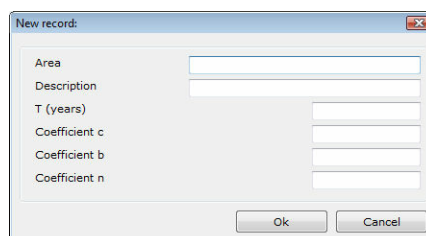
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database. Note that the database was built using data from various resources in the literature; you need to be cautious when using a curve.
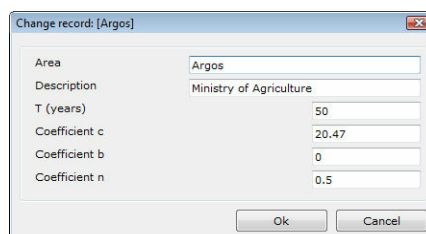

To add a new record:

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.
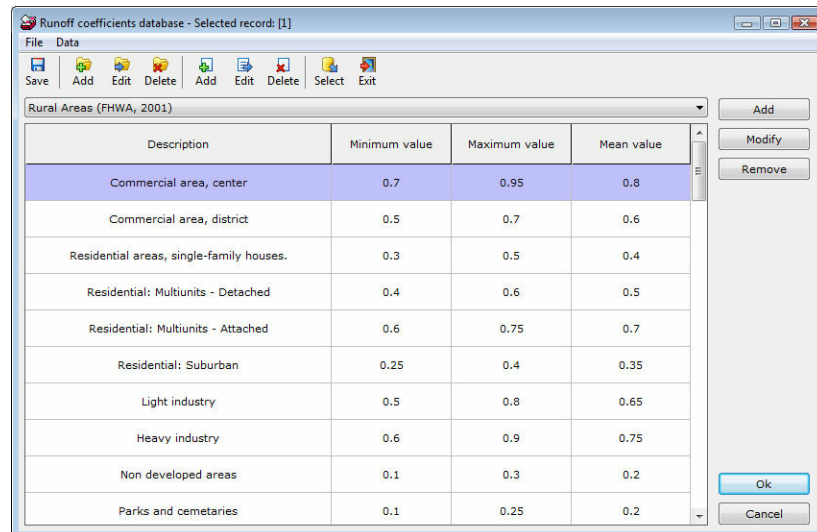
To remove an existing record:

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.

11.9 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).
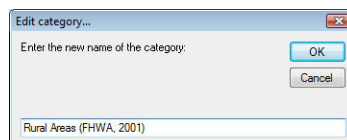
To add a new category:

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.
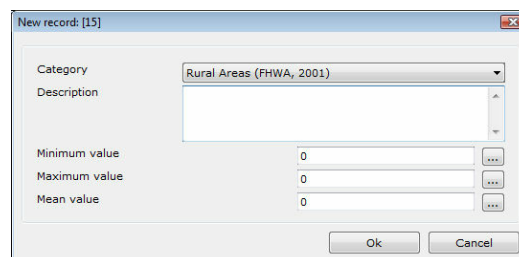
NOTE: You can calculate the mean runoff coefficient for complex areas with known runoff coefficients. Click on the buttons with the ellipses (...) next to the text boxes to invoke the following dialog box:
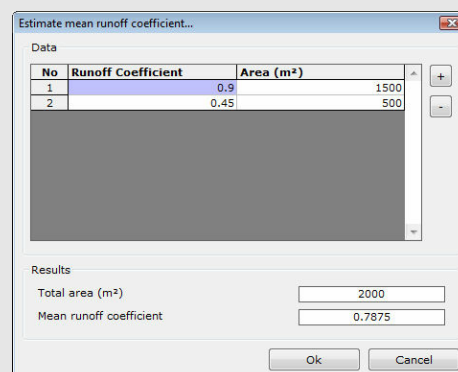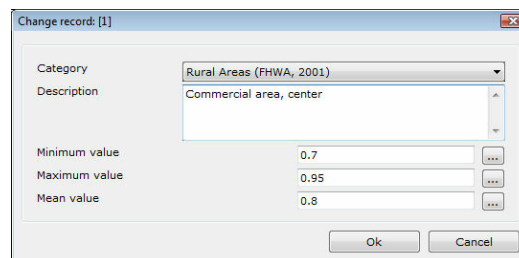

Click the plus sign **(+)** to add a new area. Type the runoff coefficient and the area in m^2. The total area and the mean runoff coefficient is displayed in the **Results** frame. Click the minus sign **(-)** to delete the selected area. The area is deleted with no

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.

11.10 Error messages

11.10.1 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

11.10.2 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.

11.10.3 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.

11.10.4 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.

11.10.5 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

- A -

about   131
above   83
actions   67
add   60, 73, 77, 87, 88, 89, 90, 91, 102
address   11
adobe   46
aluminum   138, 140
appearance   79, 84
append   54
apply   54
arcview   23, 27, 36, 44
area   112
asbestos   138, 139, 140
ASCII   34, 47
asphalt   139, 140
author   65
automatic   79

- B -

background   28, 59, 60, 61, 62
backup   15, 79
Bazin   139
bck   15, 79
behavior   83
below   83
bmp   38
branch   105
brick   138, 139, 140
broom   138
browser   131
button   84, 131

- C -

calculation   70, 79, 107, 118, 120, 122, 123
cast   138, 139, 140
CD   129
cell   83
cement   138, 139, 140
centrifugal   138, 139, 140
change   16, 49, 79

check   78, 79
clean   138
clear   54, 56, 71
climate   70
clipboard   52
close   49
CMP   138, 140
coefficient   135, 138, 139, 140, 141
column   83
command   51
comment   65
common   113
communicator   131
compatibility   15
computations   67
computer   79
concrete   138, 139, 140
conduit   89, 98, 103, 115
configuration   11
confirmation   79
connectivity   56
constant   70
content   129, 130
contents   129
contour   114
contours   30, 71
control   67
conversion   131
copper   138, 139, 140
copy   52, 53, 112
correlation   70
create   14
current   54, 78
cursor   83, 84
curve   70
cut   52

- D -

Darcy   140
data   45, 52, 53, 59, 65, 67, 83, 84, 112, 113, 134
database   134, 135, 140, 141
date   65, 79, 131
decimal   79
default   84
delete   14, 61, 76, 78, 79, 103, 113, 115
deletion   79
delimiter   53
density   134

This page was intentionally left blank.