Works 2013

Flood flows

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

Flood flows

TechnoLogismiki

Flood flows

Publisher

TechnoLogismiki

Editors

Fotis Fotopoulos
Aristotelis Charalampakis

Technical Assistance

Antigoni Egglezou

Printed: September 2012 in Athens, Greece.

Table of Contents

Chapter III File

Chapter IV Data

Chapter V Results

Chapter VI Help

Chapter VII Sections

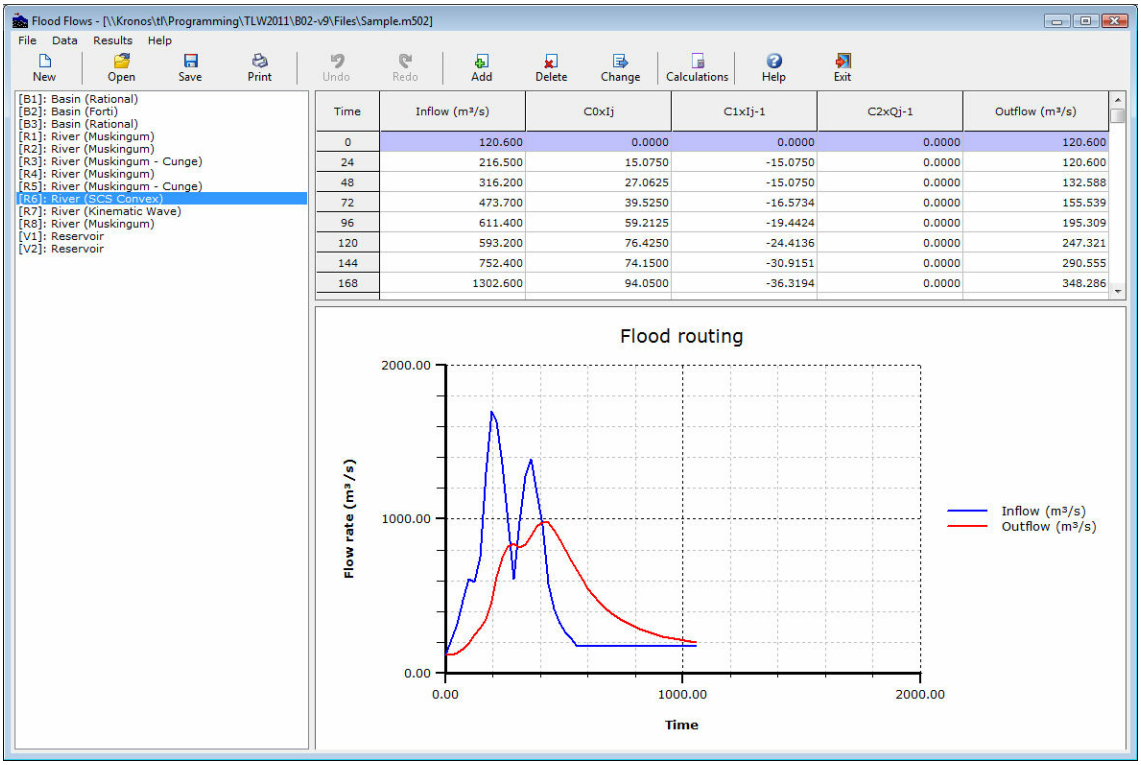
Chapter

1 About the program

1.1 What does the program do?

This programs calculates the following:

- maximum flood flow in water basins caused by rainfall.
- maximum flood flow after flood routing in rivers.
- maximum flood flow after flood routing in reservoirs.



For the calculation of the flood flows in water basins, the Rational method or other empirical formulas may be used. If the Rational method is used, one of the following formulas may be utilized to calculate the concentration time:

- Giandotti
- Kirpich / Kirpich (PA)
- Federal Aviation Administration
- Kerby / Hathaway
- Carter
- Eagleson
- Espey / Winslow
- SCS Lag
- Length and velocity
- TR-55U / TR-55P
- Direct value
- California culverts practice
- Izzard

- Kinematic wave

The concentration time is used in the IDF curve to calculate the rainfall intensity. In the case of empirical formulas, one of the following seven options may be used:

- Fuller
- Forti
- Fanning
- Iskovski
- Kuichling

The use of the above formulas is restricted by certain conditions. The program notifies the user in case one or more formulas are out of scope and should not be used.

The following synthetic hydrographs can be used:

- Snyder Synthetic Hydrograph
- SCS Unit hydrograph

For the flood routing in a river the following methods may be used:

- Muskingum
- Muskingum-Cunge
- SCS Convex
- Kinematic Wave

For the flood routing in reservoirs the Storage Indicator method is used.

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

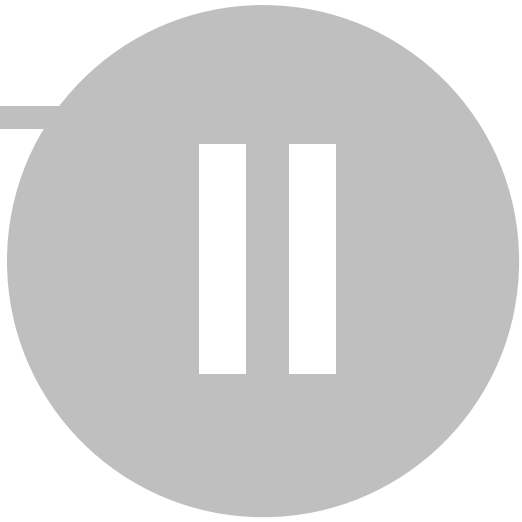
The normal response time is within two business days. If your inquiry cannot be answered by e-mail, a customer service representative will contact you by telephone.

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 Theoretical background

2.1 Introduction

In this chapter, a summary description of the necessary theoretical background is presented.

Flood flows calculation

The calculation of the maximum flood flow can be achieved using the rational method or a variety of empirical formulas.

Rational method

The time of concentration can be estimated using one of the following methods:

- Length and velocity method
- Carter formula
- Eagleson formula
- Espey / Winslow formula
- Federal Aviation Agency (FAA) formula
- Giandotti formula
- Kerby / Hathaway formula
- Kirpich and Kirpich (PA) formula
- TR55 (P or U) method
- SCS Lag method
- California Culverts Practice method
- Izzard formula
- Kinematic wave method

Empirical formulas

Forti empirical formula
Fanning empirical formula
Fuller empirical formula
Kuichling empirical formula
Iskovski empirical formula

Synthetic unit hydrograph

SCS Dimensionless Hydrograph
Snyder synthetic hydrograph

Flood routing in river

- Muskingum method
- Muskingum - Cunge method
- SCS Convex method
- Kinematic wave method

Flood routing in reservoir

- Storage Indicator method

2.2 Flood flows

2.2.1 Rational method

2.2.1.1 Rational method

According to the rational method, the maximum flood flow is proportional to the area of the basin, the runoff coefficient and the intensity of the critical storm:

$$Q = 0.278 \cdot c \cdot f \cdot I \cdot A$$

where:
- Q, maximum flood flow (m^3/s)
- c, dimensionless runoff coefficient (-)
- f, a rainfall correction factor that accounts for spatial non-uniformity (-)
- I, rainfall intensity of critical storm (mm/h)
- A, the total area of the basin (km^2)

Two important assumptions behind the rational method are a) the duration of the critical storm is equal to the basin's time of concentration and b) the return period of the calculated flood flow is equal to the critical storm's return period.

The runoff coefficient is defined as the amount of precipitation that flows to the basin's outlet to the total precipitation over the basin, during a specified time period. This time period is usually one year.

$$c = \frac{h}{P}$$

where:
- c, dimensionless runoff coefficient (-)
- P, the total precipitation (mm)
- h, the equivalent areal water depth (mm)

The runoff coefficient depends on a variety of parameters such as the vegetation, geology, land uses, etc. Characteristic values of this coefficient can be taken from tables. However, it is common that a single basin consists of several different surfaces, each of them with a different runoff coefficient. In these cases, a weighted runoff coefficient is computed using all the areas of the basin:

$$C_M = \frac{c_1 \cdot A_1 + c_2 \cdot A_2 + \ldots + c_n \cdot A_n}{A_1 + A_2 + \ldots + A_n}$$

where:
- C_M, the weighted dimensionless runoff coefficient of n areas
- c_i, the runoff coefficient of the i^{th} area
- A_i, the i^{th} area (common units for denominator and nominator)

The most common runoff coefficient values are included in the program's database.

For more information regarding the embedded database, please refer to the appendix.

The rainfall intensity of the critical storm is usually derived from intensity-duration-frequency (IDF) curves. The IDF curves are given by mathematic equations of the following form:

$$i = \frac{c \cdot T^K}{(t+b)^n}$$

where:
- i, the rainfall intensity (mm/h)
- t, time of concentration (h)
- T, return period (years)
- b, n, K, IDF coefficients (with dimensions)

The time of concentration is necessary to calculate the rainfall intensity using IDF curves. For urban areas without large basins, the Greek regulations suggest a value equal to 10 min. According to the US regulations, this time can be smaller or greater depending on the existence of water collection inlets and the percentage of coverage by constructions and infrastructure such as roads, cement areas, etc.

2.2.1.2 Time of concentration

2.2.1.2.1 Length and velocity

If the speed at which the surface water moves is known, the time of concentration is calculated by the ratio of the length the water travels to its velocity:

$$t_C = \frac{5}{18}\frac{L}{V}$$

Where:

- t_c, time of concentration (h)
- L, length of the main basin river (km)
- V, velocity of water (m/s)

2.2.1.2.2 Carter

This equation was introduced by Carter in order to be used in Washington DC sewers, to calculate the time of concentration is serviced surfaces with a total area of less than 18 km^2 and average slopes around 2%.

$$t_C = 0.2 \cdot \frac{L^{0.6}}{J^{0.3}}$$

Where:

- t_c, time of concentration (h)
- L, length of the main basin river (km)
- J, the average basin slope (m/m)

2.2.1.2.3 Eagleson

It is applied in the special case where the surface flow occurs in natural channels with constant geometry (constant cross section). This method is based on kinematic wave's theory. The time of concentration depends on the hydraulic radius of the stream and on the average Manning coefficient among others.

$$t_C = 1.25 \cdot \frac{L \cdot n}{R^{2/3} \cdot J^{0.5}}$$

where:

- t_c, time of concentration (h)
- L, length of the main basin river (km)
- J, the average basin slope (m/m)
- n, Manning's friction coefficient
- R, hydraulic radius of the main river (m)

2.2.1.2.4 Espey / Winslow

This method calculates the time of concentration based on Espey's stream coefficient, which is a function of Manning's friction factor and the percentage of impervious soil. Apart from these factors, the length of the water's path and the average ground slope are also taken into consideration:

$$t_C = 5.33 \cdot F \cdot \frac{L^{0.29}}{IMP^{0.6} \cdot J^{0.145}}$$

where:

- t_c, time of concentration (h)
- L, length of the main basin river (km)
- J, the average basin slope (m/m)
- IMP, percent of impervious soil (%)
- F, the dimensionless Espey stream coefficient

2.2.1.2.5 Federal Aviation Agency

For small water basins without formed hydrographic network (the water flows on the surface forming a wet front rather than in streams and rivers), the following formula can be used:

$$t_C = 0.37 \cdot (1.1 - c) \cdot \frac{\sqrt{L}}{J^{1/3}}$$

where:

- t_c, time of concentration (h)
- L, length of the main basin river (km)
- J, the average basin slope (m/m)

- c, runoff coefficient

2.2.1.2.6 Giandotti

This formula has been derived from experiments conducted in large basins with total area equal or greater than 170 km^2.

$$t_C = \frac{4\sqrt{A} + 1.5L}{0.8 \cdot \sqrt{H_M - H_E}}$$

where:

- t_c, time of concentration (h)
- L, length of the main basin river (km)
- A, the total area of the basin (km^2)
- H_M, the mean elevation of the basin (m)
- H_E, the elevation at the basin's outlet (m)

2.2.1.2.7 Kerby / Hathaway

If Manning's friction coefficient is known for a specific water basin, then the following equation introduced Kerby and Hathaway can be used:

$$t_C = 0.62 \cdot \frac{(L \cdot n)^{0.47}}{J^{0.235}}$$

where:

- t_c, time of concentration (h)
- L, length of the main basin river (km)
- J, the average basin slope (m/m)
- n, Manning's friction coefficient

2.2.1.2.8 Kirpich

This method comes in two variations. The first is very common in Greece. The two variations yield different results, since the PA variation is valid only for basins with very small average slopes:

$$t_C = 0.066 \cdot \frac{L^{0.77}}{J^{0.385}}$$

PA: $$t_C = 0.011 \cdot \frac{L^{0.77}}{J^{0.5}}$$

where:

- t_c, time of concentration (h)
- L, length of the main basin river (km)
- J, the average basin slope (m/m)

2.2.1.2.9 TR55 P/U

From the US TR 55 method, two variations are given in order to calculate the time of concentration: method P can be used in paved surfaces while method U can be used in undeveloped surfaces. For paved surfaces the following equation yields the time of concentration:

$$t_C = 0.042 \cdot \frac{L}{J^2}$$

while for unpaved surfaces the following equation is used:

$$t_C = 0.052 \cdot \frac{L}{J^2}$$

where:

- t_c, time of concentration (h)
- L, length of the main basin river (km)
- J, the average basin slope (m/m)

2.2.1.2.10 SCS Lag

This method was developed by U.S. Soil Conservation Service. The time of concentration based on the SCS curve number (CN - Curve Number), the length of the main river and the average basin slope:

$$t_C = 0.057 \frac{L^{0.8}\left(\frac{1000}{CN} - 9\right)^{0.7}}{\sqrt{J}}$$

where:

- t_c, time of concentration (h)
- L, length of the main basin river (km)
- J, the average basin slope (m/m)
- CN, the SCS curve number

The curve number (CN) depends on the soil's type, its status and its imperviousness. It is usually taken from tables integrated in the program.

2.2.1.2.11 California Culverts Practice

This formula was proposed by U.S. California Culverts Practice, and is similar to Kirpich's equation. The only difference is that the average slope J has been substituted by the L/DH ratio. This equation was developed in 1942 for small basins for the state of California:

$$t_C = 0.95 \frac{L^{1.155}}{\Delta H^{0.385}}$$

where:

- t_c, time of concentration (h)
- L, length of the main basin river (km)
- $DH=H_{up}-H_E$ (m)
- H_{UP}, the elevation of the most upstream point of the main river (m)
- H_E, the elevation of the basin's outlet (m)

2.2.1.2.12 Izzard

Izzard equation was developed in 1946 based on laboratory experiments from the American Bureau of Public Roads. It concerns areas covered with roads or vegetation without a developed hydrographic network. The equation is valid only for laminar flow where iL<3.87.

$$t_C = 3.46 \frac{(0.0007i + c_r)L^{1/3}}{J^{1/3} i^{2/3}}$$

where:

- t_c, time of concentration (h)
- L, length of the main basin river (km)
- J, the average basin slope (m/m)
- i, rainfall intensity (mm/h)
- c_r, Izzard's deceleration coefficient (-)

This equation is solved by trial-and-error, since the time of concentration appears in both sides of the equation. Common values of Izzard's deceleration coefficient are available through the program's database.

2.2.1.2.13 Kinematic Wave

Kinematic wave's equation was proposed by several researches during 1965-1975. It was developed after processing experimental data using equations derived from kinematic wave's theory for surface flow over developed areas.

$$t_C = 6.92 \frac{L^{0.6} n^{0.6}}{(c \cdot i)^{0.4} J^{0.3}}$$

where:

- t_c, time of concentration (h)
- L, length of the main basin river (km)
- J, the average basin slope (m/m)
- c, runoff coefficient
- i, rainfall intensity (mm/h)

- n, Manning's friction coefficient

The aforementioned equation is solved by trial-and-error, since the time of concentration appears in both sides of the equation.

2.2.1.3 Correction coefficient

2.2.1.3.1 Fornari

The reduction factor for the conversion of point rainfall to areal depends, according to Fornari, on the total area of the basin and the duration of the critical storm. It is given by the following equation:

$$f = \frac{1}{\left(1 + 0.0015 \frac{A}{t_c^{0.2}}\right)}$$

where:

- f, the reduction coefficient (-)
- A, the total area of the basin (km^2)
- t_c, time of concentration (h)

2.2.1.3.2 TP29

Leclerc and Scahhe (1972) quantified the TP-29 results using the following best-fit equation in order to estimate the value of the reduction coefficient:

$$f = 1 - \exp\left(-1.1 t_c^{0.25}\right) + \exp\left(-1.1 t_c^{0.25} - 0.25A\right)$$

where:

- f, the reduction coefficient (-)
- A, the total area of the basin (km^2)
- t_c, time of concentration (h)

2.2.2 Empirical formulas

2.2.2.1 Forti

The maximum flood flow depends only on the basin's area according to Forti's empirical formula:

$$Q = 2.35A\left(\frac{500}{A + 125} + 0.5\right)$$

Where:

- Q, the maximum flood flow (m^3/s)
- A, the total area of the basin (km^2)

2.2.2.2 Fanning

Fanning's empirical method can be applied only to basins with total area greater than 50 km^2. According to this method, the flood flow depends only on the total basin area:

$$Q = 10.26A^{0.625}$$

Where:

- Q, the maximum flood flow (m^3/s)
- A, the total area of the basin (km^2)

2.2.2.3 Fuller

Fuller's empirical method calculates the maximum flood flow in a basin as a function of its total area and the return period of the critical storm:

$$Q = 1.8A^{0.8}(1+0.8\log T)\left(1+\frac{2.66}{A^{0.3}}\right)$$

Where:

- Q, the maximum flood flow (m^3/s)
- A, the total area of the basin (km^2)
- T, the return period of the critical storm (years)

2.2.2.4 Kuichling

Kuichling's empirical formula is valid only in small and average basins with a total area up to 250 km^2. According to this method, the maximum flood flow depends only on the total basin area:

$$Q = A\left(\frac{708}{A+324}+0.164\right)$$

Where:

- Q, the maximum flood flow (m^3/s)
- A, the total area of the basin (km^2)

2.2.2.5 Iskovski

Iskovski's method is regarded as more complex and accurate compared to the other empirical methods. According to Iskovski, the maximum flood flow is calculated using the following equation:

$$Q = A \cdot H_{AN} \cdot TPG \cdot ST$$

Where:

- Q, the maximum flood flow (m^3/s)
- A, the total area of the basin (km^2)
- H_{AN}, the annual rainfall height in the basin (m)
- TPG, topography coefficient (-)
- ST, Iskovski's coefficient

The topography coefficient TPG is a function of the basin's topography and the ground type. Values of this coefficient can be found in the embedded database.

Iskovski's coefficient is given from graphs as a function of the basin's area.

2.2.3 Synthetic unit hydrograph

2.2.3.1 SCS Dimensionless Hydrograph

The SCS dimensionless hydrograph is a synthetic unit hydrograph in which the discharge is expressed by the ratio of discharge q to peak discharge q_p and the time by the ratio of time t to the time of rise of the unit hydrograph, Tp. Given the peak discharge and lag time for the duration of excess rainfall, the unit hydrograph can be estimated from the synthetic dimensionless hydrograph for the given basin. The values of q_p and Tp may be estimated using a simplified model of a triangular unit hydrograph.

From a review of a large number of unit hydrographs, the Soil Conservation Service suggests the time of recession may be approximated as 1.67Tp. As the area under the unit hydrograph should be equal to a direct runoff of 1 cm, it can be shown that:

$$q_p = \frac{2.08 \cdot A}{T_p}$$

where A is the drainage area in km^2.

Further, a study of unit hydrographs of many large and small rural watersheds indicates that the basin lag tp~0.6Tc, where Tc is the time of concentration of the watershed. Time of rise Tp can be expressed in terms of lag time tp and the duration of effective rainfall tr:

$$T_p = \frac{t_r}{2} + t_p$$

2.2.3.2 Snyder synthetic hydrograph

Snyder defined a standard unit hydrograph as one whose rainfall duration t_r is related to the basin lag t_p by:

$$t_p = 5.5 t_r$$

For a standard unit hydrograph he found that:

1. The basin lag is:

$$t_p = 0.75 \cdot C_t \cdot \left(L \cdot L_C\right)^{0.3}$$

where tp is in hours, L is the length of the main stream in km from the outlet to the upstream divide, L_C is the distance in km from the outlet to a point on the stream nearest the centroid of the watershed area and C_t is a coefficient derived from gaged watersheds in the same region.

2. The peak discharge per unit drainage area in $m^3/s \times km^2$ of the standard unit hydrograph is:

$$q_p = \frac{2.75 \cdot C_P}{t_p}$$

Where C_P is a coefficient derived from gaged watersheds in the same region.

To compute C_t and C_P for a gaged watershed, the values of L and L_C are measured from the basin map. From a derived unit hydrograph of the watershed are obtained values of its effective duration t_R in hours, its basin lag t_{PR} in hours, and its peak discharge per unit drainage area, q_{PR}, in $m^3/s \times km^2$. If $t_{PR} = 5.5\ t_R$, then $t_R = t_r$, $t_{PR} = t_P$, and $q_{PR} = q_P$ and C_t and C_P are computed by the aforementioned equations.

2.3 Routing in rivers

2.3.1 Muskingum

Muskingum method's is the simplest form of flood routing. It is based on the continuity equation and on the recursive relation between storage and flow rate at any stream or river. The storage capacity is considered to be the sum of a prismatic and a wedge component:

$$S = K \cdot Q + K \cdot [I - Q] \cdot x$$

where:

- S, the (temporary) storage in the stream (m^3)
- Q, stream outflow (m^3/s)
- I, stream inflow (m^3/s)
- x, a dimensionless parameter
- K, a time parameter (s)

If the continuity equation is taken into account, the change in storage is equal to the difference of inflow and outflow. The second equation is the formed:

$$\frac{\Delta S}{\Delta t} = I - Q$$

where:

- DS, the change in storage (m^3)
- Q, stream outflow (m^3/s)
- I, stream inflow (m^3/s)
- Dt, the time step of the calculations (s)

Parameter K expresses the time needed for a flood wave to travel the distance from the upstream control section (where the inflow takes place) to the downstream control section (where the outflow occurs). Parameter x quantifies the peak flood attenuation and it varies from 0 to 0.5. When x=0.5, there is no attenuation, that is the upstream maximum flood is equal to the downstream maximum flood. On the other hand, while x decreases towards 0, the routing phenomenon will be stronger, that is, the attenuation of the maximum flood wave will be higher. The practical problem lies in the estimation of the two parameters, K and x. A common workaround is to work backwards: if the inflow and outflow hydrographs are known, using the aforementioned equations one can compute the exact values of the two parameters.

The numerical scheme for the application of the Muskingum method dictates that the previous equation must be re-written using finite differences:

$$\frac{S_{j+1} - S_j}{\Delta t} = \frac{I_{j+1} + I_j}{2} - \frac{Q_{j+1} + Q_j}{2}$$

But according to the first equation, the change in storage may be written as:

$$S_{j+1} - S_j = K \cdot \left\{ \left[x \cdot I_{j+1} + (1-x) \cdot Q_{j+1} \right] - \left[x \cdot I_j + (1-x) \cdot Q_j \right] \right\}$$

Combining the above two equations:

$$\frac{K \cdot \left\{ \left[x \cdot I_{j+1} + (1-x) \cdot Q_{j+1} \right] - \left[x \cdot I_j + (1-x) \cdot Q_j \right] \right\}}{\Delta t} = \frac{I_{j+1} + I_j}{2} - \frac{Q_{j+1} + Q_j}{2}$$

If the last equation is re-arranged so that Q_{j+1} is the only term in its left side (which happens to be the one unknown term of the equation), then the following formula is obtained:

$$Q_{j+1} = \frac{-K \cdot x + 0.5 \cdot \Delta t}{K - K \cdot x + 0.5 \cdot \Delta t} \cdot I_{j+1} + \frac{K \cdot x + 0.5 \cdot \Delta t}{K - K \cdot x + 0.5 \cdot \Delta t} \cdot I_j + \frac{K - K \cdot x - 0.5 \cdot \Delta t}{K - K \cdot x + 0.5 \cdot \Delta t} \cdot Q_j$$

For simplicity reasons, the equation above can be written as:

$$Q_{j+1} = C_0 \cdot I_{j+1} + C_1 \cdot I_j + C_2 \cdot Q_j$$

$$C_0 = \frac{-K \cdot x + 0.5 \cdot \Delta t}{K - K \cdot x + 0.5 \cdot \Delta t}$$

$$C_1 = \frac{K \cdot x + 0.5 \cdot \Delta t}{K - K \cdot x + 0.5 \cdot \Delta t}$$
$$C_2 = \frac{K - K \cdot x - 0.5 \cdot \Delta t}{K - K \cdot x + 0.5 \cdot \Delta t}$$

From the set of equations, one can deduct that the sum of the auxiliary coefficients C_0, C_1 and C_2 is equal to unity. The first from the set of four equations is recursive. For a given time step, the y-coordinate of the outflow hydrograph is equal to the sum of: a) y-coordinate of the inflow hydrograph (this time step and the previous time step) and b) the outflow's previous value. These three coordinates are weighted by using the auxiliary coefficients.

2.3.2 Muskingum - Cunge

Cunge used the finite differences numerical scheme to convert the Muskingum equations into a wave diffusion equation. In the Muskingum – Cunge method, both inflow and outflow are represented by the letter Q and the only way to tell them apart is by their indicators Q_i the inflow and Q_{i+1} the outflow. The continuity equation is combined with Muskingum's storage equation:

$$K \cdot \frac{d}{dt}\left[x \cdot I + (1-x) \cdot Q\right] = I - Q$$

and by inserting the new notation:

$$\frac{K}{\Delta t}\left[x \cdot Q_i^{t+1} + (1-x) \cdot Q_{i+1}^{t+1} - x \cdot Q_i^t - (1-x) \cdot Q_{i+1}^t\right] =$$
$$= \frac{1}{2} \cdot \left(Q_i^{t+1} - Q_{i+1}^{t+1} + Q_i^t - Q_{i+1}^t\right)$$

Solving the above equation for $Q_{i+1}^{(t+1)}$ the final product is derived:

$$Q_{i+1}^{t+1} = \frac{\Delta t/K - 2 \cdot x}{2 \cdot (1-x) + \Delta t/K} \cdot Q_i^{t+1} + \frac{\Delta t/K + 2 \cdot x}{2 \cdot (1-x) + \Delta t/K} \cdot Q_i^t +$$
$$+ \frac{2 \cdot (1-x) - \Delta t/K}{2 \cdot (1-x) + \Delta t/K} \cdot Q_{i+1}^t$$

Same as before, for simplicity reasons, the equation can be re-written as:

$$Q_{i+1}^{t+1} = C_0 \cdot Q_i^{t+1} + C_1 \cdot Q_i^t + C_2 \cdot Q_{i+1}^t$$
$$C_0 = \frac{\Delta t/K - 2 \cdot x}{2 \cdot (1-x) + \Delta t/K}$$
$$C_1 = \frac{\Delta t/K + 2 \cdot x}{2 \cdot (1-x) + \Delta t/K}$$
$$C_2 = \frac{2 \cdot (1-x) - \Delta t/K}{2 \cdot (1-x) + \Delta t/K}$$

It is obvious that the sum of the three auxiliary coefficients C_0, C_1 and C_2 is equal to unity. Parameter x is computed using the following equation:

$$x = \frac{1}{2} \cdot \left(1 - \frac{q}{J \cdot c \cdot (\Delta x)}\right)$$

where:

- x, Muskingum – Cunge dimensionless coefficient
- J, the average bottom slope (m/m)
- q, the runoff per width unit (m^2/s)
- c, celerity - flood wave velocity (m/s)
- Dx, the spatial step (m)

The temporal coefficient K is given by the following equation:

$$K = \frac{\Delta x}{c}$$

To calculate the flood wave celerity (if no measurements exist), the uniform flow equation can be applied:

$$c = n \cdot V$$

where:

- c, celerity - flood wave velocity (m/s)
- V, the average velocity (m/s)
- n, a parameter that can be derived from the equation $Q=b \cdot A^n$

The wave celerity c, can be computed in a simpler way by applying a friction formula (such as Manning's) in a known cross section geometry. A necessary condition for this, is that the cross section geometry that will be used will remain constant along the river or stream.

2.3.3 SCS Convex

This method was developed by SCS (Soil Conservation Service) and is similar to Muskingum's method and wide spread. In every time step, the volume of the inflow hydrograph ie equal to the volume of the outflow hydrograph. If two consecutive time steps whose distance is Dt are considered and Q_1 and Q_2 the respective outflow values, then the following is true:

$$Q_2 = Q_1 + C_t \left(I_1 - Q_1 \right)$$

The aforementioned equation could be used for flood routing if the value of C_t was known:

$$C_t = \frac{Q_2 - Q_1}{I_1 - Q_1}$$

The value of the unknown coefficient can be approximated as follows:

$$C_t = \frac{\Delta t}{K}$$

The SCS Convex method can be applied only when C_t coefficient varies between 0 and 1. This can be achieved by selecting a time step Dt smaller or equal to the one fifth of the time interval required for peak flood to occur (for the inflow hydrograph). Parameter K is constant and can be approximated using the same techniques that

were introduced by Muskingum. C_t coefficient has approximately twice the value of parameter x described in Muskingum method. An estimate of parameter K can be obtained by dividing the length of the stream segment to the wave celerity. If no other indications or estimates exist regarding C_t coefficient, then the following approximation can be used:

$$C_t = \frac{V}{V+1.7}$$

where V is a velocity for constant and usual flow rate and V+1.7 is the wave celerity in the segment. Velocity units are in feet per second.

Compared to the other flood routing methods, in the Convex SCS method, the calculation of the outflow Q_2 does not depend on the simultaneous inflow I_2. Therefore the method can be used to estimate flood flows in time steps with unknown inflows. This feature may be used as a draft warning system if used in combination with electronic early warning systems, so that the flood downstream can be predicted at least a time step before it takes place.

2.3.4 Kinematic wave

In kinematic wave's routing method, the kinetic and diffusive terms are neglected. The flow is considered steady for momentum conservation and unsteady as flow depth can increase and decrease. The simplified momentum equation is:

$$\frac{\partial Q}{\partial x} + \frac{\partial A}{\partial t} = 0$$

According to the aforementioned equation, there is a unique relationship between flow and depth, i.e. Q=Q(y) or Q=Q(A). This relationship can be expressed as:

$$\frac{\partial Q}{\partial t} + c\frac{\partial Q}{\partial x} = 0$$

In which c represents the wave celerity and is given by the following formula:

$$c = \frac{dQ}{dA}$$

If the wave celerity c remains constant while the flood wave propagates from upstream to downstream, then the maximum flood wave remains constant or in other words there is no attenuation. Therefore it can be said that if c is constant, the shape of the hydrograph does not change.

2.4 Routing in reservoirs

2.4.1 Storage Indicator

Same with the flood routing in rivers, during flood routing in reservoirs the momentum and continuity equations are both true. However in reservoirs there is a unique relationship between storage and outflow. Outflow is possible through special devices such as weirs, which can be computed given the weir's geometry and the head.

Flood routing is of interest when the water level in the reservoir is equal the elevation of its crest, or when the reservoir is completely filled with water to its capacity. This is also the worst case scenario, since if the reservoir is empty, then the inflow will just increase the stored water and the outflow will be zero:

$$\frac{\Delta S}{\Delta t} = I$$

If the water level in the reservoir exceeds the elevation of the weir crest, then the outflow is non-zero, therefore the above equation is false. Puls was the first to express the continuity equation using finite differences:

$$\frac{S_{j+1} - S_j}{\Delta t} = \frac{I_j + I_{j+1}}{2} - \frac{Q_j + Q_{j+1}}{2}$$

$$\left(\frac{2 \cdot S_{j+1}}{\Delta t} + Q_{j+1}\right) = \left(I_{j+1} + I_j\right) + \left(\frac{2 \cdot S_j}{\Delta t} - Q_j\right)$$

The aforementioned equation is recursive and can be used to compute the outflow when the relationship between storage and outflow is known. Since storage S is by default uniquely dependent on the water level inside the reservoir and the outflow through a weir is computed as follows:

$$Q = c \cdot L \cdot \left(H - H_0\right)^X$$

where:

- Q,outflow through the weir (m^3/s)
- L, the weir's width (m)
- H_0, crest elevation (m)
- H, water level elevation in the reservoir (m)
- X, hydraulic head coefficient (usually equal to 3/2)
- c, flow coefficient (usually equal to 3 $m^{1/2}/s$. The flow units depend on the parameter X.)

Hence, the two variables (outflow and storage) depend on the water level, so they have a unique relationship among themselves.

Chapter

3 File

3.1 File menu

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

- New project
- Open project
- Save project
- Save project as
- Import
  - Solution from file
- Export
  - Solution to file
- Print setup
- Print
- Print to
  - Print to file
  - Print to Word
  - Print to Word (Formatted)
  - Print to Excel
- Exit

3.2 New project

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

To create a new project:

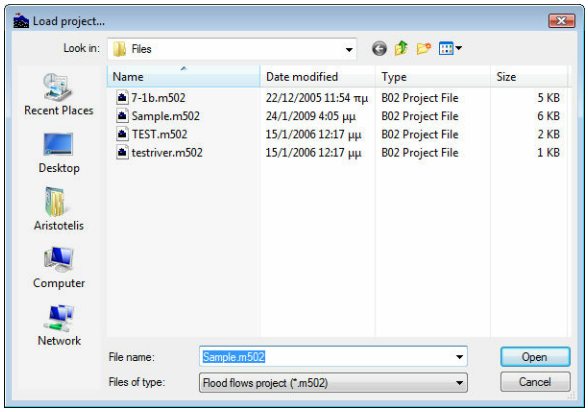
1. Select **New Project** from the **File** menu.
2. If a project is already loaded and changes have been made, a warning message will appear that asks the user whether to save the changes or not.
3. The current project is erased and a new project is started.

3.3 Open project

With this option, an existing project is loaded. The project may be 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 "Flood flows project" with the extension .m20.
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\FloodFlows

Supported file types

- **M20** (Flood flows project): Files created by version 2012 and 2013 of Flood flows.
- **M502** (Flood flows project): Files created by versions 2011, 2010, 2009, 2008, 2007 and 5.0 of Flood flows.
- **M14** (Flood flows v2.xx - v4.xx): Files created by versions 2.x, 3.x or 4.x of Flood flows. This program had been updated to version 5.0, 2007, 2008, 2009 and 2010.
- **MB6** (Flood routing v2.xx - v4.xx): Files created by versions 2.x, 3.x or 4.x of Flood routing. This program had been updated to version 5.0 but now it is merged with Hydraulic solver 2007 or later.
- **BCK** (Backup files): If you have selected from program options the creation of backup copy when a file is loaded, then the file can be loaded by selecting Backup files (*.bck) from the Files of type drop-down list.
- ***.*** (All files): Displays all files in the current folder.

Backwards compatibility

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

NOTE: If 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.

3.4 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:

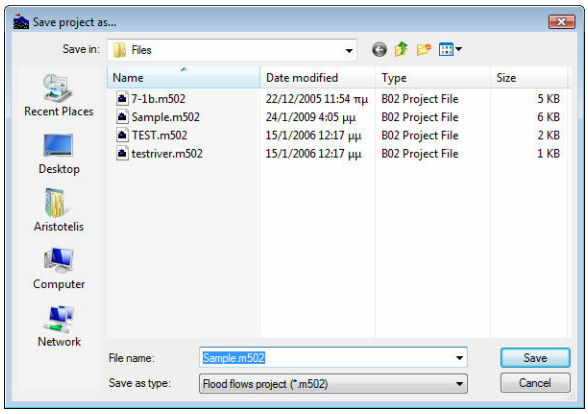
1. Select **Save project** from the **File** menu.
2. If the location of the file is already set, the project is saved to this file without any messages. If the filename is not set, a dialog box will appear that allows the selection of the filename and path.

3.5 Save project as

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

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

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



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

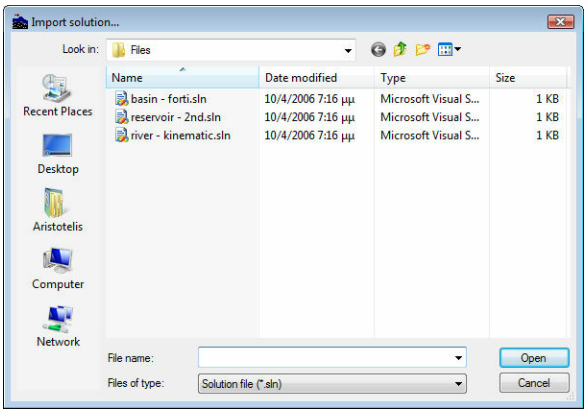
3.6 Import

3.6.1 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 "Solution 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.

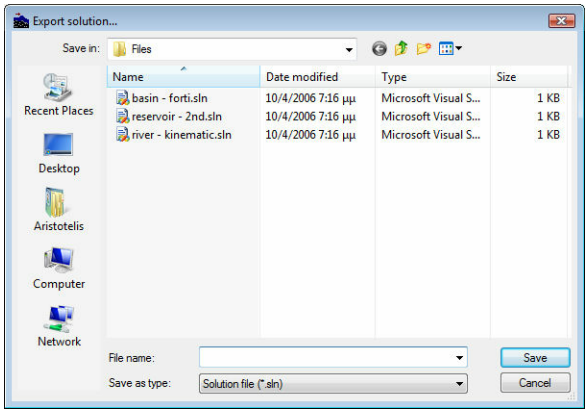
3.7 Export

3.7.1 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.

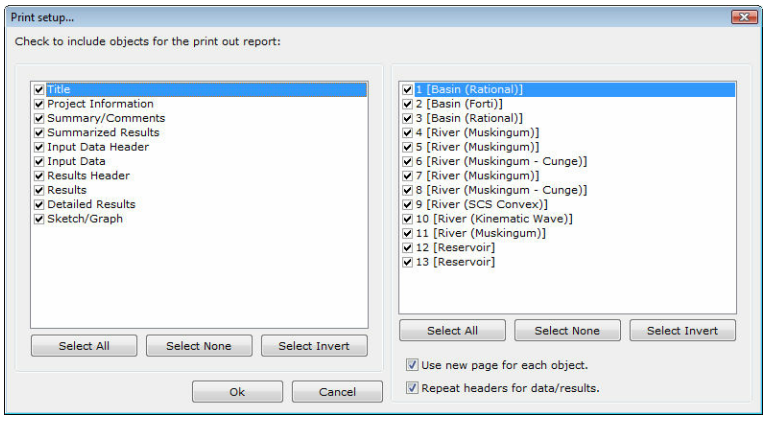
NOTE: A solution file contains a single solution only. The solution currently selected in the list of the main form will be saved in the file.

3.8 Print setup

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

To modify the print setup:

1. Select **Print setup** from the **File** menu.
2. Select the **sections** (Title, Project information etc) that will be 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.

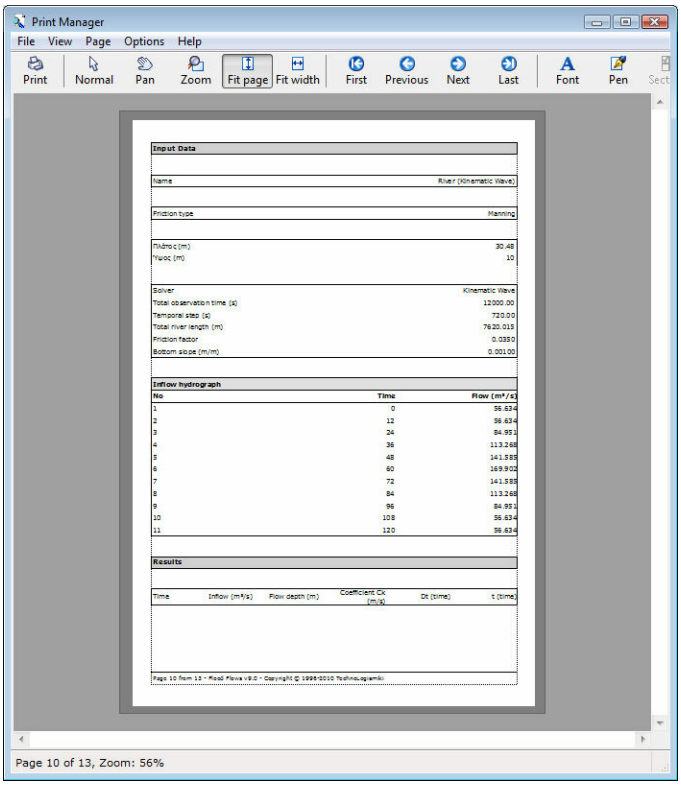
3.9 Print

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

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

To create a report:

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



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

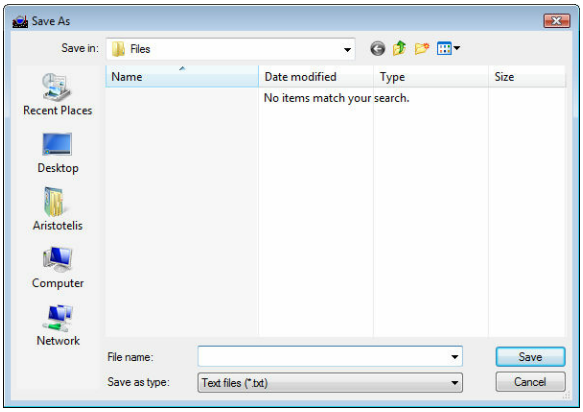
3.10 Print to

3.10.1 Print to File

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

To print to a text file:

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



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

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

3.10.2 Print to Word

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

To print the report to a Microsoft Word file:

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

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

3.10.3 Print to Word (Formatted)

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

To print the report to a formatted Microsoft Word file:

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

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

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

3.10.4 Print to Excel

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

To print the report to a Microsoft Excel file:

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

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

3.11 Exit

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

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

depending on what you have selected in General preferences.

To exit the program:

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

ignore them.
3. The program is terminated.

Chapter

4 Data

4.1 Data menu

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

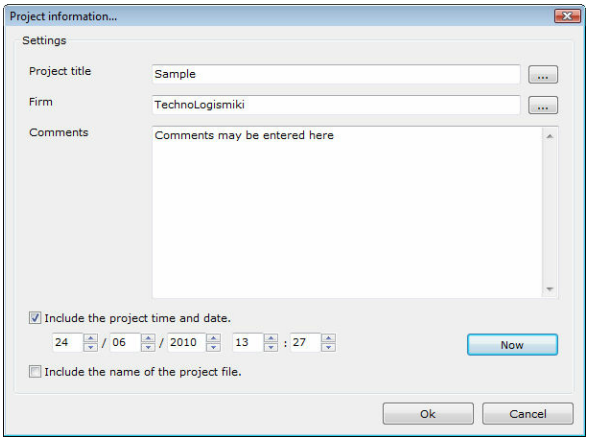
- Project info
- Undo
- Redo
- Add solution
- Remove solution
- Basins
  - Basin properties
  - IDF curve
- Rivers
  - River properties
  - Inflow hydrograph
  - Friction calculations
- Reservoirs
  - Reservoir properties
  - Inflow hydrograph
  - Storage - outflow graph
- Copy data from solution
- Options
  - General preferences
  - Cell editing
  - Customize toolbar

4.2 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.



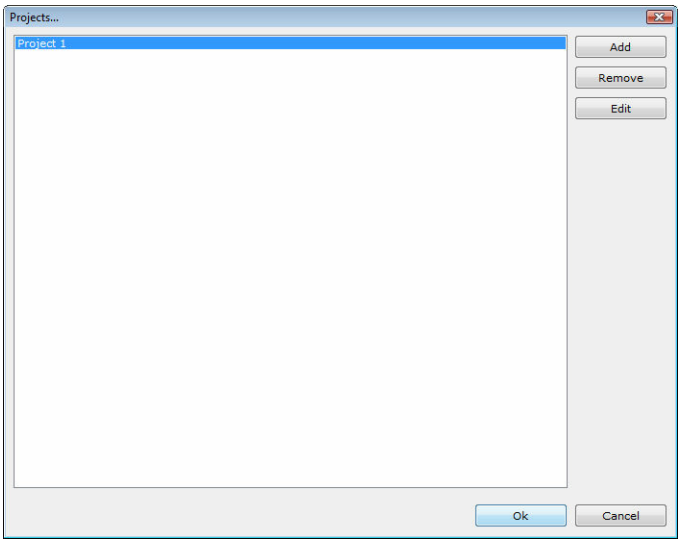
By selecting the buttons with the ellipses (...) next to the project title and author, you can access the corresponding databases.

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.

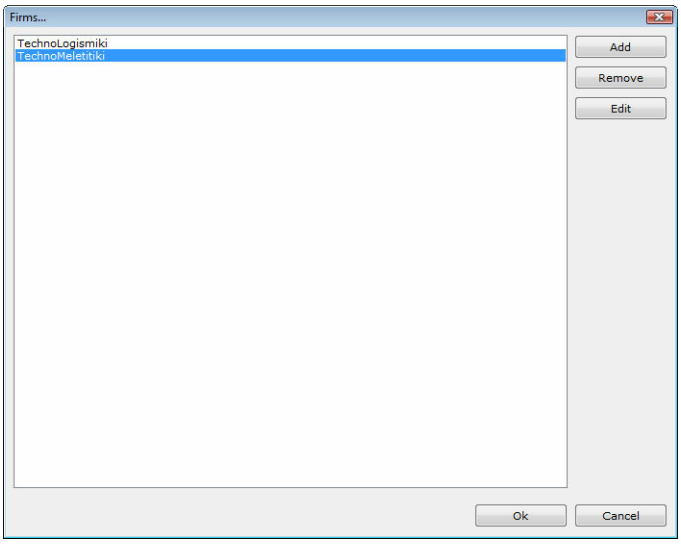


Author database

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.



4.3 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.

4.4 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.

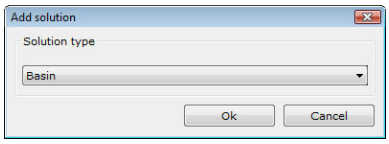
It is possible to redo more than one changes that were previously undone by following the steps described above. The number of actions that are kept in memory and may be undone or redone is 20 by default. This means that the program is able to keep track of up to 20 successive changes that are undone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

4.5 Add solution

With this option, you can add a new solution.

To add a solution:

1. Select **Add solution** from the **Data** menu.
2. Select the **type** of the problem (see below)
3. Select **Ok** to create a new solution. The solutions are grouped; the first group are the **Basins**, the second group are the **Rivers** and the last group are the **Reservoirs**. The new solution is appended at the end of the corresponding group. Select **Cancel** to close the dialog box without adding a solution.

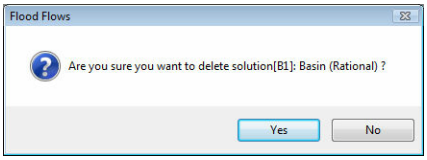


4.6 Remove solution

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.



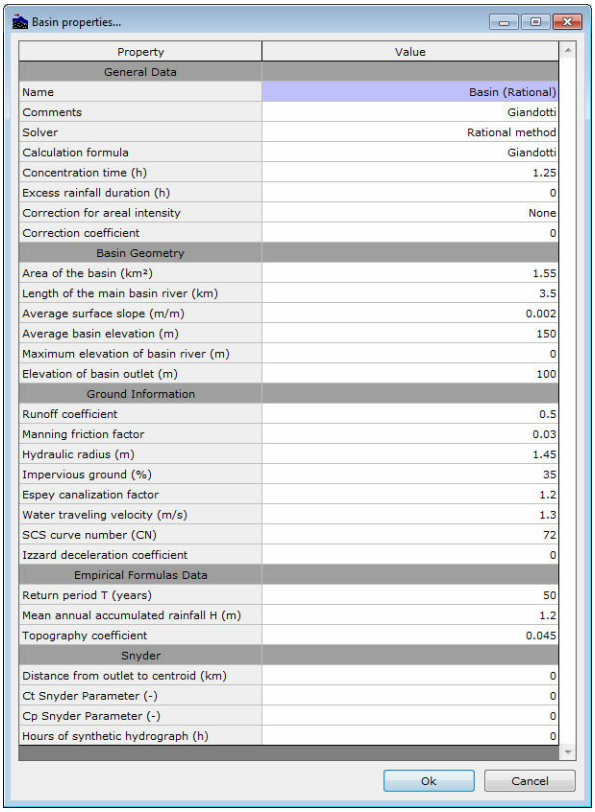
4.7 Basins

4.7.1 Basin properties

With this option, you can display and modify the properties of a basin.

To define the properties of a basin:

1. Select **Basin** from the **Data** menu.
2. Select **Basin properties** from the **Basins** menu.
3. Enter the data as described below.
4. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



NOTE: Not all data are required, as these depend on the calculation formula.

General data

- **Name**: enter the name of the basin.
- **Comments**: (optional) enter comments.

- **Solver**:
  - **Rational method**: the calculations will be performed with the Rational method.
  - **Fuller**: the area and return period are required for the Fuller empirical formula.
  - **Forti**: the area is required for the Forti empirical formula.
  - **Fanning**: the area is required for the Fanning empirical formula.
  - **Kuichling**: the area is required for the Kuichling empirical formula.
  - **Iskovski**: the area, the mean annual accumulated rainfall and the topography coefficient are required for the Iskovski empirical formula.
  - **SCS unit hydrograph**: the area, the excess rainfall duration and data for the concentration time.
  - **Snyder synthetic hydrograph**: the area, length of main basin river and all data in Snyder's category.
- **Concentration time**: if the calculations are performed with the Rational method, then the concentration time can be calculated with one of the following formulas:
  - **Direct value**: if the concentration time is known, it can be entered directly in the **Concentration time** field.
  - **Carter**: the Carter formula (Washington DC, USA) requires the length and the mean slope of the basin.
  - **Eagleson**: the Eagleson formula requires the length, mean slope, hydraulic radius and the Manning friction coefficient.
  - **Espey/Winslow**: the Espey and Winslow formula requires the length, mean slope, the Espey coefficient and the imperviousness of the area.
  - **Federal Aviation Administration**: the FAA formula requires the length and mean slope of the basin.
  - **Giandotti**: the Giandotti formula requires the length, mean slope, mean elevation and the elevation at the basin outlet.
  - **Kerby/Hathaway**: the Kerby and Hathaway formula requires the length, mean slope, and the Manning friction coefficient.
  - **Kirpich/Kirpich (PA)**: the well known Kirpich method and a variation for fields. The length and mean slope of the basin are required.
  - **Length and velocity**: the length and mean velocity are required. The concentration time is given by the result of their division.
  - **SCS Lag**: the SCS lag formula requires the length, mean slope and CN curve number.
  - **TR-55U/TR-55P**: the TR-55U/TR-55P formula requires the length and mean slope of the basin.
  - **California Culverts Practice**: the length, the maximum elevation of the main river and the elevation at the basin outlet are required.
  - **Izzard**: the length, the average basin slope and Izzard's' deceleration coefficient are required.
  - **Kinematic wave**: Manning friction coefficient, the length, the runoff coefficient and the average basin slope are required.
- **Concentration time**: In case you select **Rational method** and **Direct value,** enter the direct value in this field in h.
- **Excess rainfall duration (h)**: Used in SCS unit hydrograph method, the total excess rainfall duration in h.
- **Correction for areal intensity**: If the rational method is used, a correction factor can be optionally used in order to reduce the point rainfall intensity when converted to areal. One of the following cases can be applied:
  - **None**: the point intensity is not reduced when converted to areal.
  - **Fornari**: the Fornari method is applied to estimate the reduction factor.

  - **TP-29**: the reduction factor is estimated using the TP-29 method.
  - **Custom**: a user-supplied reduction factor is entered below.
- **Correction coefficient**: if the custom selection is chosen in the correction for areal intensity field, then a user supplied correction coefficient is expected in this field. The factor must be smaller than unity.

Basin geometry

- **Area of the basin**: enter the area of the basin in km^2.
- **Length of the main basin river**: enter the length of the main basin river in km. If the basin does not have a clearly defined main river, then you should enter the appropriate length depending on the formula you will be using e.g. Federal Aviation Administration.
- **Average surface slope**: the mean slope in m/m
- **Average basin elevation**: the mean basin elevation in m.
- **Maximum elevation of basin river**: The maximum elevation of the basin's main river is entered in m.
- **Elevation of basin outlet**: the elevation of the basin outlet in m.

Ground information

- **Runoff coefficient**: enter the runoff coefficient of the basin. If it is not known, it can be estimated based on the characteristics of the basin from the embedded database, by clicking the button with the ellipses (...).
- **Manning friction factor**: enter the Manning friction coefficient of the basin. If it is not known, it can be estimated based on the characteristics of the basin from the embedded database, by clicking the button with the ellipses (...).
- **Hydraulic radius**: enter the hydraulic radius of the main river at peak flow rate in m. You can enter a trial value and confirm it after completing the calculations. The hydraulic calculations can be performed by the Section Solver by TechnoLogismiki.
- **Espey canalization factor**: enter the Espey canalization factor for the basin. If it is not known, it can be estimated based on the characteristics of the basin from the embedded database, by clicking the button with the ellipses (...).
- **Water traveling velocity**: enter the water traveling velocity in m/s.
- **SCS curve number (CN)**: enter the SCS curve number for the basin. If it is not known, it can be estimated based on the characteristics of the basin from the embedded database, by clicking the button with the ellipses (...).
- **Izzard deceleration coefficient**: an estimate of Izzard's deceleration coefficient is entered in this field. If it is not know, then it can be estimated using various physical characteristics of the basin from the embedded database, by clicking the button with the ellipses (...).

Empirical formulas data

- **Return period T**: enter the return period in years (for the Fuller formula).
- **Mean annual accumulated rainfall** : enter the mean annual accumulated rainfall in m (for the Iskovski formula).
- **Topography coefficient**: enter the topography coefficient using the embedded database. This is required for the Iskovski formula. To use the database, click the button with the ellipses (...).

Snyder

- **Distance from outlet to centroid**: enter the total distance from the catchment's outlet to its centroid, measured along the flow path in km.
- **Ct Snyder parameter**: enter the dimensionless value of Snyder's Ct parameter.
- **Cp Snyder parameter**: enter the dimensionless value of Snyder's Cp parameter.
- **Hours of synthetic hydrograph**: the duration of the hydrograph, i.e. for a six-hour hydrograph enter 6.

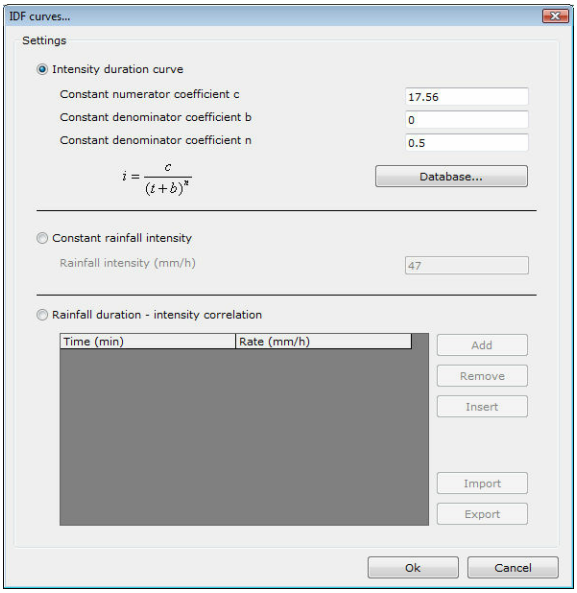
NOTE: This menu is accessible only if the selected solution is a basin-type solution.

4.7.2 IDF curve

With this option, you can enter the intensity of the rainfall. This can be calculated directly from an IDF curve, by linear interpolation from a table containing rainfall duration - intensity data or take a constant value.

To enter the rainfall intensity:

1. Select **Basin** from the **Data** menu.
2. Select **IDF curve** from the **Basin** menu.
3. Make the appropriate selections.
4. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



To use an IDF:

1. Select the **Intensity duration curve** option button.
2. Enter the dimensionless coefficients c, b and n in such a way that when t is entered in hours, the resulting intensity is given in mm/h.
3. Optionally, you can use the embedded IDF database by clicking **Database**.

To use a constant rainfall intensity:

1. Select the **Constant rainfall intensity** option button.
2. Enter the rainfall intensity in mm/h. This value is not depended on the concentration time.

To use a rainfall duration - intensity correlation:

1. Select the **Rainfall duration - intensity** option button.
2. Enter pairs of data i.e. rainfall duration in min and rainfall intensity in mm/h. The program will use linear interpolation to find the appropriate value. You can create the curve manually, using the **Add, Remove** and **Insert** buttons. You can also import a curve from a file.
3. Select **Import** to import data from an IDF or RCV file.
4. Select **Export** to export the current table data to a file.
5. Select **Add** to add a line to the table. Select **Remove** to remove the current line from the table. Select **Insert** to insert a line above the current line of the table.

NOTE: This menu is accessible only if the selected solution is a basin-type solution.

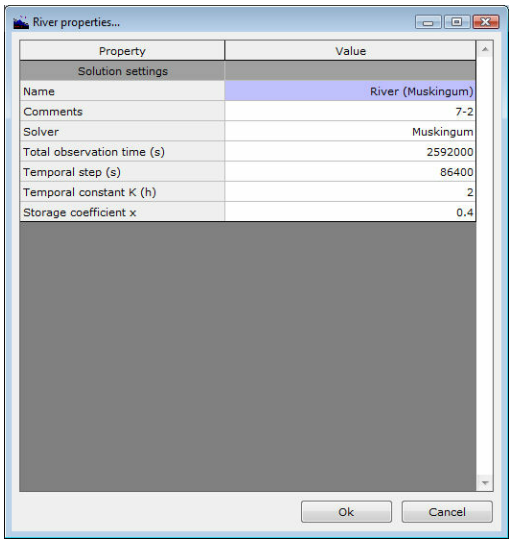
4.8 Rivers

4.8.1 River properties

With this option, you can display and modify the properties of a river.

To define the properties of a river:

1. Select **River** from the **Data** menu.
2. Select **River properties** from the **Rivers** menu.
3. Enter the data as described below.
4. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



NOTE: Not all data are required, as these depend on the solution method.

Common data

- **Name**: enter the name of the river.
- **Comments**: (optional) enter comments.
- **Solver**: select one of Muskingum, Muskingum-Cunge, SCS Convex, Kinematic Wave.
- **Total observation time**: enter the total observation time of the routing phenomenon in s. This must greater than the base time of the inflow hydrograph.
- **Temporal step**: enter the time step in s. In most cases it should be the same with the time step of the inflow hydrograph.

Muskingum method

In addition, the following data are required in the case of Muskingum method:

- **Temporal constant K**: enter the time constant K in h. This constant is related to the time lag between the upstream and downstream peak flow rates.
- **Storage coefficient x**: enter the dimensionless coefficient x. This coefficient takes a value between 0 and 0.5 and is related to the attenuation of the wave. For x=0.5 there is no attenuation whereas for x=0 the attenuation is maximum.

Muskingum-Cunge method

In addition, the following data are required in the case of Muskingum-Cunge method:

- **Total river length**: enter the total river length in m.
- **Wave celerity**: enter the wave celerity in m/s.
- **Bottom slope**: enter the mean slope of the river bed in m/m.
- **Flow rate**: enter the peak flow rate in m^3/s. This is usually set equal to the peak value of the upstream inflow hydrograph. This value is not required when the wave celerity is known.
- **Lateral inflow:** (optional) enter the lateral inflow in m^3/s. This value is used in the calculation of the C3 dimensionless coefficient of the Muskingum-Cunge formula.
- **Friction factor**: enter the friction factor. This is depended on the river characteristics. Click the button with the ellipses (...) to access the friction coefficient database containing the values corresponding to the selected friction formula.
- **Section**: enter a characteristic (constant) river section.

SCS Convex method

In addition, the following data are required in the case of SCS Convex method:

- **Total river length**: enter the total river length in m.
- **Velocity**: enter the flow velocity in m/s. This is not required if Ct or K is known.
- **SCS Ct coefficient**: enter the dimensionless SCS Ct coefficient. This coefficient is the ratio of two consecutive flow discharge values to the difference of inflow and discharge during the same time span. If the coefficient is not known then it is calculated based on K.
- **Temporal constant K**: enter the time constant K in h. This constant is related to the time lag between the upstream and downstream peak flow rates. This is not required when SCS Ct coefficient is known.

Kinematic Wave method

In addition, the following data are required in the case of Kinematic Wave method:

- **Total river length**: enter the total river length in m.
- **Bottom slope**: enter the mean bottom slope in m/m.
- **Friction factor**: enter the friction factor. This is depended on the river characteristics. Click the button with the ellipses (...) to access the friction coefficient database containing the values corresponding to the selected friction formula.
- **Section**: enter a characteristic (constant) river section.

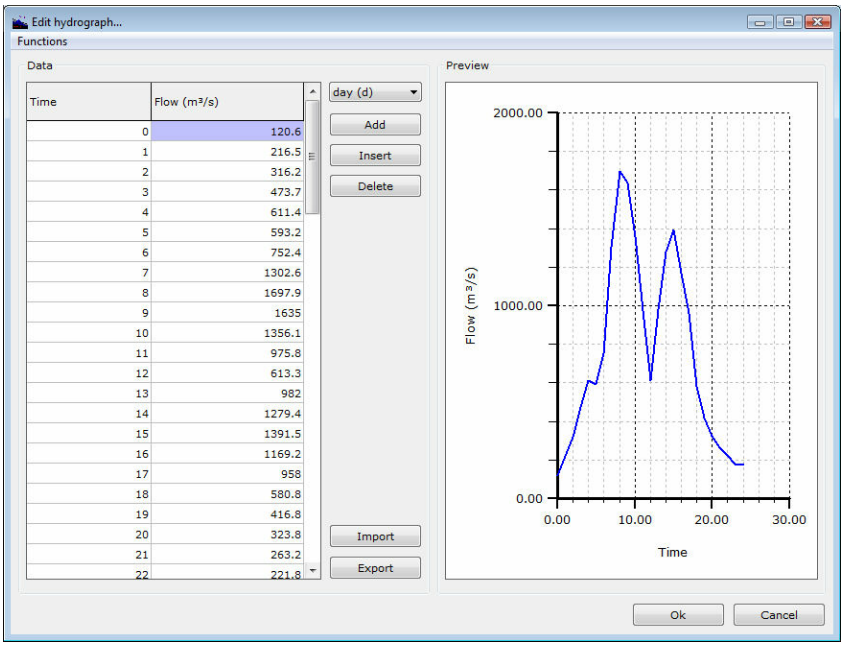
NOTE: This menu is accessible only if the selected solution is a river-type solution.

4.8.2 Inflow hydrograph

With this option, you can define the inflow hydrograph.

To define the inflow hydrograph:

1. Select **Rivers** from the **Data** menu.
2. Select **Inflow hydrograph** from the **Rivers** menu.
3. Enter the hydrograph data as described below.
4. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



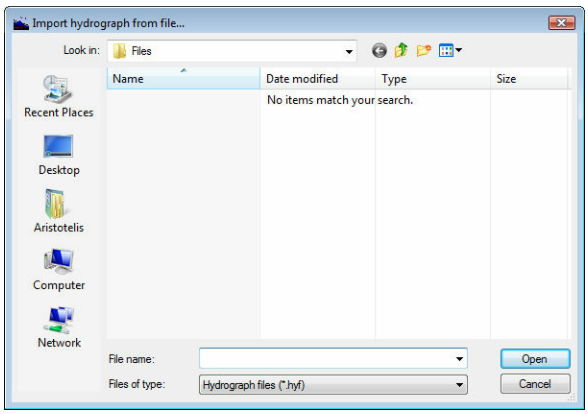
To enter the data directly:

1. Select the appropriate time unit (e.g. s, min, day, hour) from the drop-down list.
2. Click **Add** to add a line at the end of the list.
3. Click **Insert** to insert a line above the currently selected line.
4. Click **Delete** to delete the currently selected line. 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.
5. You can view the current hydrograph in the **Preview** frame.

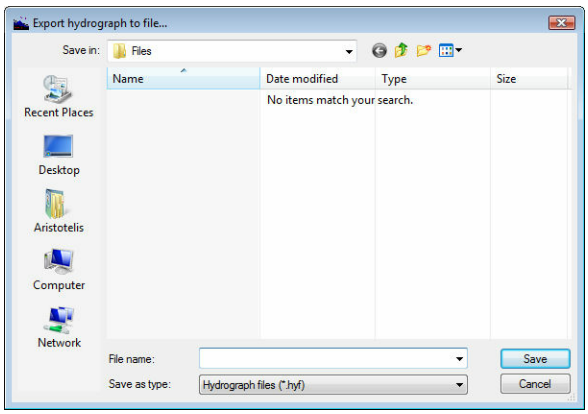
To import a hydrograph from a file:

1. Select **Import** from the **Data** frame.
2. Select the location of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Hydrograph file" with the extension .hyf.
4. Select the file by clicking on it.
5. Select **Open** to import the hydrograph. Select **Cancel** to cancel the operation.



To export a hydrograph to a file:

1. Select **Export** from the **Data** frame.
2. Select the location of the new file.
3. Type the filename in the **File name** text box.
4. Select **Save** to create the solution file with the extension .hyf. Select **Cancel** to cancel the operation.



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.

NOTE: This menu is accessible only if the selected solution is a river-type solution.

4.8.3 Friction calculations

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.

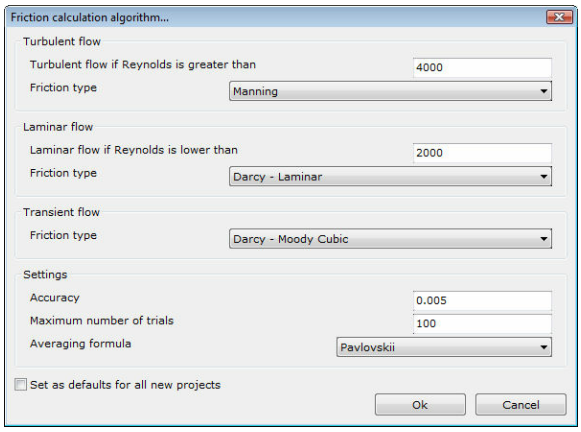
The **Accuracy** and **Maximum number of trials** ensure the stability of the algorithm. The default values are 0.0005 for the accuracy and 1000 for the maximum number of trials. It is recommended that you do not change these values.

The averaging formula may take one of the following values:

- Pavlovskii
- Colebatch

- Horton
- Cox
- Lotter

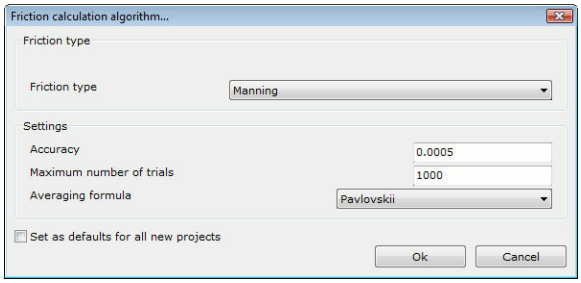
The above formulas are used when there are different friction coefficients within the same section and an average friction coefficient is needed.



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.



NOTE: This menu is accessible only if the selected solution is a river-type solution.

4.9 Reservoirs

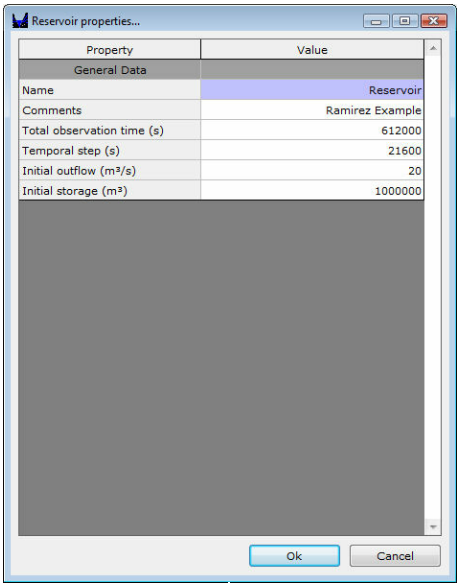
4.9.1 Reservoir properties

With this option, you can display and modify the properties of a reservoir.

To define the properties of a reservoir:

1. Select **Reservoirs** from the **Data** menu.

2. Select **Reservoir properties** from the **Reservoirs** menu.
3. Enter the data as described below.
4. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



General data

- **Name**: enter the name of the reservoir.
- **Comments**: (optional) enter comments.
- **Total observation time**: enter the total observation time in s. This must be greater than the base time of the inflow hydrograph.
- **Temporal step**: enter the time step in s. In most cases it should be the same with the time step of the inflow hydrograph.
- **Initial outflow**: enter the outflow at t=0 in m^3/s.
- **Initial storage**: enter the storage at t=0 in m^3.

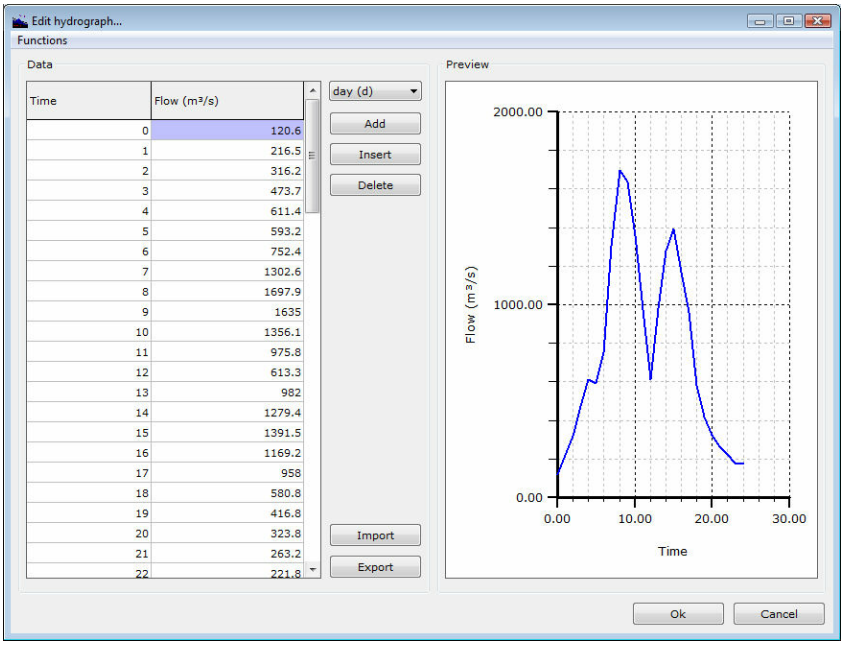
NOTE: This menu is accessible only if the selected solution is a reservoir-type solution.

4.9.2 Inflow hydrograph

With this option, you can define the inflow hydrograph.

To define the inflow hydrograph:

1. Select **Reservoirs** from the **Data** menu.
2. Select **Inflow hydrograph** from the **Reservoirs** menu.
3. Enter the hydrograph data as described below.
4. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

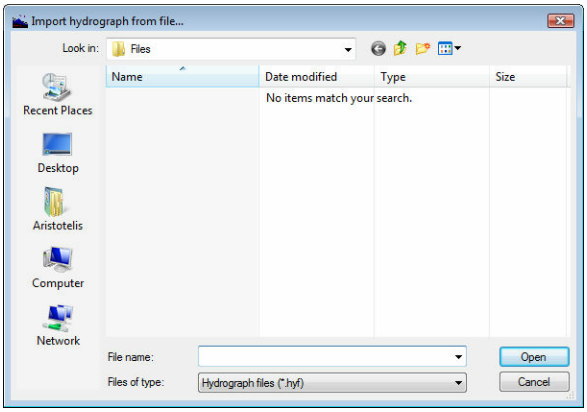


To enter the data directly:

1. Select the appropriate time unit (e.g. s, min, day, hour) from the drop-down list.
2. Click **Add** to add a line at the end of the list.
3. Click **Insert** to insert a line above the currently selected line.
4. Click **Delete** to delete the currently selected line. 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.
5. You can view the current hydrograph in the **Preview** frame.

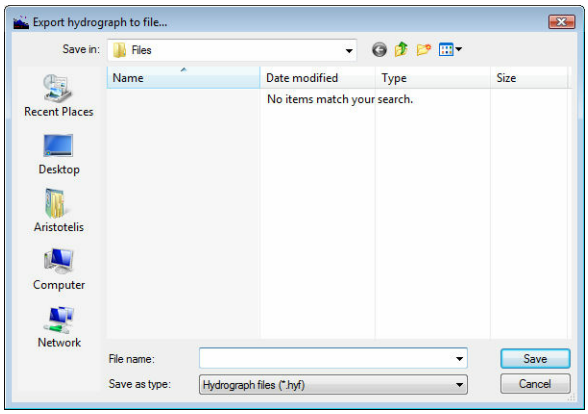
To import a hydrograph from a file:

1. Select **Import** from the **Data** frame.
2. Select the location of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Hydrograph file" with the extension .hyf.
4. Select the file by clicking on it.
5. Select **Open** to import the hydrograph. Select **Cancel** to cancel the operation.



To export a hydrograph to a file:

1. Select **Export** from the **Data** frame.
2. Select the location of the new file.
3. Type the filename in the **File name** text box.
4. Select **Save** to create the solution file with the extension .hyf. Select **Cancel** to cancel the operation.



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.

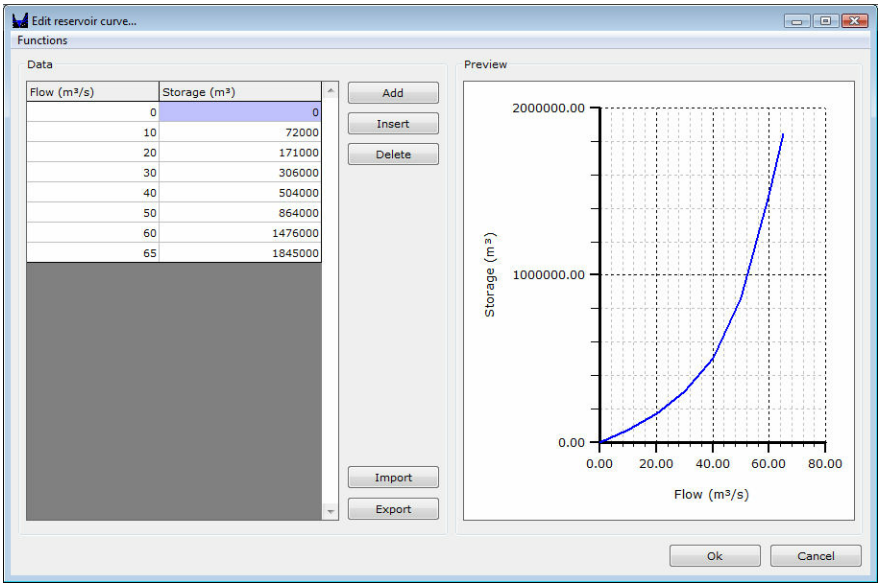
NOTE: This menu is accessible only if the selected solution is a reservoir-type solution.

4.9.3 Storage - outflow graph

With this option, you can define the storage-outflow graph of the reservoir.

To define the storage-outflow graph:

1. Select **Reservoirs** from the **Data** menu.
2. Select **Storage - outflow graph** from the **Reservoirs** menu.
3. Enter the storage-outflow data as described below.
4. Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

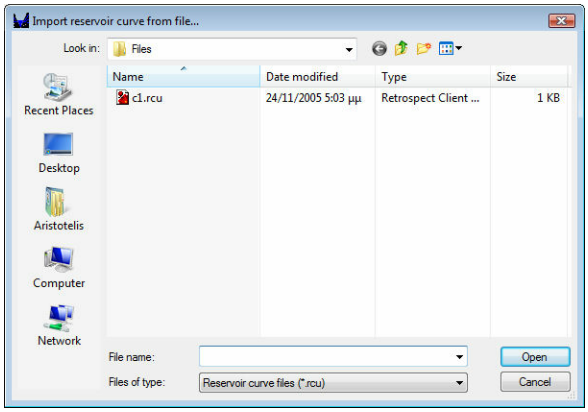


To enter the data directly:

1. Click **Add** to add a line at the end of the list.
2. Click **Insert** to insert a line above the currently selected line.
3. Click **Delete** to delete the currently selected line. 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.
4. You can view the current storage - outflow graph in the **Preview** frame.

To import a storage - outflow graph from a file:

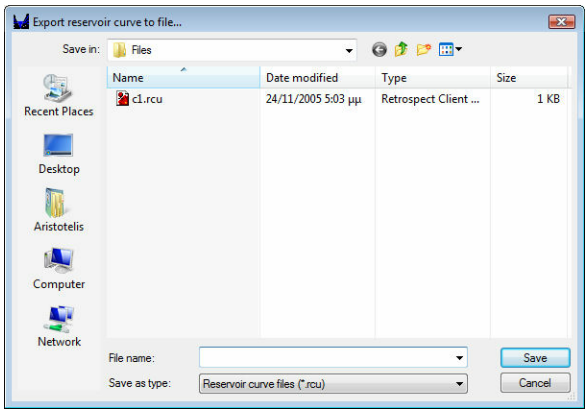
1. Select **Import** from the **Data** frame.
2. Select the location of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Reservoir curve file" with the extension .rcu.
4. Select the file by clicking on it.
5. Select **Open** to import the graph. Select **Cancel** to cancel the operation.



To export a storage - outflow graph to a file:

1. Select **Export** from the **Data** frame.

2. Select the location of the new file.
3. Type the filename in the **File name** text box.
4. Select **Save** to create the solution file with the extension .rcu. Select **Cancel** to cancel the operation.



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.

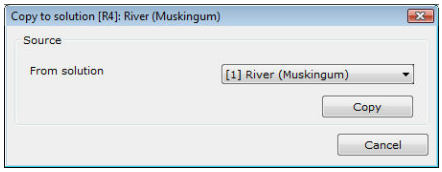
NOTE: This menu is accessible only if the selected solution is a reservoir-type solution.

4.10 Copy data 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.

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 is erased and substituted by the data of the source solution.

4.11 Options

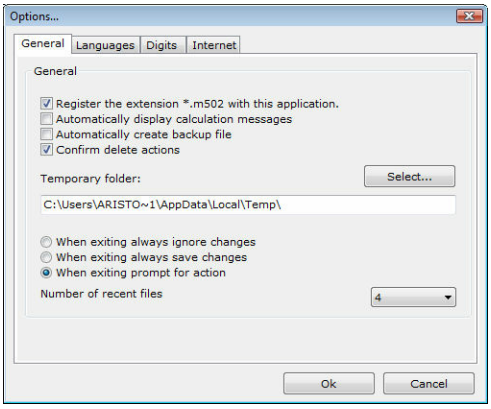
4.11.1 General preferences

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 *.m20 with this application** to associate the extension .m20 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

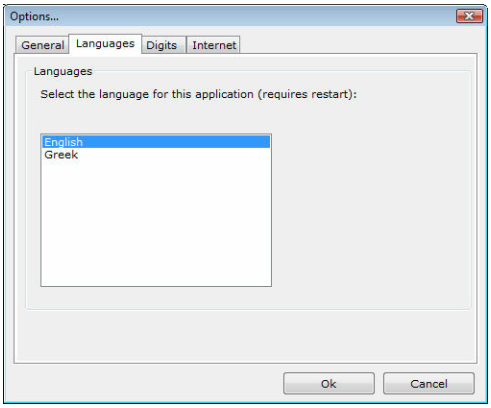
example the deletion of a solution

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.

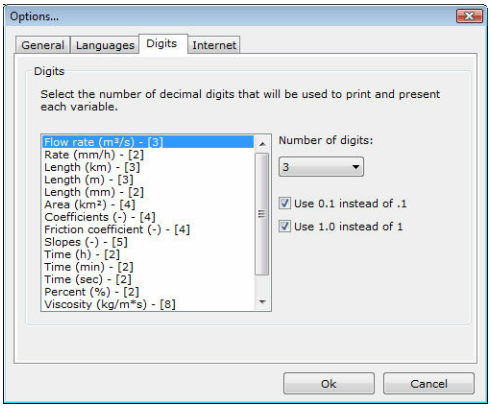
Languages Tab



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



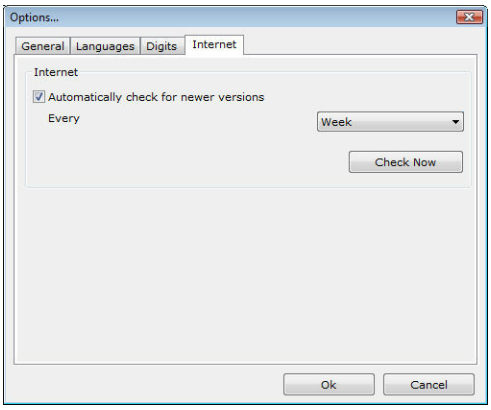
With this tab, you can modify the way the results are presented. All values used in the program are displayed in the list on the left.

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.

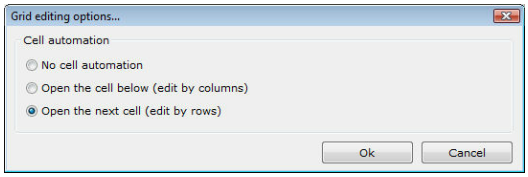
When a newer version is found, you will be prompted to download and install the latest version.

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.

4.11.2 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.

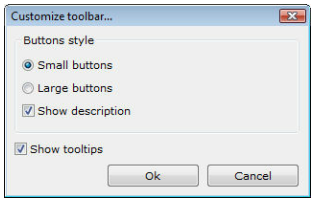
Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

4.11.3 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.

Chapter

5 Results

5.1 Results menu

With this menu, you can perform calculations and view the results. In the **Results** menu you can select one of the following options:

- Perform calculations
- Calculations report
- Basins
  - Comparative analysis
- Rivers
  - Results details
- Reservoirs
  - Results details

5.2 Perform calculations

With this option, you can manually perform calculations for all solutions.

To manually perform calculations for all solutions:

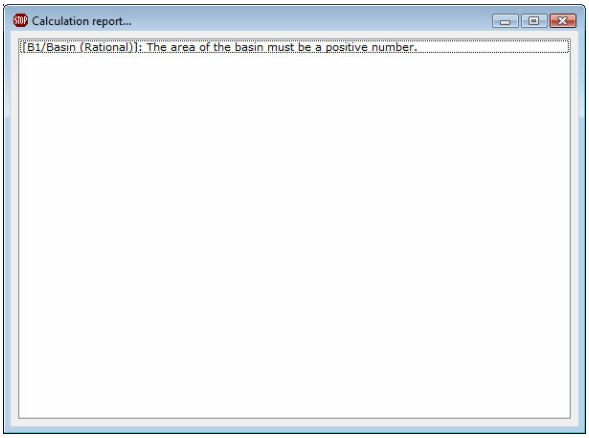
1. Select **Perform calculations** from the **Results** menu.
2. The calculations are performed. The results are displayed in the main form. If there are errors in one or more solutions, an error report is prepared. If you have selected **Automatically display calculation messages** in general preferences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

5.3 Calculations report

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 Perform calculations.

To display the report:

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



NOTE: This option is available only when one or more solutions are not completed successfully.

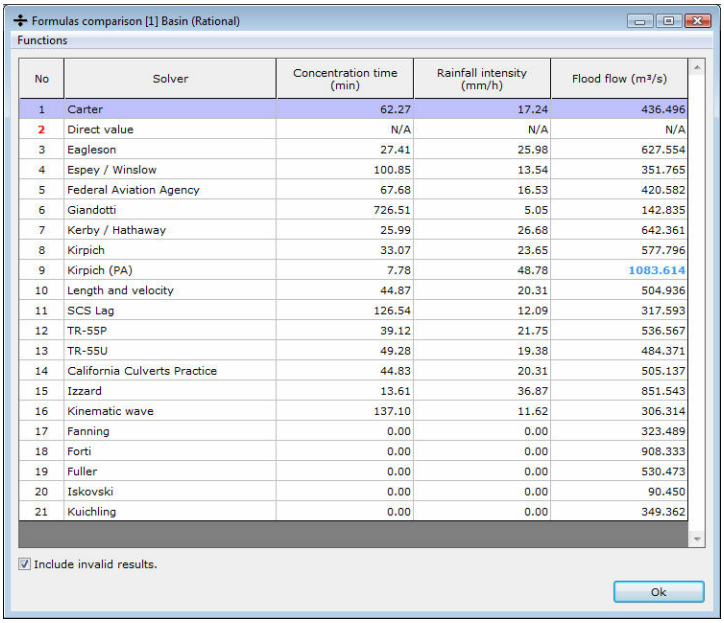
5.4 Basins

5.4.1 Comparative analysis

With this option, you can calculate the flood flow with all possible methods, whether they are applicable or not.

To perform the comparative analysis:

1. Select **Basins** from the **Results** menu.
2. Select **Comparative analysis** from the **Basins** menu.
3. Check **Include invalid results** if you want all results to be displayed. The number of the solver is printed in red if the method is not applicable.
4. Select **Ok** to close the dialog box.



No	Solver	Concentration time (min)	Rainfall intensity (mm/h)	Flood flow (m³/s)
1	Carter	62.27	17.24	436.496
2	Direct value	N/A	N/A	N/A
3	Eagleson	27.41	25.98	627.554
4	Espey / Winslow	100.85	13.54	351.765
5	Federal Aviation Agency	67.68	16.53	420.582
6	Giandotti	726.51	5.05	142.835
7	Kerby / Hathaway	25.99	26.68	642.361
8	Kirpich	33.07	23.65	577.796
9	Kirpich (PA)	7.78	48.78	1083.614
10	Length and velocity	44.87	20.31	504.936
11	SCS Lag	126.54	12.09	317.593
12	TR-55P	39.12	21.75	536.567
13	TR-55U	49.28	19.38	484.371
14	California Culverts Practice	44.83	20.31	505.137
15	Izzard	13.61	36.87	851.543
16	Kinematic wave	137.10	11.62	306.314
17	Fanning	0.00	0.00	323.489
18	Forti	0.00	0.00	908.333
19	Fuller	0.00	0.00	530.473
20	Iskovski	0.00	0.00	90.450
21	Kuichling	0.00	0.00	349.362

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 Excel.

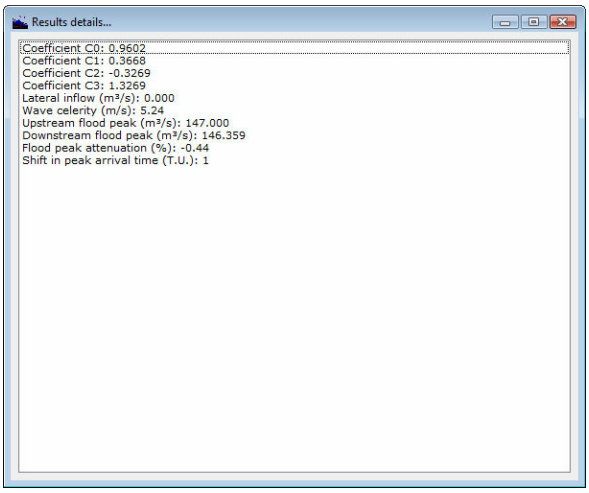
5.5 Rivers

5.5.1 Results details

With this option, you can view the result details for the selected river.

To view the results details:

1. Select **Rivers** from the **Results** menu.
2. Select **Results details** from the **Rivers** menu.
3. Hit ESC to close the dialog box.



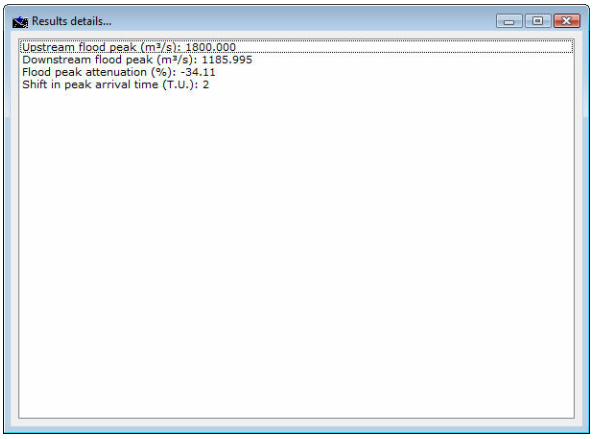
5.6 Reservoirs

5.6.1 Results details

With this option, you can view the result details for the selected reservoir.

To view the results details:

1. Select **Reservoirs** from the **Results** menu.
2. Select **Results details** from the **Reservoirs** menu.
3. Hit ESC to close the dialog box.



Chapter

6 Help

6.1 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

6.2 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.

6.3 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.

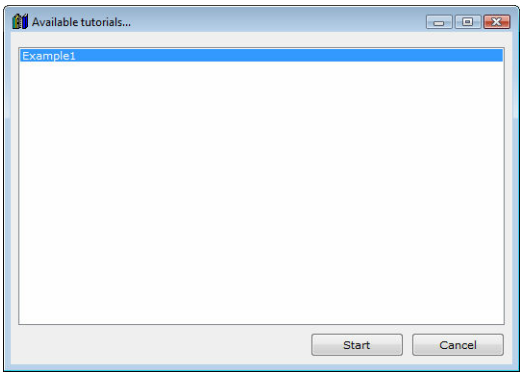
NOTE: Adobe Acrobat Reader or a similar program that can display pdf files is required in order to view or print the user guide.

6.4 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.



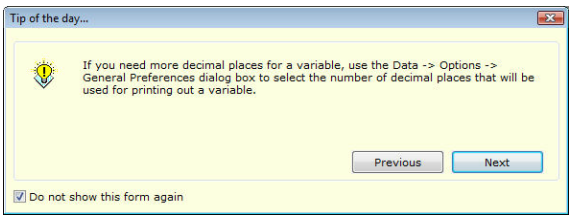
NOTE: The number and content of the tutorials is changed frequently. Use the live update system of TechnoLogismiki's products to download the latest tutorials.

6.5 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.



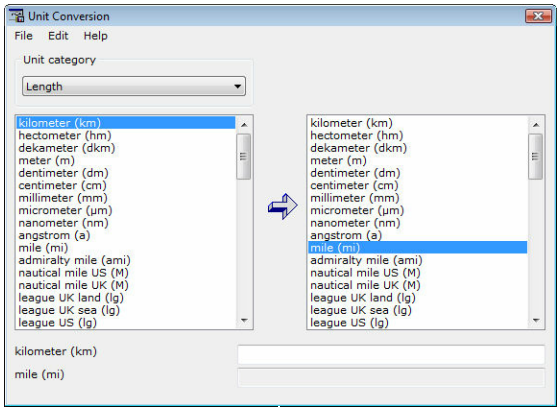
NOTE: The number and content of the tips is changed frequently. Use the live update system of TechnoLogismiki's products to download the latest tips.

6.6 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.

6.7 TechnoLogismiki website

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

6.8 Buy products

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

6.9 TechnoLogismiki NOMOS

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

6.10 TechnoLogismiki Live!

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

6.11 About the program

With this option, a form containing the name, version and licence information of the program appears.

To show this form:

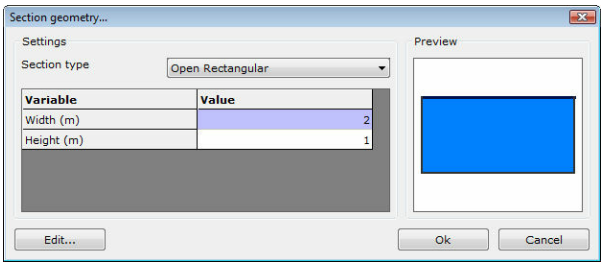
1. From the **Help** menu, select **About the program**.
2. The form appears.
3. Click anywhere on the form or hit ESC to close the form.

Chapter

7 Sections

7.1 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)

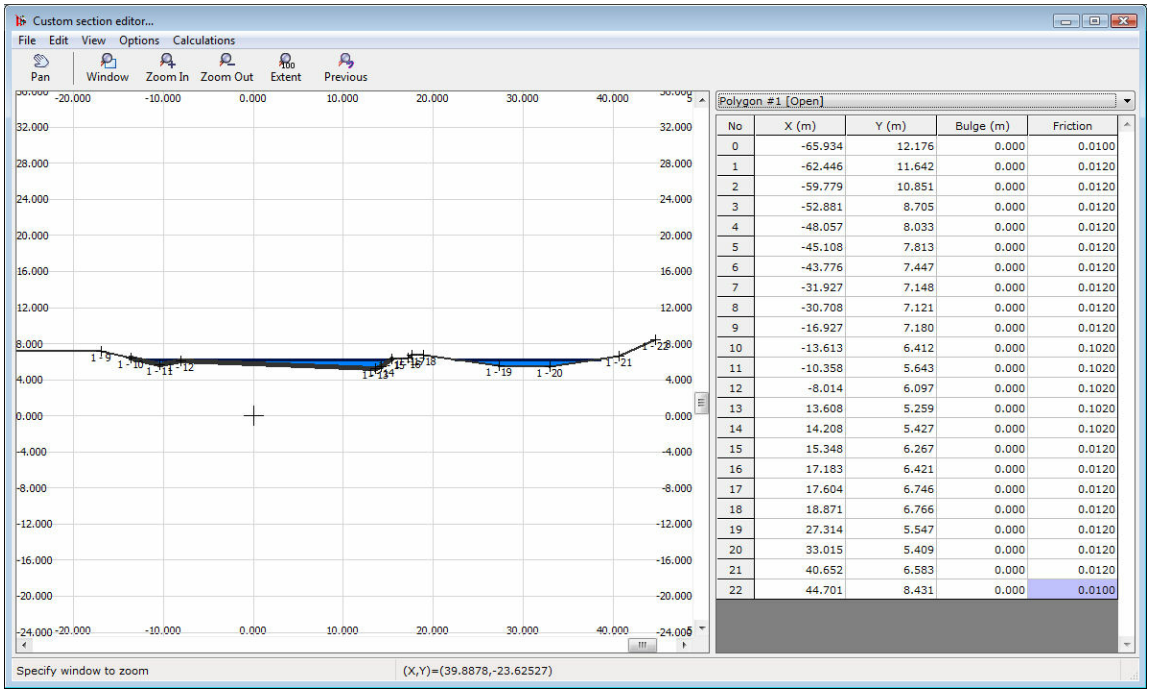
Oval	Width
Wide oval	Width
Basket handle	Width
Simple mouth-shaped	Width
Normal mouth-shaped	Width
Simple horse-shoe	Width
Normal horse-shoe	Width
Donut	External diameter, Internal diameter
Arch	Width, Height, Arc segments
Ellipse	Width, Height, Arc segments
Parabolic	Width, Height, Arc segments
Irregular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

7.2 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.

7.3 File

7.3.1 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

7.3.2 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.

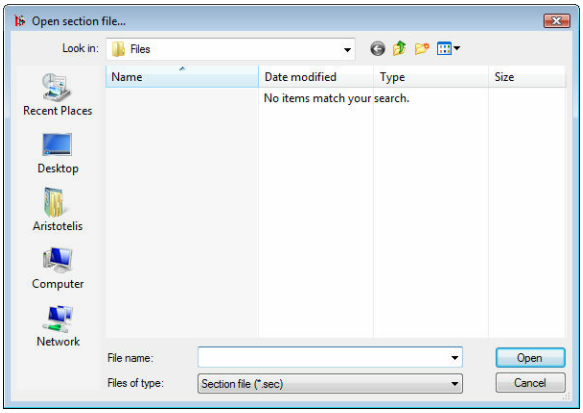
7.3.3 Open section

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.

7.3.4 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:

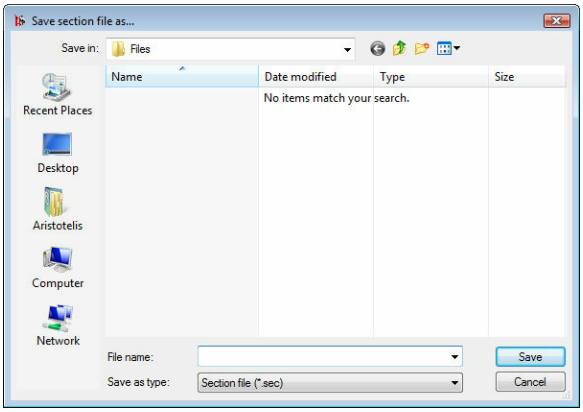
1. Select **Save Section** from the **File** menu.
2. If the filename and path are already set, the section is saved to this file without any messages. If the filename and path are not set, a dialog box will appear that allows the selection of the filename and path.

7.3.5 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.

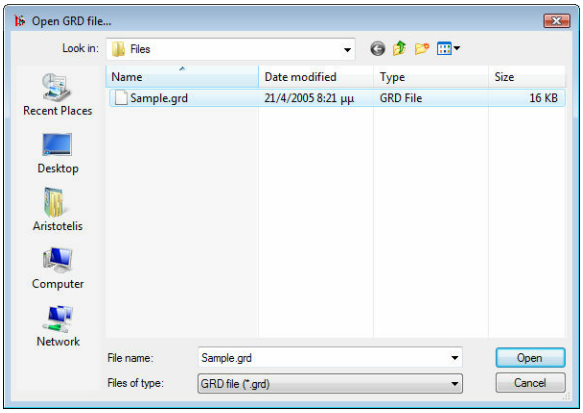
7.3.6 Import

7.3.6.1 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.

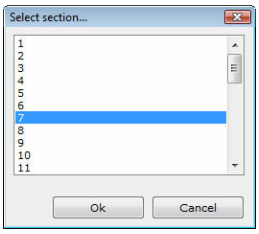


Case 1

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.



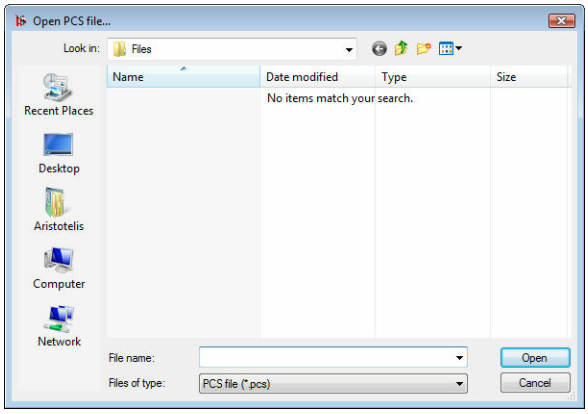
Select **Ok** to proceed. The current section data are erased and substituted by the selected section. Select **Cancel** to cancel the operation.

7.3.6.2 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.



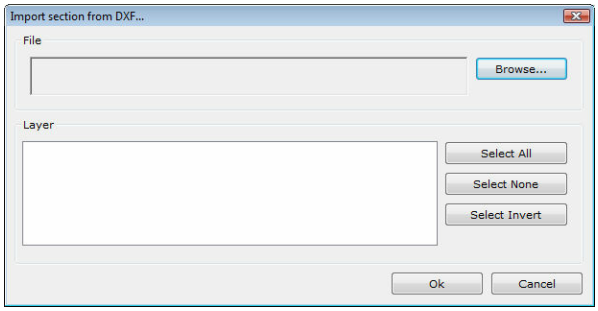
7.3.6.3 Import from DXF

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

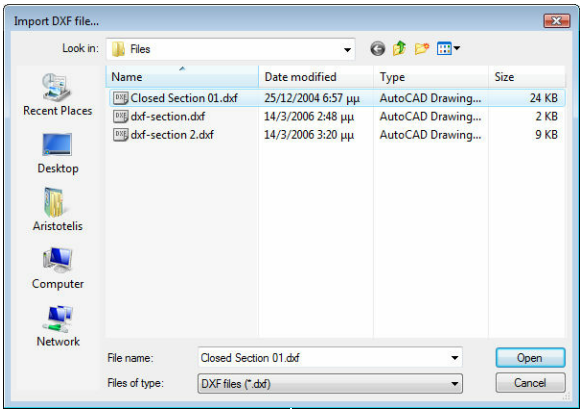
DXF files contain information in layers. You can select one or more layers that contain the section data.

To import a section from a DXF file:

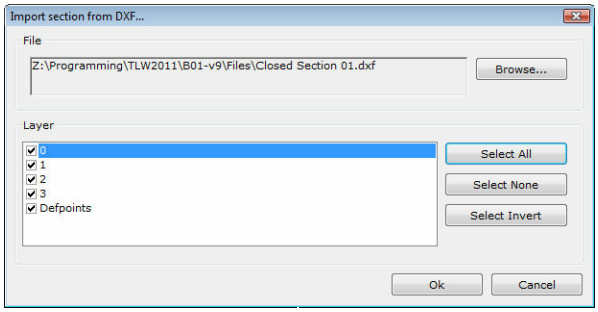
1. Select **Import** from the **File** menu.
2. Select **Import from DXF** from the **Import menu**. The following dialog box appears:



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.

7.3.6.4 Import from ArcView Shapefile

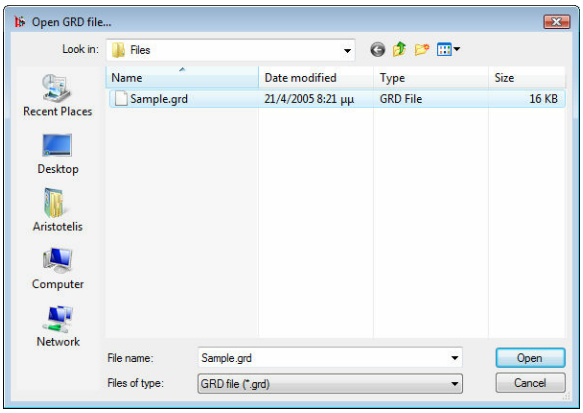
Shapefiles can be created by several programs such as ArcView GIS, MapInfo, GPS Trackmaker etc. In reality, this "file" consists of three files with the extensions shp, shx and dbf. The GIS driver recognizes the following shapefile types:

- 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.



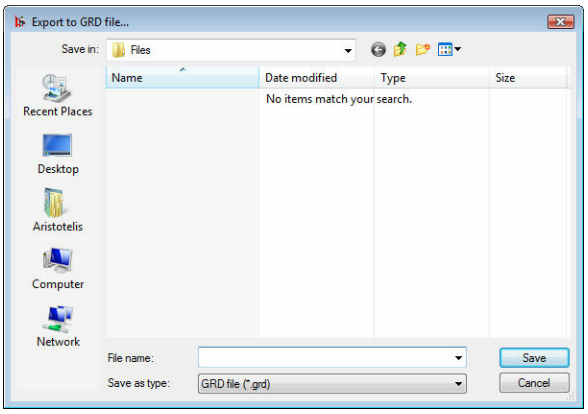
7.3.7 Export

7.3.7.1 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.



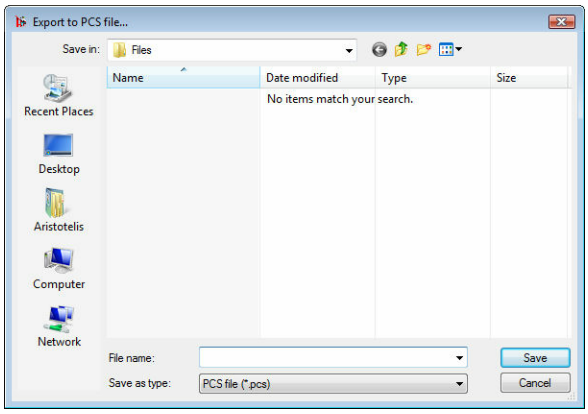
NOTE: Information on the friction coefficients and curved edges (if any) will not be exported as they are not supported by the file format.

7.3.7.2 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.



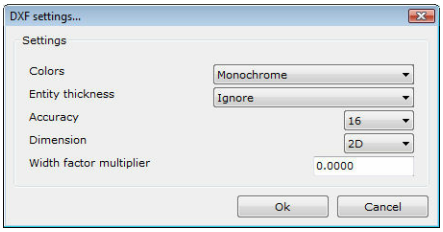
NOTE: Information on curved edges (if any) will not be exported as they are not supported by the file format.

7.3.7.3 Export to DXF

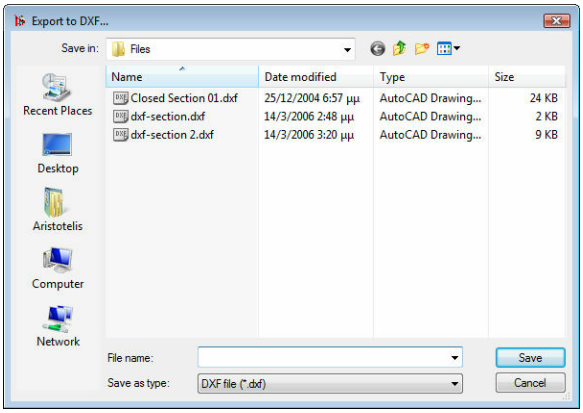
DXF files are recognized by virtually all CAD programs (AutoCAD, IntelliCAD, Microstation etc) as well as TechnoLogismiki's products.

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.

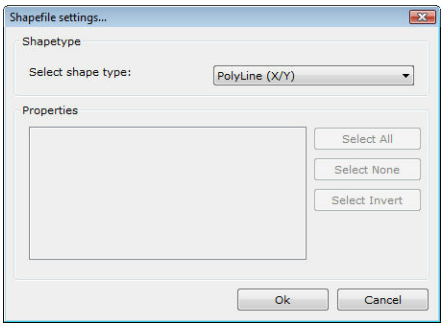


7.3.7.4 Export to ArcView Shapefile

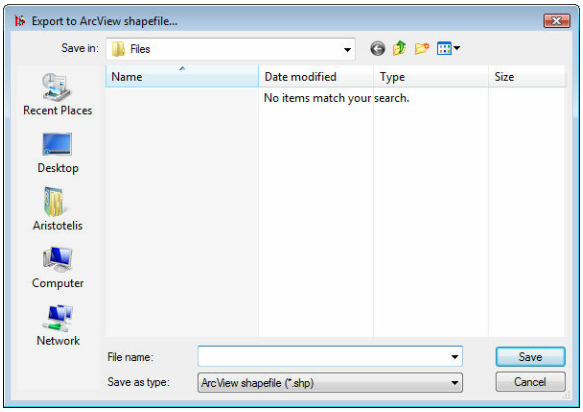
With this option, you can create a shapefile that can be used by programs such as ArcView GIS, MapInfo, GPS Trackmaker and other.

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.



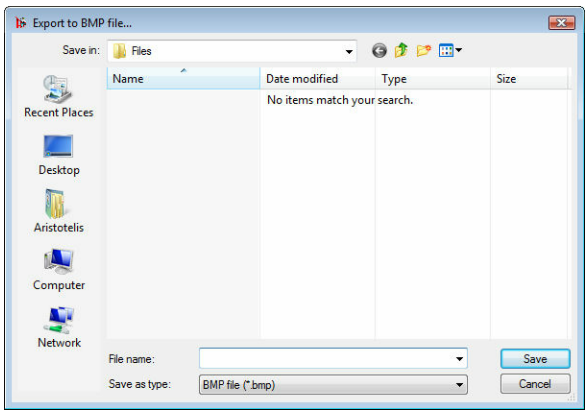
NOTE: Not all shape types are compatible with all programs.

7.3.7.5 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.



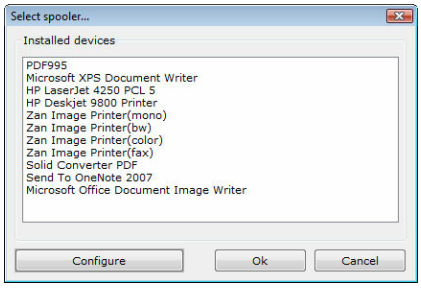
7.3.8 Print sketch

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.

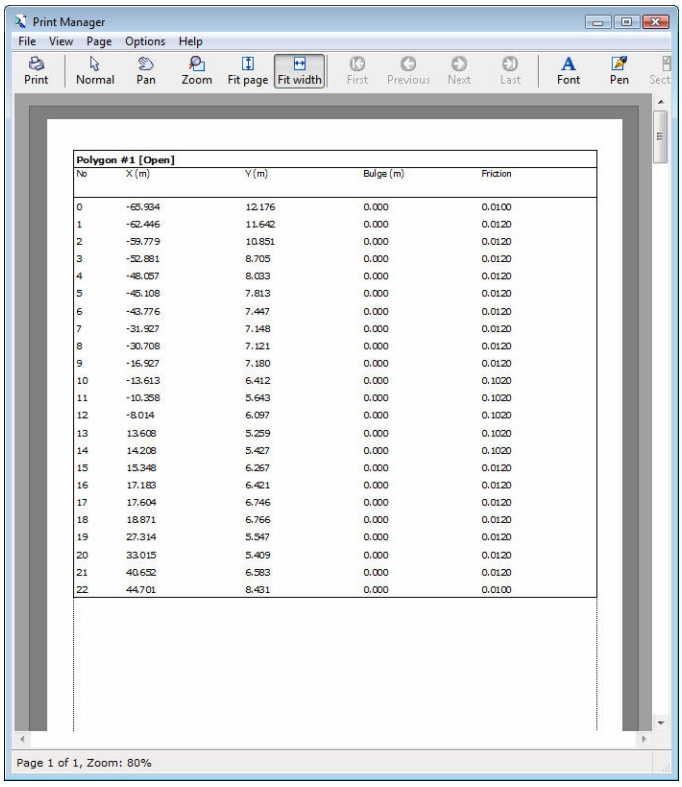


7.3.9 Print section data

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.

7.3.10 Print section data to

7.3.10.1 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.

7.3.10.2 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:

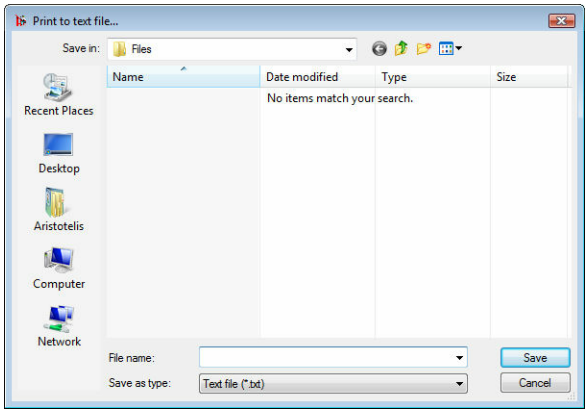
1. Select **Print section data to** from the **File** menu.
2. Select **Microsoft Word** from the **Print section data to** menu.

7.3.10.3 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.



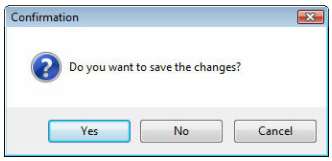
NOTE: If a file with the same name and in the same location already exists, a warning message will appear that asks whether to overwrite the file or not. If you answer Yes, then the existing file is erased and the new file takes its place. If you answer No, the existing file remains intact but the report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

7.3.11 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.**



7.4 Edit

7.4.1 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

7.4.2 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.

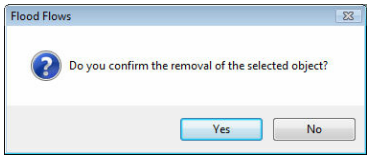
NOTE: This option is not available when entering ground and excavation data, as they must consist of a single polygon.

7.4.3 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.

7.4.4 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.

7.4.5 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 selected vertex 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 inserting a vertex.

7.4.6 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.

7.4.7 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.

7.4.8 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.

7.4.9 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.

7.4.10 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.

7.5 View

7.5.1 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

7.5.2 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.

7.5.3 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.

7.5.4 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.

7.5.5 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.

7.5.6 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.

7.5.7 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.

7.5.8 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.

7.5.9 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.

7.5.10 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.

7.5.11 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.

7.5.12 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.

7.6 Options

7.6.1 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:

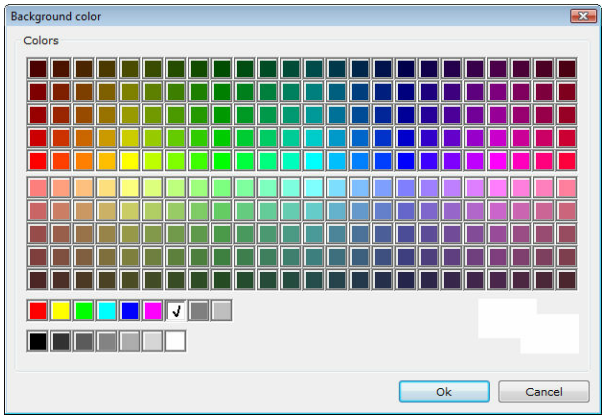
- Background color
- Interior color
- Grid
- Edge pen color
- Edge pen width
- Inactive vertices
- Water area
- Excavations

7.6.2 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.



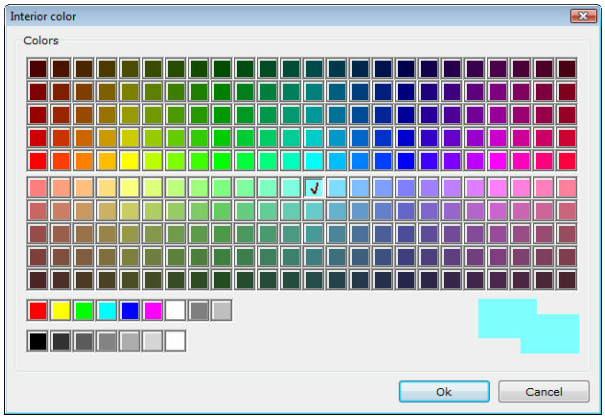
NOTE: The color palette follows standard CAD color palettes.

7.6.3 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.



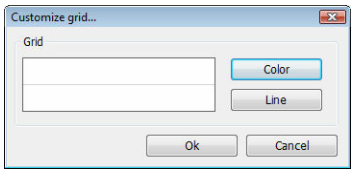
NOTE: The color palette follows standard CAD color palettes.

7.6.4 Grid

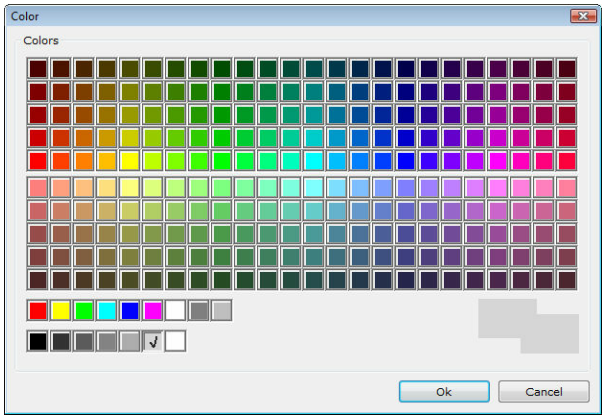
With this option, you can change the color and line style of the grid. Note that in order to view the changes the grid must be visible.

To change the color and line style of the grid:

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

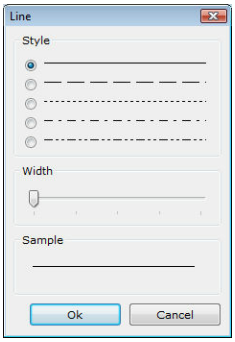


3. Select **Color** to change the color of the line.
3.1. The color selection dialog box appears.
3.2. Select the **color** from the 256 available colors. The currently selected color is marked with a tick. On top of the **Cancel** button, the old and the new color are displayed.
3.3. Click **Ok** to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



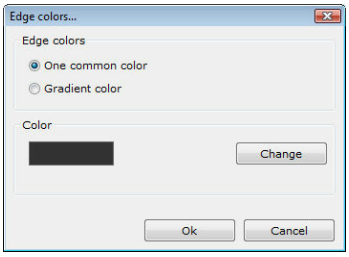
5. Click **Ok** to save the grid options and close the dialog box or click the **Cancel** button to close the dialog box without saving the changes.

7.6.5 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:

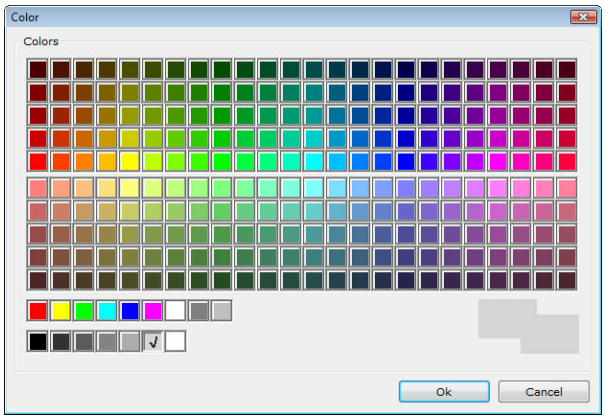
1. Select **Edge pen color** from the **Options** menu.
2. The edge pen color dialog box appears:



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.

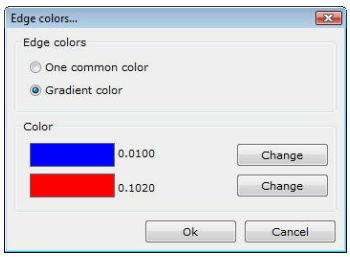


NOTE: The color palette follows standard CAD color palettes.

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

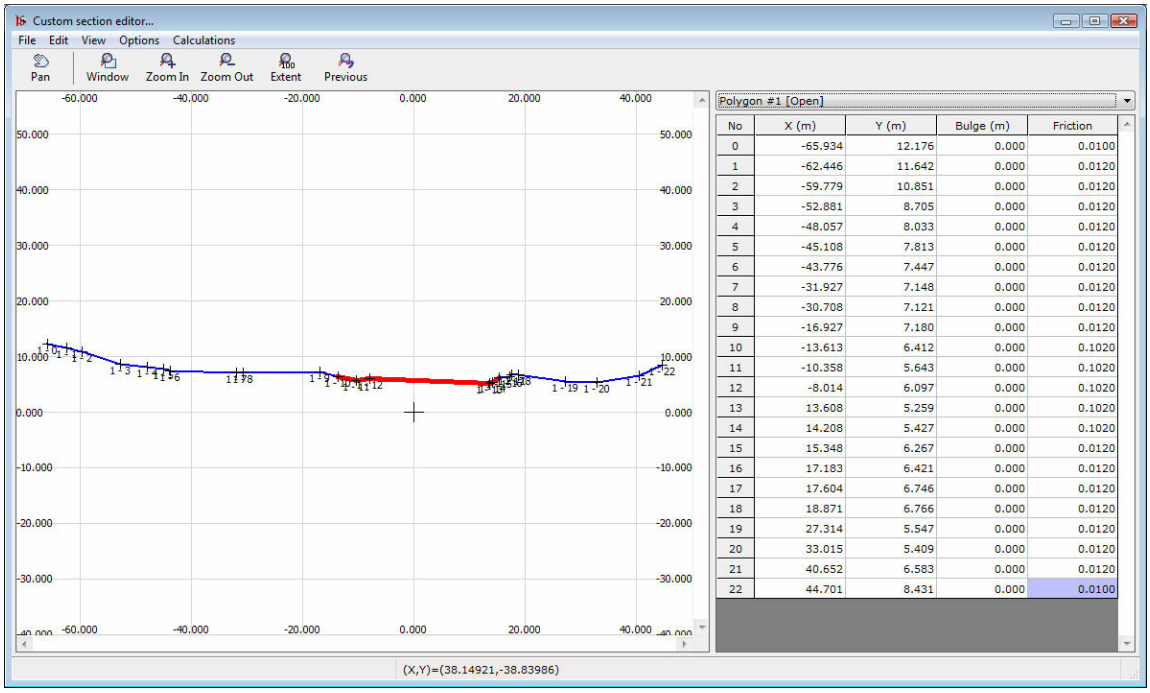
4. Two colors must be defined. The first corresponds to the minimum friction and the second to the maximum friction value. These values are displayed next to the corresponding picture boxes. Select the appropriate **Color** button to change the corresponding color.
4.1. The color selection dialog box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



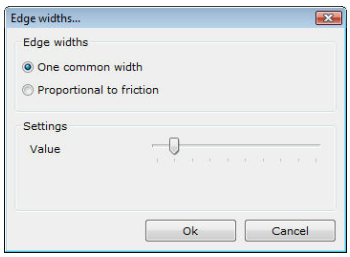
NOTE: You can combine gradient color with varying pen width.

7.6.6 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

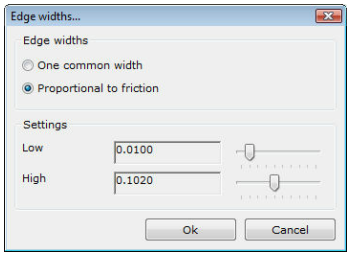
on the friction of the edge.

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.

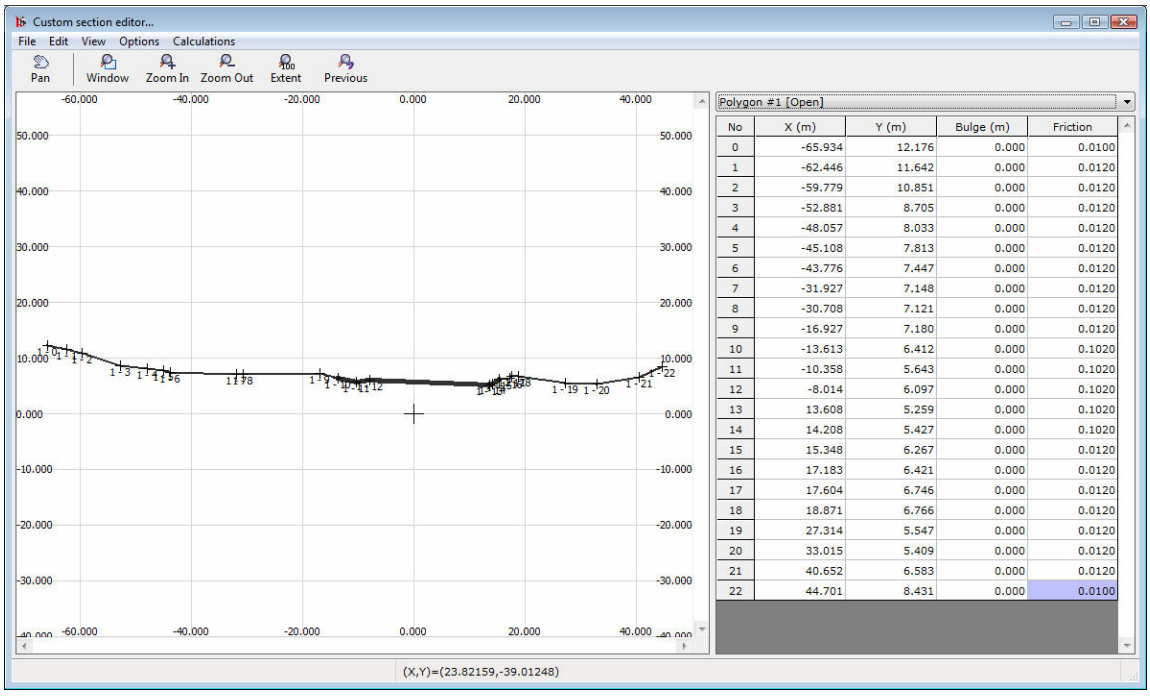
B. Proportional to friction

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



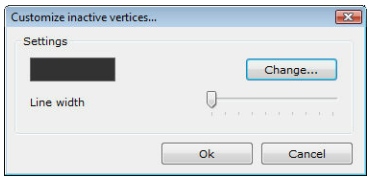
NOTE: You can combine proportional width with varying pen color.

7.6.7 Inactive vertices

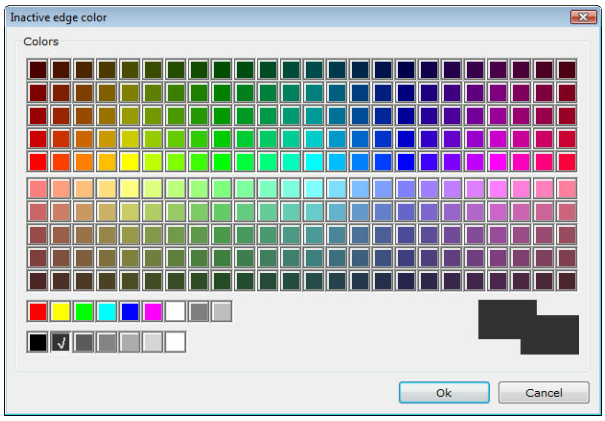
With this option, you can change the color and pen width of the edges between inactive vertices.

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.

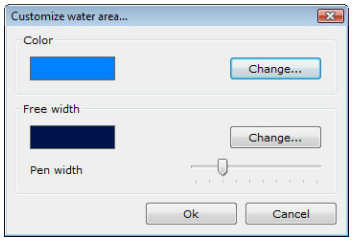
7.6.8 Water area

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.

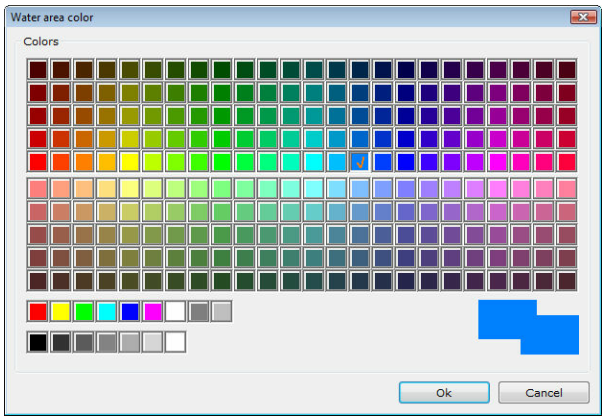
To change the interior color of water areas or the pen color and pen width of the free width:

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

2. The following dialog box appears:

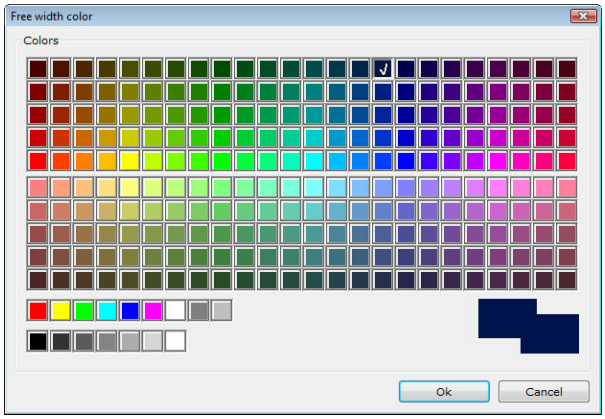


3. Select **Change** from the **Color** frame to change the interior color for water areas.
3.1. The color selection dialog box appears.
3.2. Select the **color** from the 256 available colors. The currently selected color is marked with a tick. On top of the **Cancel** button, the old and the new color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

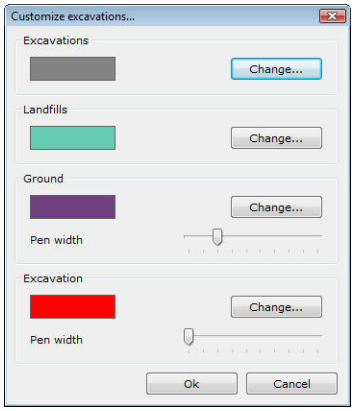
NOTE: The free width is not visible in pressure sections.

7.6.9 Excavations

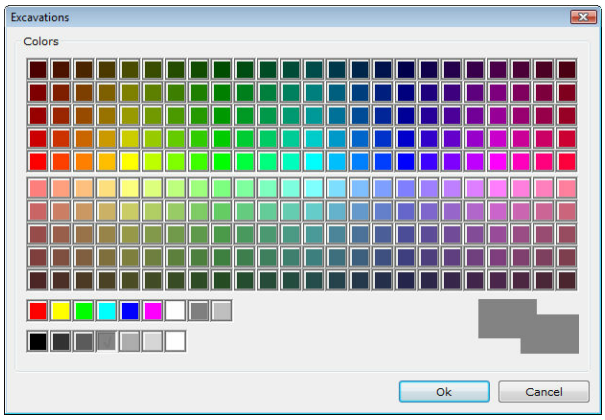
With this option, you can change the color and line styles of objects related to excavations.

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:

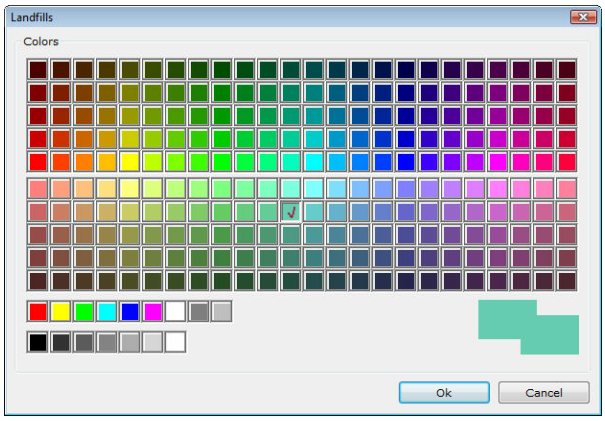


3. Select **Change** from the **Excavations** frame to change the interior color for excavations.
3.1. The color selection dialog box appears.
3.2. Select the **color** from the 256 available colors. The currently selected color is marked with a tick. On top of the **Cancel** button, the old and the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



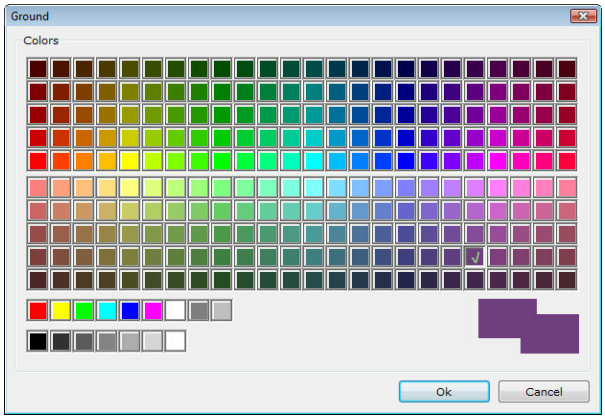
NOTE: The color palette follows standard CAD color palettes.

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.

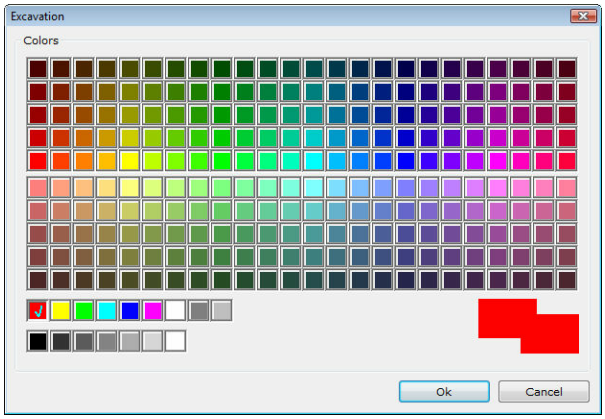
5. Select **Change** from the **Ground** frame to change the pen color for the ground line.
5.1. The color selection dialog box appears.
5.2. Select the **color** from the 256 available colors. The currently selected color is marked with a tick. On top of the **Cancel** button, the old and the new color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

7.7 Calculations

7.7.1 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

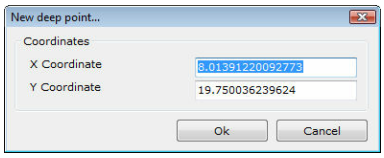
7.7.2 Origin

7.7.2.1 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.

7.7.2.2 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.

7.7.2.3 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.

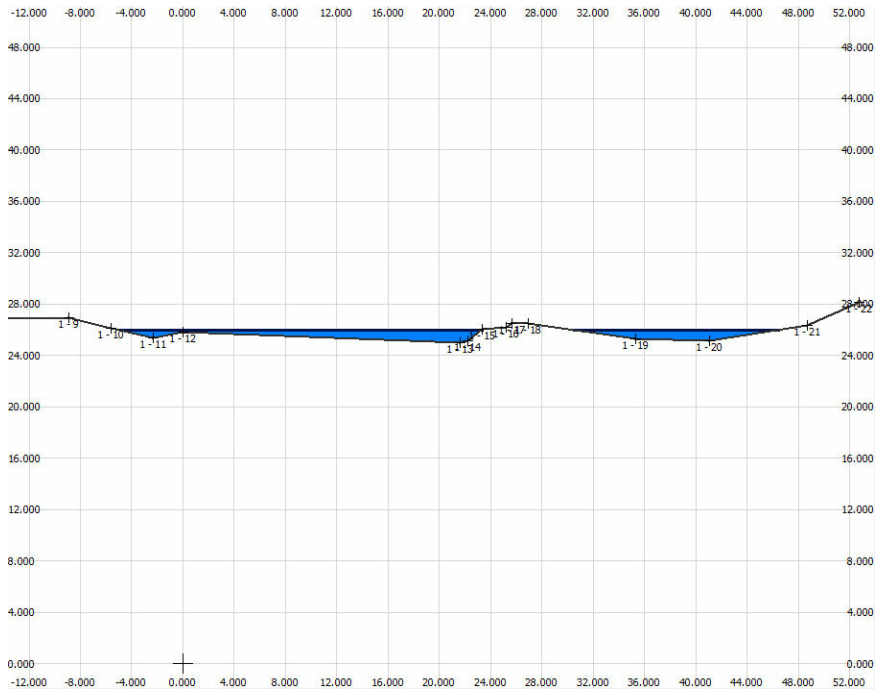
To set the coordinates of the origin equal to those of the deep point:

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.

7.7.3 Active nodes

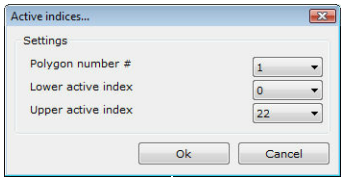
With this option, you can set the active nodes (vertices) of a section. By default, all nodes are active; therefore, the flow may occur anywhere in the section.



However, if we know that the flow occurs between vertex 4 and 10 of the above example, we must deactivate vertices 0 to 4.

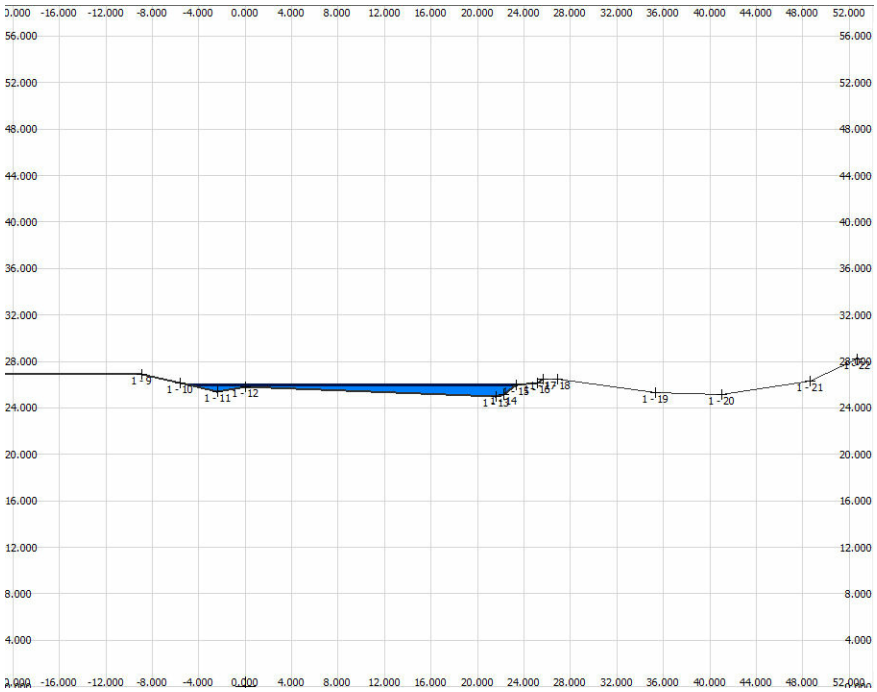
To select the active nodes (vertices):

1. Select **Active nodes** from the **Calculations** menu.
2. Select the polygon in question, the lower active index (in this case, 4 instead of 0) and the upper active index (in this case, the same node i.e. node 10 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 4 as the first active vertex in the above example, the flow is confined to the right riverbed:



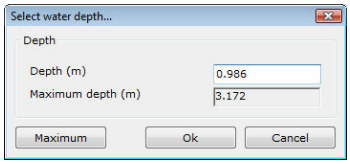
NOTE: The inactive edges are drawn with different pen color and size, according to the preferences of the program.

7.7.4 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.



7.7.5 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.

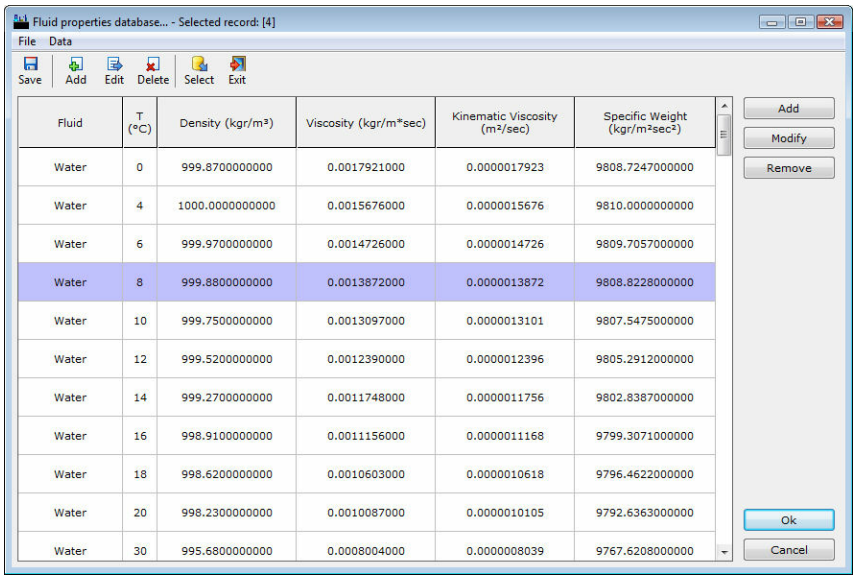
Chapter VIII

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.

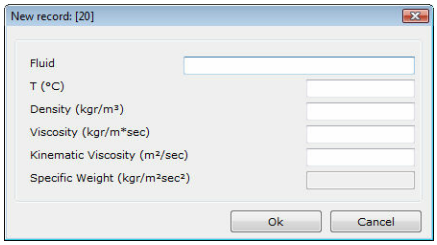
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



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:

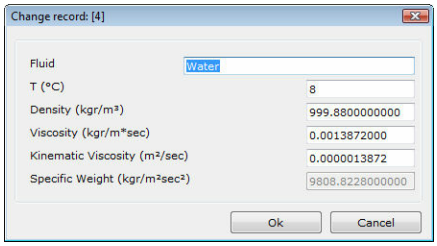
1. Click **Add** to open the new record dialog box.
2. Type the name of the fluid. This field is required.
3. Enter the temperature, density, viscosity and kinematic viscosity of the fluid.
4. The specific weight is calculated automatically.
5. Click **Ok** to close the dialog box and add a new record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



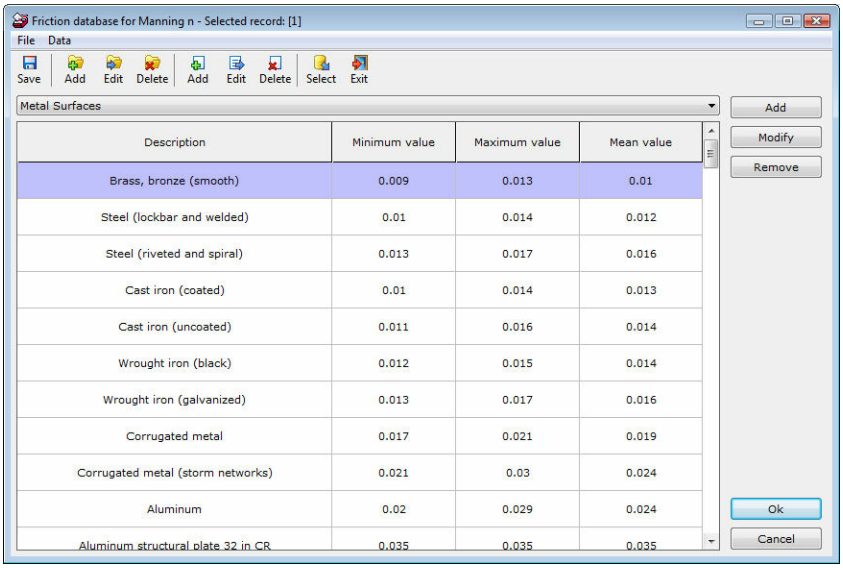
To remove an existing record:

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.

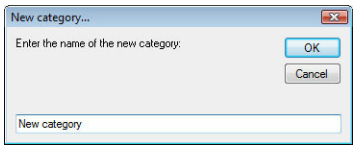
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



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

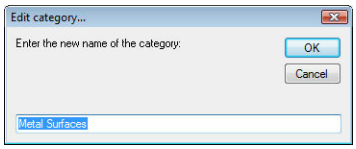
To add a new category:

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.



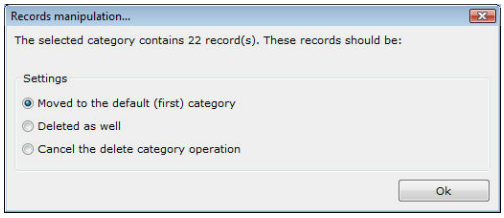
To modify the name of an existing category:

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:

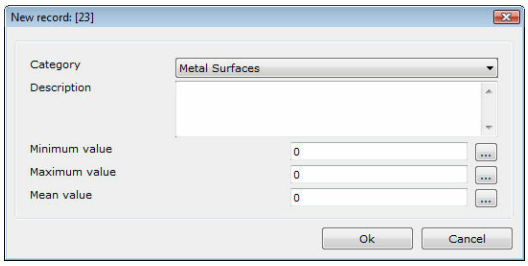


4.1. Select the first option to move the records of the category to the default (first category).
4.2. Select the second option to delete the records.
4.3. Select the third option to cancel the deletion.
5. Click **Ok** to proceed.

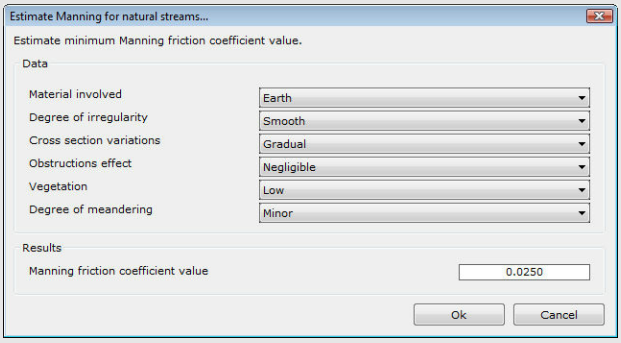
NOTE: The database must contain at least one category.

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.



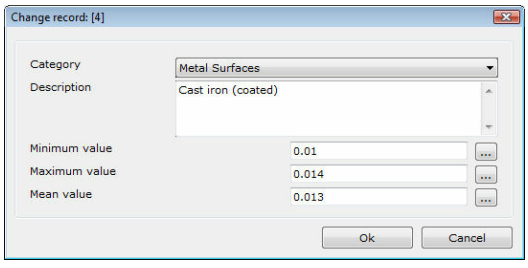
NOTE: In case of Manning friction coefficients in natural streams, you can estimate the values based on several characteristics of the stream. Click on the buttons with the ellipses (...) next to the text boxes to invoke the following dialog box:



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

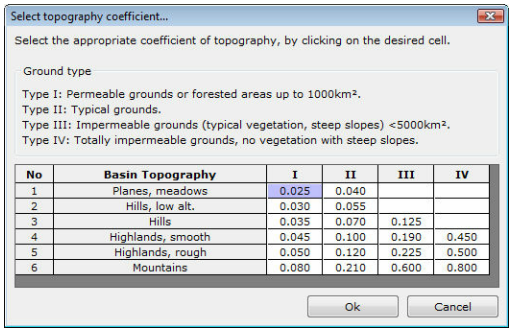
Surface / Material	Mean value (mm)
Aluminum	0.300
Asbestos cement	0.002
Asphalted cast iron	0.120
Best concrete	0.366
Brick in mortar	0.610
Sewer brick	0.610
CMP	0.305
Concrete	0.122
Centrifugal spun	0.366
Concrete (steel forms)	1.829
Concrete (wood forms)	0.610
Copper	0.002
Galvanized steel	1.520
Glass	0.001
PVC	0.122
HDPE	0.150

8.7 Topography coefficient

For your convenience, a topography coefficient database is embedded in the program.

To select the topography coefficient:

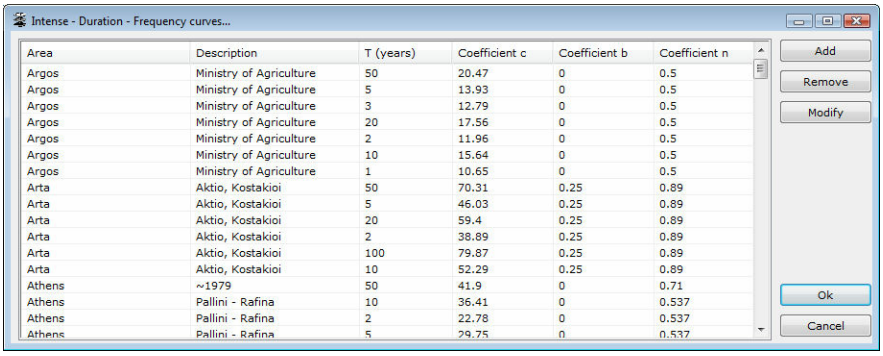
1. Click the cell containing the desired value.
2. Click **Ok** to transfer the data to the corresponding fields and close the database. Select **Cancel** to close the database without transferring any data.



8.8 IDF database

For your convenience, a fully customizable IDF (Intensity - Duration - Frequency) curves database is embedded in the program. The IDF database is invoked in various cases within the program. By selecting an appropriate record and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

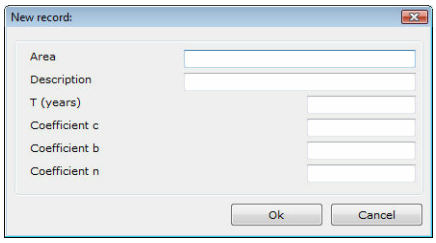
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database. Note that the database was built using data from various resources in the literature; you need to be cautious when using a curve.



To add a new record:

1. Click **Add** to open the new record dialog box.
2. Enter the **area** and optionally the description of the record.
3. Enter the return period in years. This value was used for the calculation of the IDF curve and is for reference purposes only; it is not used in the calculations.
4. Enter the dimensionless coefficients c, b, n, in such a way that when time is entered in hours, the intensity is given in mm/hr.

5. Click **Ok** to close the dialog box and add a new record at the end of the list. Click **Cancel** to close the dialog box without making any changes.



To modify an existing record:

1. Click **Modify** to open the modify record dialog box.
2. Make the appropriate changes.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.

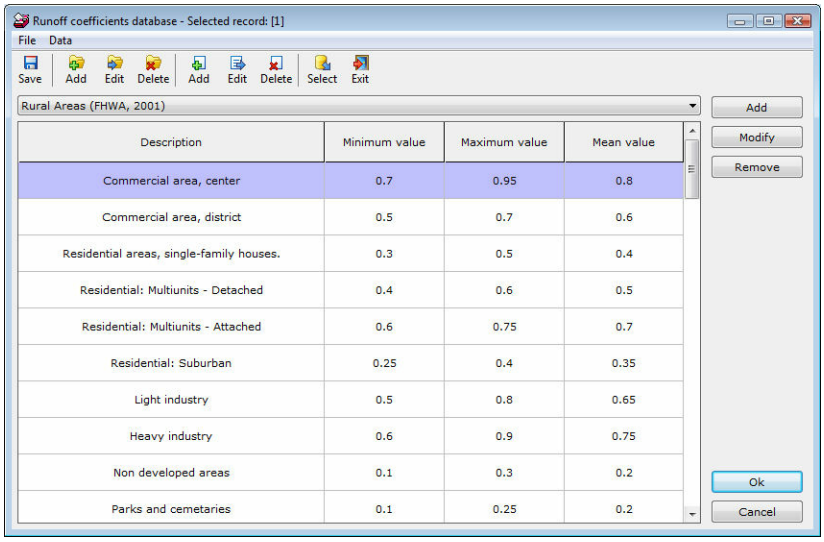
To remove an existing record:

1. Select the record you wish to remove.
2. Click **Remove** to remove the record. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.

8.9 Runoff coefficient database

For your convenience, a fully customizable runoff coefficient database is embedded in the program. The database is invoked in various cases within the program. By selecting an appropriate record and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

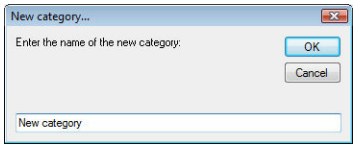
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



The database consists of several categories. Usually, the category defines the regulations (e.g. FHWA, 2001).

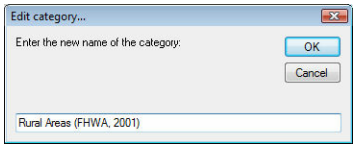
To add a new category:

1. Select **Add category** from the **Data** menu.
2. Type the name of the category in the text box. The name of the category must be unique.
3. Select **Ok** to add the category at the end of the list. Select **Cancel** to cancel the procedure.



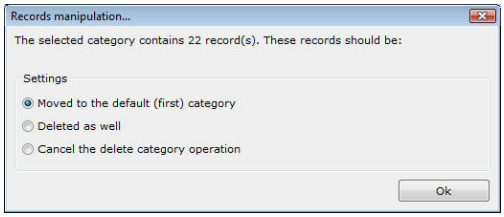
To modify the name of an existing category:

1. Click **Modify** to open the modify category dialog box.
2. Type the name of the category in the text box. The name of the category must be unique.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



To remove an existing category:

1. Select the category you wish to remove from the drop-down list.
2. Click **Remove** to remove the category. You will be asked to confirm the deletion.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.
4. If the category contains records, then the following dialog box appears:

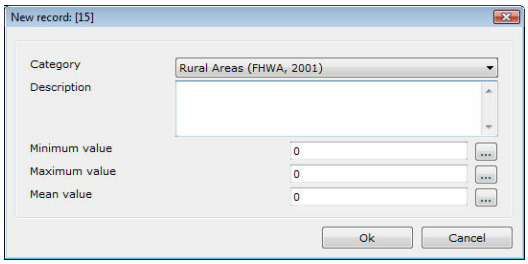


4.1. Select the first option to move the records of the category to the default (first category).
4.2. Select the second option to delete the records.
4.3. Select the third option to cancel the deletion.
5. Click **Ok** to proceed.

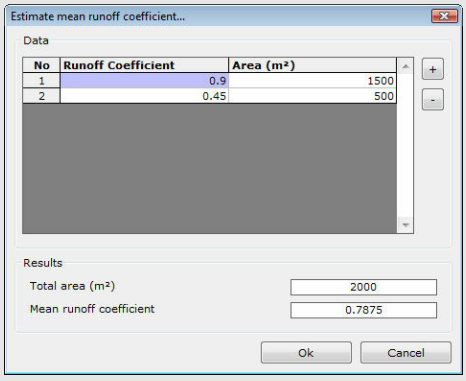
NOTE: The database must contain at least one category.

To add a new record:

1. Click **Add** to open the new record dialog box.
2. Select the category of the new record from the drop-down list.
3. Type the description of the record. This field is required.
4. Enter the minimum, maximum and mean value of the runoff coefficient.
5. Click **Ok** to close the dialog box and add a new record at the end of the list. Click **Cancel** to close the dialog box without making any changes.



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



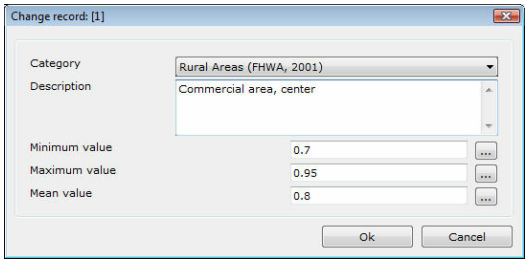
Click the plus sign **(+)** to add a new area. Type the runoff coefficient and the area in m^2. The total area and the mean runoff coefficient is displayed in the **Results** frame. Click the minus sign **(-)** to delete the selected area. The area is deleted with no

confirmation.

Make the appropriate selections. Click **Ok** to close the dialog box and transfer the data to the corresponding text box. Click **Cancel** to close the dialog box without transferring any data.

To modify an existing record:

1. Click **Modify** to open the modify record dialog box.
2. Make the appropriate changes.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.



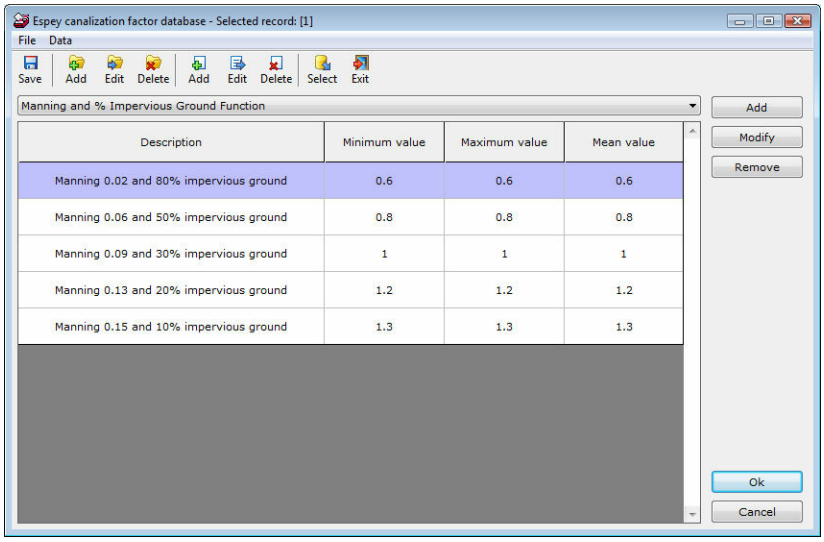
To remove an existing record:

1. Select the record you wish to remove.
2. Click **Remove** to remove the record. You will be asked to confirm the deletion.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.

8.10 Izzard deceleration coefficient

For your convenience, a fully customizable Izzard deceleration coefficient database is embedded in the program. The database is invoked in various cases within the program. By selecting an appropriate record and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

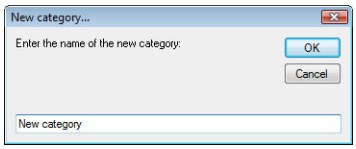
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



The database consists of several categories. Usually, the category defines the calculation formula (e.g. Manning and % impervious ground function).

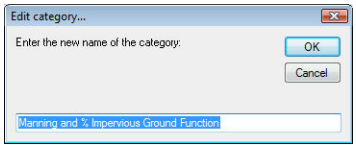
To add a new category:

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.



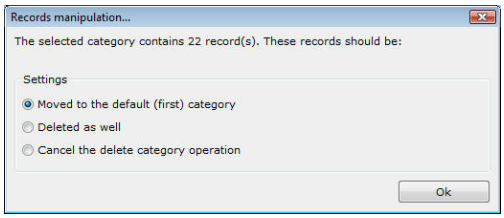
To modify the name of an existing category:

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:

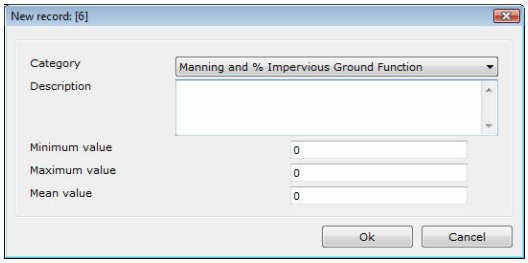


4.1. Select the first option to move the records of the category to the default (first category).
4.2. Select the second option to delete the records.
4.3. Select the third option to cancel the deletion.
5. Click **Ok** to proceed.

NOTE: The database must contain at least one category.

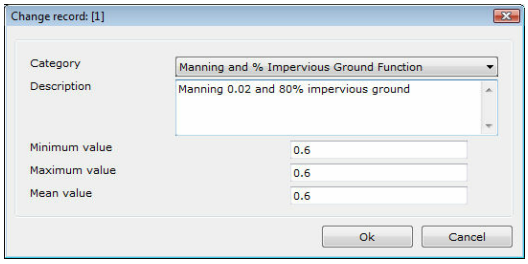
To add a new record:

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 Izzard deceleration coefficient.
5. Click **Ok** to close the dialog box and add a new record at the end of the list. Click **Cancel** to close the dialog box without making any changes.



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.



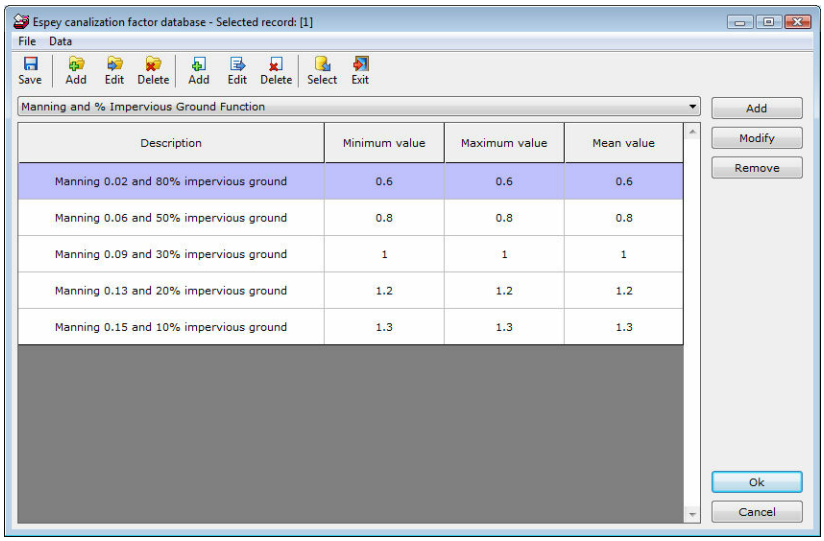
To remove an existing record:

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.

8.11 Espey channel coefficient database

For your convenience, a fully customizable Espey channel coefficient database is embedded in the program. The database is invoked in various cases within the program. By selecting an appropriate record and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

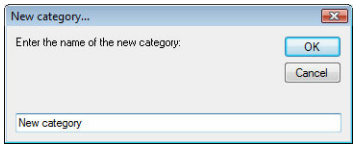
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



The database consists of several categories. Usually, the category defines the calculation formula (e.g. Manning and % impervious ground function).

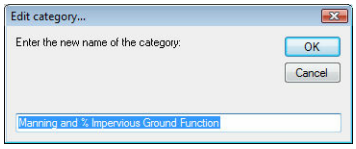
To add a new category:

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.



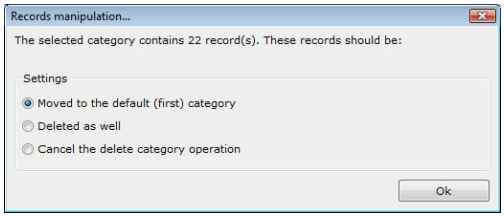
To modify the name of an existing category:

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:

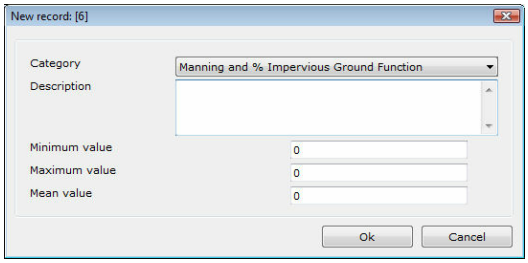


4.1. Select the first option to move the records of the category to the default (first category).
4.2. Select the second option to delete the records.
4.3. Select the third option to cancel the deletion.
5. Click **Ok** to proceed.

NOTE: The database must contain at least one category.

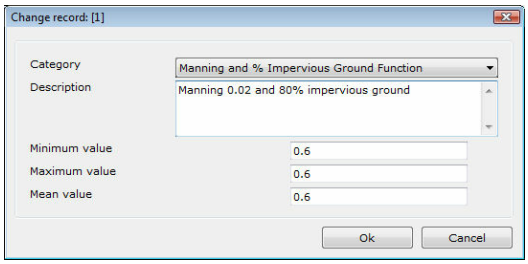
To add a new record:

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 Espey channel coefficient.
5. Click **Ok** to close the dialog box and add a new record at the end of the list. Click **Cancel** to close the dialog box without making any changes.



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.

8.12 SCS curve number database

For your convenience, a fully customizable SCS curve number database is embedded in the program. The database is invoked in various cases within the program. By selecting an appropriate record and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

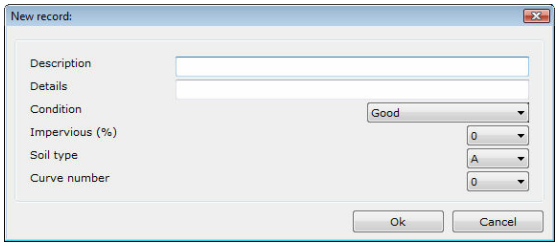
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.

To add a new record:

1. Click **Add** to open the new record dialog box.
2. Type the description and optionally the details.
3. Enter the condition, imperviousness, soil type (A, B, C or D) and curve number (0-100).

- **Soil type A**: Soils having high infiltration rates, even when thoroughly wetted and consisting chiefly of deep, well to excessively-drained sands or gravels. These soils have a high rate of water transmission.
- **Soil type B**: Soils having moderate infiltration rates when thoroughly wetted and consisting chiefly of moderately deep to deep, moderately fine to moderately coarse textures. These soils have a moderate rate of water transmission.
- **Soil type C**: Soils having slow infiltration rates when thoroughly wetted and consisting chiefly of soils with a layer that impedes downward movement of water, or soils with moderately fine to fine texture. These soils have a slow rate of water transmission.
- **Soil type D**: Soils having very slow infiltration rates when thoroughly wetted and consisting chiefly of clay soils with a high swelling potential, soils with a permanent high water table, soils with a clay pan or clay layer at or near the surface, and shallow soils over nearly impervious material. These soils have a very slow rate of water transmission.

4. Click **Ok** to close the dialog box and add a new record at the end of the list. Click **Cancel** to close the dialog box without making any changes.



To modify an existing record:

1. Click **Modify** to open the modify record dialog box.
2. Make the appropriate changes.
3. Click **Ok** to save the changes and close the dialog box. Click **Cancel** to close the dialog box without saving the changes.

To remove an existing record:

1. Select the record you wish to remove.
2. Click **Remove** to remove the record. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. Select Yes to proceed with the deletion. Select No to cancel the deletion.

Keyword index

- A -

- B -

- C -

- D -

- E -

- F -

- G -

- H -

- I -

- K -

- L -

- M -

- N -

- O -

- P -

- R -

- S -

- T -

- U -

- V -

- W -

- X -

- Y -

- Z -

This page was intentionally left blank.