Works 2013

Hydraulic solver

Version 11.0.0

USER GUIDE

TECHNO logismiki
Advanced Technical Software

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

Hydraulic solver

TechnoLogismiki

Hydraulic solver

Publisher

TechnoLogismiki

Editors

Fotis Fotopoulos
Aristotelis Charalampakis

Technical Assistance

Antigoni Egglezou

Printed: September 2012 in Athens, Greece.

Table of Contents

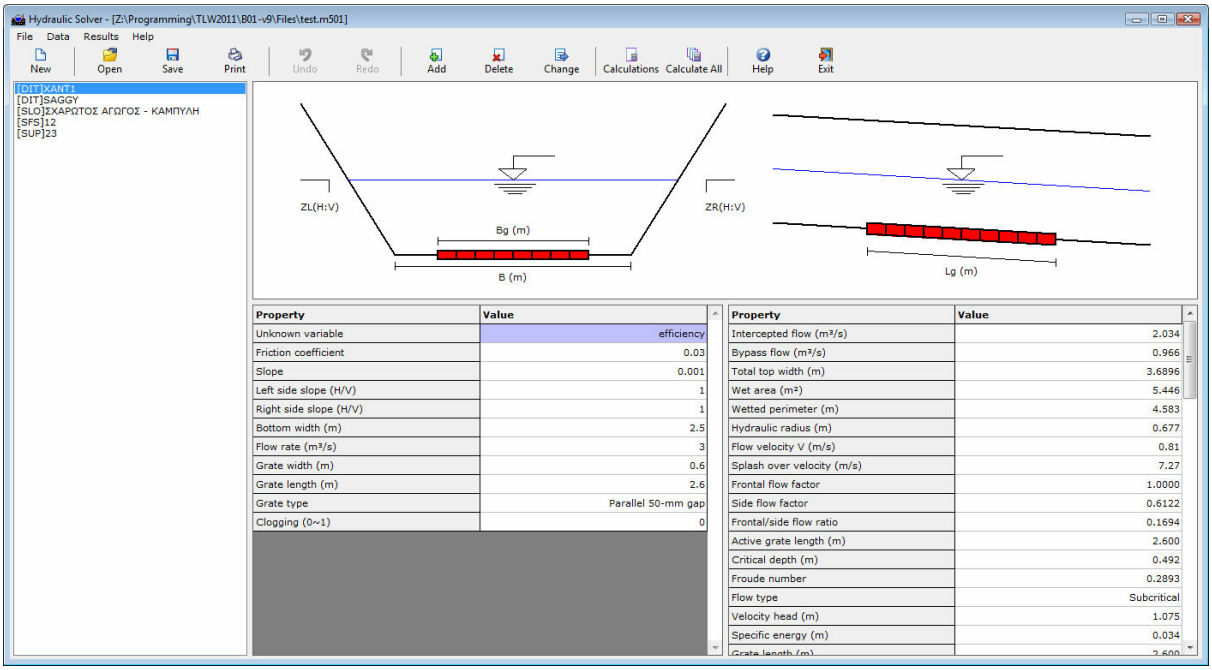
Chapter

1 About the program

1.1 What does the program do?

This program allows the calculation of flow problems in:

- sections with free surface
- sections under pressure
- orifices
- weirs
- grates
- curbs
- combination inlets
- ditches
- slots
- pumps
- reaction turbines
- impulse turbines
- hydropower pumps



For the calculation of the linear friction losses several formulas may be used, such as Hazen - Williams, Manning, Kutter, Bazin etc. The friction coefficients can also be calculated from several formulas, according to the preferences of the user.

For each type of problem a number of solvers may be invoked, based on the available data. For example, for the flow with free surface, the unknown variable may be the flow depth for given flow rate and slope or the unknown variable may be the flow rate for given flow depth and slope etc.

The rating tools (rating graphs or rating tables) allow the optimum design of sections based on custom criteria. The rating graph is created for a specific output subject to

variation of up to two variables, whereas the rating table is created for a specific output subject to variation of up to five variables.

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

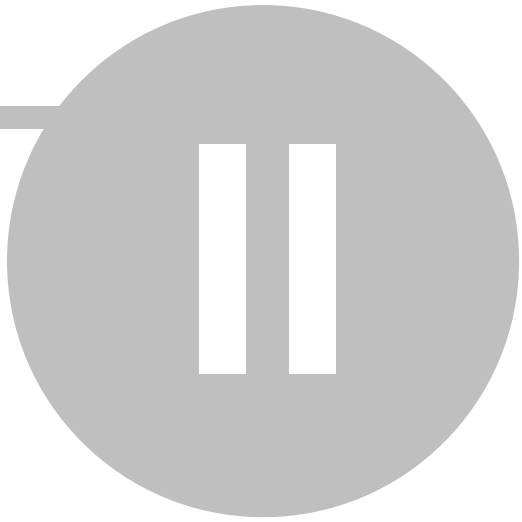
The normal response time is within two business days. If your inquiry cannot be answered by e-mail, a customer service representative will contact you by telephone.

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



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
  - Solution from file
  - From GRD file
  - From PCS file
  - From ArcView Shapefile
- Export
  - Solution to file
  - To GRD file
  - To PCS file
  - To ArcView Shapefile
  - Export sketch
- 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:

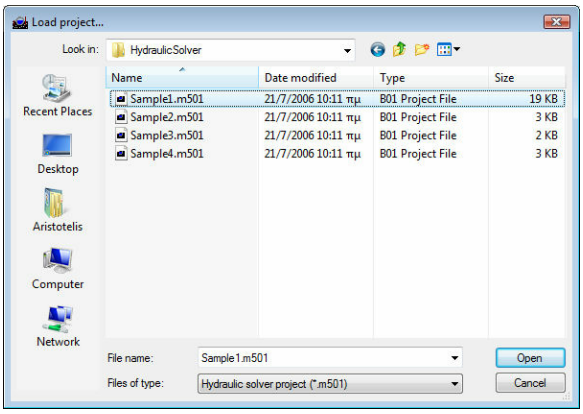
1. Select **New project** from the **File** menu.
2. If a project is already loaded and changes have been made, a warning message will appear that asks the user whether to save the changes or not.
3. The current project is erased and a new project is started.

2.3 Open project

With this option, an existing project is loaded. The project may be stored 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 "Hydraulic solver project" with the extension .m01.
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\HydraulicSolver

Supported file types

- **M01** (Hydraulic solver project): Files created by version 2012 and 2013 of Hydraulic solver.
- **M501** (Hydraulic solver project): Files created by versions 2011, 2010, 2009, 2008, 2007 and 5.0 of Hydraulic solver.
- **M521** (Hydropower works v5.xx): Files created by Hydropower works version 5.0. This program has been merged with Hydraulic solver 2007 or later and has been discontinued.
- **MB9** (Free surface sections v4.xx): Files created by Free surface sections version 4.0. This program has been merged with Hydraulic solver 2007 or later and has been discontinued.
- **M10** (Flow under pressure v4.xx): Files created by Flow under pressure version 4.0. This program has been merged with Hydraulic solver 2007 or later and has been discontinued.
- **MB1** (Hydraulic pump v4.xx): Files created by Hydraulic pump version 4.0. This program had been updated to version 5.0 but now it has been merged with Hydraulic solver 2007 or later and has been discontinued.
- **M21** (Hydropower works v4.xx): Files created by Hydropower works version 4.0. This program had been updated to version 5.0 but now it has been merged with Hydraulic solver 2007 or later and has been discontinued.
- **MB9** (Section solver v3.xx): Files created by Section solver version 3.0. This program had been updated to version 5.0 but now it has been merged with Hydraulic solver 2007 or later and has been discontinued.
- **M10** (Flow under pressure v3.xx): Files created by Flow under pressure version 3.0. This program had been updated to version 4.0 but now it has been merged with Hydraulic solver 2007 or later and has been discontinued.

- **MB1** (Hydraulic pump v3.xx): Files created by Hydraulic pump version 3.0. This program had been updated to version 4.0 but now it has been merged with Hydraulic solver 2007 or later and has been discontinued.
- **M21** (Hydropower works v3.xx): Files created by Hydropower works version 3.0. This program had been updated to versions 4.0 and 5.0 but now it has been merged with Hydraulic solver 2007 or later and has been discontinued.
- **MB2** (Section solver v2.xx): Files created by Section solver version 2.0 or earlier. This program had been updated to version 3.0 and 4.0 but now it has been merged with Hydraulic solver 2007 or later and has been discontinued.
- **MB5** (Uniform flow v2.xx): Files created by Uniform flow version 2.0 or earlier. This program had been discontinued since version 3.0 and now it has been merged with Hydraulic solver 2007 or later.
- **M10** (Flow under pressure v2.xx): Files created by Flow under pressure version 2.0. This program had been updated to version 3.0 and 4.0 but now it has been merged with Hydraulic solver 2007 or later and has been discontinued.
- **MB1** (Hydraulic pump v2.xx): Files created by Hydraulic pump version 2.0. This program had been updated to version 3.0 and 4.0 but now it has been merged with Hydraulic solver 2007 or later and has been discontinued.
- **BCK** (Backup files): If you have selected from program options the creation of backup copy when a file is loaded, then the file can be loaded by selecting Backup files (*.bck) from the Files of type drop-down list.
- ***.*** (All files): Displays all files in the current folder.

Backwards compatibility

This version implements full backwards compatibility; therefore, the loading of a project that has been created with older versions of the program or other programs that have been merged with Hydraulic solver is guaranteed. Note however that when a project is saved with the latest format, it cannot be used by previous versions.

NOTE: If the message "Could not load project. File may be corrupt or saved by an unknown or incompatible version of the program" appears, 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 can 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 attempt to save 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:

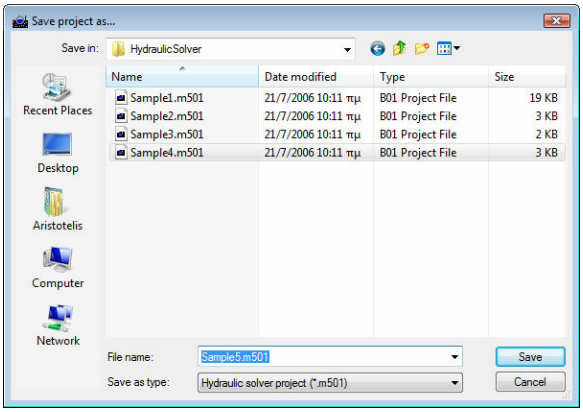
1. Select **Save project** from the **File** menu.
2. If the location of the file is already set, the project is saved to this file without any messages. If the filename is not set, a dialog box will appear that allows the selection of the filename and path.

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

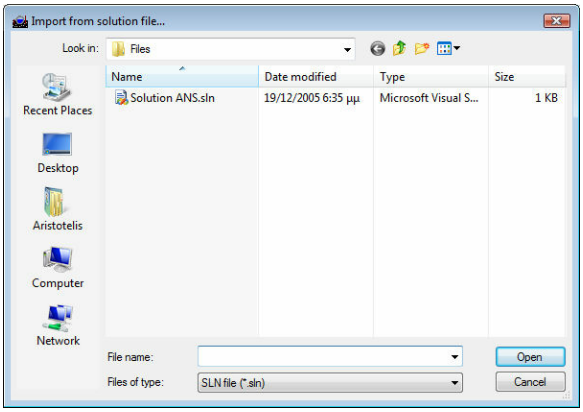
2.6.1 Solution from file

A project may contain one or more solutions. With this option, solutions in files with the extension .sln can be imported in the current project. You can create solution files by exporting a solution.

To import solutions from an existing file to the current project:

1. Select **Import** from the **File** menu.
2. Select **Solution from file** from the **Import** menu.
3. Select the location of the solution file.
4. Select the file type from the **Files of type** drop-down list. The default option is "SLN file" with the extension .sln.
5. Select the file by clicking on it.
6. Select **Open** to import the solution to the current project. The solution is appended

to the end of the list of the solutions. Select **Cancel** to cancel the operation.



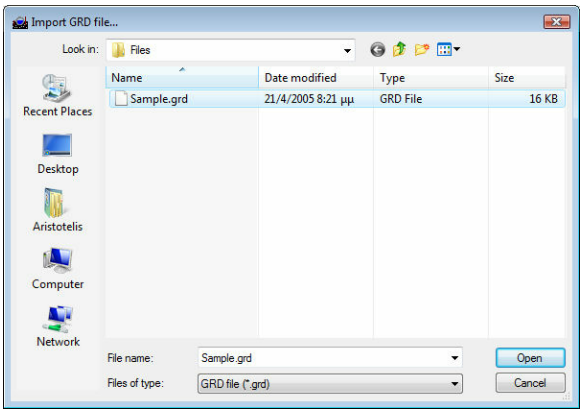
NOTE: With this option, you can import solutions exported by the same version of the program.

2.6.2 From GRD file

GRD files are created by many popular programs such as VERM, Anadelta, Odos as well as Hydraulic programs by TechnoLogismiki. They contain geometric information on a prismatic cross section with straight edges as well as the title of the section, station data etc.

To import a GRD file:

1. Select **Import** from the **File** menu.
2. Select **from GRD file** from the **Import** 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 "GRD file" with the extension .grd.
5. Select the file by clicking on it.
6. Select **Open** to open and analyze the file. Depending on the contents of the file, there are two cases.

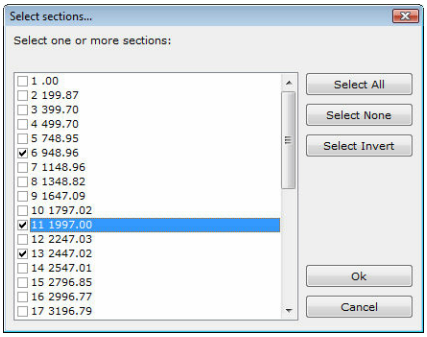


Case 1

The GRD file contains only one section. In this case, a new solution of flow with free surface with no calculations is appended to the list of solutions of the current project.

Case 2

The GRD file contains more than one sections. In this case, a dialog box appears that allows the selection of one or more sections. Select the sections that you wish to import to the project.



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

Select **Ok** to proceed. For each of the selected sections, a new solution of flow with free surface with no calculations is appended to the list of solutions of the current project. Select **Cancel** to close the dialog box with no changes.

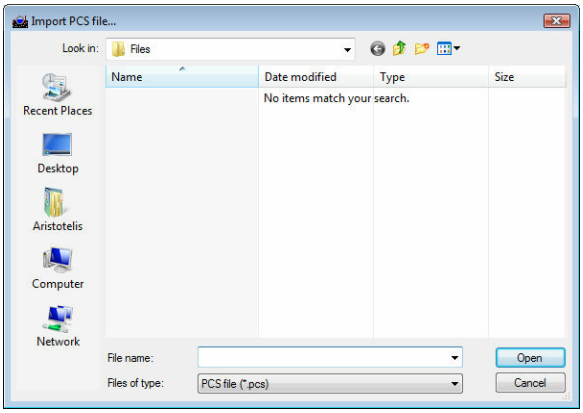
NOTE: By default, the type of the new solutions is set to flow with free surface with no calculations. You can change the type of the solution by selecting Change solution from the **Data** menu.

2.6.3 From PCS file

PCS files can be created by Hydraulic programs (by TechnoLogismiki) version 2.0 or later. It is the preferred way to exchange section data between programs and users. Each file contains a single section, with information on the prismatic section and the friction coefficient of each edge.

To import a section from a PCS file:

1. Select **Import** from the **File** menu.
2. Select **from PCS file** from the **Import** 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 "PCS file" with the extension .pcs.
5. Select the file by clicking on it.
6. Select **Open** to open and analyze the file. A new solution of flow with free surface with no calculations is appended to the list of solutions of the current project. Select **Cancel** to cancel the operation.



NOTE: By default, the type of the new solutions is set to flow with free surface with no calculations. You can change the type of the solution by selecting Change solution from the **Data** menu.

2.6.4 From ArcView Shapefile

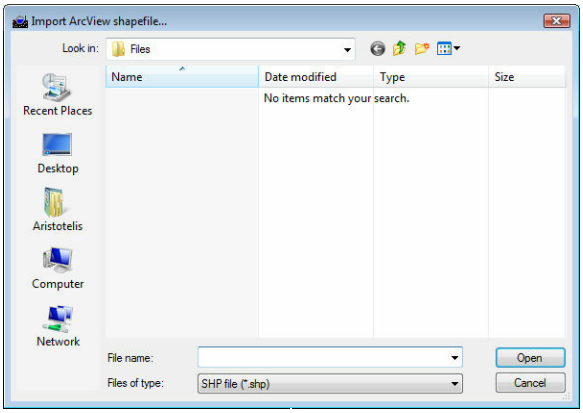
Shapefiles can be created by several programs such as ArcView GIS, MapInfo, GPS Trackmaker etc. In reality, this "file" consists of three files with the extensions shp, shx and dbf. The GIS driver recognizes the following shapefile types:

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

Shapefiles containing Polygons (simple, M and Z) and Multipatch are not recognized.

To import data from a shapefile:

1. Select **Import** from the **File** menu.
2. Select **from ArcView Shapefile** from the **Import** 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 "SHP file" with the extension .shp.
5. Select the file by clicking on it.
6. Select **Open** to open and analyze the file. A new solution of flow with free surface with no calculations is appended to the list of solutions of the current project. Select **Cancel** to cancel the operation.



NOTE: By default, the type of the new solutions is set to flow with free surface with no calculations. You can change the type of the solution by selecting Change solution from the **Data** menu.

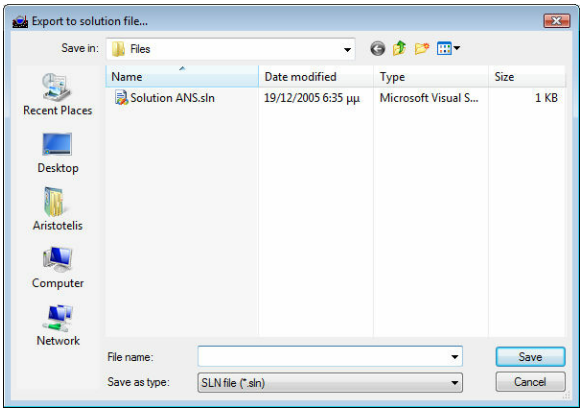
2.7 Export

2.7.1 Solution to file

A project may contain one or more solutions. With this option, a file containing a single solution from the current project can be exported. This is the preferred option when you want to send a solution to another user.

To export a solution to a file:

1. Select the solution containing the section data you wish to export from the list in the main form.
2. Select **Export** from the **File** menu.
3. Select **Solution to file** from the **Export** menu.
4. Select the location of the new file.
5. Type the filename in the **File name** text box.
6. Select **Save** to create the solution file with the extension .sln. Select **Cancel** to cancel the operation.



To import a solution from a file, select Solution from file from the **File > Import** menu.

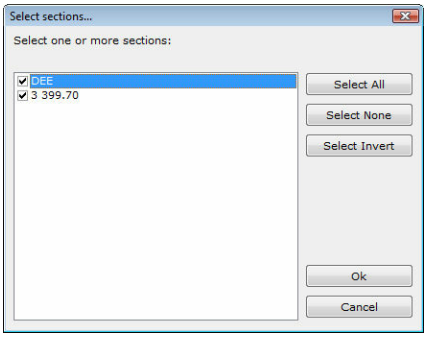
NOTE: A solution file contains a single solution only. The solution currently selected in the list of the main form will be saved in the file.

2.7.2 To GRD file

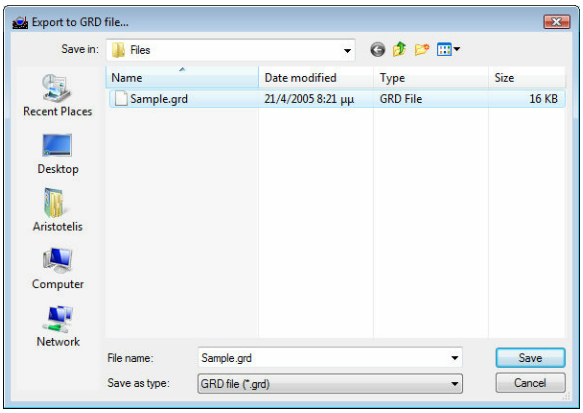
With this option, a GRD file containing the sections of one or more solutions of the current project is created. GRD files contain geometric information on a prismatic cross section with straight edges as well as the title of the section, station data etc.

To export sections to a GRD file:

1. Select **Export** from the **File** menu.
2. Select **To GRD file** from the **Export** menu.
3. The solutions included in the current project are analyzed. If there are no solutions that contain section data then the process is terminated automatically. If there are more than one solutions that contain section data, then the process is lead to step 4, or else to step 5.
4. A dialog box appears that allows the selection of the solutions that will be included in the GRD file. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all solutions, deselect all solutions and invert the current selection.



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.



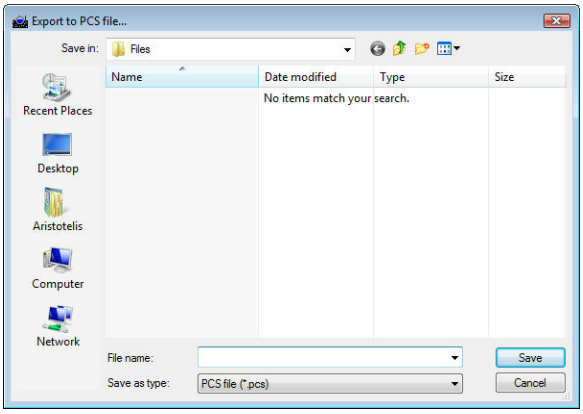
NOTE: Information on the friction coefficients and curved edges (if any) will not be exported as they are not supported by the file format.

2.7.3 To PCS file

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

To export a section to a PCS file:

1. Select the solution containing the section data you wish to export from the list in the main form.
2. Select **Export** from the **File** menu.
3. Select **To PCS file** from the **Export** menu.
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.



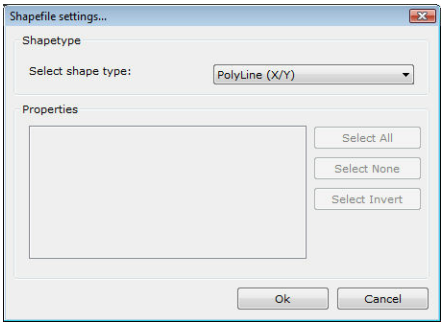
NOTE: Information on curved edges (if any) will not be exported as they are not supported by the file format.

2.7.4 To ArcView Shapefile

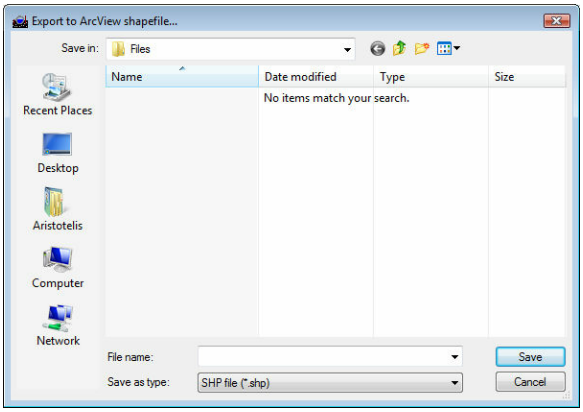
With this option, you can create a shapefile that can be used by programs such as ArcView GIS, MapInfo, GPS Trackmaker and other.

To export a section to an ArcView Shapefile:

1. Select the solution containing the section data you wish to export from the list in the main form.
2. Select **Export** from the **File** menu.
3. Select **To ArcView Shapefile** from the **Export** menu.
4. Select the **shape type** from the drop-down list.



5. Select **Ok** to proceed. Select **Cancel** to abort the operation and close the dialog box.
6. Select the path of the file.
7. Type the filename in the **File name** text box.
8. Select **Save** to create the file. Select **Cancel** to cancel the operation.



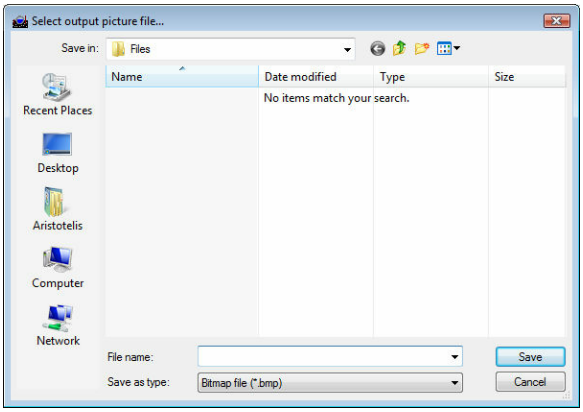
NOTE: Not all shape types are compatible with all programs.

2.7.5 Export sketch

With this option, you can export (in BMP format) the sketch of the solution as it appears in the main form of the program.

To export the sketch of the solution:

1. Select the solution from the list of the main form in order to display the sketch.
2. Select **Export** from the **File** menu.
3. Select **Export sketch** from the **Export** menu.
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.

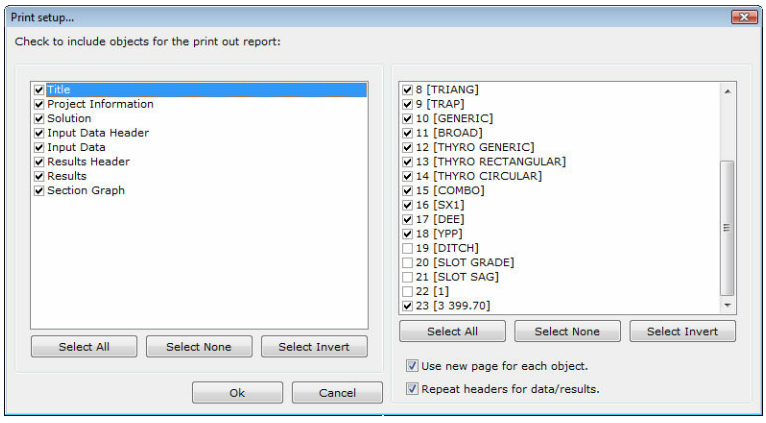


2.8 Print setup

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 printed for each solution, from the list on the left.
3. Select the **solutions** that will be included in the report from the list on the right.
4. Check **Use new page for each object** if you want to use a new page for each solution in the report.
5. Check **Repeat headers for data/results** if you want to repeat the headers each time a separate data or results table is used.
6. 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.

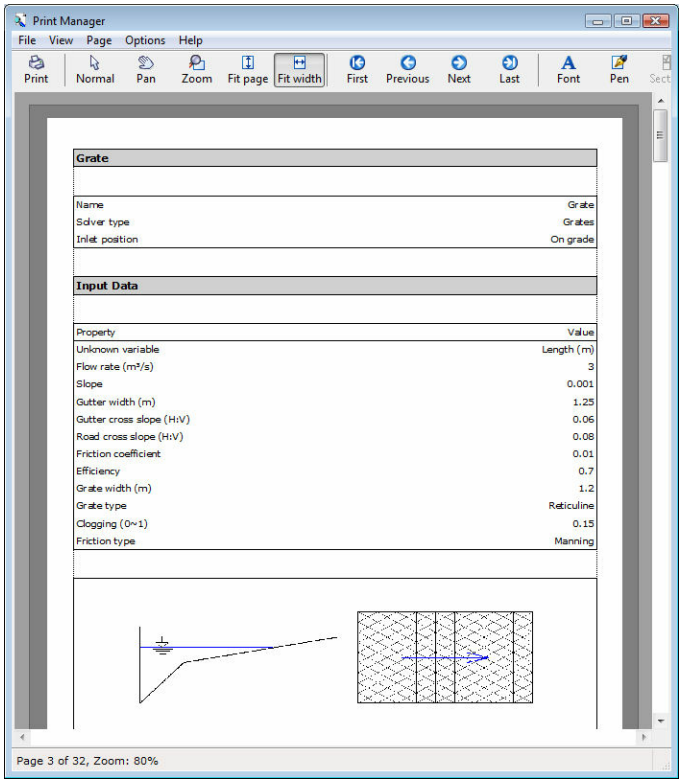
2.9 Print

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.

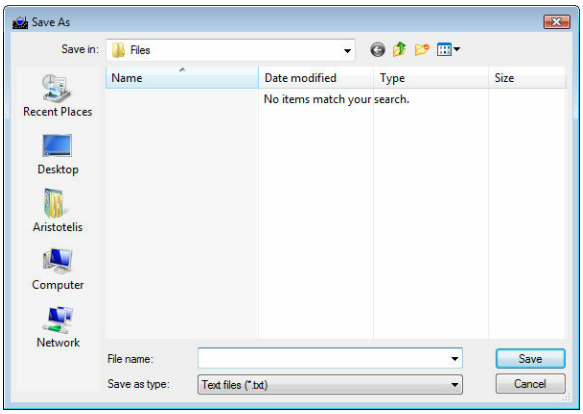
2.10 Print to

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

3 Data

3.1 Data menu

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

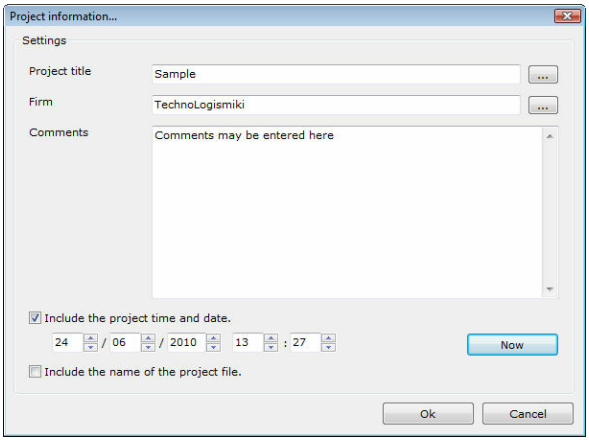
- Project info
- Undo
- Redo
- Add solution
- Change solution
- Delete solution
- Move up
- Move down
- Friction calculation
- Copy from solution
- Options
  - General preferences
  - Grid editing
  - Customize toolbar
  - Sketch

3.2 Project info

With this option, you can add project information that include, optionally, title, author 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, author and comments.
3. Check **Include project time and date** if you want to include the time and date in the project.
3.1. Type the day, month, year, hours and minutes in the corresponding text boxes. Alternatively, you may click on the up/down arrows to increase or decrease the respective value in the text box.
3.2. If you click on **Now** then all text boxes are filled with 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.



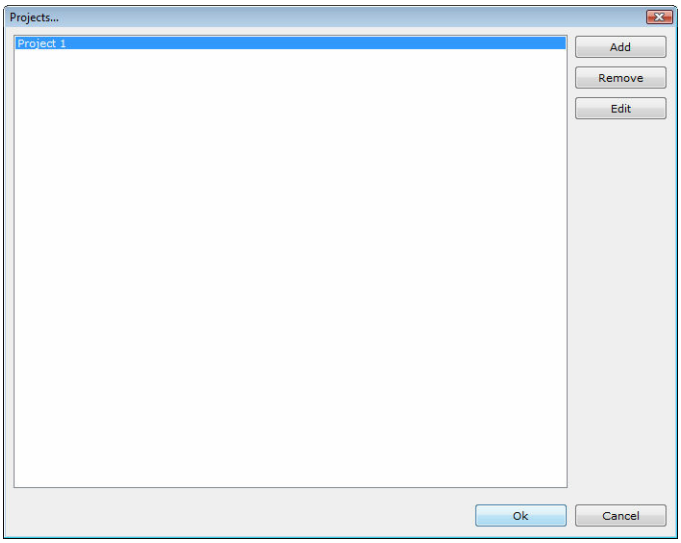
By selecting the buttons with the ellipses (...) next to the project title and author, you can access the corresponding databases.

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.

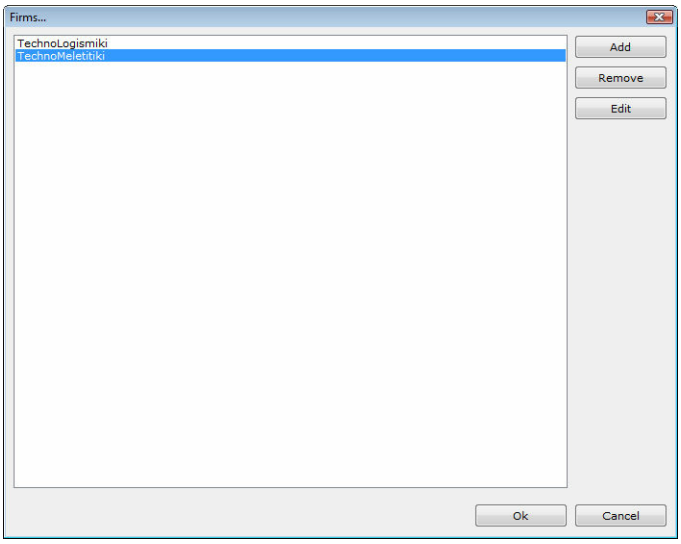


Author database

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

To use the author database:

1. Select the button with the ellipses (...) next to the author text box. The author database appears.
2. Select **Add** to add a new 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 author and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



3.3 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.

3.4 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.

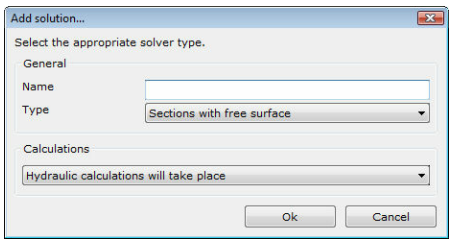
It is possible to redo more than one changes that were previously undone by following the steps described above. The number of actions that are kept in memory and may be undone or redone is 20 by default. This means that the program is able to keep track of up to 20 successive changes that are undone 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.

3.5 Add solution

With this option, you can add a new solution at the end of the list.

To add a solution:

1. Select **Add solution** from the **Data** menu.
2. Type the name of the solution in the corresponding text box. The name can be any string that has not been used in another solution of the current project.
3. Select the **type** of the problem (see below)
4. Select the appropriate calculation options. The available combinations are shown in the table below.
5. Select **Ok** to create a new solution and append it to the end of the list. Select **Cancel** to close the dialog box without any changes.



Solver	Prefix	Options	Description
Sections with free surface	SFS	• Hydraulic calculations will take place • Only draw the section	Problem of flow with free surface with calculations without flow rate or with given flow depth.
Sections under	SUP	• Hydraulic	Problem of flow under pressure with

pressure		calculations will take place • Only draw the section	hydraulic calculations or without flow rate.
Weirs	WEI	• Rectangular sharp-crested • Triangular sharp-crested • Trapezoid (cipoletti) sharp-crested • Generic • Broad	Several types of weirs. If a weir is not broad and cannot be described by one of the available sharp-crested types, it can be described by generic type.
Orifices	ORI	• Generic • Rectangular • Circular	Several types of orifices. If the orifice is not circular or rectangular, it can be described by generic type. In this case, the area of the hole can be entered directly.
Grates	GRA	• On grade • In sag	The grate can be located on grade (where the slope is non zero) or in sag, i.e. in a deep point where storm water stagnates.
Curbs	CUR	• On grade • In sag	The curb can be located on grade (where the slope is non zero) or in sag, i.e. in a deep point where storm water stagnates.
Combination inlets	CMB	• On grade • In sag	The combination inlet can be located on grade (where the slope is non zero) or in sag, i.e. in a deep point where storm water stagnates.
Slots	SLO	• On grade • In sag	The slot can be located on grade (where the slope is non zero) or in sag, i.e. in a deep point where storm water stagnates.
Ditches	DIT	• On grade • In sag	The ditch can be located on grade (where the slope is non zero) or in sag, i.e. in a deep point where storm water stagnates.
Pumps	PUM	-	Calculation of pump power needed to elevate the pressure line of a fluid through a system containing valves, curves and confusors.
Reaction turbines	RCT	-	Calculation of geometric characteristics and properties of Francis-type reaction turbines used in hydropower works.
Impulse turbines	IMT	-	Calculation of geometric characteristics and properties of impulse turbines used in hydropower works.

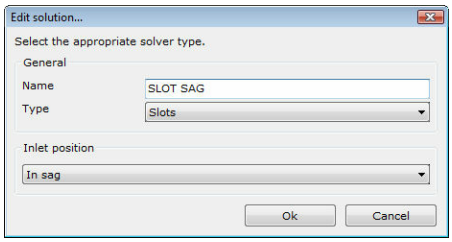
Hydropower pumps	HPP	-	Calculation of geometric characteristics and properties of pumps used in hydropower works.

3.6 Change solution

With this option, you can change the name, type and options of a solution. Since different solvers require different input data (for example, orifices and sections under pressure), if the type of the solution is changed then only the common data (in this case, the flow rate) is kept. All other data of the new type must be entered by the user.

To change the properties of an existing solution:

1. Select the solution you wish to change from the list in the main form.
2. Select **Change solution** from the **Data** 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.

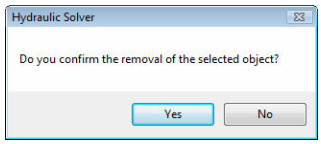


3.7 Delete solution

With this option, you can delete an existing solution.

To delete an existing solution:

1. Select the solution you wish to delete from the list in the main form.
2. Select **Delete solution** from the **Data** menu.
3. You may be prompted to confirm the deletion depending on what you have selected in General preferences. If you select No then the deletion is canceled.
4. The solution is deleted.



3.8 Move up

With this option, you can move an existing solution up in the list. This command, together with Move down, are useful when you want the solutions to be presented in the report in a specific order.

To move a solution up in the list:

1. Select the solution you wish to move up from the list in the main form.
2. Select **Move up** from the **Data** menu.
3. The selected solution is moved up in the list by one place.

NOTE: This command has no effect if the selected solution is first on the list.

3.9 Move down

With this option, you can move an existing solution down in the list. This command, together with Move up, are useful when you want the solutions to be presented in the report in a specific order.

To move a solution down in the list:

1. Select the solution you wish to move down from the list in the main form.
2. Select **Move down** from the **Data** menu.
3. The selected solution is moved down in the list by one place.

NOTE: This command has no effect if the selected solution is last on the list.

3.10 Friction calculation

With this option, you can select the formulas that will be used for the calculation of the friction losses and calibrate the solvers.

To select the friction formulas:

1. Select the solution for which you wish to select the friction formulas from the list in the main form.
2. Select **Friction calculation** from the **Data** 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.

Depending on the type of flow (under pressure or with free surface), two different dialog boxes may appear:

A. Flow under pressure

In case the flow is under pressure, you can select up to three different friction formulas, depending on the type of flow (turbulent, transient, laminar). If you check **Set as defaults for all new projects** then these values will be preselected for all new projects.

First, you define the boundaries of turbulent and laminar flow. The use of Re>4000 for turbulent flow and Re<2000 for laminar flow is recommended. For intermediate values, the flow is considered to be transient. If you set the same friction formula for all types of flow, this analysis has no effect.

Note that different friction formulas need different friction coefficients. The first three choices are variations of the well-known Manning formula. The first, named **Manning,** assumes constant friction coefficient. The other two calculate the friction coefficient as

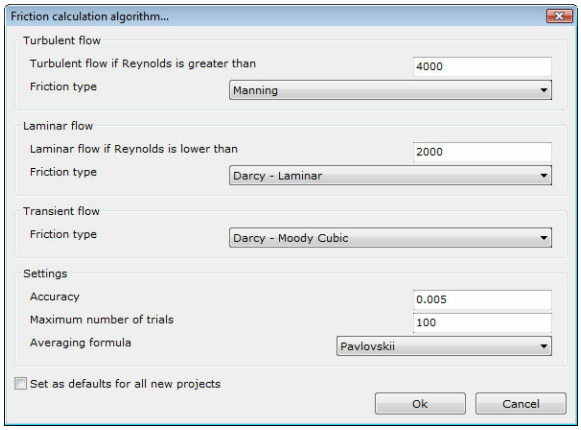
a function of the fill ratio and the coefficient that corresponds to full flow. These three formulas give the same results in flow under pressure; this is not the case for flow with free surface, where up to 30% difference may be observed.

The **Accuracy** and **Maximum number of trials** ensure the stability of the algorithm. The default values are 0.0005 for the accuracy and 1000 for the maximum number of trials. It is recommended that you do not change these values.

The averaging formula may take one of the following values:

- Pavlovskii
- Colebatch
- Horton
- Cox
- Lotter

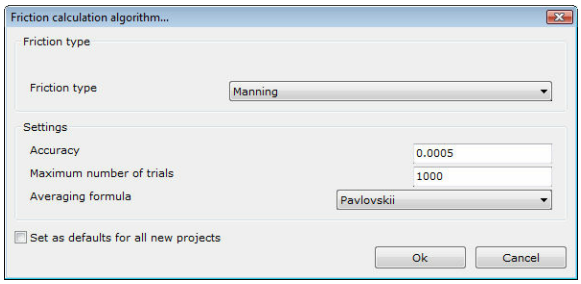
The above formulas are used when there are different friction coefficients within the same section and an average friction coefficient is needed.



B. Flow with free surface

In case the flow is with free surface, you can select a single friction formula. If you check **Set as defaults for all new projects** then this value will be preselected for all new projects. The friction formulas available are: Manning, Bazin, Kutter, Ganguillet-Kutter, Chezy, Hazen-Williams.

The **Accuracy** and **Maximum number of trials** ensure the stability of the algorithm. The default values are 0.0005 for the accuracy and 1000 for the maximum number of trials. It is recommended that you do not change these values.

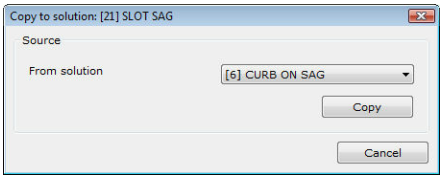


3.11 Copy from solution

With this option, you can copy all data from another existing solution of the same project. This command is particularly useful when you want to solve multiple problems that are almost the same. For example, you may want to solve the problem of flow under pressure of the same section with slight variations of the friction coefficient.

To copy all data from another existing solution:

1. Select the target solution from the list in the main form.
2. Select **Copy from solution** from the **Data** menu.
3. The name of the target solution is displayed in the caption of the dialog box.
4. Select the source solution from the drop-down list.
5. Select **Copy** to copy the data from the source solution to the target solution. Select **Cancel** to cancel the operation.



NOTE: The data of the target solution are erased and substituted by the data of the source solution.

3.12 Options

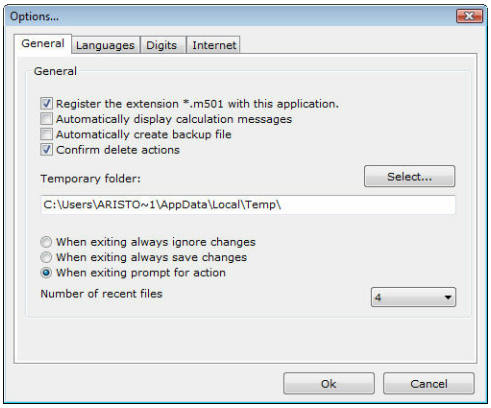
3.12.1 General preferences

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 *.m01 with this application** to associate the extension .m01 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.

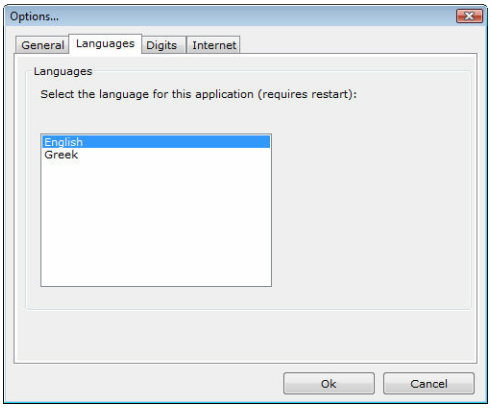
Check **Confirm delete actions** if you want to be asked for confirmation each time an object is about to be deleted. This setting affects the behavior of all delete actions, for example the deletion of a solution

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.

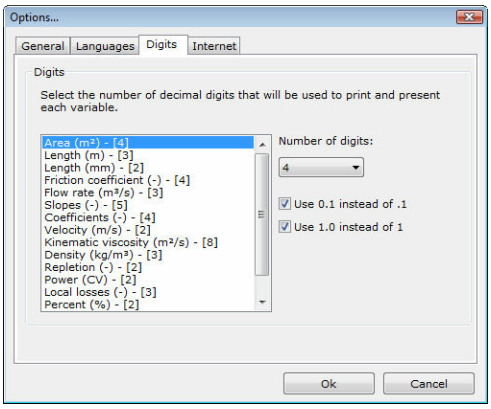
Languages Tab



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



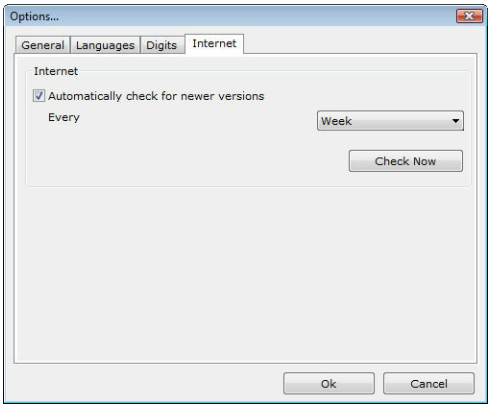
With this tab, you can modify the way the results are presented. All values used in the program are displayed in the list on the left.

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.

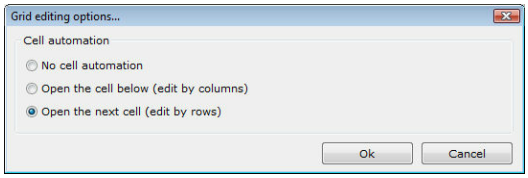
When a newer version is found, you will be prompted to download and install the latest version.

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.

3.12.2 Grid editing

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



The behavior 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.

In some cases, the program may automatically fill some missing values (for example, when performing linear interpolation). In this case, you can select a distinctive color in order to recognize these values. You can choose the color by clicking on the button in the **Auto-complete settings** frame.

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

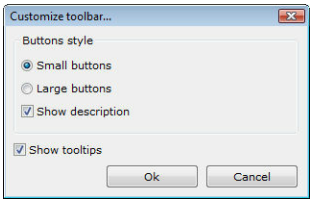
Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

3.12.3 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, old and new.

3.12.4 Sketch

With this option, you can show or hide the sketch in the main form.

To show or hide the sketch in the main form:

1. Select **Options** from the **Data** menu.
2. Select **Sketch** from the **Options** menu.
3. If the sketch is hidden, it is shown and vice versa.

Chapter IV

4 Results

4.1 Results menu

With this menu, you can perform calculations and create rating tables and curves. In the **Results** menu you can select one of the following options:

- Perform calculations
- Calculate all
- Report details
- Rating table
- Rating curves

4.2 Perform calculations

With this option, you can manually perform calculations for the current solution. Normally, this is done automatically as you enter or modify data. However, you may need to repeat the calculations manually when specific preferences have changed, such as the friction formula.

To manually perform calculations for the current solution:

1. Select **Perform calculations** from the **Results** menu.
2. The calculations are performed and the results are displayed in the main form.

4.3 Calculate all

With this option, you can manually perform calculations for all solutions in the project. This is equivalent to performing calculations for each solution separately.

To manually perform calculations for all solutions in the current project:

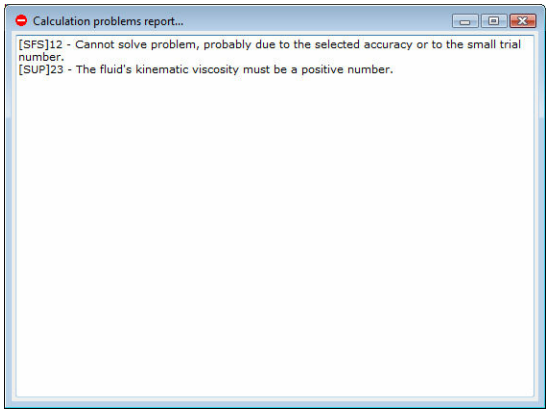
1. Select **Calculate all** from the **Results** menu.
2. The calculations are performed and the results of the current solution are displayed in the main form.
3. If there are errors in one or more solutions, an error report is prepared. If you have selected **Automatically display calculation messages** in general preferences then the report details are automatically displayed.

4.4 Report details

With this option, a report containing information on calculation problems and errors is created. If you have selected **Automatically display calculation messages** in general preferences then this report is automatically displayed when you select Calculate all.

To display the report:

1. Select **Report details** from the **Results** menu.
2. The report appears.
3. Hit ESC to close the report.



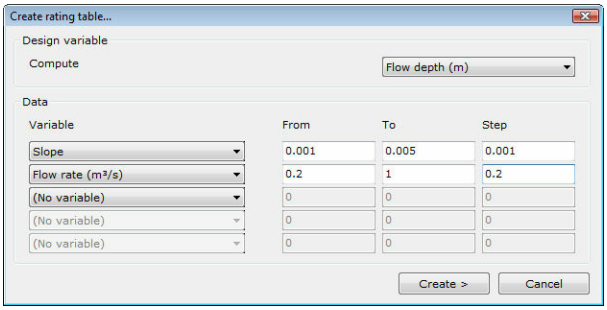
4.5 Rating table

With this option, you can create rating tables with up to 5 variables. For example, variable A may take values from A1 to A2 with step equal to AS and variable B may take values from B1 to B2 with step equal to BS. The solver is invoked for all possible combinations and the results are presented in tabulated form.

The results can be copied to the clipboard, saved in ASCII, Word or Excel format.

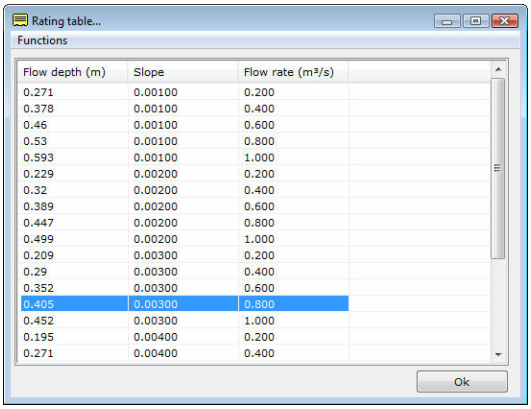
To create a rating table:

1. Select the solution you wish to use for the rating table from the list in the main form.
2. Select **Rating table** from the **Results** menu.
3. Select the values that will be used for the creation of the rating table:
3.1. Select the design variable from the drop-down list in the **Design variable** frame. If the list is disabled then the currently selected design variable is the only option.
3.2. Select up to 5 variables. To select a variable, the previous variable must have been set.
3.3. Select the lower bound, the upper bound and the step of each variable.
4. Select **Create >** to create the rating table. Select **Cancel** to close the dialog box without creating the rating table.



NOTE: This process may take several minutes, depending on the number, range and step of the variables.

5. The rating table appears. The table contains all results from the repetitive process.



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

- **Copy**: the selected cells are copied to the clipboard.
- **Select all**: all cells are selected.
- **Copy format**: select one of tab, comma, space delimited. The use of tab delimited is recommended for compatibility with Microsoft Excel.
- **Print**: a document with the selected cells is created and sent to the **Print Manager**.
- **Export to File**: the contents of the selected cells are send to an ASCII text file.
- **Export to Word**: the contents of the selected cells are send to Microsoft Word.
- **Export to Excel**: the contents of the selected cells are send to Microsoft Word.

Select **Ok** to close the rating table.

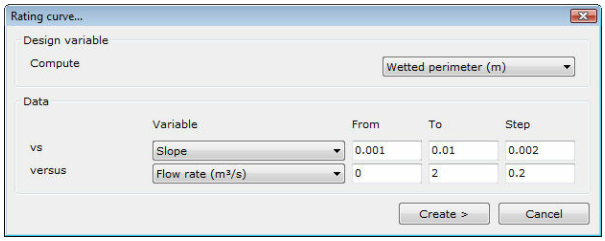
NOTE: The number and type of the available variables depends on the invoked solver.

4.6 Rating curve

With this option, you can create rating curves with up to 2 variables. For example, variable A may take values from A1 to A2 with step equal to AS and variable B may take values from B1 to B2 with step equal to BS. The solver is invoked for all possible combinations and the results are presented in a graph.

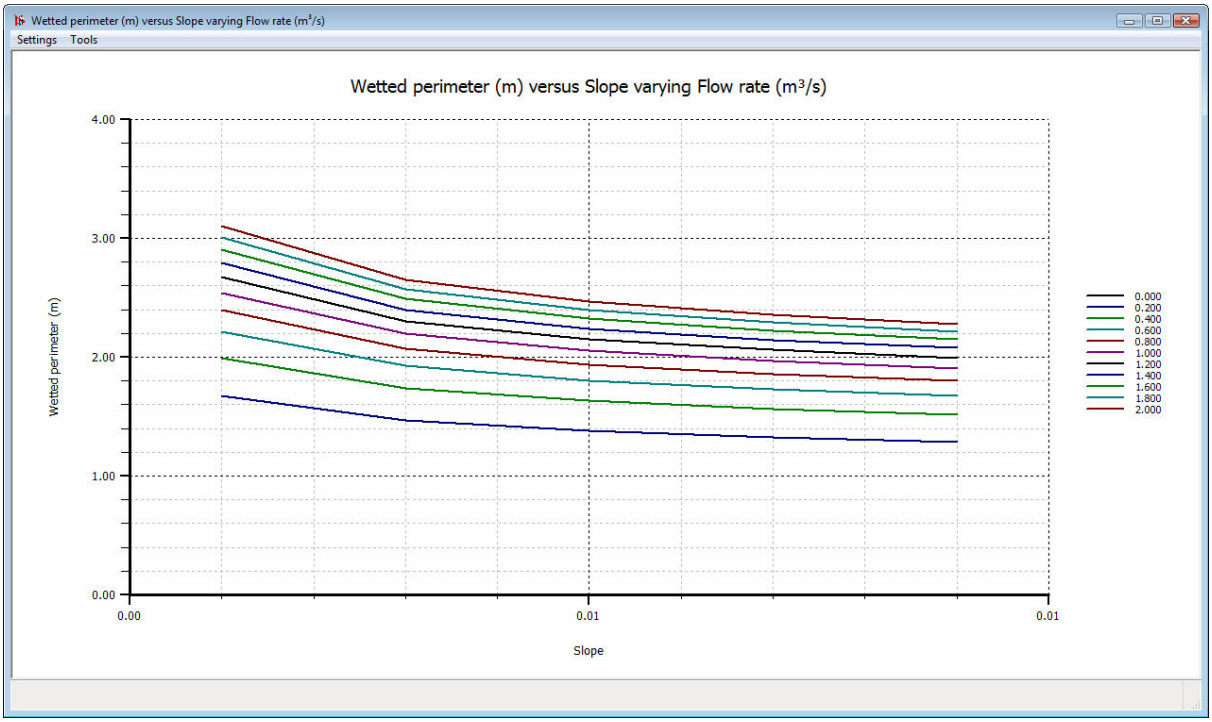
To create a rating curve:

1. Select the solution you wish to use for the rating table from the list in the main form.
2. Select **Rating curve** from the **Results** menu.
3. Select the values that will be used for the creation of the rating table:
3.1. Select the design variable from the drop-down list in the **Design variable** frame. If the list is disabled then the currently selected design variable is the only option.
3.2. Select up to 2 variables. To select a variable, the previous variable must have been set.
3.3. Select the lower bound, the upper bound and the step of each variable.
4. Select **Create >** to create the rating curve. Select **Cancel** to close the dialog box without creating the rating table.



NOTE: This process may take several minutes, depending on the number, range and step of the variables.

5. The rating curve appears. The graph contains all results from the repetitive process.



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.

Click the **X button** at the top-right corner of the form to close the rating curve.

NOTE: The number and type of the available variables depends on the invoked solver.

Chapter

5 Solvers

5.1 Sections with free surface

The variables associated with this solver are the following:

Variable	Explanation
Flow rate	The flow rate in m^3/s.
Slope	The bottom slope in (m/m).
Kinematic viscosity	The kinematic viscosity of the fluid in m^2/s. This is required in case the selected friction formula depends on this variable. You can use the database to select default values. To use the database, click the button with the ellipses (...) that appears when you double-click the cell of the property value.
Friction coefficient	The value and unit system of the friction coefficient depends on the selected friction formula. You can use the database to select default values. To use the database, click the button with the ellipses (...) that appears when you double-click the cell of the property value. In the **Database** section you can find some default values for the friction coefficient for the following formulas: Manning, Bazin, Hazen - Williams and Darcy - Weisbach.
Section	The section. The data is entered using the advanced section editor. To invoke the editor, click the button with the ellipses (...) that appears when you double-click the cell of the property value.
Flow depth	The uniform flow depth in (m).

NOTE: Not all above variables may appear simultaneously. This depends on the type of the problem and the solution settings.

5.2 Sections under pressure

The variables associated with this solver are the following:

Variable	Explanation
Flow rate	The flow rate in m^3/s.
Length	The total length of the pipe in m.
Kinematic viscosity	The kinematic viscosity of the fluid in m^2/s. This is required in case the selected friction formula depends on this variable. You can use the database to select default values. To use the database, click the button with the ellipses (...) that appears when you double-click the cell of the property value.
Specific weight	The specific weight of the fluid in N/m^3 You can use the database to select default values. To use the database, click the button with the ellipses (...) that appears when you double-click the cell of the property value.
Friction	The value and unit system of the friction coefficient depends on the

coefficient	selected friction formula. You can use the database to select default values. To use the database, click the button with the ellipses (...) that appears when you double-click the cell of the property value. In the **Database** section you can find some default values for the friction coefficient for the following formulas: Manning, Bazin, Hazen - Williams and Darcy - Weisbach.
Elevation 1	The elevation of the start point of the pipe (point 1) in m.
Elevation 1	The elevation of the end point of the pipe (point 2) in m.
Pressure 1	The pressure at the start point of the pipe (point 1) in N/m^2.
Pressure 2	The pressure at the end point of the pipe (point 2) in N/m^2.
Section	The section. The data is entered using the advanced section editor. To invoke the editor, click the button with the ellipses (...) that appears when you double-click the cell of the property value.

NOTE: Not all above variables may appear simultaneously. This depends on the type of the problem and the solution settings.

5.3 Weirs

The variables associated with this solver are the following:

Variable	Explanation
Headwater elevation	The headwater elevation in m.
Crest elevation	The crest elevation in m.
Tailwater elevation	The downstream tailwater elevation in m. This elevation can not be greater than the upstream elevation. In this case, the weir will operate as an orifice.
Discharge coefficient	A dimensionless coefficient that quantifies the effect of the boundaries friction, the headwater elevation, the opening shape etc on the flow rate. When no experimental data are available, its value can be taken from reference tables.
Crest length	The crest length normal to the flow direction in m.
Number of contractions	The number of contractions that suppress the jet stream. Enter 0 to describe a non-contracted jet stream. Enter 1 or 2 to describe a one or two-sided contraction. Contractions are possible when the upstream channel is greater than the rectangular weir opening.
Flow rate	The flow rate in m^3/s.
Angle	The angle of the base corner of a triangular weir in deg.
Crest breadth	The crest breadth of a broad weir in m.
Crest surface type	It is used by the American circular FHWA HDS-5 to calculate the submergence factor and adjusted discharge coefficient.

NOTE: Not all above variables may appear simultaneously. This depends on the type of the problem and the solution settings.

5.4 Orifices

The variables associated with this solver are the following:

Variable	Explanation
Headwater elevation	The headwater elevation in m.
Centroid elevation	The elevation of the centroid of the orifice in m.
Tailwater elevation	The downstream tailwater elevation in m. The upstream elevation must be greater than the invert elevation of the orifice. If the downstream elevation is greater than that of the invert elevation then the orifice behaves as submerged orifice.
Discharge coefficient	A dimensionless coefficient that quantifies the effect of the boundaries friction, the headwater elevation, the opening shape etc on the flow rate. When no experimental data are available, its value can be taken from reference tables.
Opening area	The area of the opening in m^2 . This is used for the calculation of the flow velocity when the orifice is not circular or rectangular.
Opening width	The width of the opening in m. This applies to rectangular orifices.
Opening height	The height of the opening in m. This applies to rectangular orifices.
Opening diameter	The diameter of the opening in m. This applies to circular orifices.
Flow rate	The flow rate in m^3/s.

NOTE: Not all above variables may appear simultaneously. This depends on the type of the problem and the solution settings.

5.5 Grates

The variables associated with this solver are the following:

Variable	Explanation
Flow rate	The flow rate in m^3/s.
Slope	The bottom slope in (m/m).
Gutter width	The gutter width in m. It may be 0 in triangular sections.
Gutter cross slope (H:V)	The dimensionless slope of the gutter. Normally, this is greater than the corresponding road cross slope. If the gutter is not configured properly, this value is set equal to the road cross slope (for example 2% in straight roads, 6~8% in curves)
Road cross slope	The dimensionless road cross slope.

(H:V)	
Friction coefficient	The friction coefficient according to Manning. Although it is possible to use other friction formulas, this is not recommended as the experimental equations used during the calculations are based on the Manning friction formula. In the **Database** section you can find some default values for the friction coefficient for the following formulas: Manning, Bazin, Hazen - Williams and Darcy - Weisbach.
Efficiency	The dimensionless efficiency ratio of the grate. This is a number between 0 and 1.
Length	The length of the grate in m.
Grate width	The width of the grate in m.
Grate type	One of seven characteristic grate types. If the grate is not in the list, select the type that is closest with respect to the arrangement of the bars. This variable is used for the calculation of the splash-over velocity.
Clogging	The dimensionless ratio of clogging. This is a number between 0 and 1 and it is used for the calculation of existing grates that are clogged by litter. This can be set to a non-zero value even for new grates when the engineer expects that the maintenance of the grates will be insufficient or non-existent.
Spread	The desired spread in m. This applies when the grate is located on sag.
Depression width	The depression width in m. A local depression may be implemented for the improvement of the hydraulic behavior of the grate. This is not common, therefore this value is usually 0.
Depression height	The depression height in m. A local depression may be implemented for the improvement of the hydraulic behavior of the grate. This is not common, therefore this value is usually 0.

NOTE: Not all above variables may appear simultaneously. This depends on the type of the problem and the solution settings.

5.6 Curbs

The variables associated with this solver are the following:

Variable	Explanation
Flow rate	The flow rate in m^3/s.
Slope	The bottom slope in (m/m).
Gutter width	The gutter width in m. It may be 0 in triangular sections.
Gutter cross slope (H:V)	The dimensionless slope of the gutter. Normally, this is greater than the corresponding road cross slope. If the gutter is not configured properly, this value is set equal to the road cross slope (for example 2% in straight roads, 6~8% in curves)
Road cross slope	The dimensionless road cross slope.

(H:V)	
Friction coefficient	The friction coefficient according to Manning. Although it is possible to use other friction formulas, this is not recommended as the experimental equations used during the calculations are based on the Manning friction formula. In the **Database** section you can find some default values for the friction coefficient for the following formulas: Manning, Bazin, Hazen - Williams and Darcy - Weisbach.
Efficiency	The dimensionless efficiency ratio of the curb. This is a number between 0 and 1.
Length	The length of the curb in m.
Curb throat type	The entrance of the curb may be horizontal, vertical or inclined.
Throat incline angle	If the entrance is inclined, then this angle (in degrees) is used. 0 degrees corresponds to horizontal entrance and 90 degrees corresponds to vertical entrance.
Spread	The desired spread in m. This applies when the curb is located on sag.
Depression width	The depression width in m. A local depression may be implemented for the improvement of the hydraulic behavior of the curb. This is not common, therefore this value is usually 0.
Depression height	The depression height in m. A local depression may be implemented for the improvement of the hydraulic behavior of the curb. This is not common, therefore this value is usually 0.

NOTE: Not all above variables may appear simultaneously. This depends on the type of the problem and the solution settings.

5.7 Combination inlets

The variables associated with this solver are the same with those of grates.

5.8 Slots

The variables associated with this solver are the following:

Variable	Explanation
Flow rate	The flow rate in m^3/s.
Slope	The bottom slope in (m/m).
Gutter width	The gutter width in m. It may be 0 in triangular sections.
Gutter cross slope (H:V)	The dimensionless slope of the gutter. Normally, this is greater than the corresponding road cross slope. If the gutter is not configured properly, this value is set equal to the road cross slope (for example 2% in straight roads, 6~8% in curves)
Road cross slope (H:V)	The dimensionless road cross slope.
Friction coefficient	The friction coefficient according to Manning. Although it is possible to use other friction formulas, this is not recommended as the experimental equations used during the

	calculations are based on the Manning friction formula. In the **Database** section you can find some default values for the friction coefficient for the following formulas: Manning, Bazin, Hazen - Williams and Darcy - Weisbach.
Efficiency	The dimensionless efficiency ratio of the slot. This is a number between 0 and 1.
Length	The length of the slot in m.
Spread	The desired spread in m. This applies when the slot is located on sag.
Depression width	The depression width in m. A local depression may be implemented for the improvement of the hydraulic behavior of the slot. This is not common, therefore this value is usually 0.
Depression height	The depression height in m. A local depression may be implemented for the improvement of the hydraulic behavior of the slot. This is not common, therefore this value is usually 0.

NOTE: Not all above variables may appear simultaneously. This depends on the type of the problem and the solution settings.

5.9 Ditches

The variables associated with this solver are the following:

Variable	Explanation
Friction coefficient	The friction coefficient according to Manning. Although it is possible to use other friction formulas, this is not recommended as the experimental equations used during the calculations are based on the Manning friction formula. In the **Database** section you can find some default values for the friction coefficient for the following formulas: Manning, Bazin, Hazen - Williams and Darcy - Weisbach
Slope	The bottom slope in (m/m).
Left side slope (H:V)	The dimensionless slope of the left side.
Right side slope (H:V)	The dimensionless slope of the right side.
Bottom width	The width of the bottom of the ditch. The ditch is considered to be trapezoidal.
Flow rate	The flow rate in m^3/s.
Efficiency	The dimensionless efficiency ratio of the grate. This is a number between 0 and 1.
Grate width	The width of the grate in m.
Grate length	The length of the grate in m.
Grate type	One of seven characteristic grate types. If the grate is not in the list, select the type that is closest with respect to the arrangement of the bars. This variable is used for the calculation of the splash-over velocity.

Clogging	The dimensionless ratio of clogging. This is a number between 0 and 1 and it is used for the calculation of existing grates that are clogged by litter. This can be set to a non-zero value even for new grates when the engineer expects that the maintenance of the grates will be insufficient or non-existent.
Depression width	The depression width in m. A local depression may be implemented for the improvement of the hydraulic behavior of the grate. This is not common, therefore this value is usually 0.
Depression height	The depression height in m. A local depression may be implemented for the improvement of the hydraulic behavior of the grate. This is not common, therefore this value is usually 0.
Spread	The desired spread in m. This applies when the slot is located on sag.

NOTE: Not all above variables may appear simultaneously. This depends on the type of the problem and the solution settings.

5.10 Pump

5.10.1 Pump

The variables associated with this solver are the following:

Variable	Description
Slope	The flow rate through the oppression and suction pipe in m^3/s. This value can be entered only when the selected solver does not refer to its calculation.
Kinematic viscosity	The kinematic viscosity of the fluid in m^2/s. This is required in case the selected friction formula depends on this variable. You can use the database to select default values. To use the database, click the button with the ellipses (...) that appears when you double-click the cell of the property value.
Specific weight	The specific weight of the fluid in N/m^3 You can use the database to select default values. To use the database, click the button with the ellipses (...) that appears when you double-click the cell of the property value.
Pump performance factor	The dimensionless performance factor of the pump. This is a value between 0 and 1.
Length of suction pipe	The total length of the suction pipe in m.
Diameter of suction pipe	The diameter of the suction pipe in m. Note that only circular non-varying pipes are supported.
Length of oppression pipe	The total length of the oppression pipe in m.
Diameter of oppression pipe	The diameter of the oppression pipe in m. Note that only circular non-varying pipes are supported.
Friction coefficient	The value and unit system of the friction coefficient depends on the selected friction formula. You can use the database to select

	default values. To use the database, click the button with the ellipses (...) that appears when you double-click the cell of the property value. In the **Database** section you can find some default values for the friction coefficient for the following formulas: Manning, Bazin , Hazen - Williams and Darcy - Weisbach.
Elevation difference	The desired elevation difference in m. This value can be entered only when the selected solver does not refer to its calculation.
Pump power	The pump power in CV. This value is provided by the manufacturer. This value can be entered only when the selected solver does not refer to its calculation.
Configure system	To configure the pump system, click the button with the ellipses (...) that appears when you double-click the cell of the property value.

NOTE: Not all above variables may appear simultaneously. This depends on the type of the problem and the solution settings.

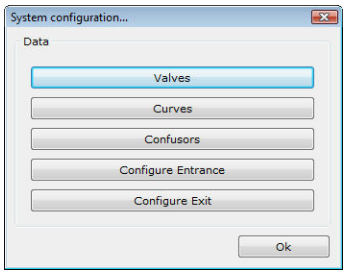
5.10.2 System configuration

With system configuration, you can enter and modify the following properties of the pump system:

- Valves
- Curves
- Confusors
- Configure entrance
- Configure exit

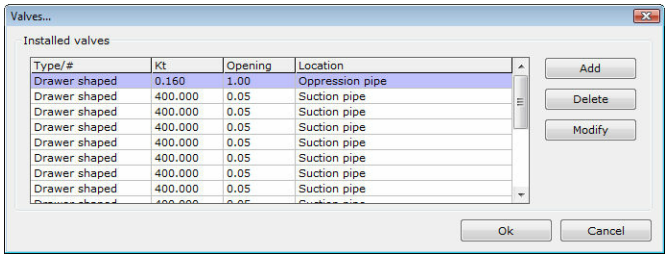
To modify the system configuration:

1. Select the appropriate **Pump** solution from the list in the main form.
2. Double-click next to **Configure system** property cell.
3. Click the button with the ellipses (...) that appears in the property value cell.
4. The pump system configuration dialog box appears.
5. Modify the properties of the system by clicking on the corresponding buttons of the dialog box.
6. Select **Ok** to close the dialog box and save the changes.



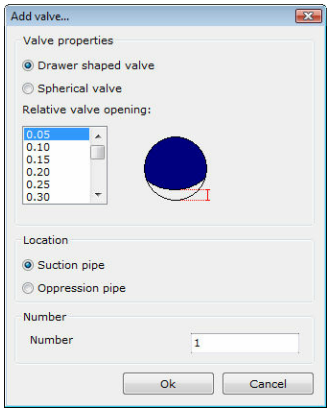
5.10.3 Valves

With this option, you can add, modify and delete the valves of the oppression and suction pipe. Valves are devices that are used in closed sections under pressure in order to modify the flow rate by introducing local energy losses. The most common types of valves are the spherical and the drawer-shaped.



To add a new valve:

1. Click **Add** button.
2. Select one of **Drawer-shaped** or **Spherical** valve.
3. Select the appropriate relative valve opening (if the valve is drawer-shaped) or the opening angle (if the valve is spherical).
4. Select the location of the valve. The valve may be in the oppression pipe or the suction pipe.
5. Select the appropriate number of valves. This can be greater than 1 if there are more that one valves with the same characteristics.
6. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

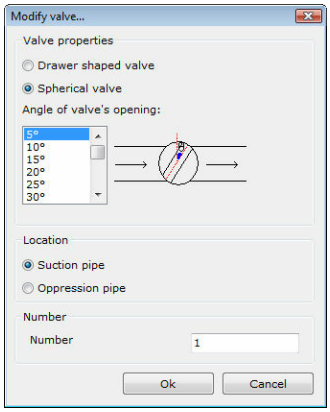


To modify an existing valve:

1. Select the valve from the list.
2. Click the **Modify** button.
3. Make the appropriate changes in the valve configuration dialog box. This dialog box is the same as in the case of a new valve.
4. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

NOTE: Alternatively to steps 1 & 2, you can double-click the name of the valve on the

list.

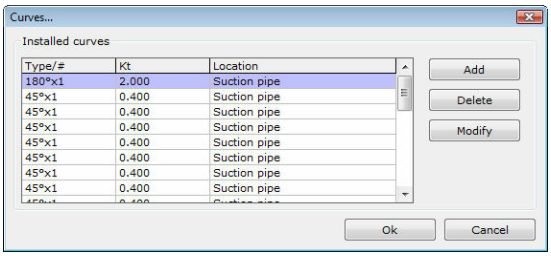


To delete an existing valve:

1. Select the valve from the list.
2. Click the **Delete** button to delete the valve from the list. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab. If you select No then the deletion of the valve is canceled.

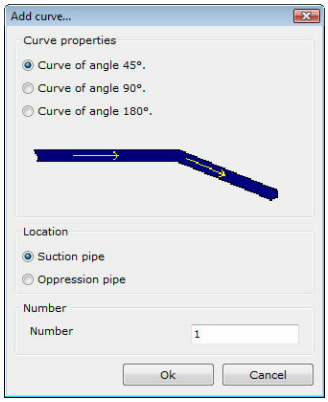
5.10.4 Curves

With this option, you can add, modify and delete curves of the oppression and suction pipe. There are curves in almost all pipes under pressure; as a consequence, local energy losses are introduced in the system due to the change in the direction of flow.



To add a new valve:

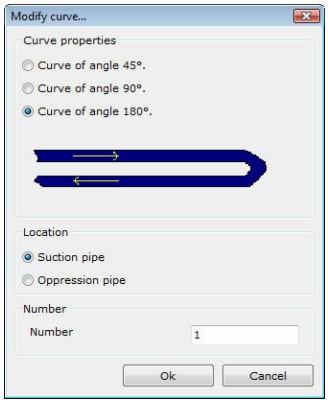
1. Click **Add** button.
2. Select one of **45 degrees, 90 degrees, 180 degrees**.
3. Select the location of the curve. The curve may be in the oppression pipe or the suction pipe.
4. Select the appropriate number of curves. This can be greater than 1 if there are more that one curves with the same characteristics.
5. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



To modify an existing curve:

1. Select the curve from the list.
2. Click the **Modify** button.
3. Make the appropriate changes in the curve configuration dialog box. This dialog box is the same as in the case of a new curve.
4. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

NOTE: Alternatively to steps 1 & 2, you can double-click the name of the curve on the list.

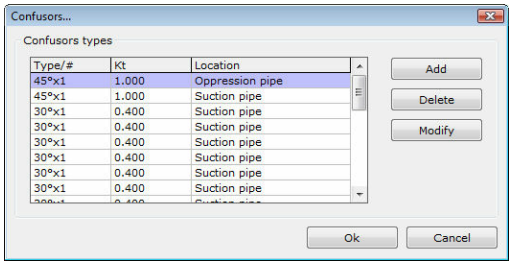


To delete an existing curve:

1. Select the curve from the list.
2. Click the **Delete** button to delete the curve from the list. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab. If you select No then the deletion of the curve is canceled.

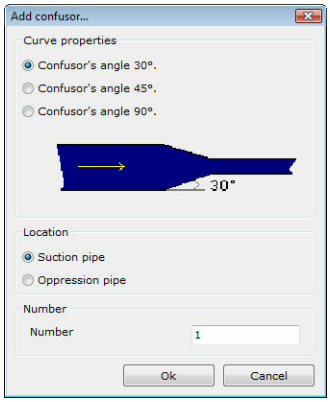
5.10.5 Confusors

With this option, you can add, modify and delete confusors of the oppression and suction pipe. The presence of confusors introduces local energy losses due to flow separation.



To add a new confusor:

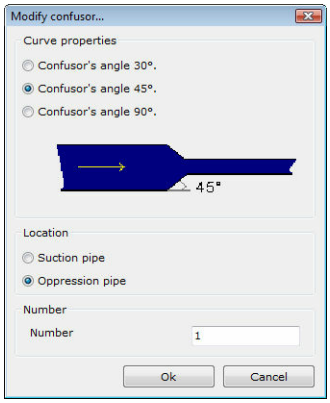
1. Click **Add** button.
2. Select one of **30 degrees, 45 degrees, 90 degrees**.
3. Select the location of the confusor. The confusor may be in the oppression pipe or the suction pipe.
4. Select the appropriate number of confusors. This can be greater than 1 if there are more that one confusor with the same characteristics.
5. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



To modify an existing confusor:

1. Select the confusor from the list.
2. Click the **Modify** button.
3. Make the appropriate changes in the confusor configuration dialog box. This dialog box is the same as in the case of a new confusor.
4. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

NOTE: Alternatively to steps 1 & 2, you can double-click the name of the confusor on the list.



To delete an existing confusor:

1. Select the confusor from the list.
2. Click the **Delete** button to delete the confusor from the list. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab. If you select No then the deletion of the confusor is canceled.

5.10.6 Configure entrance

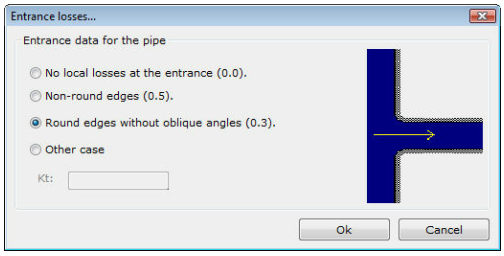
Entrance losses refer to the suction pipe. These are local losses that are introduced due to the abrupt change in the uniform flow at the entrance of the pump. Entrance losses are usually quantified by the formula ht=Kt*V^2/2*g. Kt is the local losses coefficient and it is depended on the shape of the edges of the entrance.

To configure the entrance:

1. Select **Configure entrance** from the pump system configuration.
2. Select one of the following:

- No local losses at the entrance (Kt=0).
- Non-round edges (Kt=0.5).
- Round edges without oblique angles (Kt=0.3).
- Other case: the custom coefficient text box is enabled and you can type the desired value.

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



5.10.7 Configure exit

Exit losses refer to the oppression pipe. These are local losses that are introduced due to the abrupt change in the uniform flow at the exit of the pump. Exit losses are

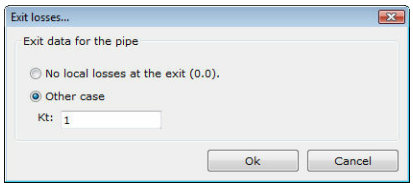
depended on the local losses coefficient Kt.

To configure the exit:

1. Select **Configure exit** from the pump system configuration.
2. Select one of the following:

- No local losses at the exit (Kt=0).
- Other case: the custom coefficient text box is enabled and you can type the desired value

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



5.11 Reaction turbines

The variables associated with this solver are the following:

Variable	Explanation
Average drop height	The average drop height in m. This can be calculated as the mean value of the maximum and minimum drop height.
Design flow	The flow rate through the turbine in m^3/s.
Number of turbines	The number of turbines installed. Enter 1 for a single turbine.
Electrical current	The electrical current frequency in Hz. Enter 60 Hz for USA and 50 Hz for Greece. For other countries, you can use the database to select the correct values. To use the database, click the button with the ellipses (...) that appears when you double-click the cell of the property value.
Operation elevation	The elevation of the installation in m.
Max operation temperature	The maximum temperature of the water going through the turbine in degrees Celsius.
Propeller opening coefficient	The dimensionless coefficient describing the opening of the turbine propeller.
# poles multiplicative	Select one of 2, 4 or 8 as the maximum number for which the number of poles is a multiplicative. For example, select 8 if the number of poles is 16.
Turbine type	Select one of Francis or Propeller - type turbine.

5.12 Impulse turbines

The variables associated with this solver are the following:

Variable	Description
Average drop height	The average drop height in m. This can be calculated as the mean value of the maximum and minimum drop height.
Turbine power	The turbine power in MW.
Best turbine efficiency level	The dimensionless best turbine efficiency level. This is a value between 0 and 1.
Electrical current	The electrical current frequency in Hz. Enter 60 Hz for USA and 50 Hz for Greece. For other countries, you can use the database to select the correct values. To use the database, click the button with the ellipses (...) that appears when you double-click the cell of the property value.
Best pump efficiency level	The dimensionless best pump efficiency level. This is a value between 0 and 1.
Pump speed times area	The speed of the pump in m/s times the area in m^2.
Max operation temperature	The maximum temperature of the water going through the turbine in degrees Celsius.
Energy losses	The total energy losses, i.e. the sum of the linear and local losses in m.
# poles multiplicative	Select one of 2, 4 or 8 as the maximum number for which the number of poles is a multiplicative. For example, select 8 if the number of poles is 16.
Tailrace elevation	The elevation of the bottom of the tailrace in m.

5.13 Hydropower pump

The variables associated with this solver are the following:

Variable	Description
Static elevation difference	The static elevation difference in m.
Design flow	The design flow rate through the pump in m^3/s.
Water level fluctuation upstream	The water level fluctuation upstream in m.
Number of pumps	The number of pumps. Enter 1 for a single pump.
# pumps for optimal efficiency	The number of pumps that should be used for optimal efficiency.
Electrical current	The electrical current frequency in Hz. Enter 60 Hz for USA and 50 Hz for Greece. For other countries, you can use the database to select the correct values. To use the database, click the button with the ellipses (...) that appears when you double-click the cell of the property value.
Losses in suction pipe and grates	The sum of the linear and local losses in the suction pipe and grates in m.
Losses in oppression pipe	The local losses in the oppression pipe at the exit of the pump in m.
Operation losses in	The linear losses in the oppression pipe at the exit of the pump

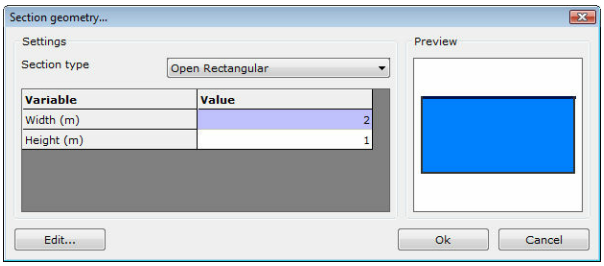
oppression pipe	in m.
Operation elevation	The elevation of the installation in m.
Max operation temperature	The maximum temperature of the water going through the pump in degrees Celsius.
Suction height	The suction height before the pump in m.
Special suction speed	The special suction speed. This values is between 153 and 155.
Motor's efficiency coefficient	The dimensionless motor efficiency coefficient. This is a value between 0 and 1.
Operation mode	Select one the following four operation modes: **A**: Clay with mean concentration less than 100 mg/L and organic material. **B**: Clay and silt with mean concentration less than 500 mg/L and fine sand for small return periods. **C**: Clay, silt and fine sand with mean concentration less than 2000 mg/L and coarse sand during floods. **D**: Clay, silt, sand and sometimes gravel with mean concentration less than 1000 mg/L.
# poles multiplicative	Select one of 2, 4 or 8 as the maximum number for which the number of poles is a multiplicative. For example, select 8 if the number of poles is 16.

Chapter VI

6 Sections

6.1 Section input

When you double-click the cell of the property value of **Section**, a command button with ellipses (...) appears. If you click it, the following dialog box appears:



In order to enter section data, follow these steps:

1. Select the **type** of the section from the drop-down list.
2. Fill the geometric data by typing onto the table. The number and description of fields depends on the type of the section.
3. You can preview the current configuration in the picture of the **Preview** frame.
4. Select **Ok** to use the currently selected configuration and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

NOTE: If you click the **Edit** button then the section editor will appear. This editor must be used in the case of prismatic (irregular) sections. However, its use is not compulsory when dealing with standard section types, as these can be fully defined by the above dialog box.

The available section types are the following:

Section type	Geometric data required
Circular	Diameter
Open rectangular	Width, Height
Rectangular	Width, Height
Open trapezoid	Width, Height, Left slope, Right slope
Trapezoid	Width, Height, Left slope, Right slope
Open reverse trapezoid	Width, Height, Left slope, Right slope
Reverse trapezoid	Width, Height, Left slope, Right slope
Open isosceles trapezoid	Width, Height, Side slope
Isosceles trapezoid	Width, Height, Side slope
Open reverse isosceles trapezoid	Width, Height, Side slope
Reverse isosceles trapezoid	Width, Height, Side slope
Twin rectangular	Width, Height (total)
Twin open rectangular	Width, Height (total)
Twin circular	Diameter (of each circular section)

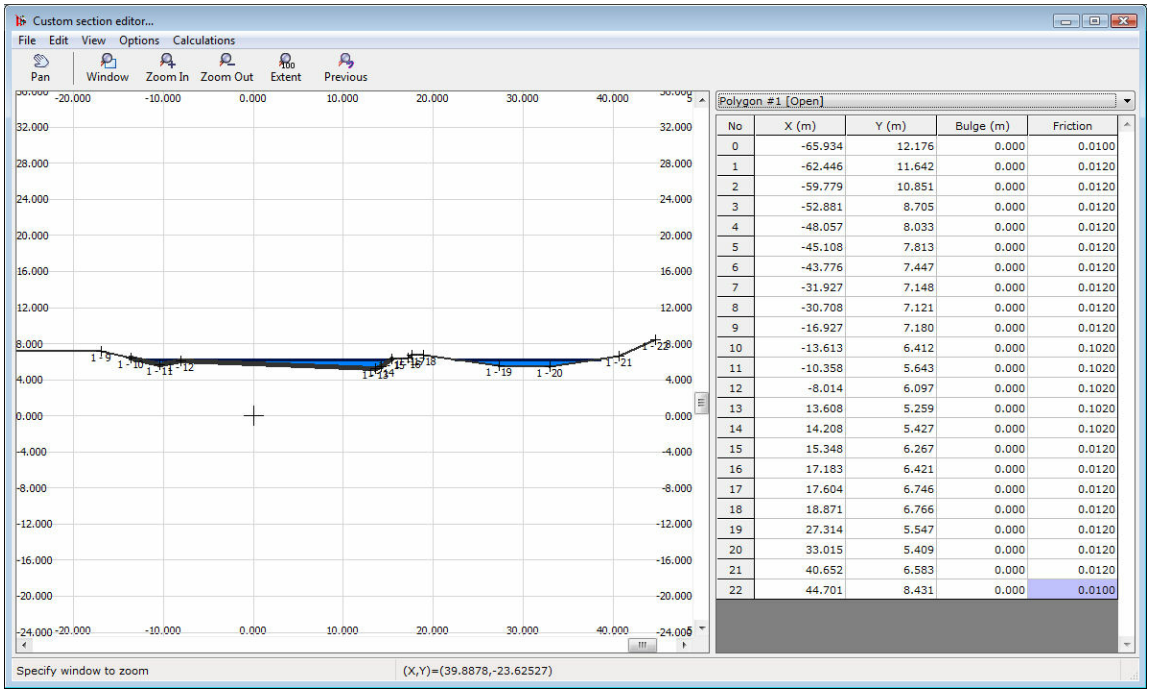
Oval	Width
Wide oval	Width
Basket handle	Width
Simple mouth-shaped	Width
Normal mouth-shaped	Width
Simple horse-shoe	Width
Normal horse-shoe	Width
Donut	External diameter, Internal diameter
Arch	Width, Height, Arc segments
Ellipse	Width, Height, Arc segments
Parabolic	Width, Height, Arc segments
Irregular	Use section editor

NOTE: The above section types are based on advanced script techniques. Therefore, this list may be changed (for example, some more types may be added) with updates that are irrelevant with the executable file of the program.

6.2 Section editor

The list of the standard types of sections is extensive; however, when the section is irregular it cannot be described by these types. In this case, you can use the section editor. With this editor, you can design a new section from scratch or modify a standard section. Note that after editing, the section will always be referred to as irregular.

The section is described by **curvilinear polygons** i.e. open or closed polylines with straight or curved edges. The curved edges are circular arcs; in this way, virtually any section can be described accurately with a minimum number of nodes.



The user interface of the section editor is shown in the above picture. It consists of four parts: the **menu,** the **toolbar**, the **drawing** and the **table** of coordinates.

A section may:

- consist of one or more curvilinear polygons
- consist of many open or closed curvilinear polygons or any combination of them
- include ground information
- include excavation information

There are five main menus:

- **File**: file operations, printing, importing from other formats
- **Edit**: data input and editing
- **View**: configuration of the drawing and the table of coordinates
- **Options**: other options (colors, line styles etc)
- **Calculations**: calculation tools for checking the section.

In order to input section data:

1. Add one or more polygons.
2. For each polygon add three or more nodes.
3. For each node, enter X coordinate, Y coordinate and friction coefficient. Optionally, you can enter the bulge, if the edge is curved. These values are explained below.
4. Check that the section is filled (with fluid) properly. If this doesn't happen, disable or enable some nodes so that the flow is correct.
5. You can optionally enter ground data. If stabilization works are necessary then this line represents the ground before dredging. In the case of natural sections, the section is the same with the ground.
6. You can optionally enter excavation data. The excavation line is located below the section and it represents the outline of the section and the level where the section will be built. For uncovered sections, the excavation line is the same with the section. For covered sections, the excavation line is displaced downwards by the thickness of the cover e.g. concrete.

To input data from the table:

Polygon list: select the active polygon by using the drop-down list. If you select to view the nodes, then the nodes of the active polygon are shown in the drawing.
N/O: the number of the node. This column is not editable.
X (m): the X coordinate of the node in meters.
Y (m): the Y coordinate of the node in meters.
Bulge (m): this is used only in cases of curved edges. It represents the distance in meters of the middle point of the segment connecting two nodes with the middle point of the arc. This value is 0 for straight lines. The bulge refers to segments, not nodes; therefore, the bulge of the first node refers to the segment connecting the first with the second node. The bulge is positive if the arc connecting two nodes is on the right side of the corresponding straight line and vice versa. You cannot enter a value for the last node, as this has no meaning.
Friction (-): the friction coefficient. This value refers to segments not nodes; therefore, the friction of the first node refers to the friction of the segment connecting the first with the second node. You cannot enter a value for the last node, as this has

no meaning. The values of friction coefficients depend on the selected friction formula.

NOTE: The bulge and friction coefficient columns are not available when entering ground and excavation data.

6.3 File

6.3.1 File menu

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

- New section
- Open section
- Save section
- Save section as
- Import
  - Import from GRD
  - Import from PCS
  - Import from DXF
  - Import from ArcView Shapefile
- Export
  - Export to GRD
  - Export to PCS
  - Export to DXF
  - Export to ArcView Shapefile
  - Export to Bitmap
- Print sketch
- Print section data
- Print section data to
  - Microsoft Excel
  - Microsoft Word
  - Text file
- Close

6.3.2 New section

With this option, you can start a new section. All data of the current section are lost.

To start a new section:

1. Select **New Section** from the **File** menu.
2. The current section data are erased and a new section is created.

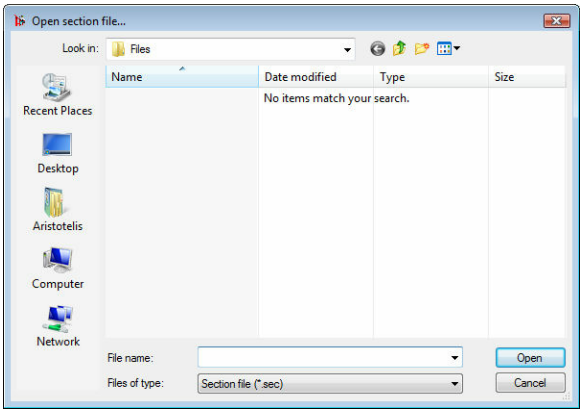
6.3.3 Open section

With this option, an existing section is loaded. The section file may be located locally, in a network or in an external media device such as a CD-Rom. When a section is loaded, all data of the previous section are lost.

To open an existing project:

1. Select **Open Section** 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 "Section file" with the extension .sec.
4. Select the file by clicking on it.
5. Select **Open** to open the selected file.



Supported file formats

- **SEC** (Section file): Section files created by versions 2010, 2009, 2008, 2007 and 5 of the program.
- ***.*** (All files): Files with any extension.

NOTE: If a message "Error while loading file" is displayed then either you are trying to load a file that doesn't contain section data or the file is used (and locked) by another process in your computer.

6.3.4 Save section

With this option, you can save all data of a section 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 attempt to save the section. 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 section as... from the **File** menu.

To save the current project:

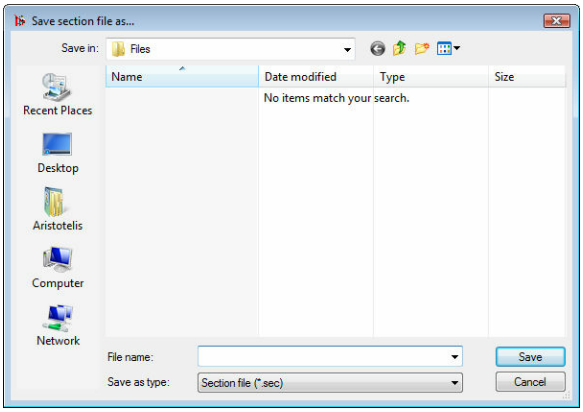
1. Select **Save Section** from the **File** menu.
2. If the filename and path are already set, the section is saved to this file without any messages. If the filename and path are not set, a dialog box will appear that allows the selection of the filename and path.

6.3.5 Save section as

With this option, the current section is saved just as in the case of Save section, 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 Section 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 section with the selected filename and path.



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 section are saved.

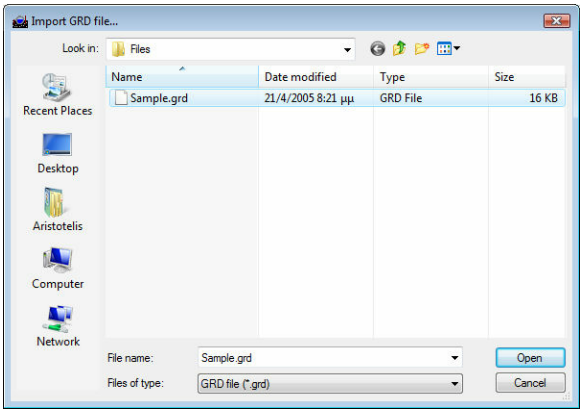
6.3.6 Import

6.3.6.1 Import from GRD

GRD files are created by many popular programs such as VERM, Anadelta, Odos as well as Hydraulic programs by TechnoLogismiki. They contain geometric information on a prismatic cross section with straight edges as well as the title of the section, station data etc.

To import a GRD file:

1. Select **Import** from the **File** menu.
2. Select **from GRD file** from the **Import** 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 "GRD file" with the extension .grd.
5. Select the file by clicking on it.
6. Select **Open** to open and analyze the file. Depending on the contents of the file, there are two cases.

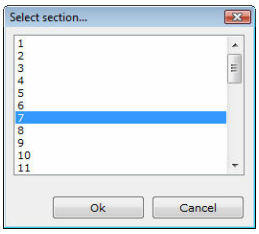


Case 1

The GRD file contains only one section. In this case, the current section data are erased and substituted by the imported section.

Case 2

The GRD file contains more than one sections. In this case, a dialog box appears that allows the selection of a single section. Select the section that you wish to import.



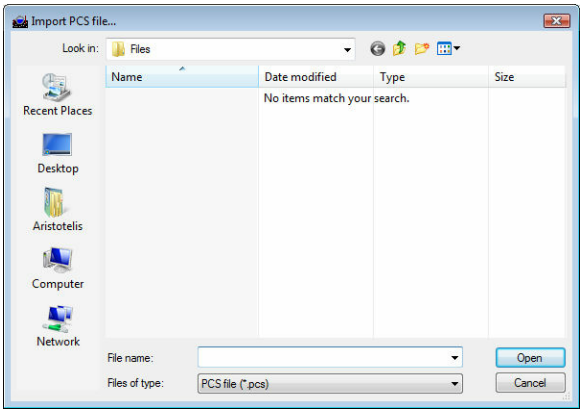
Select **Ok** to proceed. The current section data are erased and substituted by the selected section. Select **Cancel** to cancel the operation.

6.3.6.2 Import from PCS

PCS files can be created by Hydraulic programs (by TechnoLogismiki) version 2.0 or later. It is the preferred way to exchange section data between programs and users. Each file contains a single section, with information on the prismatic section and the friction coefficient of each edge.

To import a section from a PCS file:

1. Select **Import** from the **File** menu.
2. Select **from PCS file** from the **Import** 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 "PCS file" with the extension .pcs.
5. Select the file by clicking on it.
6. Select **Open** to open and analyze the file. The current section data are erased and substituted by the imported. Select **Cancel** to cancel the operation.



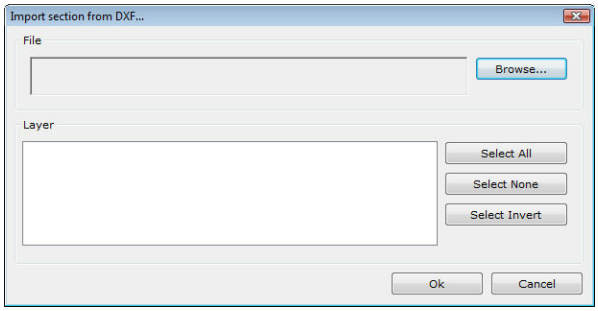
6.3.6.3 Import from DXF

DXF files are recognized by virtually all CAD programs (AutoCAD, IntelliCAD, Microstation etc) as well as TechnoLogismiki's products.

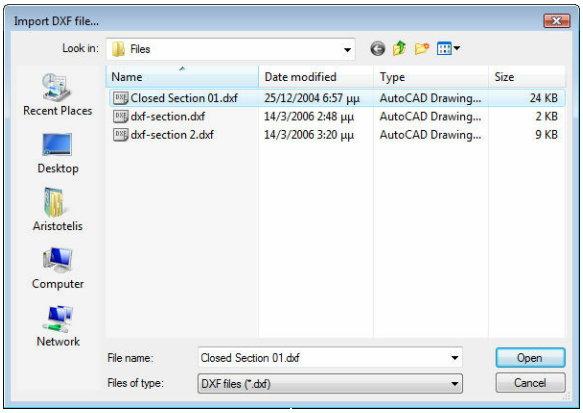
DXF files contain information in layers. You can select one or more layers that contain the section data.

To import a section from a DXF file:

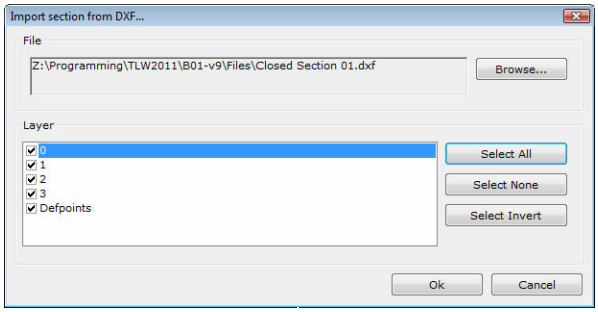
1. Select **Import** from the **File** menu.
2. Select **Import from DXF** from the **Import menu**. The following dialog box appears:



3. Click **Browse**.
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 files" 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 **Layer** frame is loaded with the layers contained in the DXF file.
8. Select one or more layers that contain section data. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all layers, deselect all layers and invert the current selection.



9. Select **Ok** to proceed. Select **Cancel** to close the dialog box with no changes.

NOTE: The current DXF driver can import the following entities:
- Polylines
- LWPolylines
- Circles

If there are no recognizable entities, an error message will be displayed.

6.3.6.4 Import from ArcView Shapefile

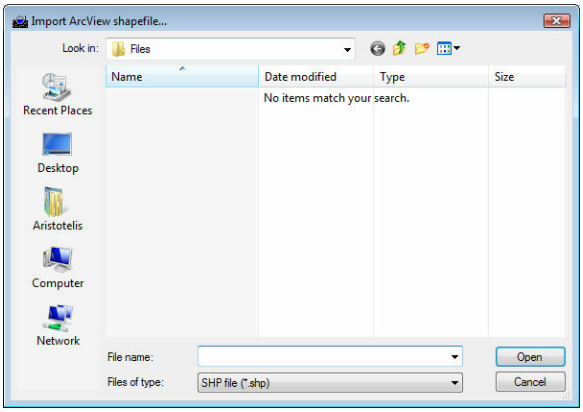
Shapefiles can be created by several programs such as ArcView GIS, MapInfo, GPS Trackmaker etc. In reality, this "file" consists of three files with the extensions shp, shx and dbf. The GIS driver recognizes the following shapefile types:

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

Shapefiles containing Polygons (simple, M and Z) and Multipatch are not recognized.

To import data from a shapefile:

1. Select **Import** from the **File** menu.
2. Select **from ArcView Shapefile** from the **Import** 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 "SHP file" with the extension .shp.
5. Select the file by clicking on it.
6. Select **Open** to open and analyze the file. The section contained in the shapefile is imported into the editor. Select **Cancel** to cancel the operation.



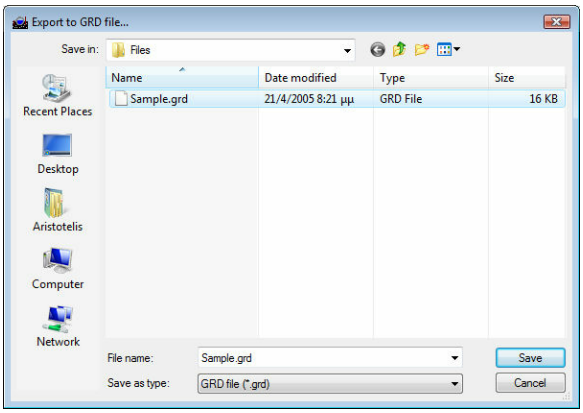
6.3.7 Export

6.3.7.1 Export to GRD

With this option, a GRD file containing the current section data 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 sections to a GRD file:

1. Select **Export** from the **File** menu.
2. Select **Export to GRD file** from the **Export** 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. Select **Cancel** to cancel the operation.



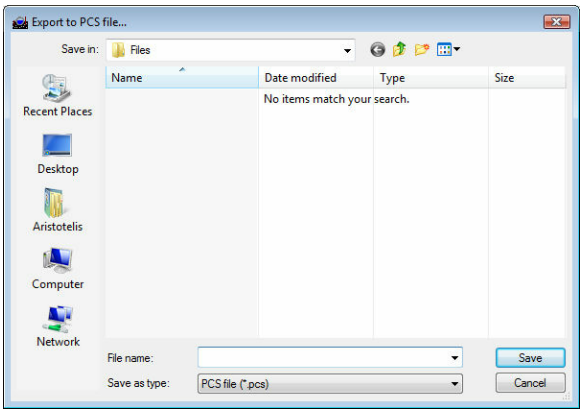
NOTE: Information on the friction coefficients and curved edges (if any) will not be exported as they are not supported by the file format.

6.3.7.2 Export to PCS

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

To export a section to a PCS file:

1. Select **Export** from the **File** menu.
2. Select **Export To PCS** from the **Export** 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. Select **Cancel** to cancel the operation.



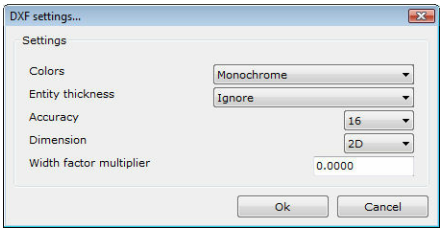
NOTE: Information on curved edges (if any) will not be exported as they are not supported by the file format.

6.3.7.3 Export to DXF

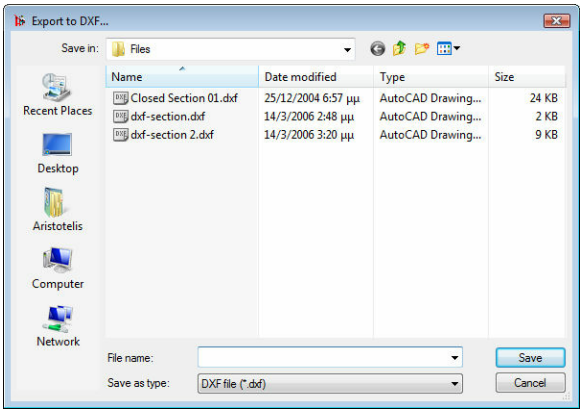
DXF files are recognized by virtually all CAD programs (AutoCAD, IntelliCAD, Microstation etc) as well as TechnoLogismiki's products.

To export a section to a DXF file:

1. Select **Export** from the **File** menu.
2. Select **Export to DXF** from the **Export** menu.
3. The DXF driver configuration form appears:



4. Make the appropriate selections. Click **Ok** to proceed to the filename selection form. Click **Cancel** to cancel the operation.
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.

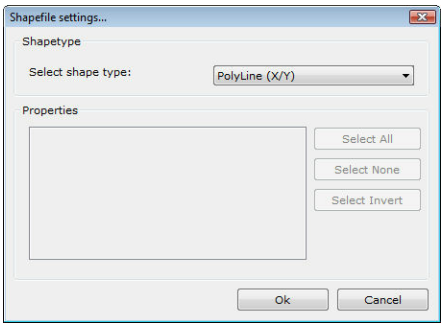


6.3.7.4 Export to ArcView Shapefile

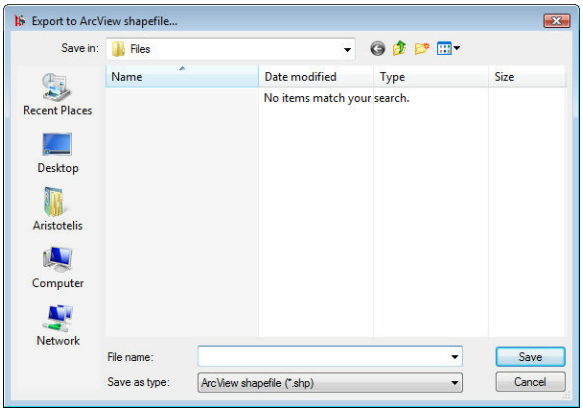
With this option, you can create a shapefile that can be used by programs such as ArcView GIS, MapInfo, GPS Trackmaker and other.

To export the current section to an ArcView Shapefile:

1. Select **Export** from the **File** menu.
2. Select **Export To ArcView Shapefile** from the **Export** menu.
3. Select the **shape type** from the drop-down list.



4. Select **Ok** to proceed. Select **Cancel** to abort the operation and close the dialog box.
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.



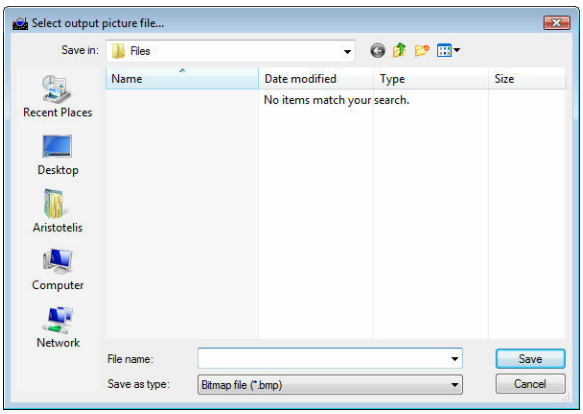
NOTE: Not all shape types are compatible with all programs.

6.3.7.5 Export to Bitmap

With this option, you can export (in BMP format) the sketch of the section, as it appears in main form of the section editor.

To export the sketch of the section:

1. Select **Export** from the **File** menu.
2. Select **Export to bitmap** from the **Export** 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. Select **Cancel** to cancel the operation.



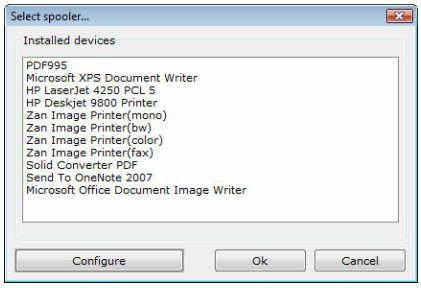
6.3.8 Print sketch

With this option, you can print the sketch of the section, as it appears in main form of the section editor, directly to a printer.

To print the sketch of the section:

1. Select **Print sketch** from the **File** menu.
2. Select the appropriate printer device.
3. You can optionally configure the printer device by clicking the **Configure** button. This will show the default printer driver configuration dialog box. Refer to the printer's

manual for more information.
4. Select **Ok** to print the sketch. Select **Cancel** to cancel the operation.

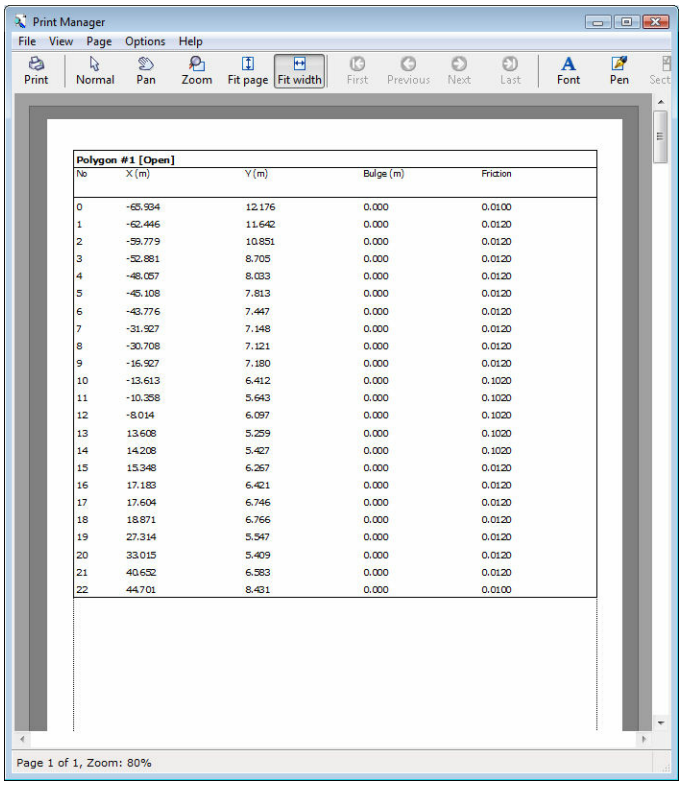


6.3.9 Print section data

With this option, you can prepare a report containing the section data. Note that with this option 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 with the section data:

1. Select **Print section data** 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.

6.3.10 Print section data to

6.3.10.1 Microsoft Excel

If Microsoft Excel (version 97, 2000, XP, 2003 or later) has been installed in the system, then a Microsoft Excel file containing the section data 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 section data to a Microsoft Excel file:

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

6.3.10.2 Microsoft Word

If Microsoft Word (version 97, 2000, XP, 2003 or later) has been installed in the system, then a Microsoft Word file containing the section data 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 section data to a Microsoft Word file:

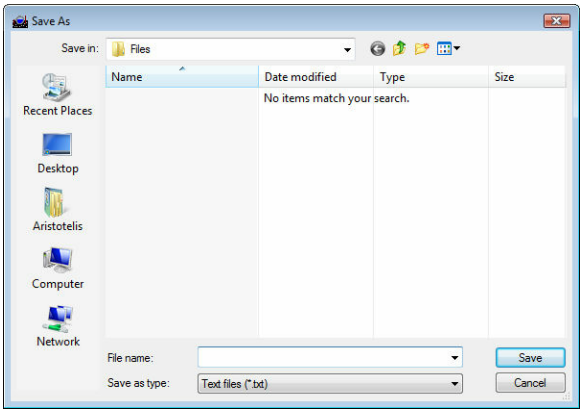
1. Select **Print section data to** from the **File** menu.
2. Select **Microsoft Word** from the **Print section data to** menu.

6.3.10.3 Text file

With this option, you can create a simple text file containing the section data. 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 section data to** from the **File** menu.
2. Select **Text file** from the **Print section data 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.



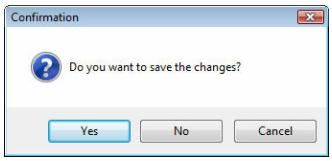
NOTE: If a file with the same name and in the same location already exists, a warning message will appear that asks whether to overwrite the file or not. If you answer Yes, then the existing file is erased and the new file takes its place. If you answer No, the existing file remains intact but the report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

6.3.11 Close

With this option, you can close the **Section editor** and return to section input.

To close the **Section editor**:

1. Select **Close** from the **File** menu.
2. Select **Yes** if you want to save the changes and return to section input. Select **No** if you want to discard the changes and return to section input. Select **Cancel** if you want to cancel the operation and return to **Section editor.**



6.4 Edit

6.4.1 Edit menu

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

- Add polygon
- Remove polygon
- Add vertex
- Insert vertex
- Remove vertex
- Select all
- Cut
- Copy
- Paste

6.4.2 Add polygon

With this option, you can add a **curvilinear polygon** to the section i.e. an open or closed polyline with straight or curved edges. A section must contain at least one polygon. There is no restriction in the number of nodes of the polygon.

To add a polygon:

1. Select **Add polygon** from the **Edit** menu.
2. A polygon is added to the drop-down list of the polygons.

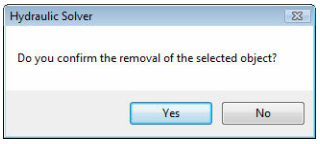
NOTE: This option is not available when entering ground and excavation data, as they must consist of a single polygon.

6.4.3 Remove polygon

With this option, you can remove a **curvilinear polygon** from the section.

To remove a polygon:

1. Select the polygon from the drop-down list.
2. Select **Remove polygon** from the **Edit** menu.
3. The polygon is deleted. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab. If you select No then the deletion is canceled.



NOTE: This option is not available when entering ground and excavation data, as they must consist of a single polygon.

6.4.4 Add vertex

With this option, you can add a vertex to the currently selected polygon. The vertex is appended to the data matrix.

To add a vertex:

1. Select the polygon from the drop-down list.
2. Select **Add vertex** from the **Edit** menu.
3. A vertex is appended to the data matrix of the currently selected polygon.

NOTE: This option is not available when there is no polygon in the section. In this case, add a polygon before adding a vertex.

6.4.5 Insert vertex

With this option, you can add a vertex to the selected polygon. The vertex is inserted before the currently selected vertex of the currently selected polygon.

To insert a vertex:

1. Select the polygon from the drop-down list.
2. Select **Insert vertex** from the **Edit** menu.
3. A vertex is inserted before the currently selected vertex of the selected polygon.

NOTE: This option is not available when there is no polygon in the section. In this case, add a polygon before inserting a vertex.

6.4.6 Remove vertex

With this option, you can remove a vertex of the currently selected polygon.

To remove a vertex:

1. Select the polygon from the drop-down list.
2. Select the vertex from the data matrix.
3. Select **Remove vertex** from the **Edit** menu.
4. The currently selected vertex of the currently selected polygon is removed.

NOTE: Each polygon must consist of at least two nodes. These nodes cannot be removed.

6.4.7 Select all

With this option, you can select all vertices of the currently selected polygon.

To select all vertices:

1. Select **Select all** from the **Edit** menu
2. All vertices of the currently selected polygon are selected.

6.4.8 Cut

With this option, you can cut the currently selected cells of the data matrix and transfer them to the clipboard.

To cut:

1. Select the cells you wish to cut from the data matrix.
2. Select **Cut** from the **Edit** menu.
3. All selected cells of the data matrix are cut and transferred to the clipboard.

6.4.9 Copy

With this option, you can copy the currently selected cells of the data matrix to the clipboard.

To copy:

1. Select the cells you wish to copy from the data matrix.
2. Select **Copy** from the **Edit** menu.
3. All selected cells of the data matrix are copied to the clipboard.

6.4.10 Paste

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

To paste:

1. Select the top left cell of the range where you want to paste the data.
2. Select **Paste** from the **Edit** menu.
3. The data are pasted from the clipboard to the data matrix.

NOTES:
- The appropriate number of columns and rows is automatically identified by the program.
- With this option you can transfer data from other programs, such as Microsoft Excel.

6.5 View

6.5.1 View menu

With this menu, you can modify the way you view the section. In the **View** menu you can select one of the following options:

- Zoom extent
- Zoom window
- Zoom previous
- Zoom in
- Zoom out
- Display vertices
- Display origin
- Display grid
- Display data matrix
- Toggle excavations / section

6.5.2 Zoom extent

With this option, you can view the whole section.

To view the whole section:

1. Select **Zoom extent** from the **View** menu.
2. The viewport is adjusted automatically to include the whole section.

NOTE: This option is also available in the toolbar.

6.5.3 Zoom window

With this option, you can zoom to a specified window.

To zoom to a window:

1. Select **Zoom window** from the **View** menu.
2. Click on the drawing to define one corner of the window or hit ESC to cancel the procedure.
3. Click on the drawing to define the opposite corner of the window or hit ESC to cancel the procedure.
4. The viewport is adjusted automatically to zoom to the specified window.

NOTE: This option is also available in the toolbar.

6.5.4 Zoom previous

With this option, you can revert to the previous zoom configuration.

To use the previous zoom configuration:

1. Select **Zoom previous** from the **View** menu.
2. The previous zoom configuration is applied.

NOTE: This option is also available in the toolbar.

6.5.5 Zoom in

With this option, you can zoom in to a specified point.

To zoom in to a specified point:

1. Select **Zoom in** from the **View** menu.
2. Click on the drawing to define the point to zoom in to or hit ESC to cancel the procedure.
3. The zoom factor is doubled and the viewport is adjusted to zoom in to the specified point.

NOTE: This option is also available in the toolbar.

6.5.6 Zoom out

With this option, you can zoom out from a specified point.

To zoom out from a specified point:

1. Select **Zoom out** from the **View** menu.
2. Click on the drawing to define the point to zoom out from or hit ESC to cancel the procedure.
3. The zoom factor is halved and the viewport is adjusted to zoom out from the specified point.

NOTE: This option is also available in the toolbar.

6.5.7 Pan

With this option, you can move the drawing within the viewport.

To pan the drawing:

1. Select **Pan** from the **View** menu.
2. Click and drag on the drawing to move it within the viewport or hit ESC to cancel the procedure.
3. The drawing is moved within the viewport.

NOTE: This option is also available in the toolbar.

6.5.8 Display vertices

With this option, you can show or hide the vertices of the currently selected polygon.

To show or hide the vertices of the currently selected polygon:

1. Select **Display vertices** from the **View** menu.
2. If the vertices are visible then they become hidden and vice versa. If the option is active then there is a tick on the left of the menu.

6.5.9 Display origin

With this option, you can show or hide the origin i.e. the point corresponding to (0,0).

To show or hide the origin:

1. Select **Display origin** from the **View** menu.
2. If the origin is visible then it becomes hidden and vice versa. If the option is active then there is a tick on the left of the menu.

6.5.10 Display grid

With this option, you can show or hide the grid.

To show or hide the grid:

1. Select **Display grid** from the **View** menu.
2. If the grid is visible then it becomes hidden and vice versa. If the option is active then there is a tick on the left of the menu.

6.5.11 Display data matrix

With this option, you can show or hide the data matrix.

To show or hide the data matrix:

1. Select **Display data matrix** from the **View** menu.
2. If the data matrix is visible then it becomes hidden and vice versa. If the option is active then there is a tick on the left of the menu.

6.5.12 Toggle excavations / section

With this option, you can set whether the data matrix will display section data or excavation data.

To toggle between section and excavation data:

1. Select **Toggle excavations / section** from the **View** menu.
2. If the data matrix contains section data then it will be loaded with excavation data and vice versa.

NOTE: When the data matrix contains excavation data, the drop-down list of polygons is loaded with two polygons, namely the **excavation line** and the **natural ground line.**

6.6 Options

6.6.1 Options menu

With this menu, you can modify the way you view the section. In the **Options** menu you can select one of the following options:

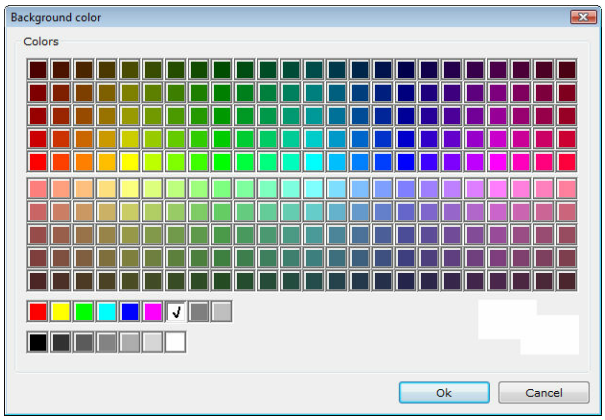
- Background color
- Interior color
- Grid
- Edge pen color
- Edge pen width
- Inactive vertices
- Water area
- Excavations

6.6.2 Background color

With this option, you can change the background color of the drawing. The default value is white.

To change the background color:

1. Select **Background color** from the **Options** menu.
2. The color selection dialog box appears.
3. 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.
4. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



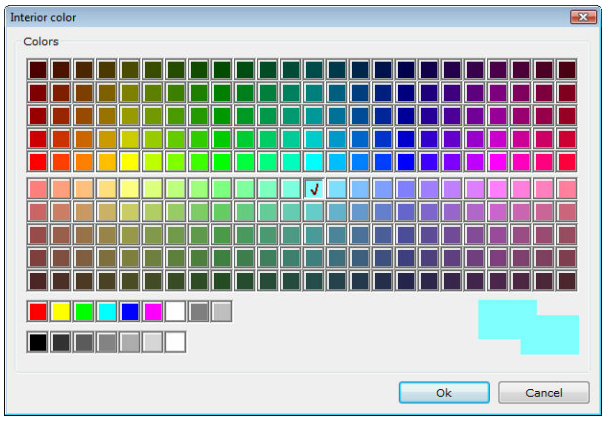
NOTE: The color palette follows standard CAD color palettes.

6.6.3 Interior color

With this option, you can change the interior color of closed polygons. The default value is grey.

To change the interior color of closed polygons:

1. Select **Interior color** from the **Options** menu.
2. The color selection dialog box appears.
3. 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.
4. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



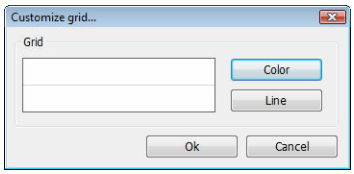
NOTE: The color palette follows standard CAD color palettes.

6.6.4 Grid

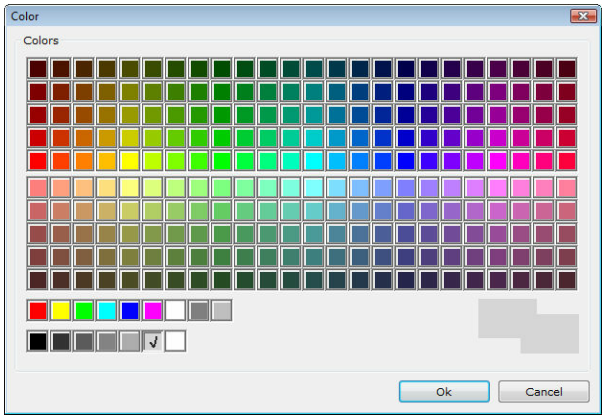
With this option, you can change the color and line style of the grid. Note that in order to view the changes the grid must be visible.

To change the color and line style of the grid:

1. Select **Grid** from the **Options** menu.
2. The grid options dialog box appears:

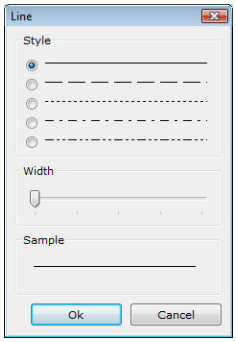


3. Select **Color** to change the color of the line.
3.1. The color selection dialog box appears.
3.2. Select the **color** from the 256 available colors. The currently selected color is marked with a tick. On top of the **Cancel** button, the old and the new color are displayed.
3.3. Click **Ok** to 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.

4. Select **Line** to change the line style.
4.1. The line style selection dialog box appears.
4.2. Select the line style and width. The currently selected color is marked with a tick. On top of the **Cancel** button, the old and the new color are displayed.
4.3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



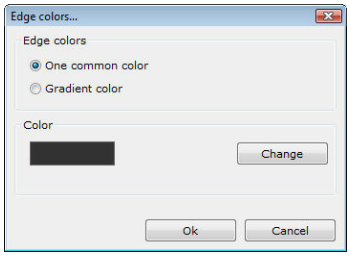
5. Click **Ok** to save the grid options and close the dialog box or click the **Cancel** button to close the dialog box without saving the changes.

6.6.5 Edge pen color

With this option, you can change the pen color of the polygon edges.

To change the pen color of the polygon edges:

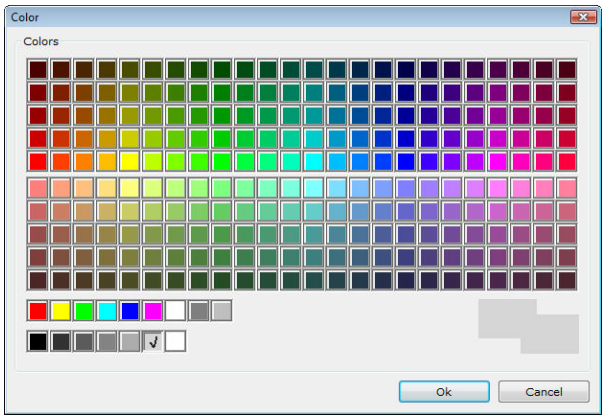
1. Select **Edge pen color** from the **Options** menu.
2. The edge pen color dialog box appears:



3. Select **One common color** to use a single color for all edges. Select **Gradient color** to use an intermediate color between two colors depending on the friction of the edge.

A. One common color

4. Select **Color** to change the color of the pen.
4.1. The color selection dialog box appears.
4.2. 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.
4.3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.

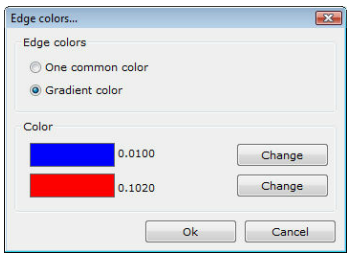


NOTE: The color palette follows standard CAD color palettes.

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

B. Gradient color

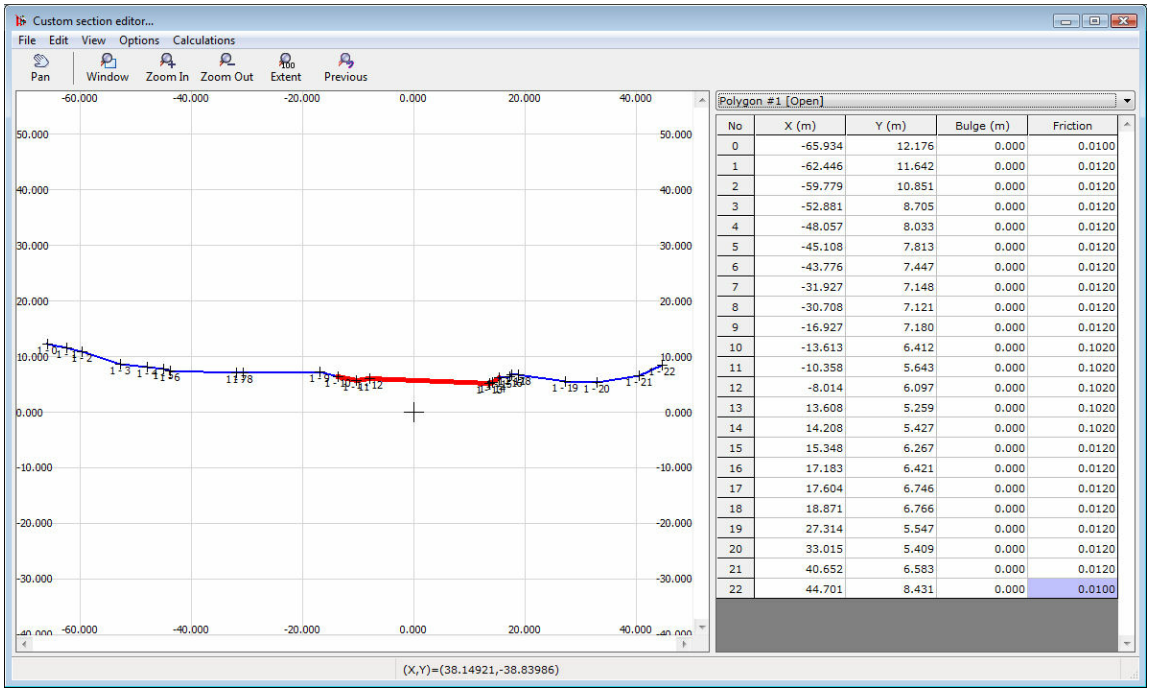
4. Two colors must be defined. The first corresponds to the minimum friction and the second to the maximum friction value. These values are displayed next to the corresponding picture boxes. Select the appropriate **Color** button to change the corresponding color.
4.1. The color selection dialog box appears.
4.2. 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.
4.3. 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.

5. Repeat step 4 with the second color, if necessary.
6. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.

Example of section with gradient edge color



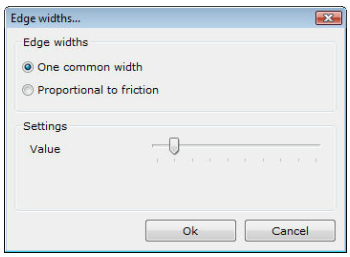
NOTE: You can combine gradient color with varying pen width.

6.6.6 Edge pen width

With this option, you can change the pen width of the polygon edges.

To change the pen width of the polygon edges:

1. Select **Edge pen width** from the **Options** menu.
2. The edge pen width dialog box appears:



3. Select **One common width** to use a single width for all edges. Select **Proportional to friction** to use an intermediate width between two values depending

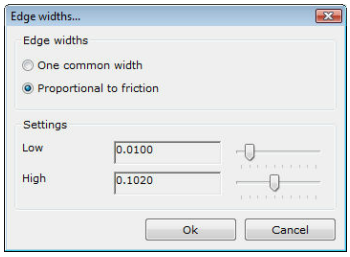
on the friction of the edge.

A. One common width

4. Select the appropriate width using the slider.
5. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.

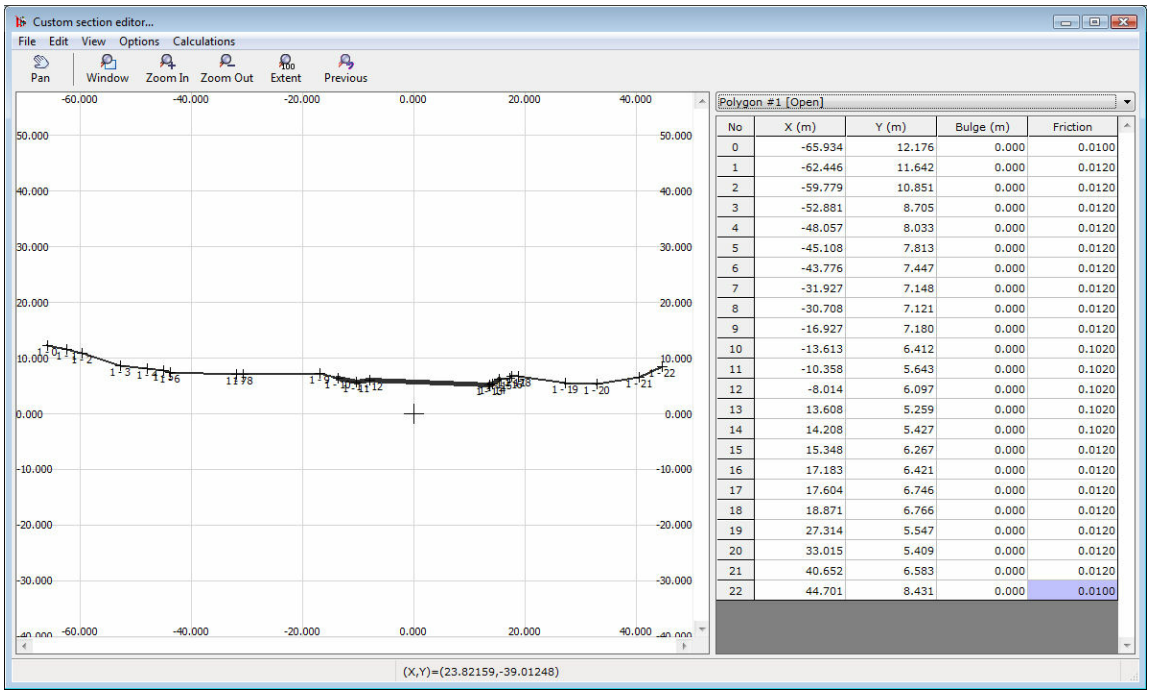
B. Proportional to friction

4. Two values must be defined. The first corresponds to the minimum friction and the second to the maximum friction. These values are displayed next to the corresponding labels. Select the appropriate width using the corresponding slider.



5. Repeat step 4 with the second width, if necessary.
6. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.

Example of section with proportional edge width



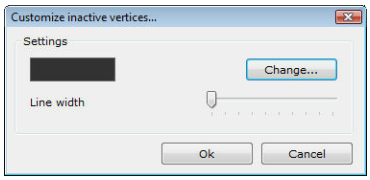
NOTE: You can combine proportional width with varying pen color.

6.6.7 Inactive vertices

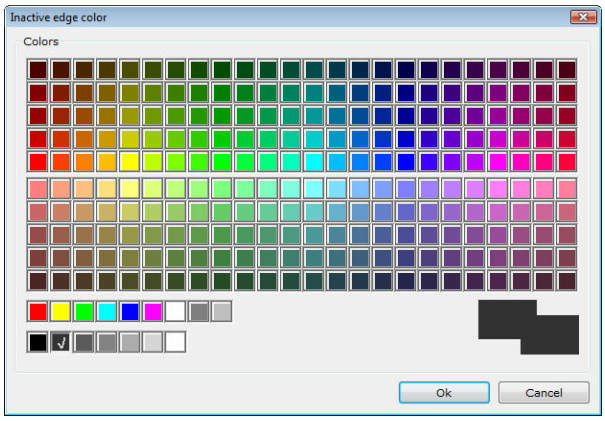
With this option, you can change the color and pen width of the edges between inactive vertices.

To change the color and pen width of the inactive edges:

1. Select **Inactive vertices** from the **Options** menu.
2. The inactive vertices dialog box appears:



3. Select **Change** to change the color of the pen.
3.1. The color selection dialog box appears.
3.2. 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.
3.3. 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.

4. Select the appropriate pen width using the slider.
5. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.

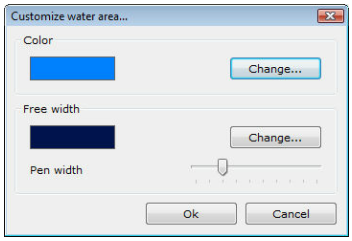
6.6.8 Water area

With this option, you can change the interior color of water areas. You can also change the pen color and width of the free width.

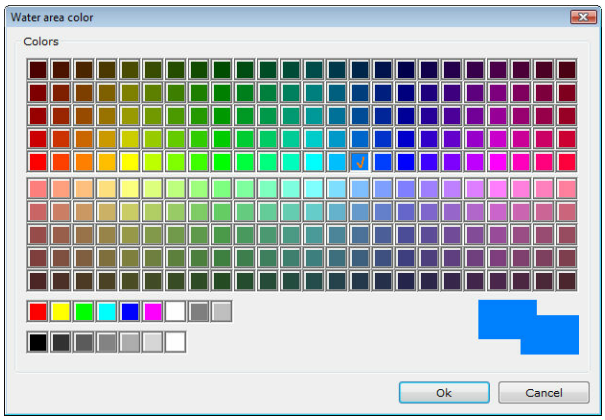
To change the interior color of water areas or the pen color and pen width of the free width:

1. Select **Water areas** from the **Options** menu.

2. The following dialog box appears:

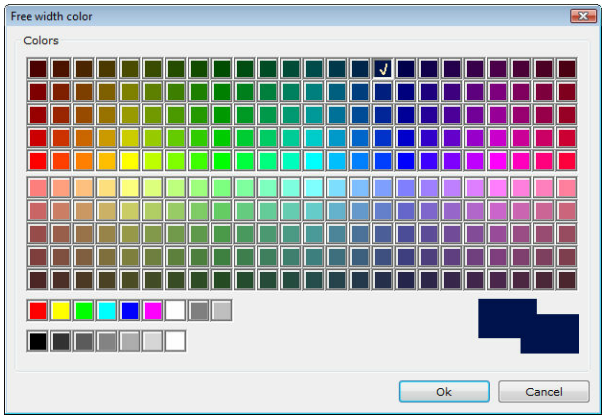


3. Select **Change** from the **Color** frame to change the interior color for water areas.
3.1. The color selection dialog box appears.
3.2. Select the **color** from the 256 available colors. The currently selected color is marked with a tick. On top of the **Cancel** button, the old and the new color are displayed.
3.3. 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.

4. Select **Change** from the **Free width** frame to change the pen color for free width.
4.1. The color selection dialog box appears.
4.2. 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.
4.3. 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.

5. Select the appropriate pen width using the slider.
6. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.

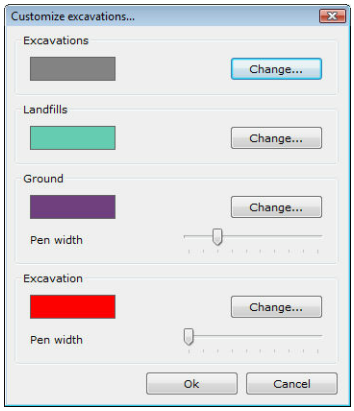
NOTE: The free width is not visible in pressure sections.

6.6.9 Excavations

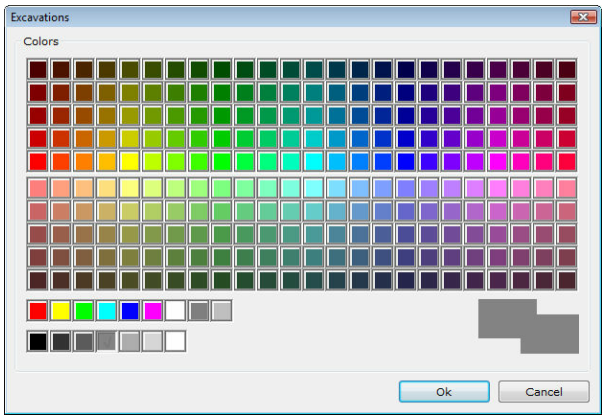
With this option, you can change the color and line styles of objects related to excavations.

To change the color and line styles of objects related to excavations:

1. Select **Excavations** from the **Options** menu.
2. The following dialog box appears:

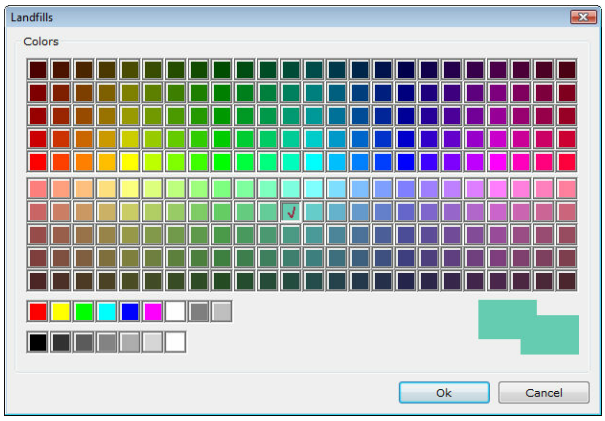


3. Select **Change** from the **Excavations** frame to change the interior color for excavations.
3.1. The color selection dialog box appears.
3.2. Select the **color** from the 256 available colors. The currently selected color is marked with a tick. On top of the **Cancel** button, the old and the new color are displayed.
3.3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



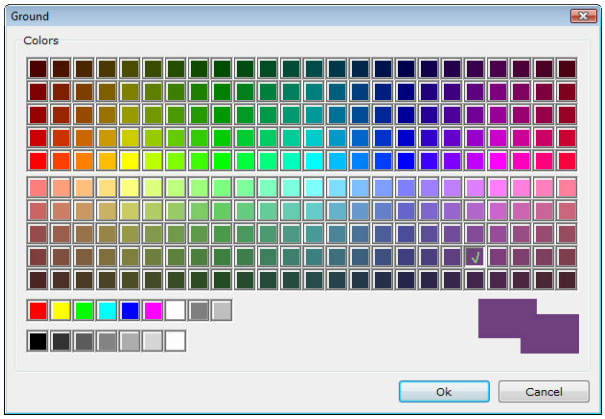
NOTE: The color palette follows standard CAD color palettes.

4. Select **Change** from the **Landfills** frame to change the interior color for landfills.
4.1. The color selection dialog box appears.
4.2. 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.
4.3. 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.

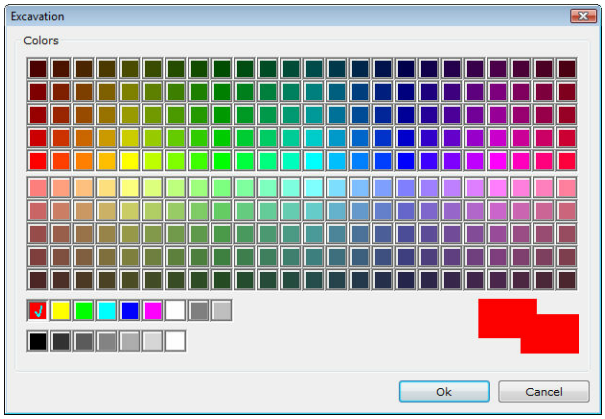
5. Select **Change** from the **Ground** frame to change the pen color for the ground line.
5.1. The color selection dialog box appears.
5.2. Select the **color** from the 256 available colors. The currently selected color is marked with a tick. On top of the **Cancel** button, the old and the new color are displayed.
5.3. 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.

6. Select the appropriate pen width of the ground line using the slider of the **Ground** frame.

7. Select **Change** from the **Excavation** frame to change the pen color for the excavation line.
7.1. The color selection dialog box appears.
7.2. 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.
7.3. 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.

8. Select the appropriate pen width of the excavation line using the slider of the **Excavation** frame.
9. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.

NOTE: In order for excavations or landfills to be calculated, the ground and excavation line must intersect each other. However, the presence of excavation or landfill areas is not compulsory.

6.7 Calculations

6.7.1 Calculation menu

With this menu, you have access to several computational tools. In the **Calculations** menu you can select one of the following options:

- Origin
  - Enter origin coordinates
  - Enter origin graphically
  - Set origin to deep point
- Active nodes
- Select flow depth
- Full flow

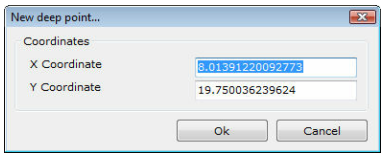
6.7.2 Origin

6.7.2.1 Enter origin coordinates

With this option, you can set the coordinates of the origin, i.e. the (0,0) point, in the WCS (World Coordinate System) analytically. The coordinates of all vertices are changed to comply with the new origin of the coordinate system.

To set the coordinates of the origin analytically:

1. Select **Origin** from the **Calculations** menu.
2. Select **Enter origin coordinates** from the **Origin** menu.
3. The origin coordinates dialog box appears.
4. Type the appropriate coordinates in the text boxes.
5. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



NOTE: To return to the WCS, set both coordinates of the origin equal to 0.

6.7.2.2 Enter origin graphically

With this option, you can set the coordinates of the origin, i.e. the (0,0) point, in the WCS (World Coordinate System) graphically. The coordinates of all vertices are changed to comply with the new origin of the coordinate system.

To set the coordinates of the origin graphically:

1. Select **Origin** from the **Calculations** menu.
2. Select **Enter origin graphically** from the **Origin** menu.
3. Click the desired point on the drawing or hit ESC to cancel the operation.
4. The coordinates of the origin are changed.

NOTE: To return to the WCS, set both coordinates of the origin equal to 0 analytically.

6.7.2.3 Set origin to deep point

With this option, you can set the coordinates of the origin, i.e. the (0,0) point, in the WCS (World Coordinate System) equal to those of the deep point of the section. The coordinates of all vertices are changed to comply with the new origin of the coordinate system.

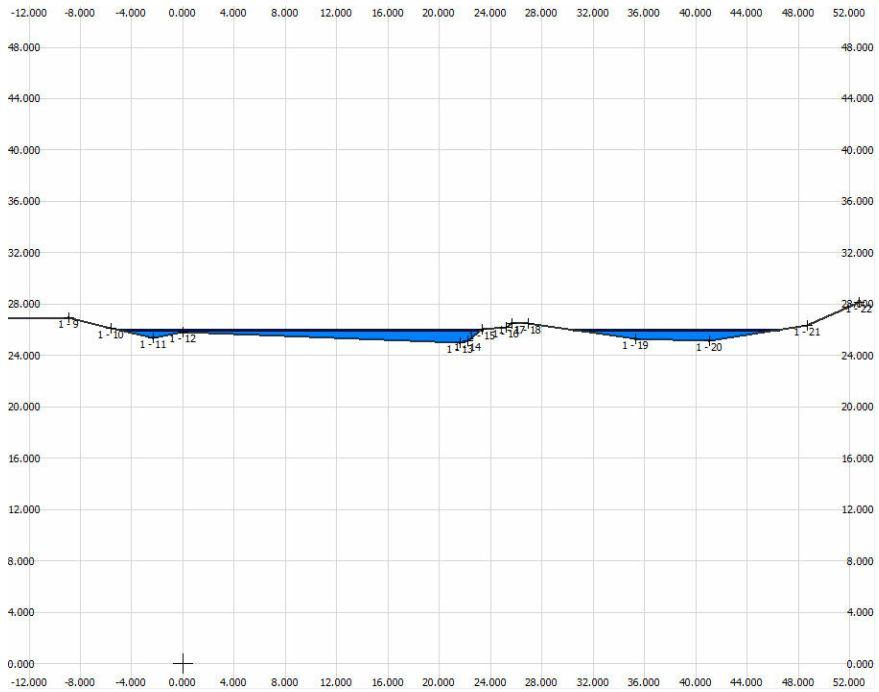
To set the coordinates of the origin equal to those of the deep point:

1. Select **Origin** from the **Calculations** menu.
2. Select **Set origin to deep point** from the **Origin** menu.
3. The coordinates of the origin are set equal to those of the deep point of the section.

NOTE: To return to the WCS, set both coordinates of the origin equal to 0 analytically.

6.7.3 Active nodes

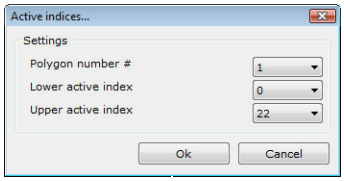
With this option, you can set the active nodes (vertices) of a section. By default, all nodes are active; therefore, the flow may occur anywhere in the section.



However, if we know that the flow occurs between vertex 0 and 17 of the above example, we must deactivate vertices 17 to 22.

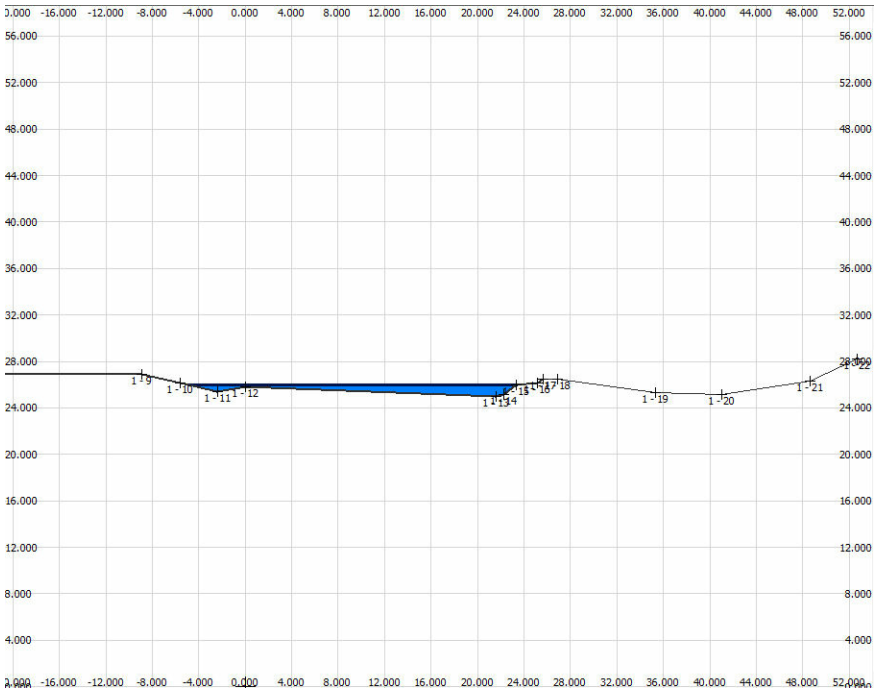
To select the active nodes (vertices):

1. Select **Active nodes** from the **Calculations** menu.
2. Select the polygon in question, the upper active index (in this case, 17 instead of 22) and the lower active index (in this case, the same node i.e. node 0 is selected).



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

If you select 17 as the last active vertex in the above example, the flow is confined to the left riverbed:



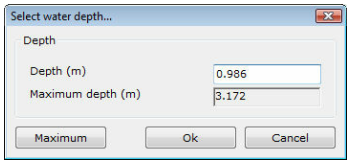
NOTE: The inactive edges are drawn with different pen color and size, according to the preferences of the program.

6.7.4 Select flow depth

With this option, you can set a trial flow depth. This may be a value between 0 and the maximum flow depth. This option is used for demonstrating the way the section is filled for various depths and it is not used in calculations.

To set a trial depth:

1. Select **Select flow depth** from the **Calculations** menu.
2. The flow depth selection dialog box appears.
3. Type the trial flow depth in the text box. The maximum flow depth is displayed in the label beneath the text box.
4. Alternatively, click **Maximum** to use the maximum flow depth.
5. Click **Ok** to close the dialog box and apply the selected flow depth. Click **Cancel** to close the dialog box without changing the flow depth.



6.7.5 Full flow

With this option, you can view the way the section is filled in an animated way. After the animation, the full flow configuration of the section is displayed.

To view the way the section is filled in an animated way and view the full flow configuration of the section:

1. Select **Full flow** from the **Calculations** menu.
2. The flow depth is increased from 0 to its maximum value.

Chapter

7 Help

7.1 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

7.2 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.

7.3 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.

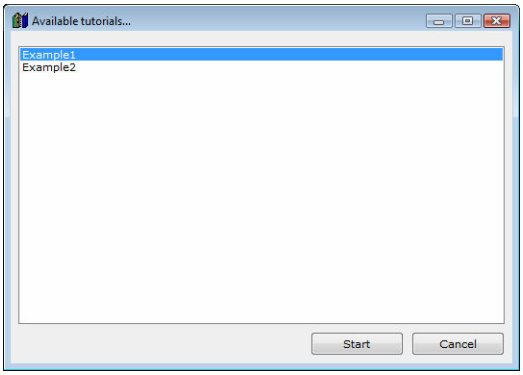
7.4 Tutorials

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.



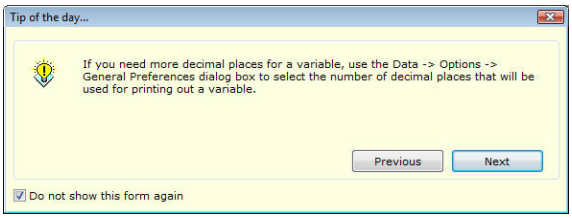
NOTE: The number and content of the tutorials is changed frequently. Use the live update system of TechnoLogismiki's products to download the latest tutorials.

7.5 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.



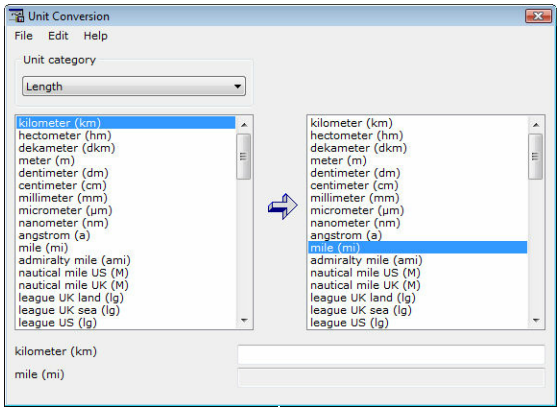
NOTE: The number and content of the tips is changed frequently. Use the live update system of TechnoLogismiki's products to download the latest tips.

7.6 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.

7.7 TechnoLogismiki website

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

7.8 Buy products

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

7.9 TechnoLogismiki NOMOS

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

7.10 TechnoLogismiki Live!

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

7.11 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.

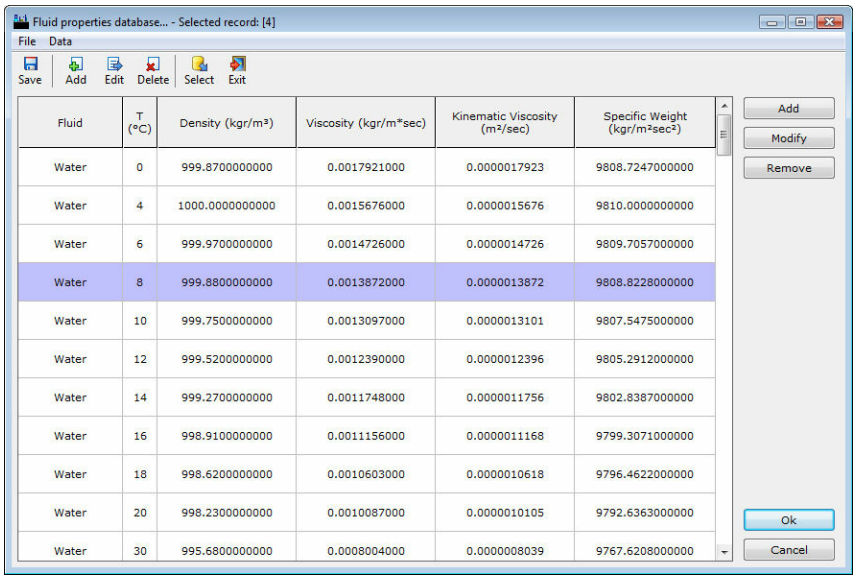
Chapter VIII

8 Databases

8.1 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.

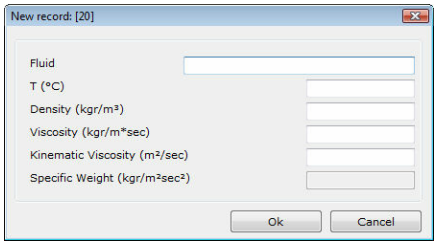
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



Fluid	T (°C)	Density (kgr/m³)	Viscosity (kgr/m*sec)	Kinematic Viscosity (m²/sec)	Specific Weight (kgr/m²sec²)
Water	0	999.8700000000	0.0017921000	0.0000017923	9808.7247000000
Water	4	1000.0000000000	0.0015676000	0.0000015676	9810.0000000000
Water	6	999.9700000000	0.0014726000	0.0000014726	9809.7057000000
Water	8	999.8800000000	0.0013872000	0.0000013872	9808.8228000000
Water	10	999.7500000000	0.0013097000	0.0000013101	9807.5475000000
Water	12	999.5200000000	0.0012390000	0.0000012396	9805.2912000000
Water	14	999.2700000000	0.0011748000	0.0000011756	9802.8387000000
Water	16	998.9100000000	0.0011156000	0.0000011168	9799.3071000000
Water	18	998.6200000000	0.0010603000	0.0000010618	9796.4622000000
Water	20	998.2300000000	0.0010087000	0.0000010105	9792.6363000000
Water	30	995.6800000000	0.0008004000	0.0000008039	9767.6208000000

To add a new record:

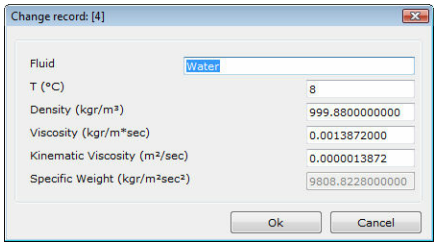
1. Click **Add** to open the new record dialog box.
2. Type the name of the fluid. This field is required.
3. Enter the temperature, density, viscosity and kinematic viscosity of the fluid.
4. The specific weight is calculated automatically.
5. Click **Ok** to close the dialog box and add a new record 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.



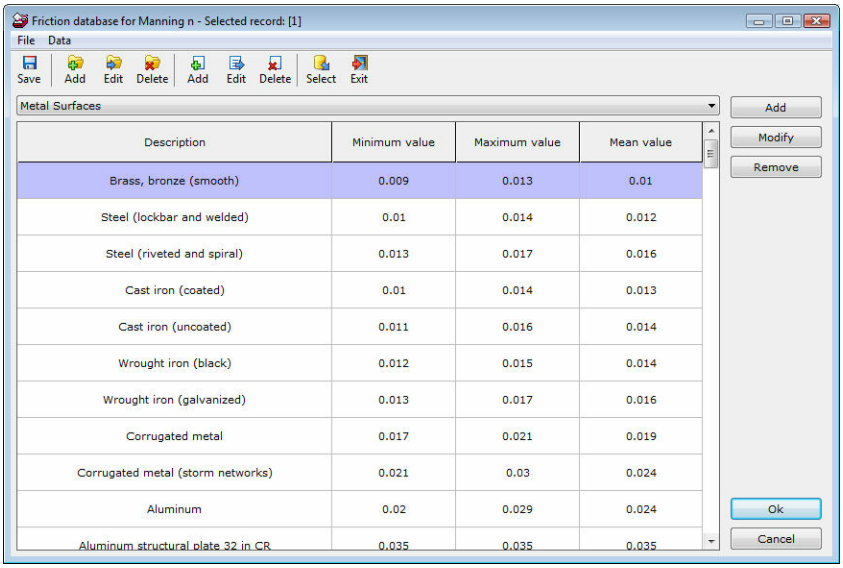
To remove an existing record:

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.

8.2 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.

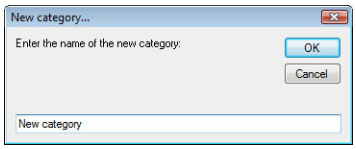
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



The database consists of several categories. Usually, the category defines the material of the surface (e.g. Metal surfaces).

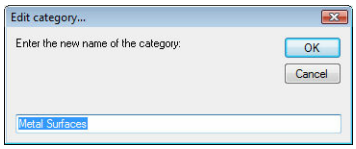
To add a new category:

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.



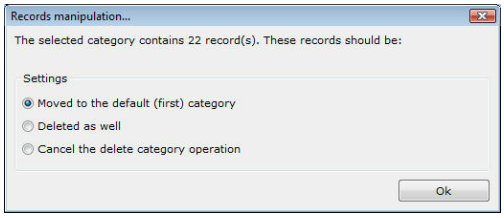
To modify the name of an existing category:

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:

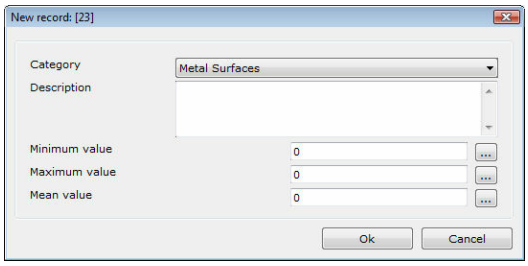


4.1. Select the first option to move the records of the category to the default (first category).
4.2. Select the second option to delete the records.
4.3. Select the third option to cancel the deletion.
5. Click **Ok** to proceed.

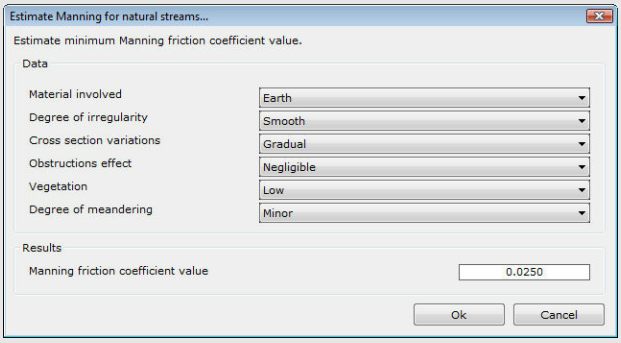
NOTE: The database must contain at least one category.

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.



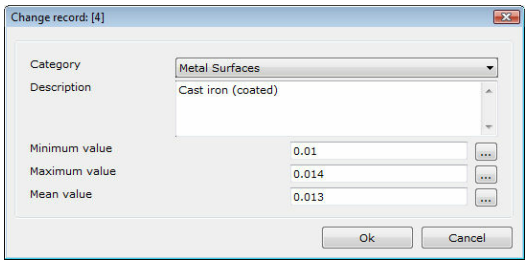
NOTE: In case of Manning friction coefficients in natural streams, you can estimate the values based on several characteristics of the stream. Click on the buttons with the ellipses (...) next to the text boxes to invoke the following dialog box:



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.

8.3 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

8.4 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

8.5 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

8.6 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

8.7 Local energy losses friction coefficients

Fitting	K-value	Fitting	K-value
Pipe entrance		**90° smooth bend[1]**	
Bellmouth	0.00 ~ 0.05	r/D=4 (r radius)	0.16 ~ 0.18
Rounded	0.18	r/D=2	0.19 ~ 0.25
Sharp edged	0.50	r/D=1.5	0.26 ~ 0.34
Projecting	0.80 ~ 1.00	r/D=1	0.35 ~ 0.40
Contraction - Sudden		**Mitered bend**	
D_2/D_1 = 0.20	0.41 ~ 0.50	15° angle	0.05
0.20 < D_2/D_1 = 0.40	0.30 ~ 0.41	30° angle	0.10
0.40 < D_2/D_1 = 0.60	0.18 ~ 0.30	45° angle	0.20
0.60 < D_2/D_1 = 0.80	0.06 ~ 0.18	60° angle	0.35
D_2/D_1 = 0.80	0.00 ~ 0.06	90° angle	0.80
Contraction - Conical		**Tee[2]**	
15° angle	0.02	Line flow	0.30 ~ 0.40
22.5° angle	0.04	Branch flow	0.60 ~ 2.10
45° angle	0.07	**45° Wye[2]**	
Expansion - Sudden		Line flow	0.20 ~ 0.35
D_2/D_1 = 0.20	0.92 ~ 1.00	Branch flow	0.45 ~ 0.55
0.20 < D_2/D_1 = 0.40	0.71 ~ 0.92	**Cross[2]**	
0.40 < D_2/D_1 = 0.60	0.41 ~ 0.71	Line flow	0.40 ~ 0.60
0.60 < D_2/D_1 = 0.80	0.13 ~ 0.41	Branch flow	0.60 ~ 0.90
D_2/D_1 = 0.80	0.00 ~ 0.13	**Spherical valves[3]**	
Expansion - Conical		90° angle	0.05

Fitting	K-value	Fitting	K-value
15° angle	0.03	60° angle	1.20
22.5° angle	0.07	45° angle	10.00
45° angle	0.14	30° angle	50.00

NOTES:

1.These values are valid for Reynolds number around $2x10^5$.
2. Typical values for commercial parts.
3. The angle of spherical valves is the complementary angle defined by the pipe axis and the valve opening axis.

Keyword index

- 1 -

- 3 -

- 4 -

- 9 -

- A -

- B -

- C -

- D -

- E -

- F -

- G -

- H -

- I -

- J -

- K -

- L -

- M -

- N -

- O -

- P -

- R -

- S -

- T -

- U -

- V -

- W -

- X -

- Y -

- Z -

This page was intentionally left blank.